Comparing Optimized Perimeter Steel Bracing of Tall Buildings under Different Seismic Regions

by

Chelsea Karina Medina

B.S. Civil Engineering, University of Southern California, 2022

Submitted to the Department of Civil and Environmental Engineering in partial fulfillment of the requirements for the degree(s) of

Master of Engineering in Civil and Environmental Engineering

at the

Massachusetts Institute of Technology

June 2023

©2023 Chelsea Karina Medina. All rights reserved.

The author hereby grants to MIT a nonexclusive, worldwide, irrevocable, royalty-free license to exercise any and all rights under copyright, including to reproduce, preserve, distribute and publicly display copies of the thesis, or release the thesis under an open-access license...

Authored by: Chelsea Karina Medina
Department of Civil and Environmental Engineering
May 12, 2023

Certified by: John A. Ochsendorf
Class of 1942 Professor of Civil and Environmental Engineering
Thesis Supervisor

Vittoria Laghi
Lecturer of Civil and Environmental Engineering
Co-Thesis supervisor

Accepted by: Colette L. Heald
Professor of Civil and Environmental Engineering
Chair, Graduate Program Committee
Comparing Optimized Perimeter Steel Bracing of Tall Buildings under Different Seismic Regions

by

Chelsea Karina Medina

Submitted to the Civil and Environmental Department on May 12, 2023, in Partial Fulfillment of the Requirements for the Degree of Master of Engineering in Civil Engineering

ABSTRACT

There are challenges associated with building high-rise structures sustainably and safely, especially in seismic regions. These types of structures face extreme loading conditions. One promising solution for these challenges is topology optimization, which involves determining the optimal material distribution to achieve desired performance criteria under certain constraints. However, implementing topology optimization for real-life structures under seismic design codes is challenging due to multiple nonlinear constraints, discrete variables, and high computational cost. Recently, there have been several attempts to use topology optimization for seismic design. Considered groundbreaking in this regard is research proposed by Amory Martin in 2020. This author’s work proposed a method called the sum of modal compliances to optimize a steel lateral frame system in tall buildings for seismic design. The focus of this work is to expand upon this method, generating lateral frame systems for tall buildings from response spectra in different seismic regions rather than from an idealized design spectrum. The structural performance of the various optimized framing layouts produced were further verified through a nonlinear analysis, which indicated that they had the potential to outperform traditional bracing systems under seismic excitation. This was a trend observed in multiple seismic regions in North America. This research has important implications as the use of topology optimization in designing lateral brace frames for tall buildings under seismic excitation could help develop safer and more sustainable structures, reducing embodied carbon while maximizing construction revenue.

Thesis supervisor: John Ochsendorf
Class of 1942 Professor of Civil and Environmental Engineering
Thesis Supervisor

Co-Thesis supervisor: Vittoria Laghi
Lecturer of Civil and Environmental Engineering
Co-Thesis supervisor

3
Acknowledgements

I would first like to thank my external supervisor, lecturer Vittoria Laghi. The advice and guidance that lecturer Laghi provided has been invaluable to this work and has shaped my experience at MIT. It has truly been an honor to work with her throughout the year.

I would also like to thank the Department of Civil and Environmental Engineering at MIT, and Professor John Ochsendorf, for providing me the opportunity to study at Massachusetts Institute of Technology. Additionally, I would like to thank Josephine Carstensen for exposing me to the topic of topology optimization through her course, which inspired this research.

I want to acknowledge Professor Tigran Ayrapetyan, Professor Burcin Becerik and the Civil Engineering Department at The University of Southern California. They provided me with the academic foundation, as well as the encouragement and support, to pursue graduate studies at MIT.

It has been a privilege to work alongside such a talented group of scholars in the Civil Engineering department. I wish everyone well on their exciting career journeys.

I am grateful for the support of my family in this journey and the encouragement from my closest friends. I appreciate every single one of them.

Finally, I also want to thank some of the students in the Carstensen Group for their help in developing the coding skills necessary to complete this project, and to Sarah Smith for helping me navigate this process.

I am extremely grateful to my parents, Carmelina and Elmer and my sister, Yvette. Their sacrifice, love, and support have allowed me to have experiences and achievements far beyond my dreams. I am so blessed to have them in my life.
Table of Contents

Chapter 1: Introduction........................................................................................................ 9
  1.1 Motivation.................................................................................................................. 12

Chapter 2: Literature Review........................................................................................... 13
  2.1 Seismic Activity and High-Rise Steel Structures...................................................... 13
  2.2 Topology Optimization............................................................................................ 14
  2.3 Previous Attempts of Using Topology Optimization for Dynamic Responses........ 14

Chapter 3: Methodology................................................................................................. 20
  3.1 Conceptual Overview............................................................................................... 20
      3.1.1 Topological Optimization Process in MATLAB.............................................. 20
      3.1.2 Creating a SAP2000 Nonlinear Analysis Model........................................... 22

Chapter 4: Results & Discussion................................................................................. 25
  4.1 Trends in the Optimized Topologies produced in MATLAB.................................... 25
  4.2 Baseline Nonlinear Comparison.............................................................................. 28
  4.3 Limitations to Research & Future work..................................................................... 30

Chapter 5: Conclusion.................................................................................................. 32

References....................................................................................................................... 33

Appendix A- MATLAB Scripts Utilized ......................................................................... 37

Appendix B- Earthquake Graphs.................................................................................... 54

Appendix C- 36 Optimized Topologies Results from MATLAB....................................... 56

Appendix D- SAP2000 Models....................................................................................... 74
Chapter 1
Introduction

High-rises are becoming increasingly popular as there is a need for more efficient land use as cities become more densely populated (Reddy, 2009). Building upwards, rather than outwards, helps with this need. Another reason high-rise buildings are becoming more popular is that in many parts of the world, these types of projects are viewed as a way to enhance a country’s prestige and economic power. There are many different definitions for high-rises. In the context of this paper, the term high-rise refers to a building that has a height that exceeds 36 m or more than 12 stories with a mixed-use program (Imad Shakir et al., 2021). Overall, the popularity of high-rises reflects the changing needs and preferences of urban populations.

High-Rises can have advantages over low-rise buildings. For starters, the views in comparison to low-rise infrastructure is unmatched. Moreover, high-rise structures also provide an environment of central living that is appealing for single-people and couples seeking a greater sense of community and shared space (Ali and Al-Kodmany, 2012). Furthermore, the U.S. Urban Land Institute believes that high-rises have the capacity to make cities more sustainable (Urban Land Institute, 2008). This is a statement that is supported by Foster et al.’s research (2008) in which the following quote was stated: “Manhattan can be considered the greenest place in America, if measured by energy use per inhabitant.” It’s that a tall building has access to natural light and ventilation, that allow for the incorporation of green technologies such as solar panels, photovoltaic cells, and wind turbines. In concept, high-rises also preserve open and natural spaces. Consider the amount of land that a 5 story building needs in comparison to a building with 50 stories to house the same amount of people. However, it is important to acknowledge that as a built form, high-rises do consume more material and energy (Ali and Al-Kodmany, 2012). This is a problem that can be solved using emerging digital technologies.

Research is exploring ways in which high-rises can be built in a more “greener” way. Some of these technologies and practices are looking into transforming construction materials and adjusting the system. In terms of material, due to steel’s high embodied carbon factor, the feasibility of using recycled steel is being investigated. Also being explored is the use of composite materials, nanomaterials, and carbon fiber in the construction process (Ali and Al-Kodmany, 2012). Furthermore, in terms of structural systems, hybrid slabs are being incorporated into high-rises with the potential of reducing embodied carbon footprint by up to 15% (Berger et al., 2020). Perhaps, a more interesting technological method being investigated in the design of high-rises is topology optimization. Topology optimization is a framework that has the capacity to find design solutions through optimizing the structural layout to satisfy needs such as minimizing material used while maintaining high performance (Zuo et al., 2012). Topology Optimization has recently been implemented in the design of reinforced concrete beams and steel frames (Changizi and Jalalpour, 2018; Nahum et al., 2023). With new methodologies in topology optimization and material research, high-rises can be designed in a more sustainable way.

Building high-rise structures presents unique structural challenges, and safety concerns. As a structure becomes taller, its structural integrity becomes a concern due to natural forces such as gravity, wind, and earthquakes (Ali and Al-Kodmany, 2012). Therefore, a lateral force resisting system must be designed to provide crucial structural stability to the building in the case of lateral loads caused by earthquakes or wind (Schuller, 1990). For tall buildings, wind is the main force driving the design (Ali and Al-Kodmany, 2012). Nevertheless, buildings in seismic regions require additional considerations. In seismic regions, an important factor to consider in design is ductility.
This is why most structural systems in these regions are composed of steel as it is a material that can bend and flex without breaking, making it more resistant to earthquake forces (Uang and Bruneau, 2018).

Designing a building to resist structural damage during an earthquake can be complex. For starters, earthquakes are a release of energy from the earth’s tectonic plates during movement. This energy release can be in different frequency ranges and is unexpected. The duration can last from a mere 10 seconds to about a minute. Soil conditions as well can influence an earthquake’s intensity (Sarkisian, 2012). There are many factors that are unknown in an earthquake. However, one thing is known: most infrastructure damages occur in the excitations close to the first few modes (Aly and Abburu, 2015). Due to the unpredictable nature of an earthquake, building codes factor in the return period of a seismic event (Ladjinovic and Folic, 2004). A commonly used method featured in the ASCE 7-16 design code is the Design Response Spectrum. The ASCE Design Response spectrum is a piecewise plot that represents the expected acceleration response of a structure to ground motion, as a function of the period of vibration. It is based on probabilistic seismic hazard analysis, derived from historical earthquake data (Response spectrum curves (ASCE) - 2016). However, it is important to realize that this is different from the response spectrum as shown in Figure 1.1. The response spectrum on the other hand is a tool that can be used to analyze the dynamic response of a specific structure. In other words, they do not yield the same results because the design spectrum is a predicted response while the response spectrum corresponds to the behavior of a specific earthquake (Rajasekaran, 2009).
As briefly mentioned before, the design code is influenced by previous earthquakes. One earthquake that influenced the U.S. design codes tremendously was the 1994 Northridge Earthquake. The Northridge earthquake prompted codes to require higher seismic loads and more stringent detailing requirements in steel-framed buildings because many multistory steel buildings featured brittle fracture of beam-to-column moment connections (Uang and Bruneau, 2018). The Northridge earthquake was one of the most expensive earthquakes, with a structural damage cost estimated at $20 billion (Seismic Safety Commission, 2020).

Another earthquake that was significant for structural design is the 1989 Loma Prieta Earthquake, which affected the San Francisco area. It is considered one of the most important earthquakes to hit the United States since the 1906 earthquake. Loma Prieta helps in understanding how different types of soils and geological conditions can impact shaking intensity. It also highlighted the importance of retrofitting older concrete and masonry buildings in the Bay area (Hanks and Krawinkler, 1991).

