The Thesis Committee for Umit Can Oksuz
Certifies that this is the approved version of the following thesis:

Simplified Modeling for Assessing Collapse Resistance of Steel Gravity Frames with Composite Floor Systems

APPROVED BY
SUPERVISING COMMITTEE:

Eric B. Williamson, Supervisor

Michael D. Engelhardt

Simplified Modeling for Assessing Collapse Resistance of Steel Gravity Frames with Composite Floor Systems

by

Umit Can Oksuz, BS, BS

Thesis

Presented to the Faculty of the Graduate School of

The University of Texas at Austin

in Partial Fulfillment

of the Requirements

for the Degree of

Master of Science in Engineering

The University of Texas at Austin

May 2014

Dedication

Dedicated to my family.

Acknowledgements

First of all, I would like to thank my family, Hayal Oksuz, Nalan Golal, Gamze Golal, Zarife Golal, and Yakup Golal, for the constant support and guidance they provided me throughout my life. They always believed me, supported me in toughest times, and helped me to achieve my highest goals. They spent so much effort on making me the person I am proud to be today. I couldn't have gotten this far without their help.

This project is supported by the Science & Technology Directorate, U.S. Department of Homeland Security under Award Number: 2010-ST-108-000014. I want to thank the Department of Homeland Security (DHS) for their support of this research program. Their support is greatly appreciated.

Finally, I want to thank my advisor, Dr. Eric B. Williamson, for his supervision and guidance throughout the research. Thank you for inviting me to participate in this research project, which I've learned and enjoyed so much, and assigning me as a teaching assistant for the first year engineering course. Also, I want to thank Dr. Engelhardt for his support and guidance. He has spent so much time discussing my research with me. I am grateful to my professors for sharing their structural engineering knowledge and teaching me as much as possible on various topics.

May, 2014

Abstract

Simplified Modeling for Assessing Collapse Resistance of Steel Gravity Frames with Composite Floor Systems

Umit Can Oksuz, MSE

The University of Texas at Austin, 2014

Supervisor: Eric B. Williamson

Progressive collapse is a structural failure that is initiated by the failure of a primary structural member due to manmade or natural reasons and causes a disproportionately large portion of the structure to damage and/or collapse. This thesis is focused on the computational assessment of the performance of steel gravity frames with composite floor systems under column loss scenarios. The ultimate goal is to provide step-by-step guidance to practicing civil/structural engineers on modeling and analyzing full-size structures by using simple structural analysis software with the purpose of determining progressive collapse resistance.

In this research project, a steel frame structure with simple framing connections and a composite floor system was tested, modeled, and analyzed under an interior column loss scenario. For the computational analysis part of the research, a simplified modeling approach was developed and verified by comparing the analysis results with detailed finite element model results and available experimental data. Next, the test specimen was

modeled with the proposed approach using the SAP2000 software, and an analysis was performed. Results of the analysis were compared with the test data to verify that the model accurately simulates the measured behavior of the structure. In the end, it was concluded that steel gravity frame structures with composite floor systems can be accurately simulated by using the proposed simplified modeling approach up to the point of first element failure. Moreover, it was shown that practicing civil/structural engineers can do quick and simple checks for their structure's ability to resist progressive collapse by using the methods and approaches that are described in this thesis.

Table of Contents

List of Tables

List of Figures

CHAPTER 1

Introduction

1.1 OVERVIEW

This thesis focuses on the computational analysis portion of a detailed and comprehensive research program, which is officially titled "Keeping Our Structures Standing and Our People Alive: The Next 25 Years". The research program is funded by the Department of Homeland Security, and the primary objective is to investigate the progressive collapse resistance of steel structures both experimentally and computationally. This project is implemented through a collaborative research team including personnel from The University of Texas at Austin, Imperial College of London, Walter P. Moore consulting structural engineers, and Protection Engineering Consultants.

The definition of progressive collapse has varied with time and location. First, if we review the definition in the United Kingdom Building Regulations, the following requirement has been given for disproportionate collapse: "The building shall be constructed so that in the event of an accident the building will not suffer collapse to an extent disproportionate to the cause." It should be noted that British Standards use the term *disproportionate collapse* instead of *progressive collapse*. Second, in 1975, the National Building Code of Canada stated that "Progressive collapse is the phenomenon in which the spread of an initial local failure from element to element eventually results in collapse of a whole building or disproportionately large parts of it." In 1990, this statement was changed to "Structural integrity is defined as the ability of the structure to absorb local failure without widespread collapse." These examples illustrate how the definition of progressive collapse has evolved with time and how it differs by location. It should be noted that Canadian Standard used the term *structural integrity* instead of *progressive collapse* in 1990. For the purpose of this thesis, the American Society of Civil Engineers Standard *Minimum Design Loads for Buildings and Other Structures* (ASCE 7-10) definition is accepted as a reference point, which states that "Progressive collapse is defined as the spread of an initial local failure from element to element, resulting eventually in the collapse of an entire structure or a disproportionately large part of it."

The first U.S. consensus document that added discussion on progressive collapse was ANSI A58.1 (1972), which was prepared four years after the Ronan Point Apartment collapse. This 22-story high tower block in Newham, UK partly collapsed after a gas stove explosion in 1968 and made structural integrity a significant concern (BBC News, 16 May 1968). ANSI A58.1 (1972) included a brief general statement about progressive collapse and used the terms *structural integrity, overloads and abnormal loads*, which approached progressive collapse as an extraordinary loading problem. In 1999, after the Oklahoma City bombing and collapse of the Alfred P. Murrah Federal Building in 1995, the U.S. Department of Defense released *Interim Department of Defense Antiterrorism / Force Protection Construction Standards* (1999) to reduce the risk of mass causalities and to change the direction of the requirements to Anti-terrorism criteria. Although, the document was not threat specific, the initiating event was assumed to be an explosive attack. Finally, the terrorist attack on the World Trade Center towers on September 11, 2001 directed public attention toward the topic of progressive collapse and spurred on significant research interest. Current U.S. Department of Defense guidelines in Unified Facilities Criteria *Design of Buildings to Resist Progressive Collapse* (UFC 4-023-03,

2010) define progressive collapse consistently with the commentary of ASCE 7-10 as described previously. The guideline refers to ASCE 7 and states that "except for specially designed protective systems, it is usually impractical for a structure to be designed to resist general collapse caused by severe abnormal loads acting directly on a large portion of it. However, structures can be designed to limit the effects of local collapse and to prevent or minimize progressive collapse." Accordingly, the main goal of the guideline is to limit the distribution of local damage/collapse, and to stop or minimize progressive collapse by ensuring redundancy and continuity in structural systems.

The primary objective of all the standards mentioned above is to ensure that structures have sufficient progressive collapse resistance. Otherwise, a structure can experience a partial or total collapse that is disproportionate to the initiating event. Although, progressive collapse is a complicated dynamic phenomenon, mitigation of failure can be achieved by providing redundancy and local resistance to structural components. An effective way to provide redundancy is to provide alternative ways to redistribute load. By this method, the progressive collapse risk can be minimized because loads on a failed member can be carried by the remaining members. Further, designing the critical locations of a structure for enhanced capacity provides local resistance that can prevent local failure. Thus, preventing a potential initiating event from occurring, global collapse can be avoided. In addition to these approaches, ASCE 7-10 Section C1.4 states that providing better resistance for uplift loads, not relying on transfer girders for support of upper floors, and considering greater ductility, redundancy and continuity in the design of connections between structural components, columns, and beams can decrease damage, increase energy absorption, and allow redistribution of loads.

Although, there are a variety of finite element computer programs available for conducting high-fidelity simulations of structural response, this thesis presents the computational assessment of the behavior of steel structures under progressive collapse and column removal scenarios using the SAP 2000 analysis software (Computers and Structures, Inc., 2013). A simplified modeling approach is used throughout the thesis, while comparisons with detailed finite element analysis results and available experimental data are presented to establish confidence in the simplified models.

Unlike much past progressive collapse research, the current study does not focus on individual components of a structural system. Rather, the experimental test data presented in this thesis has been obtained from a three-dimensional steel structure, which was tested under pseudo-static internal and external column removal scenarios at the Ferguson Structural Engineering Laboratory of University of Texas at Austin. This structure was constructed with all structural components that an actual floor system would have, including a composite floor slab with corrugated decking, shear studs, connection bolts, shear tabs, double angle connections, and reinforcement bars. This unique test data is also helpful for understanding three-dimensional response of actual structures.