Other notable earthquakes include the 1988 Saguenay earthquake that hit eastern Canada, a region not previously known for seismic activity (Du Berger et al., 1991). This earthquake prompted the evaluation of the seismic design base shear provisions and the codified minimum strength for masonry systems in the Canadian Building Code (Tso and Zhu, 1991). Another earthquake that caused extreme damages was the 2002 Denali Earthquake. Most of the damages caused by the 2002 Denali Earthquake were highways and bridges, as it triggered numerous landslides causing a phenomenon known as liquefaction. Overall, it could be said that the earthquake caused little structural damage despite being comparable in size and type to the 1906 San Francisco earthquake as it struck a sparsely populated region of south-central Alaska. What was shocking in the engineering world about this earthquake is that, despite its magnitude, the Trans-Alaska Oil Pipeline managed to survive unscathed. Another notable fact about this earthquake, was that the effects were widespread and even extended to other states such as Seattle and Louisiana (Fuis and Wald, 2003). As you can see, there is a lot to learn from previous earthquakes that can be used in helping build a structure that is more resistant to future earthquake forces.

Recently, a method has been proposed to use topology optimization for engineering steel-framed tall buildings to resist earthquake forces. The method, known as the “proposed sum of modal compliances,” was developed by Amory Martin (2020) with the framework based on the response spectrum analysis method. The original framework incorporates the use of the idealized Chilean Standard design spectrum, as the design response spectrum is smoother in comparison to others. This design spectrum used is shown in Figure 1.2. The overall goal of the framework is to design the lateral steel frame system by maximizing the stiffness while constraining the amount of material used to achieve this. This is significant because it is a potential cost-saving solution that can help minimize embodied carbon usage without sacrificing structural integrity in terms of earthquake safety (Martin and Deierlein, 2020). However, despite the framework being a step in the right direction in applying topology optimization for dynamic responses, it assumes linear analysis and thus cannot be directly implemented yet.
1.1 Motivation

In recent years, topology optimization has developed rapidly to explore efficient structural geometries (Beghini et al., 2014; Galjaard et al., 2014; Bialkowski, 2016). One such method is PolyTop SMC proposed by Martin and Deierlein (2020), which through considering multiple eigenmodes, minimizes structural vibration for seismic excitation, described by a design response spectrum in a tall building. However, there exists a gap between using the results produced from PolyTop SMC and implementing it for earthquake design, which requires an extensive look into the differences produced through linear and nonlinear analysis. This is significant because the original optimization method explored by Martin does not account for differences in modal participation factors, and in real seismic loading, the forces vary through the structure from inertial affects rather than from a discrete force (Martin and Deierlein, 2020).

The scope of the project is to modify Martin’s PolyTop SMC Code (2020) to optimize the domain in response to a generated idealized response spectrum from an earthquake’s accelerogram. The results would later be translated into SAP2000 for a nonlinear analysis to assess the robustness of the response through a nonlinear analysis. This would be evaluated through past earthquakes of varying frequencies from different seismic regions in North America. Currently, PolyTop SMC produces an optimized topology from the idealized Chilean Standard, which is not an accurate representation of every earthquake (Martin and Deierlein, 2020). By updating the coding process to yield responses that are more representative of the accelerogram behavior, subjecting the optimized design to a nonlinear earthquake response in SAP2000 would help understand the robustness of the optimal design produced. Furthermore, subjecting the optimized topology to a nonlinear spectrum would also allow for a better understanding in the structural failure produced with the PolyTop SMC approach to bridge the gap in using topology optimization for seismic design.

This research is important for several reasons. In general, the construction industry is driven by cost, and topology optimization can be a tool that reduces the amount of material used on a project without sacrificing safety. By reducing material usage, buildings that use this method could have the potential to reduce their embodied carbon footprint, which is also an important factor to consider as the built environment accounts for a significant portion of CO₂ emissions (Donofrio, 2016).
Chapter 2
Literature Review

2.1 Seismic Activity in High-Rise Steel Structures

As previously mentioned, high-rise construction is a solution to the issue of overpopulation and a popular building material for this type of infrastructure has been steel. In general, structural design is heavily influenced by earthquake loading (Zakian and Kaveh, 2022). Due to its properties, steel is more resistant to these types of forces in comparison to other materials which is why it is such a popular building material.

Steel has a rich history in being used in earthquake engineering. For starters, it is a very ductile material which is important in seismic resistant designs because it allows for structures to absorb, and dissipate energy caused by earthquake waves preventing failure and/or collapse. When it comes to the seismic design of a structure, ductility and capacity design are at its foundation (Uang and Bruneau, 2018). As a result, steel is primarily used for constructing the lateral structural system.

Overall, Steel Lateral Systems can be grouped into two main categories: Moment and Braced Frames. Moment Frames were previously considered to be the leading system for earthquake resistance. However, that all changed with the Northridge Earthquake of 1994. During the Northridge event, several steel moment frames in multi-story buildings experienced brittle failure costing millions in damage causing a push towards braced beams (Uang and Bruneau, 2018). Under seismic loading, brace frames tend to have better structural performance than moment frames (Choi et al., 2008). However, even after the Northridge event Steel Moment Frames were still considered as a design choice due to their open-space floor plans. Also, it is important to note that although drift is reduced in braced frames, the columns are subjected to high axial loads causing a significant moment. Both systems have their benefits and show how steel columns can play a significant role in preventing failure during an earthquake (Uang and Bruneau, 2018).

When designing a steel structure subjected to earthquake ground motions, important parameters to consider in the design process is mass, damping, and stiffness since structures undergo inelastic deformations under seismic conditions. In general, earthquake resistant design can be divided into two main approaches: force-based design and ductility-based design. The difference is that force-based design is an approach that focuses on designing structure to withstand seismic forces and is typically used in traditional design codes while ductility-base design focuses on designing structures for a higher ductility capacity. Ductility-base design is in more modern codes specifically for seismic retrofit of existing structures. All these methods need seismic analysis to go in hand. One kind of analysis is linear analysis which estimates the response of a structure to seismic forces using linear elastic behavior assumptions. In this analysis, the structure is assumed to undergo small deformations under seismic loads, and the material behavior is assumed proportional to the applied load. Under this category falls the Response Spectrum which is a dynamic method, where modal analysis is performed, and contribution of each mode is considered in the structural response. To elaborate, the response of a structure to a representative ground motion spectrum is computed by decomposing the structure into a series of single degree of freedom systems whose number is equal to the number of selected modes. It is wildly used in seismic design. The other method is a nonlinear analysis, and it is used to predict the structural response in buildings or infrastructure under large deformations and where the material behavior...
is not linear. This method can be combined with previous analysis procedures under linear analysis (Zakian and Kaveh, 2022). For example, SAP2000 is a software that has the capabilities of performing a nonlinear spectrum analysis for a structural model. Nevertheless, it is important to consider that both analysis methods can contain inaccuracies which is why it is important to consider each scenario carefully.

2.2 Topology Optimization

Topology optimization is a mathematical method that seeks to optimize the design of a system by determining the optimal distribution of materials within a given design space (Zuo et al., 2012). The goal of topology optimization is to find the best design that meets certain performance criteria, such as minimizing weight while adhering to constraints focused on maintaining structural integrity, maximizing stiffness, or minimizing deformation under load. This is achieved by iteratively analyzing and modifying the design, until the desired performance criteria are met. Usually, this process involves the use of computer algorithms and mathematical models to explore the design space, and the resulting optimized designs can often be quite complex and non-intuitive (Zakian and Kaveh, 2022). In general, topology optimization has been widely used and applied to static loads (Sigmund and Maute, 2013). Recently, there has been a push to apply it to problems centered around dynamic loads since most design building design codes are governed by these (Martin and Deierlein, 2020).

2.3 Previous Attempts of using Topology Optimization for dynamic response.

Previously, dynamic loads have been a challenge to implement in topology optimization because of their complex and unpredictable nature which can lead to suboptimal designs. In general, the issue is that dynamic loads in comparison to static loads exhibit time-varying behavior, and result in a high-frequency response that can be computationally expensive and inaccurate (Zakian and Kaveh, 2022). However, most design in the structural engineering realm is governed by dynamic loads such as wind and earthquake (International Code Council, 2018). This is why some researchers have attempted in previous years to minimize this knowledge gap.

In 2011, the Innovative Structures Group at RMIT University in Melbourne, Australia proposed a method of topology optimization that would be able to address multiple constraints of displacement and frequency (Zuo et al., 2012). These are important factors to consider in earthquake-resistant design. The method proposed serves as an extension of the bi-directional evolutionary structural optimization or BESO method; using an evolutionary algorithm to iteratively optimize the topology of the structure that satisfies multiple displacement and frequency values as well as simultaneously minimize the weight of the structure. What was significant about using evolutionary algorithm in this case, was that it allows for a more efficient search of the design space, leading to quicker and more accurate designs. To test the validity of the proposed method, the Innovative Structures Group attempted to apply the method to a reinforced hybrid steel concrete frame building. In this research project, the dynamic load taken into consideration was wind in two different directions. The algorithm resulted in a hybrid shear wall/brace frame topology to replace the original system in place utilizing the same amount of material whilst reducing the drift by approximately 80% through increasing the stiffness (Zuo et al., 2012). The final design is shown in Figure 2.1 below. The success of this research project showcased the
capabilities iterative algorithms could have on solving other dynamic problems such as earthquake design, if further researched and modified.

In earthquake engineering, it is important to design structures that can withstand strong ground motions without collapsing, and this requires an understanding of the structure's natural frequencies and their corresponding modes of vibration. Researchers in Denmark have presented a new approach to topology optimization of vibrating structures, with the goal of finding optimal topology that maximizes multiple eigenfrequencies and frequency gaps (Du and Olhoff, 2007). The approach utilizes a level set and penalty method to handle the frequency constraints. By optimizing the structural design for maximum eigenfrequencies and frequency gaps, structural performance can be improved. This has several implications, including: the structure may have improved dynamic performance and be less susceptible to collapse or damage during an earthquake; reduced material usage: by using topology optimization to achieve a desired eigenfrequencies and frequency gaps, the resulting structure can be designed to use only the minimum amount of material necessary to meet required performance specifications; Increased design flexibility: the ability to optimize the topology to achieve desired eigenfrequencies and frequency gaps provides greater design flexibility, enabling engineers to tailor the structure to specific performance requirements; and computational efficiency: Aalborg University’s proposed method can be considered for the time as a highly efficient approach to topology optimization for vibrating structures reducing thus cost and time. Aalborg’s University approach was successfully validated on single and bi-material 2D and plate-like 3D structures. This method presents some limitations as the method is only applicable to linear elastic structures, assumes that the material properties are homogeneous and isotropic which may not be the case in real-world structures, the method assumes all design loads are previously known and can be accurately modeled which may not be possible, and lastly the method does not consider other design criteria such as cost, weight, or manufacturability (Du and Olhoff, 2007). Nevertheless, the method developed is a step in the right direction in understanding how to apply topology optimization for earthquake engineering.

More recently, researchers at John Hopkins developed and used a methodology for topology optimization of frame structures under linear stationary stochastic dynamic loading (Zhu et al., 2017). Linear stationary stochastic dynamic loading refers to loads that involve random variations of loads over time, where the statistical properties remain constant. This loading is seen in structures subjected to environmental phenomena such as wind, waves, and earthquakes.

The way that the proposed method from Hopkins University works is that it begins with generating a set of random excitations with prescribed mean and standard deviation using the spectral representation method. Then, the response of the structure subjected at each random excitation is calculated using finite element. Afterwards, the power spectral density for the response of each random excitation is calculated and a stochastic optimization problem is

Figure 2.1: Optimized Steel Bracing Results the Innovative Structures Group from RMIT University (Zuo et al., 2012)
formulated to maximize the expected value of the objective function subjected to the constraints of the expected values from the power spectral density. The optimization problem is later solved using the method of moving asymptotes. Results should be further subjected to a reliability and feasibility analysis to be verified. This method was found to be effective in achieving optimal design that satisfy both displacement and acceleration constraints under stochastic dynamic loads.

Zhu et al. (2017) applied their optimization method to a hypothetical 30-story building with a reinforced concrete core and steel perimeter frame with the objective to minimize the maximum displacement of the building under wind loads while satisfying constraints on the maximum stresses. In this scenario, two different design criteria are considered: an optimized version with outriggers and one without. The optimization favored the design with outriggers as it demonstrated significantly lower stresses and displacement values, showing that outriggers can effectively reduce the dynamic response of tall buildings. It was also observed that the optimal design, have higher natural frequencies and are more robust to stochastic loading. In terms of limitations, a linear elastic material behavior is assumed, small strains, and linearly stochastic loads as well. These assumptions are not representative of real-world scenarios, presenting an issue of accuracy. Additionally, with this formulation a uniform density distribution is utilized which is not adequate for all design scenarios. More complex structures would require high-performance computing resources.

This proposed method by Zhu et al. (2017) could be extended in application as earthquakes are a significant source of stochastic excitations on structures. Topology optimization techniques can be used to design earthquake-resistant structures that are optimized for their dynamic response under stochastic loading through minimizing the response variance of the structure subjected to constraints on the structural mass and compliance. In earthquake-resistant design, minimizing the response variance is significant because it can help reduce the potential for damage or failure during a seismic event (Zhu et al., 2017).

Liu et al. (2015) compare three dynamic analysis methods for structural topology optimization. The three methods they consider are Direct Harmonic Response, Frequency Response Function, and Time Domain Response. In the Direct Harmonic Response method, the structure is subjected to a harmonic force excitation, and the response is computed directly in the frequency domain. The resulting response is then used to evaluate the objective function and constraints in the optimization process. Harmonic force excitations are a type of dynamic load that is periodic and is represented through a sum of sinusoidal waves with different amplitudes, frequencies, and phases. The second method assessed is the frequency response function method where the frequency response function is computed for the structure, relating the harmonic excitation to the response in the frequency domain with the frequency response function used to evaluate the objective function and constraints in the optimization process afterwards. The last method evaluated at Northwestern Polytechnical University has the response of the structure computed in the time domain using a time-stepping algorithm. The resulting response is used then to evaluate the objective function and constraints. While the three methods researched aren’t directly focused on earthquake engineering, Earthquake Forces can be estimated through harmonic force excitations. The methods analyzed may not accurately capture structural behavior under large earthquakes with nonlinear effects. Also, the methods use harmonic force excitations which simplify loading conditions. These methods also are troubled with model complexity and high computational costs for large-scale structures. Overall, the methods presented in the study provide valuable insights into the dynamic behavior of structures under harmonic force excitations. It is important to consider the assumptions and limitations of the methods and complement them with
other analysis techniques to ensure the safety and reliability of structures under earthquake loading (Liu et al., 2015).

One of the first steps in extending topology optimization to seismic design in a linear sense has been the proposed Sum of Modal Compliances (Martin and Deierlein, 2020). As a method, it is based upon the concept of modal analysis which is discussed in ASCE Section 12.9.1 (ASCE, 2007). When optimizing through the Sum of Modal Compliance, the method is based on the inherent elastic analysis assumption of response spectrum. Yet, this methodology does not account for inelastic behavior or energy dissipation. This leaves a gap in directly implementing the results in structural usage.

Moreover, the Sum of Modal Compliance or SMC methodology was developed as a modified version of the original PolyTop code developed by Talischi et. al. in 2011 that incorporates four supplemental functions: Eigenanalysis, Response Spectrum, FEAnalysis, and LocalM. PolyTop serves as the foundation in implementing the SMC methodology because it uses a stable randomly generated polygon mesh, can handle arbitrary boundary domains with curves and edges and is efficient in taking the vectorization in MATLAB. Moreover, one of the most important distinctions to note between SMC PolyTop and PolyTop is that SMC must include mass and dampening matrices. Also, another important aspect is that the supplemental Response Spectrum function that is incorporated into SMC PolyTop is based on the Chilean Standard NCh422.0f96. This can present an issue in terms of accurate representation as the Chilean Standard was chosen because the response spectrum was smooth in comparison to other design spectrum which are non-smooth piecewise continuous functions as shown in the figure below. Nevertheless, it still offers an insight into what could be a finalized topology for a tall building in terms of seismic excitation.

Figure 2.2: Comparison of the U.S. ASCE Design Spectrum vs. the Chilean Standard (Zeynalov et al., 2013)

As mentioned before, the SMC framework is based upon response spectrum analysis where the frequency content of an earthquake is represented through a minimized weighted average of modal compliance. The formulation for the proposed SMC methodology is shown below in Figure 2.3. In the formulation, \( \Phi_{SMC} \) represents the sum of modal compliances, \( \mathbf{K} \) is the stiffness matrix, \( \mathbf{M} \) is the mass matrix, \( V \) is the volume, \( \bar{\mathbf{V}} \) is the volume fraction, \( \mathbf{\rho} \) is the element density vector, \( \mathbf{U}_j \) is the modal displacement vector for mode \( j \), and \( Sa(T_j) \) represents the pseudo-acceleration for the period \( T_j = \frac{2\pi}{\omega_j} \) with \( \omega_j \) being the natural frequency of mode \( j \) in rad/s.
minimize: \[ \Phi_{SMC}(\rho, U) = \sum F_j^T(\rho)U_j(\rho) \] (objective function)

subject to: \[ \int_{\Omega} \rho dV - V \leq 0 \] (constraints)
\[ 0 \leq \rho_e \leq 1, \quad \forall e \in \Omega \]

with:
\[ U_j = K^{-1}F_j, \quad \text{for } j = 1, ..., m \] (state equations)
\[ F_j = \Gamma_j M \phi_j S a(T_j) \quad \text{for } j = 1, ..., m \]
\[ [K - \omega_j^2 M]\phi_j = 0 \quad \text{for } j = 1, ..., m \]
\[ \phi_j M \phi_k = \delta_{ik} \quad \text{for } j, k = 1, ..., m \]

Figure 2.3: Framework of the proposed SMC method (Martin and Deierlein, 2020).

Overall, the SMC computational procedure is described through Figure 2.4. In general, the procedure consists of an outer loop to iteratively compute the modal load vectors and an inner loop for the stiffness and design variables. The final topology is only obtained after the seismic modal load vectors converge. As mentioned previously, this procedure requires the input of assumed structural properties such as density to kick start the process.

Figure 2.4: Chart showing the PolyTop SMC Process (Martin and Deierlein, 2020).

Martin (2020) had initially used the proposed topology formulation, for a high-rise of 165 m height and a base of 33 m. Other parameters defined in the setup include the modulus of elasticity set to 200 GPa, Poisson’s ratio at 0.3, and a .10 m thickness. A volume fraction of .30 and a filtering radius of .04b was also used. The results of the code incorporate the use of a passive
domain with width ratio of .12 in respect to the base of the high-rise and a modulus of elasticity defined through the variable $E_c$ at a ratio value of 50. In the formulation of this problem, SIMP material interpolation is considered every 50 iterations with an increased .5 exponential penalization that ranges from $p=1$ to $p=4$. In Martin’s (2020) initial proposal, two soil types A or B can be defined representing stiff or soft soil conditions. Both soil types had a PGA= 0.6g. Martins (2020) results for both soil types are shown in the Figure 2.5.

Figure 2.5: SMC Topolgy Optimization results for a tall building on stiff (a & b) and soft soil (c & d) (Martin and Deierlein, 2020).
Chapter 3
Methodology

3.1 Conceptual Overview

The methodology of this research can be divided into two parts: modifications to Martin’s (2020) existing MATLAB code, and conducting a nonlinear analysis using the CSI software, SAP2000. This thesis focuses on modifying Martin’s (2020) MATLAB code to represent the response spectrum of an earthquake, instead of an idealized design spectrum. Subsequently, the modified code will generate different lateral frames from various seismic regions within North America. The objective is to compare the generated lateral frames and identify trends in layout prompted by property changes, such as dimensions and core density. The next step is to translate the lateral frames into SAP2000 for a nonlinear analysis to test the robustness of the design. However, transferring the model from one software to another presents a crisis of replication between the two solutions produced. Therefore, a separate subsection is dedicated to detailing these challenges and process involved.

3.1.1 Topological Optimization Process in MATLAB

Martin’s (2020) original script consists of a main function called PolyTopSMC and four supplemental functions: EigenAnalysis, Response Spectrum, FEAnalysis, and LocalM. The objective was to modify these functions to allow for the input of the response spectrum of a specific earthquake, rather than using the idealized Chilean Design Spectrum. Once these modifications were made, four earthquakes were chosen to test the new framework out. The earthquakes selected were Northridge, Loma Prieta, Denali, and Saguenay Earthquakes, with their locations shown in the Figure 3.1. As stated in the introduction, Northridge and Loma Prieta were selected because they triggered U.S. building code modifications. The 2002 Denali earthquake was selected for its comparison to the 1906 San Francisco. Lastly, the Saguenay Earthquake was selected because it was one of the largest earthquakes to hit the North-Eastern Region. All four response spectrum data were obtained from the Pacific Earthquake Engineering Research Center (PEER) ground motion database, which is a web-based tool that allows you to search and download ground motion data.

Figure 3.1: Location of Earthquakes Selected for exploration.
In Appendix A, the modifications and files mentioned are shown. Martin’s (2020) framework is called in one main file titled, PolyScript.m, which defines parameters such as the thickness of the domain and the response spectrum properties. It creates a “fem” structure, containing information pertaining to the finite element mesh and the properties of the solid material that would depict the core in the results. It also includes parameters related to the dynamic response of the structure, such as loads and supports. The optimization problem is solved in this file using a nested optimization loop that continues until the convergence tolerance is reached or the maximum number of iterations is exceeded, with the final design than being plotted and displayed.

The objective of this study is to modify the code to import the response spectrum data from the four earthquakes mentioned, to yield more accurate results than Martin’s (2020) original code. In comparing the steel topologies produced from the four different earthquakes, material and dimension properties were altered to determine if certain properties triggered characteristics of a certain topology optimized design. The modifications aimed produce thirty-six different layouts, each with a unique combination of structural height, structural base width, ratio of core density, and earthquake data. In PolyScript.m, it was determined that the code would remain with the structure for soil type B, which represents softer soil, as tall buildings founded on rocks or hard soils generally perform better (Ali and Al-Kodmany, 2012). Therefore, the only modifications made in PolyScript.m were to the base width value and the density ratio value. Three base values (13 m, 33 m, and 69 m) were used, chosen to maintain the same core width to base ratio as in Martin’s (2020) examples. Core density ratio values were varied from 10, 50, and 100, based on Martin’s (2020) examples as well. These values were used in PolyScript.m to determine passive elements and to define the domain of the mesh to be generated. Another slight modification to PolyScript.m, was the inclusion of the ‘rng’ function to set the seed of the random number generation to a fixed value, ensuring the same results were generated when the response spectrum was imported.