The results of various portions and tasks of this detailed and comprehensive research program are documented throughout this thesis. In addition, other students working on the project have developed a series of MS theses and PhD dissertations that are referenced throughout the current document. This thesis focuses primarily on the development of simplified computational models for simulating collapse of typical steel-framed structures.

1.2 RESEARCH PROJECT SIGNIFICANCE

Progressive collapse can cause significant mass casualties in a very short period of time, although it is a rare event in the United States and Western Europe. For instance, 168 people died during the collapse of the Alfred P. Murrah Federal Building in 1995 (BBC News, 19 April 1995), and almost 3,000 people died during the collapse of World Trade Center Towers in 2001 (BBC News, 11 September 2001). Many of those casualties were due to the progressive collapse of the structure. Today, terrorist threats to the United States are higher than before (The Guardian News, 1 December 2013), and there is concern about an increasing risk of progressive collapse for US government structures due to bombing and other violent methods of attack. Occupants of these structures can be protected by slowing or preventing total structural collapse, which requires a better understanding of the progressive collapse phenomenon.

There are number of factors that contribute to the risk of damage propagation in modern structures (Breen 1976). The first factor listed in ASCE 7-10 Section C1.4 is the lack of general awareness among engineers that structural integrity against collapse is important enough to be regularly considered in design. By using the simple and tested methods provided in this thesis for analyzing three-dimensional structures under progressive collapse and column removal scenarios, practicing civil/structural engineers can check their design in a quick and accurate way using SAP2000 or other structural analysis software with similar capabilities. Moreover, this simple approach enables large multi-bay systems to be analyzed much more efficiently than the detailed finite element modeling approaches used in previous studies, while keeping good consistency with the experimental data.

Learning more about computational analysis of progressive collapse can help civil/structural engineers prevent full structural collapse after an initiating event has occurred. Moreover, many lives can potentially be saved and more safety can be provided to the occupants of structures, if those structures are designed to resist progressive collapse.

1.3 RECENT COMPUTATIONAL STUDIES ON PROGRESSIVE COLLAPSE OF STEEL GRAVITY FRAMING SYSTEMS

There are a significant number of previous studies in which researchers developed and used simplified/reduced modeling approaches to simulate the progressive collapse resistance of structures. Nonetheless, a major limitation of these past efforts is the lack of experimental data used to validate model predictions. While some researchers were able to validate individual components within their model (e.g., connection response), the interaction of components in a typical floor system has not been studied and compared with test data. An important feature of the current study is the availability of test data to assess the suitability of different modeling approaches. Recent computational research on progressive collapse that has influenced the models developed in this thesis is briefly described below.

Marjanishvili and Agnew (2006) compared linear-elastic static, nonlinear static, linear-elastic dynamic, and nonlinear dynamic methodologies for progressive collapse analyses using SAP2000. The research included the computational analysis of a nine-story steel moment-resisting frame building. The main objective of the study was to "provide clear conceptual step-by-step descriptions of various procedures for progressive collapse analysis." At the end of the research, it was concluded that the dynamic analysis procedures for progressive collapse determinations are fairly simple to perform and are readily available to practicing engineers through finite element computer programs, which are capable of nonlinear dynamic analysis, such as SAP2000. Moreover, it was found that the evaluation criteria for linear analysis procedures are non-conservative compared to nonlinear analysis procedures.

Izzudin et al. (2008) developed an energy-based modeling procedure to evaluate the progressive collapse resistance of building structures under sudden column loss scenarios. During the research, the system limit state was defined as the failure of a single connection based on the assessment of ductility demand. This method is applicable to both detailed and reduced modeling approaches of the nonlinear structural response, and it enables the determination of structural capacity by using only a single quasi-static pushdown analysis.

A reduced model of seismically designed steel moment-resisting connections was developed and validated by Main et al. (2010). Data for the model validation was obtained from an experimental study conducted by Sadek et al. (2010) in which moment-resisting connections were tested to failure under vertical column displacement. The computational analysis part of the research included detailed and reduced modeling approaches, and both methods demonstrated good agreement with the experimental data. This research also revealed that the loads under sudden column loss scenarios are resisted by tensile forces in beams that result from catenary action until the connection reaches its combined axial and flexural stress limit. This study improved the knowledge about the capacity of seismically designed moment frames under column loss scenarios, and it provided a thorough understanding about the behavior and failure modes of the connections.

Using reduced connection models consisting of nonlinear springs and rigid links, a computational model of a 10-story building was developed by Main et al. (2010). The model developed by the researchers demonstrated good agreement with the experimental data, captured the behavior and failure modes observed during testing, and showed that this type of connection is able to support the structure under the case of sudden loss of multiple columns.

The extensive computational assessments that were done by Sadek et al. (2008) and Alashker et al. (2010) concerning composite floor systems with simple shear connections have provided insight into the collapse sensitivity of gravity frames under column loss scenarios. A number of finite element models were used to represent every structural member of the structure. The studies showed that composite floor slabs increase the collapse resistance of structures over that predicted for a bare steel frame by preventing the exterior columns from being pulled inward and by providing compressive membrane action. This result is notable because most prior studies in the research literature did not directly account for the contribution of the floor system and simply considered the primary structural framing. Alashker et al. (2010) directed their research to parametric studies after realizing that the structure they were analyzing was not capable of sustaining the required gravity loads under the considered column loss scenario. These parametric studies concluded that the steel deck is the most influential component of the structure regarding collapse resistance.

Recent tests by Thompson (2009) and Weigand et al. (2012) changed the previous modeling approach of single-plate shear connections. These studies showed that single plate connections are not able to transform shear and flexure into catenary tension. Further, these researchers showed that these connections have insufficient capacity to support shear design loads under column loss scenarios. Although, previous studies by Sadek et al. (2008) and Alashker et al. (2010) assumed a gradual softening behavior in single-plate shear connections during the post-ultimate response, results from Thompson (2009) and Weigand et al. (2012) proved that single-plate shear connections experience a sudden fracture during the post-ultimate response. As a result, connections immediately lost their axial and shear resistance capacity and stopped contributing to the system strength. The information obtained from these recent research studies has been incorporated into the modeling approach proposed in this thesis.

1.4 THESIS OBJECTIVES AND SCOPE

The scope of this thesis is to document the computational analysis work that was conducted from January 2013 to May 2014 for determining the progressive collapse resistance of the three-dimensional steel structure that was constructed and tested during the current research project. All tests were conducted at the Ferguson Structural Engineering Laboratory of University of Texas at Austin. During this research, SAP2000 was used to model and simulate every component of the steel gravity frame with composite floor system. The experimental test data was obtained from a 2-bay × 2-bay specimen with steel frames, composite floor system, and connections between structural components. During the experiment, the interior column of the prototype structure, which was loaded by uniformly distributed floor loading, was removed pseudo-statically.

Throughout the computational analysis work, time efficient and easy-to-converge methods were searched and verified by comparing the results with available experimental data. By using these methods, it is possible to decrease the analysis time from a few hours to a few minutes. These simplifications include modeling shear studs and bolts as spring elements, beams and columns as frame elements, and corrugated decking as thick and thin nonlinear layered shell elements. Specific details of these models are given later in this thesis. The high accuracy of the adopted approach was proven with experimental data by Sadek et al. (2010) and computational models by Main et al. (2010) and Alashker et al. (2011). All of these modeling approaches and procedures are described and verified step-by-step in the following chapters of the thesis.

The objective of this thesis is to provide a step-by-step description of the methods that have been used throughout this research and to document the final results from the computational analysis work. The information provided in this thesis will allow practicing civil/structural engineers to check the progressive collapse resistance of their preliminary and final designs in a quick and accurate way with SAP2000 or any similar structural engineering software.

A detailed description of the experimental research program and the observed results are published elsewhere and can be found in the MS thesis by Hull (2013) and PhD dissertation by Hadjioannou (2014).

1.5 THESIS ORGANIZATION

Other than the Introduction Chapter (Chapter 1) and Summary, Conclusions and Recommendations Chapter (Chapter 4), this thesis includes two major chapters. Chapter 2 and Chapter 3 provide the description and verification of the methods and approaches that are used throughout this study to evaluate the progressive collapse resistance of structures. In addition, these chapters document the usage of these methods to determine the progressive collapse resistance of the structure constructed and tested during this research project. A more detailed description of the chapters is given in the following paragraphs.