The file PolyScript.m calls the main function PolyTop_SMC2.m, which implements the Sum of Modal Compliances. The function PolyTop_SMC2.m uses finite element analysis to solve the optimization problem, implementing an optimality criterion for updating the design variables in the iterations produced in the loop. It also includes functions that define the objective and constraint functionals and their respective sensitivities with respect to the design variables. Therefore, the only modifications required for this function were related to the plot domain to ensure that the results produced are fully displayed.

The file PolyScript.m also calls the supplemental function ResponseSpectrum.m as one of the parameters used in the optimization loop. Originally, the file ResponseSpectrum.m was structured to calculate the idealized design response spectrum (Sa) and its derivative with respect to period (dSadT) based on the Chilean Standard NCh422.Of96. Its parameters are T, which represents the period of the seismic event, the peak ground acceleration (PGA), the reference period (T0), and k, which is the damping ratio exponent. However, this file was modified so that the function would import a .csv table containing the response spectrum values for a specific earthquake event and then fit a polynomial curve that best fits the earthquake data to represent the response spectrum, as the framework is only able to interpret smooth curves. The degree of the polynomial curve for each earthquake was determined manually by changing the value in the file called PolyFit.m and comparing the curve generated to the original data imported for that earthquake, as shown in Figure 3.2 for the 1994 Northridge Earthquake. The polyfit curve in ResponseSpectrum.m is then evaluated at the input T period to represent Sa, and then the derivative of the curve is also calculated using the polynomial derivative in order to be used in
obtaining results in *PolyScript.m*. Appendix B contains graphs showing the polyfit curve and the response spectrum for all four earthquakes used.

The last file modified that *PolyScript.m* utilizes is *TallBuildingDomain.m*, which generates a domain representing the tall building structure with a fixed base. This file was changed to accommodate varying base and height values. The other functions utilized by *PolyScript.m*, namely *LocalM.m*, *FEAnalysis.m*, and *EigenAnalysis.m* were left untouched. *LocalM.m* calculates the element mass matrix for a finite element analysis, whereas *FEAnalysis.m* conducts the analysis of the structural system subjected to seismic dynamic excitation. It computes the natural frequencies and mode shapes required for the modal response spectrum analysis. Finally, *EigenAnalysis.m* performs an eigenvalue analysis of the finite element model using the mass and stiffness matrices produced in the previous functions. In other words, it computes the natural frequencies and mode shapes of the structure.

![Figure 3.2: Response Spectrum and Polyfit Curve for Northridge Earthquake](image)

3.1.2 Creating a SAP2000 Nonlinear Analysis Model

Once the 36 thirty-six optimized topologies were generated, certain lateral systems were modeled in the SAP2000 program. The topologies selected to be modeled in SAP2000 were those that did not display signs of failed convergence and seemed feasible to construct, as they resembled existing steel frames in the industry. It is important to note that translating results from a 2D platform to a 3D model requires assumptions to be made, posing a replication crisis, which can be considered a main factor that stands in the way of applying topology optimization to seismic design.

To transfer the results, the framing members were drawn first in SAP2000. This was done through image scaling, as the result from MATLAB presents a lack of clarity, as it does not clearly state the length or dimensions of a steel brace, as shown in Figure 3.3. Since the height and base are known, the core height in the MATLAB result could be measured to form a proportion that is used to find out how far a brace extends or where the starting and ending points are in reference to the height. Even then, the result does not mention how many steel members are used to form each brace; it just shows the general layout. That is why it was assumed when constructing the SAP2000 model that the bracing extended from the middle point of the face to the edge unless there was another member that connected to that brace, then it was split at that point. Also as seen in Figure 3.3, the MATLAB script forces the bracing to appear to meet at the core which may not necessarily
be accurate for a lateral framing system. As such, the MATLAB result was interpreted as a representation of the perimeter of one of four identical sides, as seen in Figure 3.4. The connections of the bracing in the middle of the face were assumed as the portion was covered in the MATLAB result. Also, the edge columns were modeled with a height of 4.12 m based on Martin’s (2020) example, and a concrete floor plate was added at each of these divisions. The concrete core was also modeled in SAP2000 using the PolyArea tool. The dimensions and location of the core were determined through a similar proportion process using the width to core ratio of 0.12 that Martin (2020) defined in the original PolyScript.m file. The last element modeled in SAP2000 were the restraints. This was modeled as pinned bases because it was defined as fixed in the MATLAB formulation and a previous experiment conducted that showed effectiveness in avoiding weak story failure (Uang and Bruneau, 2018). The pinned bases could also be seen in the model face displayed in Figure 3.4.
Once all elements of the SAP2000 model were drawn, material properties were defined for the elements. A992Fy50 grade steel was chosen for its high strength to weight ratio, and 4000 psi concrete was used for the core and slabs due to its commonality in building construction. Image scaling was again used to determine the cross-sections to be used in the model. In the MATLAB framework, it was stated that for high-rises, the thickness into the page would generally be about 0.10 m. Therefore, this value used for the depth of all the beams used in the model. This implied that the value being scaled for the brace members displayed on the generated optimized result would correspond to the width if using rectangular members for simplicity. Since there is a lack of clarity in the optimized result regarding where a member begins or ends, the concept that the dimensions of a column’s cross section at upper story of a building structure should be equal to or smaller than the column beneath were applied (Zakian and Kaveh, 2022). To determine the width value to be used in SAP2000, the width of the generalized layout produced from the optimization algorithm was measured and used in a proportion relating to the width of the structure, as this information was known. For members in the SAP2000 model, that are not illustrated in the MATLAB-produced results, such as the floor beams and columns, the default rectangular steel member size in the software was used.

Regarding the concrete elements, as previously mentioned, 4000 psi concrete was used and drawn as thin shelled elements. However, for the concrete core, the modulus of elasticity was changed to reflect the ratio used in the MATLAB optimized result. For example, if the optimized result had used a ratio of 50, the modulus of elasticity used for concrete core would have to be set to 10,000 GPa in SAP2000, since the modulus of elasticity for steel is 200 GPa. No other properties from the default settings in SAP2000 were changed for the concrete used in the core. For the concrete used to model the floor slabs in SAP2000, the default modulus of elasticity for 4000 psi concrete was used.

The overall goal of modeling the optimized configurations on SAP2000 was to compare the performance to a standard used X-braced system. Keeping the same height, base, and floor divisions as the modeled optimized version, the X-brace model features symmetrical faces with equally spaced bays corresponding to the overall structural height. Figure 3.5 shows this configuration for a 165 m high-rise. Furthermore, the core and floor slabs had the same material definitions as the optimized model being compared to.

Regarding the steel cross-section size used for bracing, this was determined through a backhand calculation using the density function. In order to be comparable to the optimized model, the weight of the bracing was set to be equivalent. Using the optimized modeled brace mass and the known density of the A992 grade steel of 7850 kg/m³, the volume of the steel to be used was determined. From there, assuming the 0.10 m thickness for the steel members was still true and the lengths of the braces were known, the width of the cross-section to be used could be determined. It is to be noted that unlike the optimized model that featured bracing with different cross-sectional sizes, the standard brace frame was modeled assuming the same size would be use throughout as typical frames do in the industry.
The purpose of using SAP2000 is to conduct a response spectrum analysis comparing the performance of the unoptimized bracing and the optimized layout. For any model requiring this type of analysis, the earthquake data must be imported into the software. Using the .txt data files obtained from the PEER database, the response spectrum function must be defined in SAP2000. Once the function has been defined, the .txt file containing the earthquake data points can be added. The data must then be specified as either representative of a period v. value or frequency v. value trend, depending on the type of earthquake data obtained from the PEER database. After the data has been defined as a function, the load case must be defined for the model in SAP2000. When defining the load case, select the dynamic response spectrum type. Once the type of load is selected, you can add the previously defined function to the load type. Finally, you will need to adjust the scale factor of 9.81, as PEER data is in units of g. Once this has been completed for a selected earthquake, the analysis could be run on the SAP2000 model being analyzed.

Chapter 4
Results & Discussion

4.1 Trends in the Optimized Topologies

In total, thirty-six combinations were produced from the altered MATLAB optimization. These combinations were formed by analyzing four different earthquakes, three different building heights, and three different core density ratios. As previously mentioned, the seismic events for which lateral systems were optimized are the 1988 Saguenay earthquake, the 1989 Loma Prieta earthquake, the 1994 Northridge earthquake, and the 2002 Denali earthquake. Furthermore, as mentioned, the dimensions of the structure were varied, with the first having a height of 65 m and a 13 m base, the second having a total height of 165 m and a 33 m base, and the third having a height of 345 m and a base of 69 m. The core density ratio was also varied from 10, 50, and 100.
Several trends were observed within the thirty-six combinations. As seen in Figure 4.1, there were little variations when comparing the lateral systems produced within the same earthquake and core density ratio. The overall layout was similar, the bracing ended at relatively the same place. There were subtle changes, such as minor braces added, or the angle of the braces was slightly modified. Each version resembles the previous within the same earthquake and core density ratio.

![Figure 4.1: A.) 3 Different Lateral systems with varying dimensions for the Loma Prieta earthquake featuring a core density ratio of 10. B.) Lateral Systems for the Northridge Earthquake with a core density ratio of 50 and varying structure dimensions. For both models Left to Right: Smallest to Tallest.](image)

Furthermore, it was observed that varying the core density ratio for a fixed height and a certain earthquake was a major factor in changing the optimized structural lateral frame, as depicted in Figure 4.2. Unlike the previous observation, no trend was observed for the ending position of the bracing, or the quantity of members used when the core density was varied across all four earthquakes. Each layout was completely different from the previous, showing the importance of the core stiffness in earthquake design.

![Figure 3.2: Steel Bracing layout focusing changing core density ratio for A.) For 165 m Structure exposed to the Saguenay Earthquake B.) For 345 m tall building exposed to Loma Prieta](image)
As predicted, different optimized layouts were produced when analyzing structures for different seismic phenomena featuring the same core density ratio and building dimensions. This is shown in Figure 4.3 for the 165 m high-rise with a core density ratio of 50. This was predicted because each earthquake analyzed, had varying intensities and periods. However, through this comparison, one trend observed was that the layouts exposed to the Northridge earthquake generally have more thicker members than the optimized results from the other earthquakes, despite having a lower magnitude.

As mentioned beforehand, the optimization continues until convergence is reached or the maximum number of iterations is reached. Some of the results produced showed asymmetrical bracing or faint lines. These are indicators that the framework reached the iteration limit before reaching the convergence tolerance. Examples of these solutions are shown in Figure 4.4. As such, this limit needs to be increased for these solutions until convergence is met.

Other notable optimized results featured member thickness that were incredibly large, as shown in Figure 4.5. These results showed that for that earthquake, a lateral system would not suffice with that particular core stiffness. This also highlighted the importance of a core in designing a structural lateral system for earthquake resistance.

---

Figure 4.3: Four identically sized structures (165 m x 33 m) with a core density ratio of 50 subjected to 4 different earthquakes:
A.) Loma Prieta B.) Northridge C.) Saguenay D.) Denali

Figure 4.4: Examples of Solutions that reached Max Number of Iterations A.) Denali Earthquake for a 65 m High-rise with a 50-core density ratio B.) Loma Prieta for a 165 m building with a 100- core density ratio C.) Saguenay Earthquake for a 65 m structure with a 100- core density ratio.
4.2 Baseline Nonlinear Comparison

A couple of the optimized layouts produced from MATLAB that converged successfully were chosen to be modeled and analyzed in MATLAB to verify and compare their performance under seismic excitation to a regular base-frame. To better understand the overall performance, other structural components such as slabs and columns were included, as 3-D models in research typically do not include all structural components (Zakian and Kaveh, 2022). These other components were modeled with the same properties to be able to compare the two models. Another interesting note in comparing models is that the bracing on the standard braced-frame system is made with the same material volume as the bracing in the optimized frame. This was done to better analyze the difference in performance.

One of the optimized layouts analyzed in SAP2000 was a 165 m high-rise with a 50-density core ratio exposed to excitations from the 1989 Loma Prieta Earthquake. The performance of this model was compared to an equal spaced 8-bay X brace lateral frame. It was observed that there was a 0.018 m difference in the maximum displacement values produced between the two systems. The results of both models can be observed in Figure 4.6. As seen, the optimized lateral frame performed about 15% better than the X-bracing system for the same amount of material. However, it is important to note that the minimum displacement value observed in the optimized frame was higher than the other model. This is expected because the bracing follows a consistent pattern throughout the lateral face, unlike the optimized version in which the bracing is strategically place.
Supporting the performance of the optimized layout’s resistance to earthquake performance was the lateral system for a 165 m high-rise with a 50-density core ratio produced for the 1989 Saguenay Earthquake. Assembled through similar assumptions such as the optimized bracing system for Loma Prieta, it was observed that the performance improved by 60% when comparing the displacement values from an 8-bay X-braced frame with the same material volume. Overall, for the optimized layout generated for the Saguenay Earthquake, the displacement values were consistent throughout the building length, with the difference between the maximum and minimum displacement values reported in SAP2000 being less than 0.010 m. These results are shown in Figure 4.7, along with a general idea of how the lateral system was modeled in the program, which is an important factor that can influence the results.

Furthermore, analyzing the performance of the optimized layouts for the 1994 Northridge seismic event formed interesting results. This is believed to be because, compared to the other layouts generated, the bracing was concentrated at a lower height. Figure 4.8 shows this optimized bracing for a building with a 165 m height. However, other models exposed to Northridge had a generally similar configuration. What is interesting is that the optimized layout for Northridge reported a maximum displacement value of 0.308 m and the lowest displacement value was 0.022 m, which is a relatively small change in values. However, the displacement values for the comparable X-braced frame were 10 times larger, with a range between 2.86 m. This was a trend found in several Northridge models that were compared to a standard X-brace. Lastly, this extreme displacement range was also different from previous comparisons conducted in SAP2000, which had displacement values that were relatively similar.
4.3 Limitations to Research and Future Work

There are certain limitations to the research conducted that need to be further evaluated both in the MATLAB code optimization process and the modeling conducted in SAP2000 for constructability. In terms of the process in generating topology optimization, additional parameters should be studied to determine how they influence results. For example, several lateral framing systems failed to converge due to a specified iteration limit of four to help with computing power. If this number is increased, there is a possibility that these frames would be complete at the expense of longer time. Another tradeoff between accuracy and computational cost is changing the mesh in the current topology optimization framework. The current mesh being used has y-axis symmetry with 20,000 polygon elements (Martin and Deierlein, 2020). In topology optimization, the mesh is the discretion of the design domain into small finite elements. Thus, this value affects the quality of the results. Taking time to refine the value of the mesh reduces the size of the finite elements and captures more details of the layout. This value could be the reason in which certain layouts produced by the current framework appeared with abnormally thicker sections. If the current mesh is considered coarse for certain earthquake spectrums, then those designs are suboptimal and could lead to a lower structural performance than anticipated. Another issue with the MATLAB Framework is that it is dependent on response spectrums that are smooth. As mentioned previously, this is why the original framework was setup to use the idealized Chilean Standard (Martin and Deierlein, 2020). Therefore, the framework was modified to use the polyfit function to import the response spectrum directly as design spectrums are predictions. However, through using the polyfit function to generate a smooth curve, certain extreme data points in the response spectrum may not be captured. Also, the polyfit function may not generate a curve close enough to resemble the behavior of the original response spectrum imported.

The process of transferring the results from MATLAB to an SAP2000 model had a considerable number of assumptions that need to be further explored in more depth. For example, member locations and sizes were determined through image scaling. However, this may not be an accurate representation of the sizes of members that MATLAB is assuming as through image scaling some cross-sections measured 1m thick or greater. Also, the type of steel member assumed
by the generated layout in MATLAB is unknown. Therefore, several member shapes should be further compared in SAP2000 to determine which leads to the most optimal design with the generated layout. This is important to be further looked at as the current measure of success in the scope of this project does not factor in effective slenderness nor local buckling, which are factors that are known to play a key role in the overall response of a brace (Uang and Bruneau, 2018). Also, in modelling the optimized result in SAP2000, it is unknown how many members are assumed in the topology optimization framework. This should be further investigated as having smaller length but more members to compose the bracing layout places a greater emphasis on the connections, which was not investigated during the scope of this analysis. Column-to-column and beam-to-column joints can influence structural performance as experienced with the Northridge earthquake in 1994 (Zakian and Kaveh, 2022). However, longer spanning members can be difficult to construct, which should be considered in trying to implement the MATLAB results. Another limitation in transferring the result from MATLAB to SAP2000 is that the result was assumed to represent a lateral face in which braced members were not directly connected to the core. However, the result could also represent a section of the system. In other words, a system in which the members are connected to the core. Therefore, it is recommended that both are further model and compared.

During the modeling of optimized layouts exposed to the Northridge earthquake in SAP2000, certain limitations were identified. Specifically, the displacement values of the optimized frame were found to be significantly different from those of a standard brace system. Further analysis revealed that this discrepancy was due to the concentration of brace members towards the bottom. To address this issue, it may be worth exploring modifications to the MATLAB framework that allow for a more uniform distribution of bracing throughout the domain. Additionally, the work on SAP2000 should be extended to consider exposing the frames to other earthquake analysis techniques such as time-history as each method has its limitations and assumptions. Through comparing the results from different techniques, the overall structural performance of the optimized frame could be verified and better understood.
Chapter 5
Conclusion
Seismic design optimization

High-rises are becoming increasingly popular due to the many benefits they offer. However, there are unique challenges when building them. These challenges include how to build them sustainably, both in the environmental sense and in an economic sense, and the extreme loading conditions to which they are exposed. These concerns are amplified in seismic regions, which require additional considerations. One form of advancement that has the potential to address these challenges is topology optimization. However, the optimization of real-life structures under seismic design code provisions can be a challenging process due to the presence of multiple highly nonlinear constraints and discrete design variables. One recent attempt to use topology optimization for seismic design, known as the proposed sum of modal compliances, has been presented recently by researcher, Amory Martin (2020). Nevertheless, his research is limited, as the framework assumes a linear response and utilizes the idealized Chilean design spectrum. The focus of this work is to expand upon this method, modifying the framework to generate an optimized lateral frame system for a tall building from a response spectrum curve rather than a design spectrum. Using earthquakes from different regions, optimized bracing systems were produced for various heights and core properties. The optimized layouts were then verified for constructability through a nonlinear analysis.

As expected, optimized layouts varied in response to the exposed seismic region. Regardless of building dimension, the layouts had a relatively similar configuration within the same region. However, the optimized steel lateral frames for tall buildings showed the greatest change in response to the density of the core within the same region. Additionally, it was observed that the optimized layouts for earthquakes with higher accelerations had thicker members. Despite these differences, a comparison of the optimized layouts from the four selected seismic regions, analyzed using SAP2000, showed that these optimized layouts had the potential to outperform traditional X-bracing observed in tall buildings by at least 15%. This is significant for several reasons. By using topology optimized lateral brace frames for tall buildings under seismic excitation, we could design safer buildings, while also reducing embodied carbon, and maximizing revenue in construction.
References


Sarkisian, Mark P. *Designing tall buildings: Structure as architecture*, 2012.


Appendix A
MATLAB Scripts Utilized

Filename: PolyScript.m

%%%% Parameters
rng(0);

% thickness of domain
PGA = 0.6; T0 = 0.4; k = 1.5; % Response spectrum properties (Soil Type A)
PPGA = 0.6; T0 = 1.2; k = 1.5; % Response spectrum properties (Soil Type B)
Sa = @(T) ResponseSpectrum(T, PGA, k, T0);

%%%% CREATE 'fem' STRUCT

[Node,Element,Supp,Load,Mass] = PolyMesher_YSym(@TallBuildingDomain, 8000, 100);
fem = struct(
    'NNode', size(Node,1), ...
    'NElem', size(Element,1), ...
    'Node', Node, ...
    'Element', Element, ...
    'Supp', Supp, ...
    'Load', Load, ...
    'Nu0', 0.3, ...
    'E0', 2e11*t, ...
    'Reg', 0, ...
    'Mass', Mass, ...
    'rho', 7800*t, ...
    'Ec', 100, ...
);

type = 'SMC';
switch type
    case 'Eigen'
        fem.omega = []; fem.optMode = 1;
    case 'DynComp'
        fem.omega = 5.5; xi = 0.05; omegai = 10;

end
omega_j = 100;
fem.aDamp = [x_i*(2*omega_i*omega_j)/(omega_i+omega_j),

x_i*(2/(omega_i+omega_j))];
case 'TotDynComp'
    fem.Freq = [0, 100];
    x_i = 0.05;
    omega_i = 10;
    omega_j = 100;
    fem.aDamp = [x_i*(2*omega_i*omega_j)/(omega_i+omega_j),

x_i*(2/(omega_i+omega_j))];
    fem.Nint = 11; % number of integration points for Gauss quadrature
    case 'SMC'
        fem.rho = 2300*t;
        % B = 33; dx = 0.12*B; - Values to be changed in correspondance to
        % Earthquake Case study #
        % B = 69; dx = 0.12*B;
        B = 13; dx = 0.12*B;
        centroid = zeros(fem.NElem,2);
        for ele=1:fem.NElem
            centroid(ele,:) = mean(fem.Node(fem.Element{ele},:));
        end
        index_pass = find(centroid(:,1)>-dx/2 & centroid(:,1)<dx/2);
        fem.passive = [fem.passive; index_pass];
end