Chapter 2 presents the step-by-step descriptions and verification of the methods and approaches that have been used throughout this thesis to evaluate the progressive collapse resistance of structures. The behavior of bare steel frames, single-plate shear connections, composite/partially composite/non-composite beams, shear studs, and other components of a typical floor system in a steel-framed building are verified by comparing results from SAP2000 with available experimental data and/or results from detailed finite element software.

Chapter 3 includes a brief description of the actual specimen tested during this research program. Then, it documents how the methods and approaches outlined in Chapter 2 apply to the prototype structure. Moreover, this chapter provides all computational analysis results related to the test specimen.

Chapter 4 includes a discussion of the computed results, summarizes the main research findings, and offers conclusions about the research. Also, future research is suggested based on the limitations of the methods proposed in this study.

CHAPTER 2

Description and Verification of Modeling Approach

2.1 GENERAL

This chapter includes step-by-step descriptions of the methods used throughout this thesis to determine the progressive collapse resistance of structures and verification of these methods by using available experimental data or results from detailed finite element models. The list below gives the methodology used throughout this thesis to create and verify the simplified component models.

1. Creating a simplified model for a single structural component (e.g., Shear Tab, Bolt Spring, and Floor Slab) by taking into consideration models described in the research literature.
2. Verifying the response of the simplified component models by using experimental data from the literature. The experimental data should belong to a single structural component in an isolated environment from the rest of the structure to neglect the effects of other structural components and mechanisms.
3. Attaching the single simplified component models to a full structural model, after completing the first two steps for all structural components.
4. Verifying the response of the full structural model by using the experimental data collected during the current research project.

Although, other similar structural analysis programs can be used to model and analyze the structures that are given in this thesis, the software used for modeling and analysis was SAP2000 Ultimate 15.1.0. This software was intentionally chosen by the research team because it is a widely available, easy-to-use, runs on a personal computer, and is a popular civil/structural analysis program used by many design offices, universities, and researchers around the world. The assumptions used during the modeling are also listed in this chapter.

2.2 CONNECTION MODELS

Two types of steel connection models were used in this research. The first one was designed to simulate the behavior of single-plate shear connections (Figure 2.1(b)), following the recommendations by Main and Sadek (2012), and a second one was developed during this research to simulate double angle connections (Figure 2.2(b)). Both of these connections are modeled using a series of springs and rigid links. In addition, both models use the same spring types—only the configuration of the springs was adjusted to represent the connections.

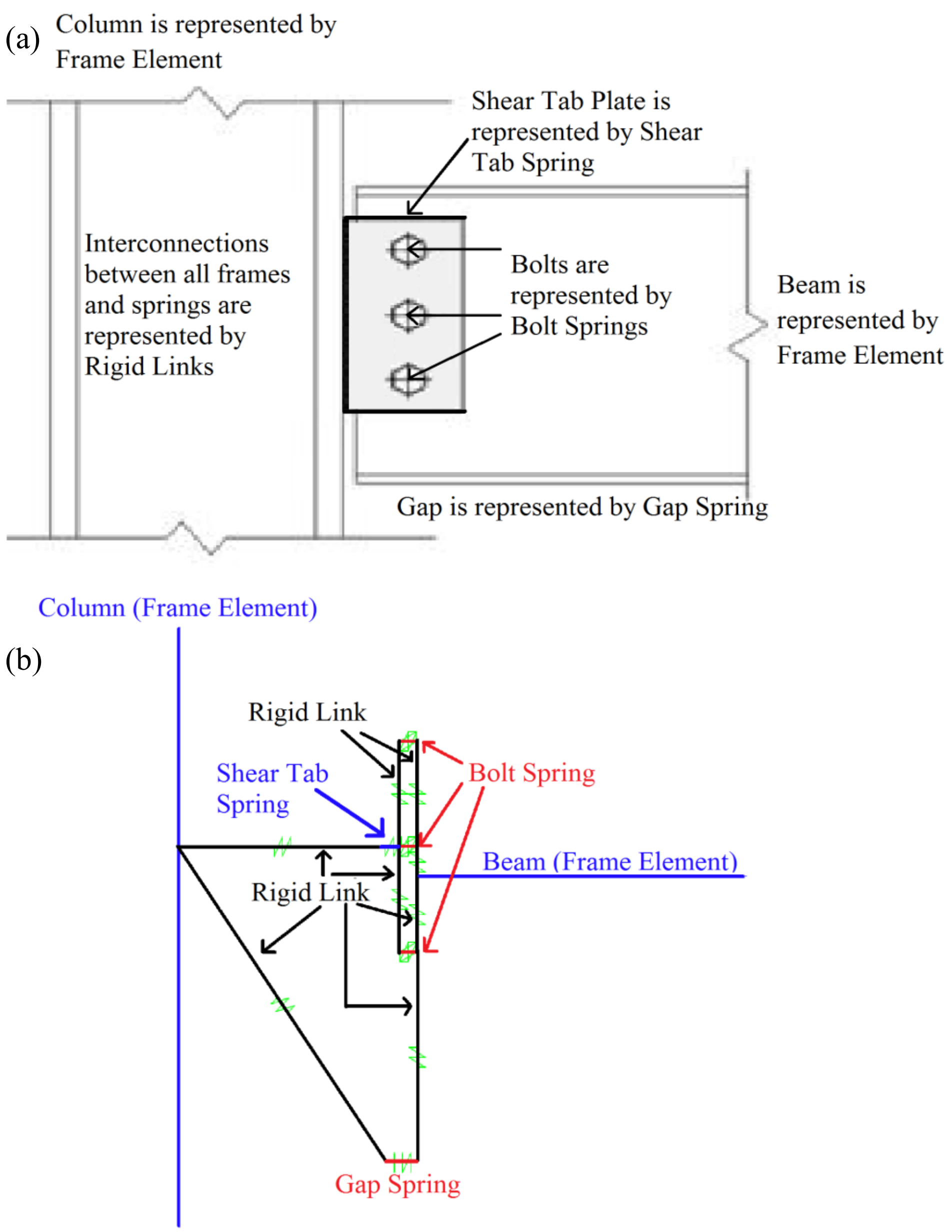

Figure 2.1: Beam-to-Column Single-Plate Shear Connection: (a) drawing of the connection; (b) simplified model of the connection

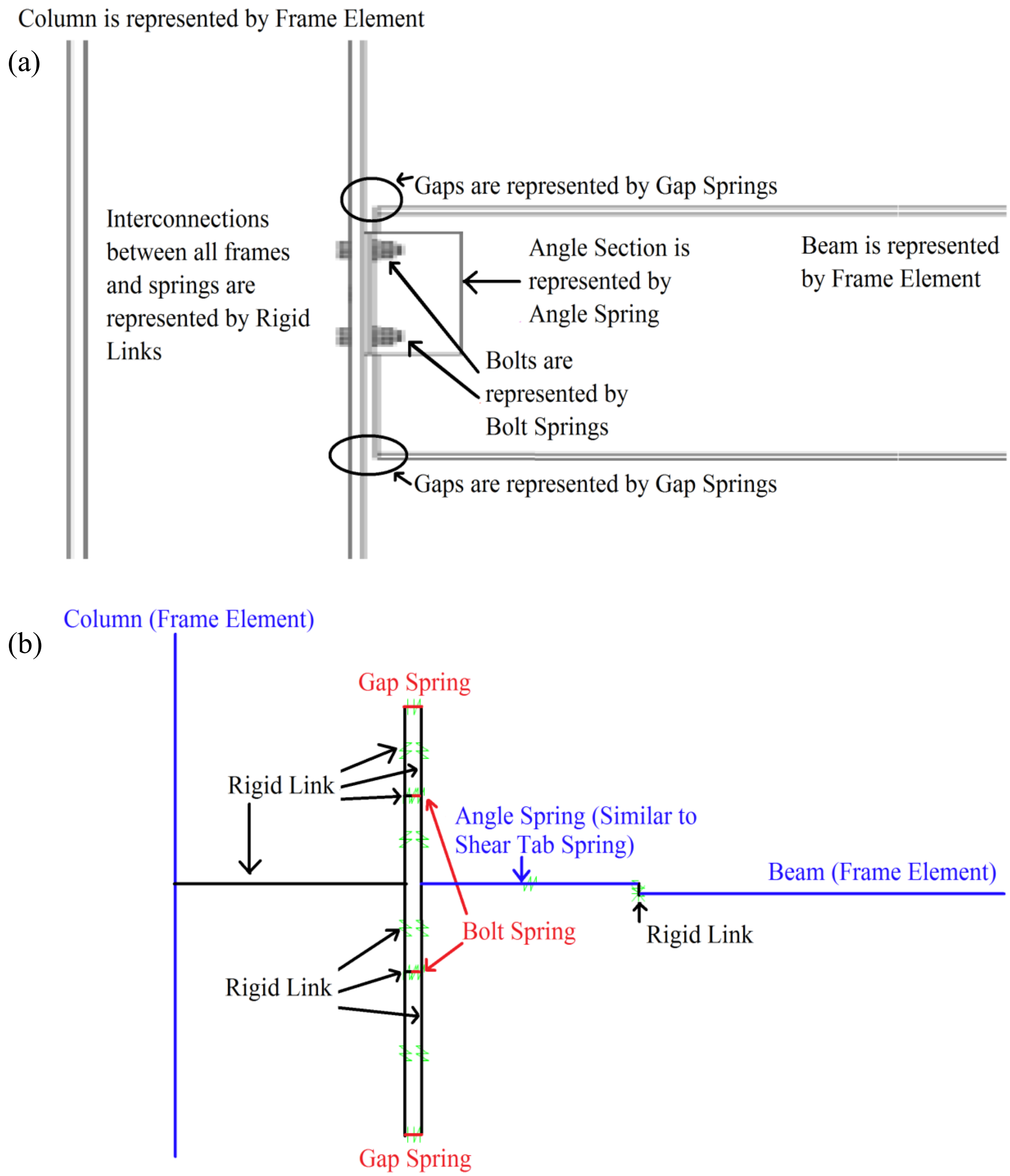

Figure 2.2: Beam-to-Column Double Angle Connection: (a) drawing of the connection; (b) simplified model of the connection

2.2.1 Description of Connection Models

As can be seen from Figure 2.1 and Figure 2.2, the models that were used to simulate connection behavior are composed of uniaxial springs. Each of these springs represents either a specific component of the connection or the cross-sectional properties of the connection. The springs that were used to simulate connection bolts, shear tabs, gaps between frames, and interconnections are created using the Link/Support Element tool within SAP2000.

Basically, the Link Element is a tool to connect two joints together, and the relative translation and rotation of these joints in all three dimensions can be adjusted by the user. Three types of Link Elements are used during the computational analysis, including MultiLinear Elastic, Gap, and Linear Links.

The Link type that was used to model the connection bolts and shear tab is a MultiLinear Elastic Link. This link type allows the user to input non-linear load-displacement data to define the spring response. Alternatively, a constant stiffness value can be assigned to the directional property of the spring rather than load-displacement data if further simplification is needed.

The bolt springs, which represent each bolt row within a connection, consider the interaction of axial force, shear force and moment. Behavior of the connection under these loads can be simulated by assigning non-linear load-deformation data to each bolt spring in the axial and transverse directions. As a result, bolt springs can act in combination to resist the overall axial load and bending moment that act on a connection. The non-linear load-deformation curve of each bolt row can be uniform or different than the other rows depending on the possible failure modes and cross-sectional properties of the connection. For instance, bearing strength of interior bolt holes can be different than exterior bolt holes under pure shear loading because bearing strength of bolt holes depends on the clear distance from the edge of the bolt hole to the edge of the adjacent bolt hole or to the edge of the material. Conversely, all bolt holes could have the same bearing strength, if pure tension load is applied to the connection, which has a single column of bolts.

To be able to assign the non-linear load-deformation data to the bolt spring of a single-plate shear connection, it is necessary to approximate the load-deformation behavior of a single bolt row under various failure modes. For this purpose, monotonic force-deformation relationships from previous research by Main and Sadek (2012) and Rex and Easterling (2003) were taken as a starting point. Validation of these models using the proposed modeling approach is described in Section 2.2.2.

The axial load-deformation behavior of a single bolt row, controlled by bolt shear failure, is based on the model by Main and Sadek (2012), and shown in Figure 2.3. In this curve, t_y and t_u represent the yield and ultimate capacities of each spring in tension, respectively, while c_y and c_u represents the yield and ultimate capacities of each spring in compression, respectively. These terms can be computed using the AISC Specification (AISC 2010, Section J3.6) with a resistance factor of Φ=0.75 for yielding and Φ=1 for ultimate.

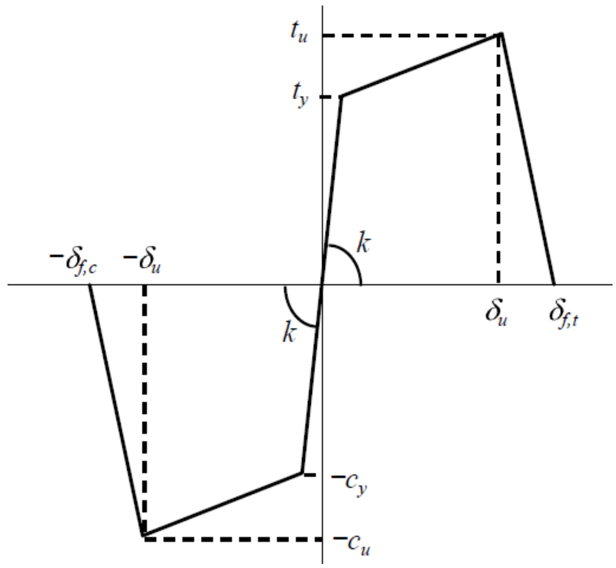

Figure 2.3: Axial load-deformation relationships for bolt springs controlled by bolt shear failure. (From Main and Sadek (2012))

The initial translational stiffness of a bolt spring can be calculated using Eq. (2.1), where K is the initial rotational stiffness of a shear tab connection and can be calculated using Eq. (2.2) from FEMA 355D (2000), and y_i is the vertical distance of the i^{th} bolt row from the center of the bolt group. In Eq. (2.2), s is the vertical spacing between bolt rows and N is the number of bolt rows.

$$k = \frac{K}{\Sigma_i y_i^2} \tag{2.1}$$

$$K = \begin{cases} 124{,}550(s(N\text{-}1)\text{-}142 \text{ mm}) & (\text{kN}\cdot\text{mm/rad}) \\ 28{,}000(s(N\text{-}1)\text{-}5.6 \text{ in}) & (\text{kip}\cdot\text{in/rad}) \end{cases} \tag{2.2}$$

Following the recommendations by Main and Sadek (2012), the failure displacement in tension ($\delta_{f,t}$) and the failure displacement in compression ($\delta_{f,c}$) should be set equal to $1.15\delta_u$, where δ_u is the displacement at ultimate load and can be obtained from Eq. (2.3).

$$\delta_u=\begin{cases} 0.085\cdot s(N\text{-}1)\text{-}0.00007\cdot\left(s(N\text{-}1)\right)^2 & \text{(mm)} \\ 0.085\cdot s(N\text{-}1)\text{-}0.0018\cdot\left(s(N\text{-}1)\right)^2 & \text{(in)} \end{cases} \tag{2.3}$$

In addition to the modeling approach for bolt shear failure described above, a load-deformation relationship was developed by Rex and Easterling (2003) to approximate the behavior associated with plate bearing, which is given in Eq. (2.4). In this equation, R is the plate load, R_n is the nominal plate strength, and $\overline{\Delta}$ is the normalized deformation=$\Delta\beta K_i/Rn$, where Δ is the hole elongation, β is the steel correction factor, and K_i is the initial stiffness.

$$\frac{R}{R_n}=\frac{1.74\overline{\Delta}}{(1+(\overline{\Delta})^{0.5})^2}\text{-}0.009\overline{\Delta} \tag{2.4}$$

Although, the overall response of a structure under a column removal scenario depends primarily on the axial and bending deformations of the connections due to the development of catenary action, there are also analytical methods to approximate the vertical shear deformations of a connection. The first method is to calculate the vertical shear behavior of each bolt spring by using Eq. 2.4 when the expected failure mode is bearing/tearout failure. The second method was developed by Main and Sadek (2012) to calculate the vertical shear behavior of each bolt spring when the expected failure mode is shear yielding/rupture of the connecting elements under shear. The vertical load-deformation relationship of each bolt can be obtained by using the curve in Figure 2.4. In this curve, v_y and v_u represent the shear yielding of the gross section of the connecting element and shear rupture of the net section of the connecting element, respectively. These terms can be calculated using the equations in AISC Specification (AISC 2010, Section J4.2) with a resistance factor of Φ=1.