% ----------------------------------------------- CREATE 'opt' STRUCT
% B = 33; - Base Value Changes
% B = 69;
% B = 13;
R = 0.05*B;
VolFrac = 0.3;
m = @(y)MatIntFnc(y,'SIMP',1);
P = PolyFilter(fem,R);
zIni = VolFrac*ones(size(P,2),1);
zIni(fem.passive) = 1;
opt = struct(
    'zMin',0.0,... % Lower bound for design variables
    'zMax',1.0,... % Upper bound for design variables
    'zIni',zIni,... % Initial design variables
    'MatIntFnc',m,... % Handle to material interpolation fnc.
    'P',P,... % Matrix that maps design to element vars.
    'VolFrac',VolFrac,... % Specified volume fraction constraint
    'Tol',0.01,... % Convergence tolerance on design vars.
    'MaxIter',25,... % Max. number of optimization iterations
    'OCMove',0.2,... % Allowable move step in OC update scheme
    'OCEta',0.5 ... % Exponent used in OC update scheme
);
%opt.VolFrac = dx/B + (1-dx/B)*VolFrac; %Modify VolFrac to account for
%passive domain
% ----------------------------------------------- RUN 'PolyTop'
figure;
T = []; F1 = []; F2 = []; F3 = [];
for penal = [1:4] %Continuation on the penalty parameter
disp(['current p: ', num2str(penal)]);
    opt.MatIntFnc = @(y)MatIntFnc(y,'SIMP', penal);
    if penal>1; fem=rmfield(fem,'F'); end
    [opt.zIni,V,fem] = PolyTop_SMC2(fem,opt);
T = [T fem.T];
F1 = [F1, nonzeros(fem.F(:,1))];
F2 = [F2, nonzeros(fem.F(:,2))];
F3 = [F3, nonzeros(fem.F(:,3))];
end

% %
Filename: PolyTop_SMC2.m

%----------------------------- PolyTop version: 1.1 (Aug13) -----------------------------
% Ref: C Talischi, GH Paulino, A Pereira, IFM Menezes, "PolyTop: A Matlab %
% implementation of a general topology optimization framework using %
% unstructured polygonal finite element meshes", Struct Multidisc Optim, %
% DOI 10.1007/s00158-011-0696-x
% %
% Sum of Modal Compliances SMC2 (Amory Martin, amorym@stanford.edu)
% -------------------------------------------------------------------------

function [z,V,fem] = PolyTop_SMC2(fem,opt)
Iter=0; Tol=opt.Tol*(opt.zMax-opt.zMin); Change=2*Tol; z=opt.zIni; P=opt.P;
[E,dEdy,V,dVdy] = opt.MatIntFnc(P*z);
[FigHandle,FigData] = InitialPlot(fem,V);
while (Iter<opt.MaxIter) && (Change>Tol)
    Iter = Iter + 1; fem.f=[];
    %Compute cost functionals and analysis sensitivities
    [f,dfdE,dfdV,fem,Cm] = ObjectiveFnc(fem,E,V);
    [g,dgdE,dgdV,fem] = ConstraintFnc(fem,E,V,VolFrac);
    %Compute design sensitivities
    dfdz = P'*(dEdy.*dfdE + dVdy.*dfdV);
    dgdz = P'*(dEdy.*dgdE + dVdy.*dgdV);
    %Update design variable and analysis parameters
    [z,Change] = UpdateScheme(dfdz,g,dgdz,z,opt,fem);
    [E,dEdy,V,dVdy] = opt.MatIntFnc(P*z);
    %Output results
    fprintf('It: %i  t Obj: %.3f  t Cm: %.1f  Change: %.3f  Vol: %.3f
n',Iter,f,Cm(1),Change,(g+1)*opt.VolFrac);
    set(FigHandle,'FaceColor','flat','CData',1-V(FigData)); drawnow; hold on;
    plot([0,0],[0,65],'-','linewidth',8,'color',[0 0.4470 0.7410]);
    %Change plot domain to 165, 345 & 65 respectively - Chelsea Medina
end
%------------------------------------------------------------------ OBJECTIVE FUNCTION

function [f,dfdE,dfdV,fem,Cm] = ObjectiveFnc(fem,E,V)
[U,fem] = FEAnalysis(fem,E,V);
f = 0; temp = 0; NModes = fem.NModes; Cm = zeros(NModes, 1);
for ii=1:NModes
    Cm(ii) = dot(fem.F(:,ii),U(:,ii)); % modal compliance
    f = f + dot(fem.F(:,ii),U(:,ii));
    temp = temp + cumsum(-U(fem.i,ii).*fem.k.*U(fem.j,ii));
end
temp = temp(cumsum(fem.ElemNDof.^2));
dfE = [temp(1);temp(2:end)-temp(1:end-1)];

function [g,dgdE,dgdV,fem] = ConstraintFnc(fem,E,V,VolFrac)
if ~isfield(fem,'ElemArea')
fem.ElemArea = zeros(fem.NElem,1);
for el=1:fem.NElem
    vx=fem.Node(fem.Element(el),1); vy=fem.Node(fem.Element(el),2);
    fem.ElemArea(el) = 0.5*sum(vx.*vy([2:end 1])-vy.*vx([2:end 1]));
end
end
g = sum(fem.ElemArea.*V)/sum(fem.ElemArea)-VolFrac;
dgdE = zeros(size(E));
dgdV = fem.ElemArea/sum(fem.ElemArea);

------------------------------ OPTIMALITY CRITERIA UPDATE
function [zNew,Change] = UpdateScheme(dfdz,g,dgdz,z0,opt,fem)
zMin=opt.zMin; zMax=opt.zMax;
move=opt.OCMove*(zMax-zMin); eta=opt.OCEta;
l1=0; l2=1e6;
while 12-l1 > 1e-4
    lmid = 0.5*(l1+l2);
    B = -(dfdz./dgdz)/lmid;
    zCnd = zMin+(z0-zMin).*B.^eta;
    zNew = max(max(min(min(zCnd,z0+move),zMax),z0-move),zMin);
    if (g+dgdz'*zNew>0), l1=lmid; end
    else
        l2=lmid; end
end
Change = max(abs(zNew-z0))/(zMax-zMin);

------------------------------ FE-ANALYSIS
function [U,fem] = FEAnalysis(fem,E,V)

% assemble mass and stiffness matrices
if ~isfield(fem,'m')
    fem.ElemNDof = 2*cellfun(@(x)length(x),fem.Element); % # of DOFs per element
    fem.i = zeros(sum(fem.ElemNDof.^2),1);
    fem.j=fem.i; fem.k=fem.i; fem.e=fem.i; fem.m=fem.i; fem.c=fem.i;
    index = 0;
    if ~isfield(fem,'ShapeFnc'), fem=TabShapeFnc(fem); end
    for el = 1:fem.NElem
        Ke=LocalK(fem,fem.Element{el});
        if ismember(el,fem.passive); Ke = fem.Ec*Ke; end
        Me=LocalM(fem,fem.Element{el});
        NDof = fem.ElemNDof(el);
        eDof = reshape([2*fem.Element{el}-1;2*fem.Element{el}],NDof,1);
        I repmat(eDof,1,NDof); J=I';
        fem.i(index+1:index+NDof^2) = I(:);
        fem.j(index+1:index+NDof^2) = J(:);
        fem.k(index+1:index+NDof^2) = Ke(:);
        fem.m(index+1:index+NDof^2) = Me(:);
        fem.e(index+1:index+NDof^2) = el;
        index = index + NDof^2;
    end
    NSupp = size(fem.Supp,1);
    FixedDofs = [fem.Supp(1:NSupp,2).*(2*fem.Supp(1:NSupp,1)-1);
                 fem.Supp(1:NSupp,3).*(2*fem.Supp(1:NSupp,1))];
    FixedDofs = FixedDofs(FixedDofs>0);
    AllDofs = [1:2*fem.NNode];
    fem.FreeDofs = setdiff(AllDofs,FixedDofs);
end
K = sparse(fem.i,fem.j,E(fem.e).*fem.k);
K = (K+K')/2;

% lumped masses
M = sparse(fem.i,fem.j,V(fem.e).*fem.m);
if ~isempty(fem.Mass)
NMass = size(fem.Mass,1);
sumM = sum(M(:)); % total mass of domain
MassDofsX = 2*fem.Mass(1:NMass,1)-1; %x-direction
MassDofsY = 2*fem.Mass(1:NMass,1); %y-direction
end
M = (M+M')/2;

% eigenvalue analysis
if ~isfield(fem, 'F')
    fprintf('Mass tributary ratio: %.3f \n', 1-sumM/sum(M(:)));
    fem = EigenAnalysis(fem,M,K);
    fprintf('Eigen analysis: T:%.2f, %.2f, %.2f s \n',fem.T(1:3));
end

% equivalent modal static analysis
U = zeros(2*fem.NNode,fem.NModes);
dofs = fem.FreeDofs(fem.FreeDofs>0);
U(dofs,:) = K(dofs,dofs)
        fem.F(dofs,:);
fem.U = U;

%--------------------------------------------------------- EIGENVALUE ANALYSIS
function [fem] = EigenAnalysis(fem,M,K)

% nodal masses (x-direction only)
NModes = fem.NModes;
NMass = size(fem.Mass,1);
MassDofs = 2*fem.Mass(1:NMass,1)-1; % x-dof
%MassDofsY = 2*fem.Mass(1:NMass,1); % y-dof
mm = diag(diag(M(MassDofs, MassDofs)));
freedofs = fem.FreeDofs(fem.FreeDofs>0);

% eigenvalue analysis
Phi = zeros(fem.NNode*2, NModes);
switch fem.analysis
    case 'eigen'
        [V,D] = eigs(K(freedofs,freedofs),M(freedofs,freedofs),NModes,'SM');
        Phi(freedofs,:) = flip(V, 2);
        Phi = Phi/max(abs(Phi(:,1))); % normalize by first mode peak roof
        T = 2*pi./sqrt(flip(diag(D)));
    case 'static'
        T = [3.58, 0.61, 0.24, 0.13, 0.09];
        Phi(MassDofs,:) = cantileverModes(NMass,NModes,0.5,false);
end
Lh = Phi'*M*ones(fem.NNode*2,1);
Mh = diag(Phi'*M*Phi);
Gamma = (Lh./Mh);

% equivalent modal vectors
F = zeros(2*fem.NNode,NModes);
Sa = fem.SaFnc(T); % Response spectrum function
for i=1:NModes
    F(MassDofs,i) = Gamma(i)*mm*Phi(MassDofs,i)*Sa(i)*fem.g;
end
F = F/1000; % scale for optimization convergence
% store values
fem.T = T;
fem.Phi = Phi;
fem.Gamma = Gamma;
fem.F = sparse(F);

%---------------------------------------------------------------
% ELEMENT STIFFNESS MATRIX
% function [Ke] = LocalK(fem,eNode)
D=fem.E0/(1-fem.Nu0^2)*[1 fem.Nu0 0;fem.Nu0 1 0 0 0 (1-fem.Nu0)/2];
nn=length(eNode); Ke=zeros(2*nn,2*nn);
W = fem.ShapeFnc{nn}.W;
for q = 1:length(W)  % quadrature loop
  dNdxi = fem.ShapeFnc{nn}.dNdxi(:,:,q);
  J0 = fem.Node(eNode,:)'*dNdxi;
  dNdxi = dNdxi/J0;
  B(1,1:2:2*nn) = dNdxi(:,1)';
  B(2,2:2:2*nn) = dNdxi(:,2)';
  B(3,1:2:2*nn) = dNdxi(:,2)';
  B(3,2:2:2*nn) = dNdxi(:,1)';
  Ke = Ke+B'*D*B*W(q)*det(J0);
end
%---------------------------------------------------------------
% ELEMENT MASS MATRIX
% function [Me] = LocalM(fem,eNode)  % Amory edit 7/6/18 (mass matrix)
nn = length(eNode);
Me = zeros(2*nn,2*nn);
W = fem.ShapeFnc{nn}.W;
for q=1:length(W)
  Ni = fem.ShapeFnc{nn}.N(:,:,q);
  dNdxi = fem.ShapeFnc{nn}.dNdxi(:,:,q);
  J0 = fem.Node(eNode,:)'*dNdxi;
  N(1,1:2:2*nn) = Ni;
  N(2,2:2:2*nn) = Ni;
  Me = Me + fem.rho*(N'*N)*W(q)*det(J0);
end
%---------------------------------------------------------------
% TABULATE SHAPE FUNCTIONS
% function fem = TabShapeFnc(fem)
ElemNNode = cellfun(@length,fem.Element);  % number of nodes per element
fem.ShapeFnc = cell(max(ElemNNode),1);
for nn = min(ElemNNode):max(ElemNNode)
  [W,Q] = PolyQuad(nn);
  fem.ShapeFnc{nn}.W = W;
  fem.ShapeFnc{nn}.N = zeros(nn,1,size(W,1));
  fem.ShapeFnc{nn}.dNdxi = zeros(nn,2,size(W,1));
  for q = 1:size(W,1)
    [N,dNdxi] = PolyShapeFnc(nn,Q(q,:));
    fem.ShapeFnc{nn}.N(:,:,q) = N;
    fem.ShapeFnc{nn}.dNdxi(:,:,q) = dNdxi;
  end
end
%---------------------------------------------------------------
% POLYGONAL SHAPE FUNCTIONS
% function [N,dNdxi] = PolyShapeFnc(nn,xi)
N=zeros(nn,1); alpha=zeros(nn,1); dNdxi=zeros(nn,2); dalpha=zeros(nn,2);
sum_alpha=0.0; sum_dalpha=zeros(1,2); A=zeros(nn,1); dA=zeros(nn,2);
\[ [p, Tri] = \text{PolyTrnglt}(nn, xi); \]
for \( i = 1:nn \)
    \( scTr = \text{Tri}(i,:); pT = p(scTr,:); \)
    \( A(i) = 1/2 \cdot \det([pT, \text{ones}(3,1)]); \)
    \( dA(i,1) = 1/2 \cdot (pT(3,2) - pT(2,2)); \)
    \( dA(i,2) = 1/2 \cdot (pT(2,1) - pT(3,1)); \)
end
\( A = [A(nn,:); A]; dA = [dA(nn,:); dA]; \)
for \( i = 1:nn \)
    \( \alpha(i) = 1/(A(i) \cdot A(i+1)); \)
    \( d\alpha(i,1) = -\alpha(i) \cdot (dA(i,1)/A(i) + dA(i+1,1)/A(i+1)); \)
    \( d\alpha(i,2) = -\alpha(i) \cdot (dA(i,2)/A(i) + dA(i+1,2)/A(i+1)); \)
end
\( \sum_\alpha = \sum_\alpha + \alpha(i); \)
\( \sum_d\alpha(1:2) = \sum_d\alpha(1:2) + d\alpha(i,1:2); \)
end

\%----------------------------------------------------
\% POLYGON TRIANGULATION
\%----------------------------------------------------
function \([p, Tri] = \text{PolyTrnglt}(nn, xi)\)
\( p = [\cos(2 \cdot \pi \cdot ((1:nn))/nn); \sin(2 \cdot \pi \cdot ((1:nn))/nn)]; \)
\( p = [p; xi]; \)
\( Tri = \text{zeros}(nn, 3); Tri(1:nn, 1) = nn + 1; \)
\( Tri(1:nn, 2) = 1:nn; Tri(1:nn, 3) = 2:nn + 1; Tri(nn, 3) = 1; \)
\%----------------------------------------------------
\% POLYGONAL QUADRATURE
\%----------------------------------------------------
function \([weight, point] = \text{PolyQuad}(nn)\)
\( [W, Q] = \text{TriQuad}; \) %integration pts & wgts for ref. triangle
\( [p, Tri] = \text{PolyTrnglt}(nn, [0 0]); \) %triangulate from origin
\( \text{point} = \text{zeros}(nn \cdot \text{length}(W), 2); \)
\( \text{weight} = \text{zeros}(nn \cdot \text{length}(W), 1); \)
for \( k = 1:nn \)
    \( scTr = \text{Tri}(k,:); \)
    for \( q = 1: \text{length}(W) \)
        \( \text{[N, dNds]} = \text{TriShapeFnc}(Q(q,:)); \) %compute shape functions
        \( J0 = p(scTr,:)'' \cdot \text{dNds}; \)
        \( l = (k-1) \cdot \text{length}(W) + q; \)
        \( \text{point}(l,:) = N'' \cdot p(scTr,:); \)
        \( \text{weight}(l) = \det(J0) \cdot \text{W}(q); \)
    end
end
\%----------------------------------------------------
\% TRIANGULAR QUADRATURE
\%----------------------------------------------------
function \([weight, point] = \text{TriQuad}\)
\( \text{point} = [1/6,1/6;2/3,1/6;1/6,2/3]; \)
\( \text{weight} = [1/6,1/6,1/6]; \)
\%----------------------------------------------------
\% TRIANGULAR SHAPE FUNCTIONS
\%----------------------------------------------------
function \([N, dNds] = \text{TriShapeFnc}(s)\)
\( N = [1-s(1)-s(2); s(1); s(2)]; \)
\( \text{dNds} = [-1,-1;0,0,1]; \)
\%----------------------------------------------------
\% INITIAL PLOT
\%----------------------------------------------------
function \([\text{handle}, map] = \text{InitialPlot}(fem, z0)\)
\( \text{Tri} = \text{zeros}([\text{fem.Element(:)}] - 2 \cdot \text{fem.NEleml, 3}); \)
\( \text{map} = \text{zeros}(\text{size}((\text{Tri}, 1), 1)); \)
\( \text{index} = 0; \)
for \( \text{el} = 1: \text{fem.NEleml} \)
    \( \text{for enode} = 1: \text{length}( \text{fem.Element{el}}) - 2 \)
        \( \text{map}(\text{index} + 1) = \text{el}; \)
        \( \text{Tri}(\text{index} + 1,:) = \text{fem.Element{el}}([1, \text{enode} + 1, \text{enode} + 2]); \)
        \( \text{index} = \text{index} + 1; \)
    end
end
handle = patch('Faces',Tri,'Vertices',fem.Node,'FaceVertexCData',... 1-z0(map), 'FaceColor', 'flat', 'EdgeColor', 'none');
axis equal; axis off; axis tight; colormap(gray); caxis([0 1]);
%---------------------------------------------------------------------------------- DISPLACEMENTS PLOT

function [handle,map] = DisplacementPlot(fem,z0,U)
Tri = zeros(length({fem.Element{:}})-2*fem.NElem,3);
map = zeros(size(Tri,1),1); index=0;
for el = 1:fem.NElem
    for enode = 1:length(fem.Element{el})-2
        map(index+1) = el;
        Tri(index+1,:) = fem.Element{el}([1,enode+1,enode+2]);
        index = index + 1;
    end
end
Ursa = sqrt(sum(U.^2, 2));
Ux = Ursa(1:2:end); Uy = Ursa(1:2:end); scale = 1000;
Ux = scale*Ux;
z0 = opt.P*z0;
dispE = zeros(fem.NElem, 1);
for el = 1:fem.NElem
    nodes = fem.Element{el};
    %if z0(el) < 0.01
    %dispE(el) = 0;
    dispE(el) = mean(Ux(nodes))*z0(el);
end
handle = patch('Faces',Tri,'Vertices',fem.Node,'FaceVertexCData',... 1-dispE(map), 'FaceColor', 'flat', 'EdgeColor', 'none'); hold on;
axis equal; axis off; colormap('jet'); caxis([0, 1]);
%colormap(gray); caxis([0 1]);
%---------------------------------------------------------------------------------- PolyTop - History --------------------------
% version: 1.1 (Aug13)
% history: Created:  8-Jan-12   Anderson Pereira & Cameron Talischi
%          Supervised by:         Ivan Menezes & Glaucio Paulino
% Modified: 16-Jul-13  Tomas Zegard
%            Fix the color axis scaling by calling caxis([0 1]);
% Modified: 16-Aug-13  Anderson Pereira
%            Removed one line of code by collapsing two "if's" for checking
%            if the element is constant or not (fem.Reg flag)
% Modified: 5-Feb-17  Amory Martin
%            Added multiple load cases functionality.
% Modified: 10-Feb-23  Chelsea Medina
%            Changed plot function in correspondence to new earthquake domain

%----------------------------------------------------------------------------------
Filename: ResponseSpectrum.m

%-------------------------------------------------------- RESPONSE SPECTRUM
% based on Chilean standard NCH433 (1996)
function [Sa, dSadT] = ResponseSpectrum(T, PGA, To, k)

% DATA to be imported set up
%     TABLE = readtable('small_Data.csv');
%     x = table2array(TABLE(:,1));
%     y = table2array(TABLE(:,2));
%     p = polyfit(x,y,10);
%     TABLE = readtable('Northridge1.csv');
%     x = table2array(TABLE(:,1));
%     y = table2array(TABLE(:,2));
%     p = polyfit(x,y,18);
%     TABLE = readtable('Alaska Denali.csv');
%     x = table2array(TABLE(:,1));
%     y = table2array(TABLE(:,2));
%     p = polyfit(x,y,25);
%     TABLE = readtable('Saguenay.csv');
%     x = table2array(TABLE(:,1));
%     y = table2array(TABLE(:,2));
%     p = polyfit(x,y,24);

% CALCULATION Modified from Chilean Idealized Curve to a better rep. curve
%     x_min = 0.32;
%     x_max = 4.0;
%     x_range = x(x >= x_min & x <= x_max); % limit x values to the desired range
%     Sa = polyval(p, x_range); % evaluate the polynomial using only the desired x values
%     dSadT = polyval(polyder(p), x_range); % evaluate the derivative of the polynomial using only the desired x values