According to the recommendations by Main and Sadek (2012), initial translational stiffness of a bolt spring and the displacement at ultimate load can be calculated with Eq. 2.1 and Eq. 2.3 respectively, which are used for the axial load-deformation relationship in the absence of specific empirical equations for shear stiffness and deformation capacity if the expected failure mode is shear yielding/rupture of the connecting elements under shear. Finally, the failure displacement in vertical shear ($\delta_{f,v}$) should be set equal to $1.15\delta_u$ based on the recommendations by Main and Sadek (2012).

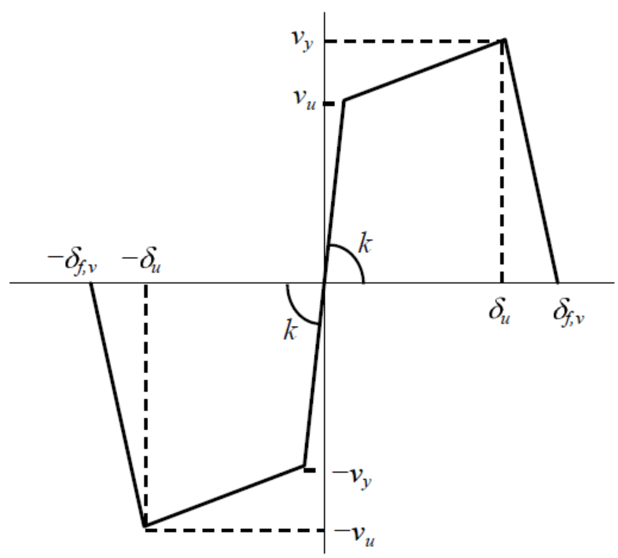

Figure 2.4: Vertical shear load-deformation relationship for bolt springs. (From Main and Sadek (2012))

The shear tab spring is responsible for representing the load-deformation response of the single plate of the shear tab. It primarily carries the torsional and out-of-plane loads that are applied to the connection. Instead of specifying non-linear load-deformation data for the shear tab springs, constant stiffness values can be assigned to all rotational and translational directions, which are calculated according to the cross-sectional and material properties of the shear tab plate.

Gaps between the end of a beam member and column are simulated with the Gap Links. This Link type only works for compression after reaching a constant shortening value, which needs to be defined by the user. It is a powerful tool for simulating the bearing forces generated by the system when the gap between the beam flange and the column closes and the members come into contact.

Finally, Rigid Links are modeled with Linear Links where all rotational and translational stiffness are fixed (Infinitely Stiff) to directly transfer the loading from one end of a link to the other. Rigid Links are used as interconnections to maintain the proper connection geometry.

2.2.2 Verification of Connection Models

This section includes the work done to verify the load-deformation behavior of the simplified connection models by comparing the results from SAP2000 with the available experimental data from previous research papers. To understand the advantages and limitations of simplified connection models, a single-plate shear connection model (Figure 2.5) is considered. For simplicity, the model is subjected first to pure tension and then to pure shear, and the shear tab and gap springs are removed. Although it is known that connections experience cyclic response under sudden column removal scenarios when no failures occur, only monotonic loading is considered for the current analysis.

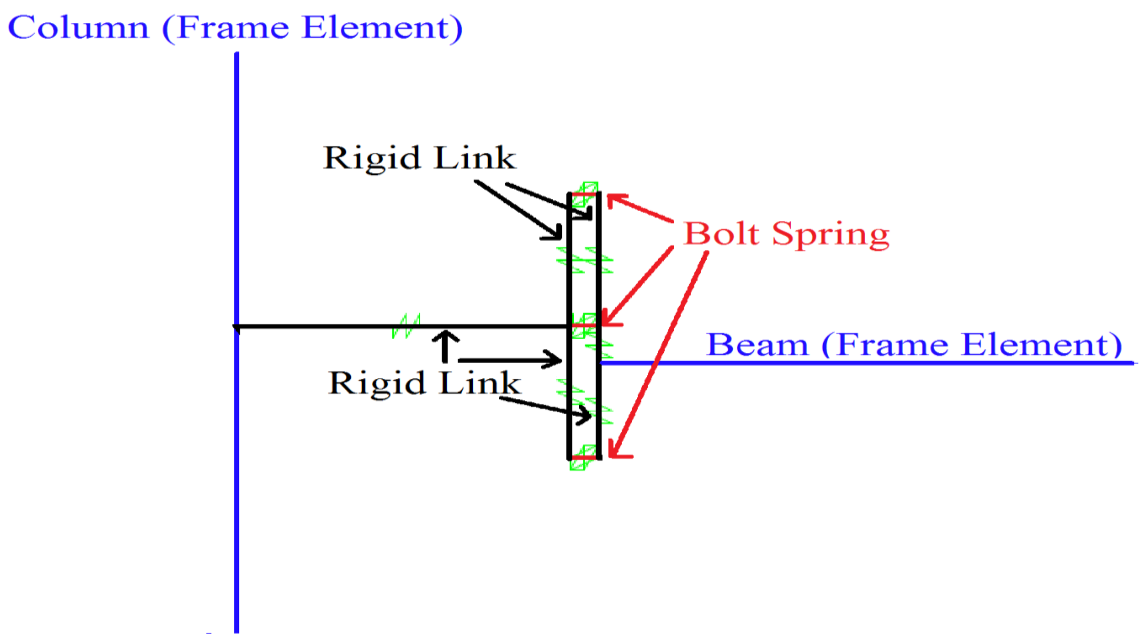

Figure 2.5: Simple Single-Plate Shear Connection under Axial-Loading

Because the connection is modeled in an isolated environment from the rest of the structure, the influence of the floor slab, shear studs, and other beams and columns are completely neglected. To be consistent with the modeling approach, only experimental data from a similar test setup should be used for validation. Consequently, the experimental data are taken from previous research by Guravich et al. (2002), which uses the test setup illustrated in Figure 2.6. The analysis model is consistent with the experiment.

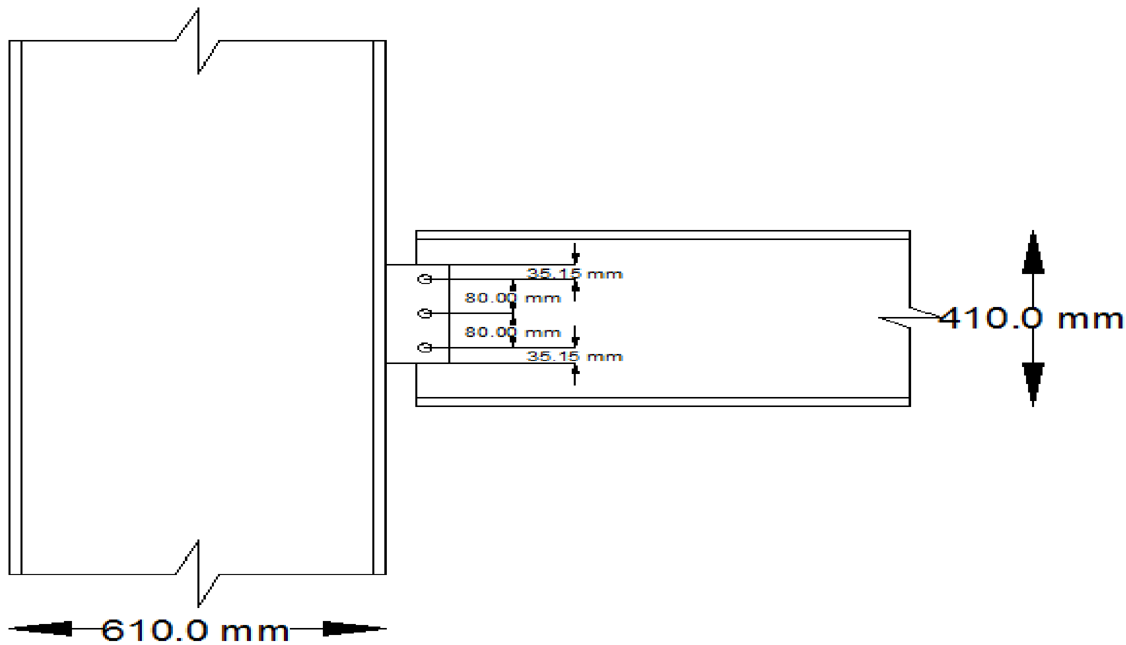

Figure 2.6: Test Setup for Single-Plate Shear Connections

During the lab experiments by Guravich et al. (2002), a pure tension load was applied at the right end of the beam, and the horizontal displacement was measured using linear strain converters (LSCs), which were located on the beam. A second experiment was done by applying pure shear to the beam, and the vertical displacement was measured beneath the location where the point load was applied. The load-displacement results for both of these experiments were reported in Guravich et al. (2002).