Sa = polyval(p,T);
    %Sa = PGA*(1+4.5*(T/To).^k)./(1+(T/To).^3);
dp = polyder(p);
DdSadT = polyval(dp,T);
    %dSadT = PGA*((9*k*(T/To).^k - 1))/(2*(T.^3/To^3 + 1)) - ...
        % (3*T.^2.*((9*(T./To).^k)/2 + 1))/(To^3*(T.^3/To^3 + 1).^2));
end
%Modification History
% Modified Feb-2023 Chelsea Medina
function [x] = TallBuildingDomain(Demand,Arg)
% B = 33; - Domain Modifications for Case Study
% B = 69;
B = 13;
BdBox = [-B/2 B/2 0 5*B];
switch (Demand)
  case ('Dist');  x = DistFnc(Arg,BdBox);
  case ('BC');    x = BndryCnds(Arg{:},BdBox);
  case ('BdBox'); x = BdBox;
  case ('PFix');  x = FixedPoints(BdBox);
end
% --------------------------------------------- COMPUTE DISTANCE FUNCTIONS
function Dist = DistFnc(P,BdBox)
Dist = dRectangle(P,BdBox(1),BdBox(2),BdBox(3),BdBox(4));
% --------------------------------------------- SPECIFY BOUNDARY CONDITIONS
function [x] = BndryCnds(Node,Element,BdBox)
eps = 0.1*sqrt((BdBox(2) - BdBox(1))*(BdBox(4) - BdBox(3))/size(Node,1));
MassNodes = find(abs(Node(:,1))<eps);
BottomMidNode = find(abs(Node(:,1))<eps & abs(Node(:,2))<eps);
BottomNodes = find(abs(Node(:,2))<eps);
FixedNodes = BottomNodes;
  % Supports
  Supp = zeros(length(FixedNodes),3);
  Supp(:,1)=FixedNodes; Supp(:,2:3)=1;
  Supp = repmat(Supp,1,1,3);
  % Wind
  % Load = [MassNodes(1:end-2), 1000*ones(2*n-2,1), zeros(2*n-2,1);
  %         MassNodes(end-1:end), [500;500], [0;0]];
  % ELF Mode 1
  % Load = [MassNodes(1:2:end), (1:n)'*1000, zeros(n,1);
  %         MassNodes(2:2:end), (1:n)'*1000, zeros(n,1)];

  nmass = length(MassNodes);
  Load = [BottomMidNode, 1, 0; MassNodes, ones(nmass, 1), zeros(nmass, 1)];
  % Load = [MassNodes(1), 1000, 0];
  g = 9.81;
  mtrib = (40*4448*1000/g)/nmass;  %(1000 kips/floor)
  Mass = [MassNodes, mtrib*ones(nmass,1), zeros(nmass,1)];
  % Mass = [];
  x = {Supp,Load,Mass};
% --------------------------------------------- SPECIFY FIXED POINTS
function [PFix] = FixedPoints(BdBox)
n = BdBox(4);
nx = 40;
PFix = [zeros(nx+1,1), linspace(0,n,nx+1)'];
%----------------------------------------------------------
% ELEMENT MASS MATRIX
% Suppl. Func Created by A.Martin, 2020

function [Me] = LocalM(fem, eNode)
nn = length(eNode);
Me = zeros(2*nn, 2*nn);
W = fem.ShapeFnc{nn}.W;
for q=1:length(W)
    Ni = fem.ShapeFnc{nn}.N(:, :, q);
    dNdxi = fem.ShapeFnc{nn}.dNdxi(:, :, q);
    J0 = fem.Node(eNode, :)'*dNdxi;
    N = zeros(2, 2*nn);
    N(1,1:2:2*nn) = Ni;
    N(2,2:2:2*nn) = Ni;
    Me = Me + fem.rho*(N'*N)*W(q)*det(J0);
end
function [U,fem] = FEAnalysis(fem,E,V)

% assemble mass and stiffness matrices
if ~isfield(fem,'m')
    fem.ElemNDof = 2*cellfun(@(length,fem.Element); % # of DOFs per element
    fem.i = zeros(sum(fem.ElemNDof.^2),1);
    fem.j=fem.i; fem.k=fem.i; fem.e=fem.i; fem.m=fem.i; fem.c=fem.i;
    index = 0;
    if ~isfield(fem,'ShapeFnc'), fem=TabShapeFnc(fem); end
    for el = 1:fem.NElem
        if ~fem.Reg || el==1,
            Ke=LocalK(fem,fem.Element{el});
            Ke = fem.Ec*Ke;
        end
        Me=LocalM(fem,fem.Element{el});
        NDof = fem.ElemNDof(el);
        eDof = reshape([2*fem.Element{el}-1;2*fem.Element{el}],[NDof,1]);
        I repmat(eDof ,1,NDof); J=I';
        fem.i(index+1:index+NDof^2) = I(:);
        fem.j(index+1:index+NDof^2) = J(:);
        fem.k(index+1:index+NDof^2) = Ke(:);
        fem.m(index+1:index+NDof^2) = Me(:); % store element mass matrices
        fem.e(index+1:index+NDof^2) = el;
        index = index + NDof^2;
    end
    NSupp = size(fem.Supp,1);
    FixedDofs = [fem.Supp(1:NSupp,2).*(2*fem.Supp(1:NSupp,1)-1);
        fem.Supp(1:NSupp,3).*(2*fem.Supp(1:NSupp,1))];
    FixedDofs = FixedDofs(FixedDofs>0);
    AllDofs = [1:2*fem.NNode];
    fem.FreeDofs = setdiff(AllDofs,FixedDofs);
end
K = sparse(fem.i,fem.j,E(fem.e).*fem.k);
K = (K+K')/2;

% mass matrix assembly
M = sparse(fem.i,fem.j,V(fem.e).*fem.m);
if ~isempty(fem.Mass) % tributary masses
    NMass = size(fem.Mass,1);
sumM = sum(M(:)); % total mass of domain
MassDofsX = 2*fem.Mass(1:NMass,1) - 1; % x-direction
MassDofsY = 2*fem.Mass(1:NMass,1); % y-direction
end
M = (M+M')/2;

% eigenvalue analysis
if ~isfield(fem, 'F')
    fprintf('Mass tributary ratio: %.3f \n', 1-sumM/sum(M(:)));
    fem = EigenAnalysis(fem, M, K);
    fprintf('Eigen analysis: T:%.2f, %.2f, %.2f s \n', fem.T(1:3));
end

% modal response spectrum analysis
U = zeros(2*fem.NNode, fem.NModes);
dofs = fem.FreeDofs(fem.FreeDofs>0);
U(dofs,:) = K(dofs,dofs)em.F(dofs,:);
fem.U = U;
Filename: EigenAnalysis.m

% Supp. Function Created by A. Martin, 2020  
function [fem] = EigenAnalysis(fem,M,K)

% nodal masses
NModes = fem.NModes;
NMass = size(fem.Mass,1);  % tributary floor masses
MassDofs = 2*fem.Mass(1:NMass,1)-1;  % x-dof
%MassDofsY = 2*fem.Mass(1:NMass,1);  % y-dof
mm = diag(diag(M(MassDofs, MassDofs)));
freedofs = fem.FreeDofs(fem.FreeDofs>0);

% eigenvalue analysis
Phi = zeros(fem.NNode*2, NModes);  % eigenvector
[V,D] = eigs(K(freedofs,freedofs),M(freedofs,freedofs),NModes,'SM');
Phi(freedofs,:) = flip(V, 2);
Phi = Phi/max(abs(Phi(:,1)));  % normalize by first mode peak roof
T = 2*pi./sqrt(flip(diag(D)));  % periods of vibration
Lh = Phi'*M*ones(fem.NNode*2,1);
Mh = diag(Phi'*M*Phi);
Gamma = (Lh./Mh);  % modal participation factors

% equivalent modal vectors
F = zeros(2*fem.NNode,NModes);
Sa = fem.SaFnc(T);  % Response spectrum function
for i=1:NModes
    F(MassDofs,i) = Gamma(i)*mm*Phi(MassDofs,i)*Sa(i)*fem.g;
end

% store values
fem.T = T;
fem.Phi = Phi;
fem.Gamma = Gamma;
fem.F = sparse(F);
Filename: PolyFit.m
% Earthquake idealized plot data
% File created by Chelsea Medina to help determine degree of curve

TABLE = readtable('Saguenay.csv');
x = table2array(TABLE(:,1));
y = table2array(TABLE(:,2));
p = polyfit(x,y,24);

% CALCULATION

yp = polyval(p,x);

% plot(x,y)
% hold on
% plot(x,yp)

plot(x,y)
hold on
plot(x(x>=0 & x<=4),yp(x>=0 & x<=4))
Appendix B
Earthquake Graphs

1. Loma Prieta Seismic Event affecting San Francisco Bay Area, 1989

2. Northridge Seismic Event affecting Southern California Region, 1994
3. Denali Earthquake affecting interior Alaska, 2002

4. Saguenay Earthquake affecting Quebec, Canada & Northeastern United States, 1988
Appendix C
Optimized Topologies from MATLAB

1. Earthquake: Loma Prieta, 1989
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

2. Earthquake: Loma Prieta, 1989
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:
3. Earthquake: Loma Prieta, 1989
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:

4. Earthquake: Loma Prieta, 1989
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:
5. Earthquake: Loma Prieta, 1989
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:

   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:
7. Earthquake: Loma Prieta, 1989  
   Height: 345 m  
   Base: 69 m  
   Ratio of Core Density, $E_c/E$: 10  
   MATLAB Output:

8. Earthquake: Loma Prieta, 1989  
   Height: 345 m  
   Base: 69 m  
   Ratio of Core Density, $E_c/E$: 50  
   MATLAB Output:
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:

10. Earthquake: Northridge Earthquake, 1994
    Height: 65 m
    Base: 13 m
    Ratio of Core Density, $E_c/E$: 10
    MATLAB Output:
11. Earthquake: Northridge Earthquake, 1994
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:

12. Earthquake: Northridge Earthquake, 1994
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:
15. Earthquake: Northridge Earthquake, 1994
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 100

![Diagram of building with height and base dimensions]

   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

![Diagram of building with height and base dimensions]
17. Earthquake: Northridge Earthquake, 1994
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:

18. Earthquake: Northridge Earthquake, 1994
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

20. Earthquake: Denali Earthquake, 2002
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:

22. Earthquake: Denali Earthquake, 2002
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:
23. Earthquake: Denali Earthquake, 2002
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, \( E_c/E \): 50
   MATLAB Output:

24. Earthquake: Denali Earthquake, 2002
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, \( E_c/E \): 100
   MATLAB Output:
25. Earthquake: Denali Earthquake, 2002
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 50
27. Earthquake: Denali Earthquake, 2002
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:

   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:
29. Earthquake: Saguenay Earthquake, 1988
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:

30. Earthquake: Saguenay Earthquake, 1988
   Height: 65 m
   Base: 13 m
   Ratio of Core Density, $E_c/E$: 100
   MATLAB Output:
31. Earthquake: Saguenay Earthquake, 1988
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:

32. Earthquake: Saguenay Earthquake, 1988
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 50
33. Earthquake: Saguenay Earthquake, 1988
   Height: 165 m
   Base: 33 m
   Ratio of Core Density, $E_c/E$: 100

34. Earthquake: Saguenay Earthquake, 1988
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 10
   MATLAB Output:
35. Earthquake: Saguenay Earthquake, 1988
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 50
   MATLAB Output:
   
   36. Earthquake: Saguenay Earthquake, 1988
   Height: 345 m
   Base: 69 m
   Ratio of Core Density, $E_c/E$: 100
Appendix D
SAP2000 Models

1. Response Spectrum for optimized brace lateral framing system for a 165 m high-rise with a 50-density core ratio subjected to Loma Prieta.

2. Response Spectrum for traditional braced lateral framing system for a 165 m high-rise with a 50-density core ratio subjected to Loma Prieta.
3. Response Spectrum for optimized lateral bracing framing system for a 165 m high-rise with a 50-density core ratio subjected to Saguenay Earthquake

4. Response Spectrum for Traditional lateral bracing framing system for a 165 m high-rise with a 50-density core ratio subjected to Saguenay Earthquake
5. Response Spectrum for optimized lateral bracing framing system for a 165 m high-rise with a 50-density core ratio subjected to Northridge Earthquake

6. Response Spectrum for traditional lateral bracing framing system for a 165 m high-rise with a 50-density core ratio subjected to Northridge Earthquake