The test setup in Figure 2.6 was modeled using the simplified modeling approach, described in Section 2.2. Displacement-controlled loading was used in two separate analyses—corresponding to pure tension and pure shear—and the load-displacement data was recorded for both cases. The SAP2000 software is capable of considering geometric nonlinearity in the form of either P-delta or P-delta and large-displacement/rotation effects. As a result, the software allows the user to choose between available geometric nonlinearity parameter options (P-Delta, P-Delta plus Large Displacement) before analyzing the model. At this stage of the research, it was unclear which geometric nonlinearity parameter option would provide better agreement with the measured data. Accordingly, nonlinear analyses were conducted considering both the P-Delta and P-Delta plus Large Displacement options. The computational and experimental load-deformation curves under pure tension and pure shear loading are shown in Figure 2.7 and Figure 2.8, respectively. Computational analysis results for both P-Delta and P-Delta plus Large Displacement options are also provided for comparison purposes.

The good fit of the computed and experimental load-deformation curves in Figure 2.7 shows that the simplified modeling approach can accurately represent the actual tensile behavior of single plate shear connections. The computational results for both geometric nonlinearity parameter options also match well. The computational load-deformation curve in Figure 2.8, which was obtained by using P-Delta plus Large Displacement analysis, shows a satisfactory match with the experimental load-deformation curves. It should be noted that the initial stiffness and peak load is the same for both the experimental and computational results. However, as can be seen from Figure 2.8, the P-Delta option gives a lower peak load than the experimental results, which shows that this analysis type has limited accuracy. Accordingly, in the remainder of this research, analyses of the frame and link elements are conducted considering only P-Delta plus Large Displacements.

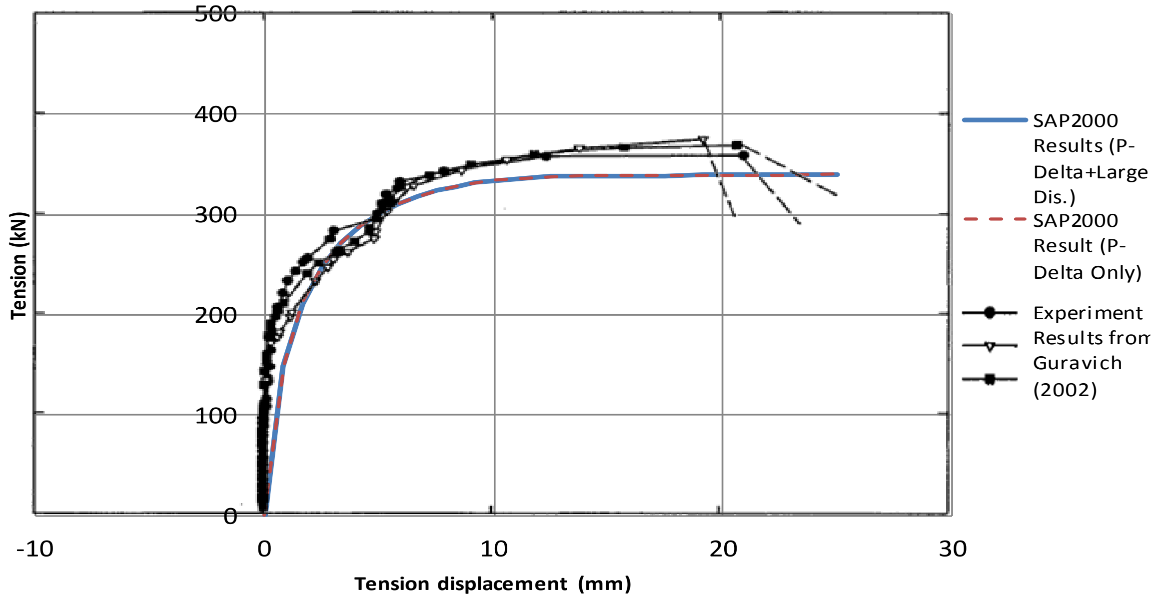

Figure 2.7: Tension versus displacement curves under pure tension loading

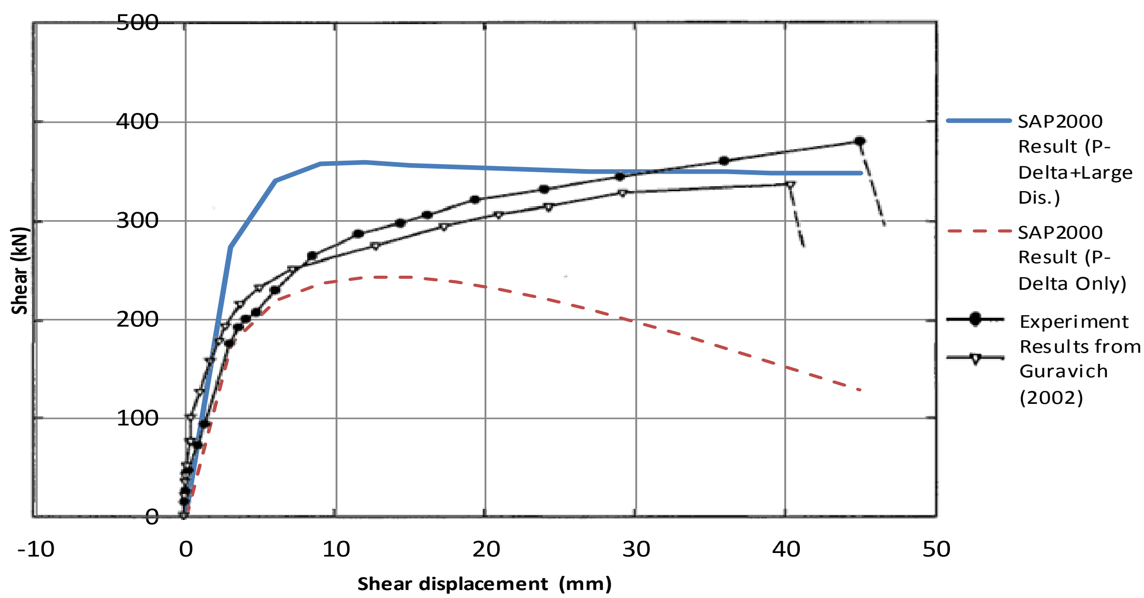

Figure 2.8: Shear versus displacement curves under pure shear loading

The difference between the P-Delta plus Large Displacement analysis results and the experimental results in Figure 2.8 is due predominantly to the complexity of the failure mode in the experiment. Although, Guravich et al. (2002) state that the ductility of the connection under pure shear was provided primarily by yielding in bearing around the bolt holes, the failure mode was stated as out-of-plane buckling. It should be noted that the simplified modeling approach is not capable of capturing all kinds of complex behaviors and should be used with caution. Nevertheless, because the structural response for progressive collapse scenarios is more sensitive to axial and bending deformations than shear deformation, and because a small amount of uncertainty in vertical shear deformation has a negligible effect on the overall response, the simplified modeling approach is believed to be suitable for computational analyses.

2.3 FLOOR SLAB MODEL

The experimental setup that is used in the current research project includes a composite floor system with corrugated decking, concrete slab, shear suds, welded wire reinforcement, and extra reinforcing bars. Although, it is hard to model every component of the floor slab due to the limitations of the SAP2000 software, a simplified modeling approach by Main and Sadek (2012) is used to approximate the complex behavior of the composite floor system. Also, the modeling approach is slightly modified during the computational analysis research to improve the response. Details of the proposed model are provided in the following sections.

2.3.1 Description of Floor Slab Model

The actual composite floor slab and the simplified floor slab modeling approach proposed by Main and Sadek (2012) are shown in Figure 2.9. The computational model utilizes rectangular shell elements with different thicknesses. Because the deep corrugations in Figure 2.9(b) have a higher moment of inertia than the thinner regions of the slab and steel decking at the bottom, they were named “strong strips.” The thinner regions of the slab were named “weak strips”. To simulate the anisotropic behavior of the steel decking, Main and Sadek (2012) recommended placing steel decking on the strong strips only.

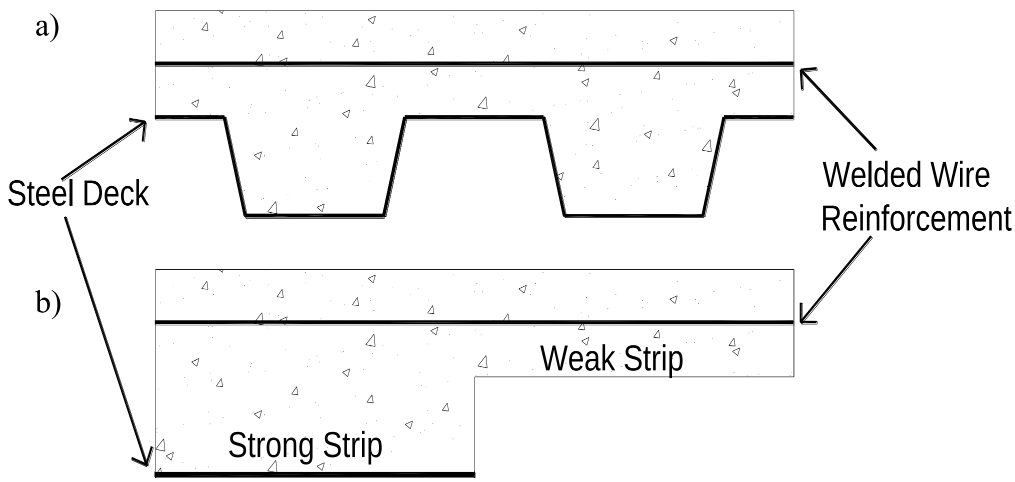

Figure 2.9: Simplified modeling of composite floor slab: (a) actual floor section; (b) simplified floor section

The computational study by Main and Sadek (2012) concluded that the load-deformation response of the structure is not sensitive to mesh size as long as the weak strips are located along the girders. Interestingly, the same study showed that placing strong strips along the girders results in an overestimation of the peak load if the rib widths are modeled larger than the actual size. As a result of this recommendation by Main and Sadek (2012), weak strips are placed along the girders for the rest of this thesis. Although, increasing the strip size will decrease the number of shell elements and increase the computational efficiency, using strips that are too wide can lead to an inaccurate solution. In their study, Main and Sadek (2012) did not require the shell element geometry used to define the strips to correspond with the exact geometry of the slab. Thus, they allowed the use of fewer strips than actual corrugations to simplify the modeling, provided the element properties were adjusted to account for the discrepancy in geometry between the actual slab and the modeled one.

All components of the floor slabs in this thesis are modeled using the Nonlinear Layered Shell Element tool within SAP2000. This tool enables the user to define nonlinear area sections with multiple layer rows, which can be defined separately by the user. For example, for the strong strip of Figure 2.9(b), two main layers must be defined: (1) a concrete membrane and plate layer for the concrete slab, and (2) a steel membrane and plate layer for the welded wire reinforcement and steel decking. Figure 2.10 illustrates a sample SAP2000 Shell Layer Definition window with these two main layers.

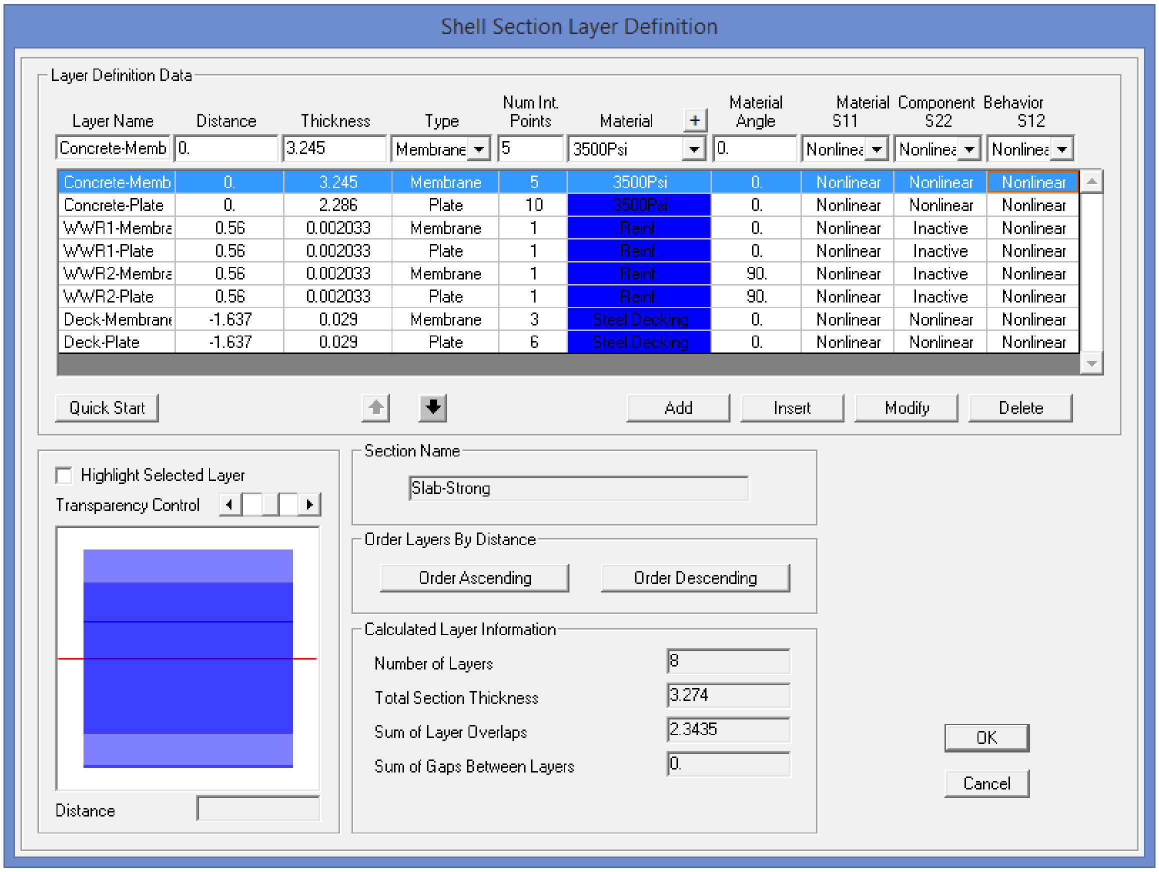

Figure 2.10: Sample SAP2000 Shell Layer Definition window

SAP2000 does not have a built-in tool to model concrete members using cracked section properties. For this reason, users should adjust concrete section properties manually to take into account concrete cracking. The ACI Building Code (ACI 318-11, Section 9.5.2.3) states that deflections of the reinforced concrete members shall be computed with the effective moment of inertia I_e, which can be calculated by using Eq. (2.5).

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^3 \cdot I_g + \left[1 - \left(\frac{M_{cr}}{M_a}\right)^3\right] \cdot I_{cr} \tag{2.5}$$

In this equation, I_e is the effective moment of inertia, I_g is the moment of inertia of the gross concrete cross-section, I_{cr} is the moment of inertia of the cracked section, M_a is the maximum moment in the member at the stage for which the deflection is being computed, and M_{cr} is the moment that would initially crack the cross-section.

Eq. (2.5) has a major disadvantage for the current study. Because the displacement-controlled load increases continuously during the computational analysis, the M_a term of the equation should be updated at each loading step, which is not allowed by the SAP2000 software. The only possible way of including concrete cracking in the SAP2000 Shell Layer Definition window is to multiply concrete layer thickness with a constant coefficient. Although there is not a specific way of finding this coefficient for concrete sections, a method was developed during this research by taking ACI Building Code (ACI 318-11, Section 8.8) as a starting point, which will be described step-by-step in the following paragraphs. To verify the proposed method, previous research and experimental data were investigated in great detail, and it was concluded that the new method can simulate concrete member behavior with acceptable accuracy. Also, the good agreement between the load-deformation behavior of the simplified models, detailed finite element program results, and available experimental data in Section 2.3.2, Section 2.4.2, and Section 2.5.2 shows that the proposed method can be used to predict the response of concrete slabs under transverse loading.

The ACI Building Code (ACI 318-11, Section 8.8.1 and Section 8.8.2(a)) states that lateral deflections of reinforced concrete building systems shall be computed by using linear analysis with flexural stiffness defined using Eq. (2.6) for determining cracked moment of inertia of flexural members (Floor Slab). In this equation, I_{cr} and I_g stand for cracked and gross moment of inertia, respectively. If a section is loaded with service lateral loads, the right side of the Eq. (2.6) should be multiplied by 1.4 according to the ACI Building Code (ACI 318-11, Section 8.8.1).

$$I_{cr}=0.25I_g \tag{2.6}$$

After calculating the cracked moment of inertia, the flexural stiffness of the floor slab should be adjusted accordingly. Because the plate component of the concrete layer is responsible for bending stiffness, users can take into account cracking by simply changing the thickness of the concrete plate according to Eq. (2.7). In this equation, t_{plate} and t_{gross} stand for the adjusted thickness of the plate layer and the actual thickness of concrete member, respectively.

$$t_{plate}{}^3=1.4\cdot 0.25\cdot t_{gross}{}^3 \tag{2.7}$$

It should be noted that cracking in concrete slabs does not propagate uniformly under uniform loading conditions (Figure 2.11). The picture by Park et al. (1964) shows a concrete slab with major diagonal cracks, and negligible cracking over the remainder of the surface. Because modifying the moment of inertia of all shell elements by using Eq. (2.7) means cracking the concrete homogenously, the result of the analysis will be an approximation of the total response. Although it is not possible to get the exact response of the system due to the inherent limitations of the simplified modeling approach available within SAP2000, the computational analysis results show acceptable agreement with experimental data. Section 2.3.2 includes a comparison between test data and model predictions.

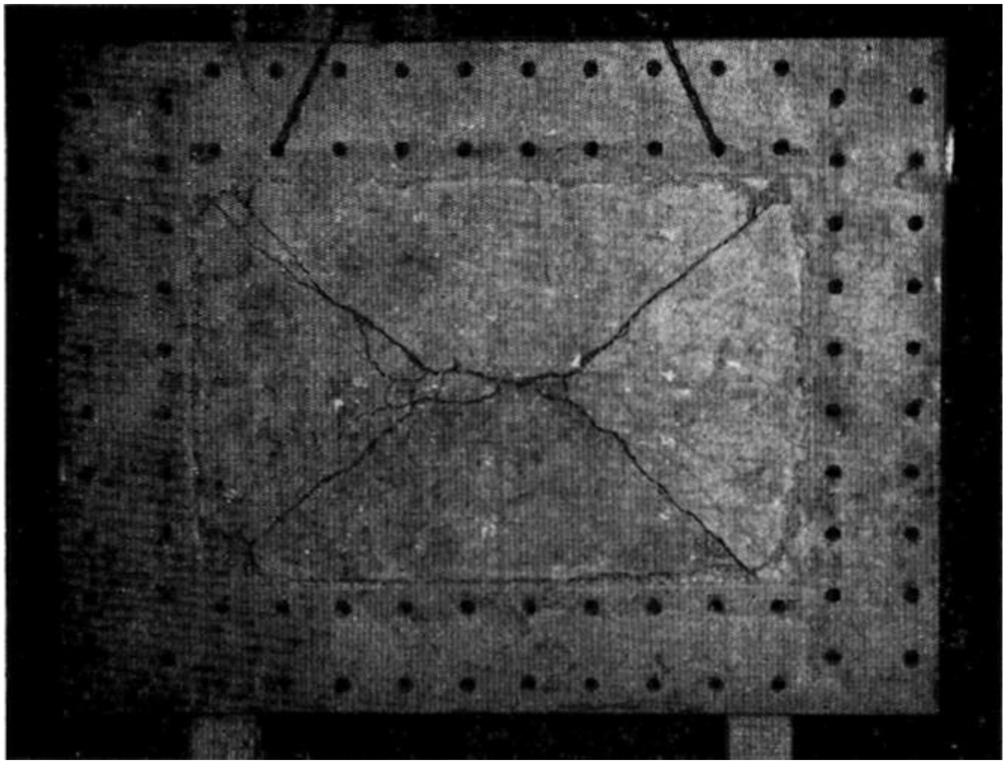

Figure 2.11: Cracked concrete slab. (From Park et al. (1964))

While conducting nonlinear analyses in SAP2000 using shell elements, it is important to select Material Component Behavior as Nonlinear to get more accurate results than linear analyses. Material Component Behavior options are located at the right side of the SAP2000 Shell Layer Definition window, which can be seen in Figure 2.10. This tool enables the user to assign nonlinear material properties to layered shells. In addition, the number of integration points for shell elements should be adjusted by the user for each layer because stresses are first calculated at integration points and then extrapolated to the joints by the software. Although, two points are enough to capture both membrane and plate behavior in many cases, nonlinear behavior may require more integration points to capture yielding near the top and bottom surfaces. For nonlinear materials, the recommended procedure is to choose four or five integration points through the thickness of the shell layer (LS-DYNA Support: Elements Tutorial (2014)). Alternatively, the required number of integration points can be found by increasing the number of integration points until the response of the structure has reached convergence. It should be noted that, using fewer integration points than required will increase the amount of error due to linear extrapolation, while using too many integration points will decrease the computational efficiency. Section 2.3.2 includes an example, which shows the application of these recommendations.

As shown in Figure 2.9, although strong and weak strips have different thicknesses, the top surfaces of both strips are level. Because SAP2000 defines shell element layers symmetrically about their mid-surface, floor slab strips are always placed on horizontal surfaces as shown in Figure 2.12. This error can be solved by separating the shared nodes of each strip and raising the weak strip nodes by half of the thickness difference between the weak and strong strips. Because each strip has unrestrained nodes after this procedure, it is important to reconnect each separated node with body constraints before analysis.

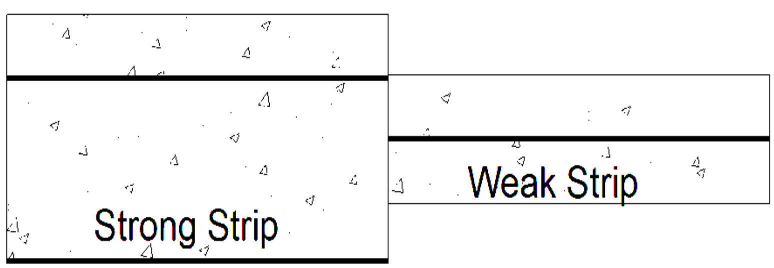

Figure 2.12: Weak strip placement error in SAP2000

2.3.2 Verification of Floor Slab Model

This section includes the work done to validate the load-deformation behavior of the simplified floor slab models by comparing the results from SAP2000 to available experimental data. To be able to understand the advantages and limitations of the model, floor slabs should be isolated and analyzed separately from other structural components. In addition, because the actual test specimen evaluated during the experimental portion of the current study includes a floor slab with a steel perimeter ring beam, it is advantageous to use experimental data with a similar test setup. As such, previous research by Park et al. (1964) is used for validation purposes.

The research by Park et al. (1964) includes uniformly loaded rectangular concrete slabs that are restrained around their perimeter. Although, various specimens with different properties were tested, only results from floor slabs—which have fully fixed edges, no steel decking, and no reinforcement bars—are used for validation purposes. Because such test specimens have limited interaction with other structural components, the benefits and limitations of the proposed floor slab models can be directly evaluated.

Figure 2.13 shows how the slab edges were fixed against rotation and translation in the experiment by Park et al. (1964). According to the figure, 1-inch diameter bright steel studs were installed to prevent rotation, and the same size high-tensile screws were used to prevent horizontal spread. In the SAP2000 model, fixed supports were placed around the perimeter of the floor slab to simulate the boundary conditions in the test. Table 2.1 shows the properties of the two uniformly loaded rectangular concrete slabs used to validate the simplified floor slab models.

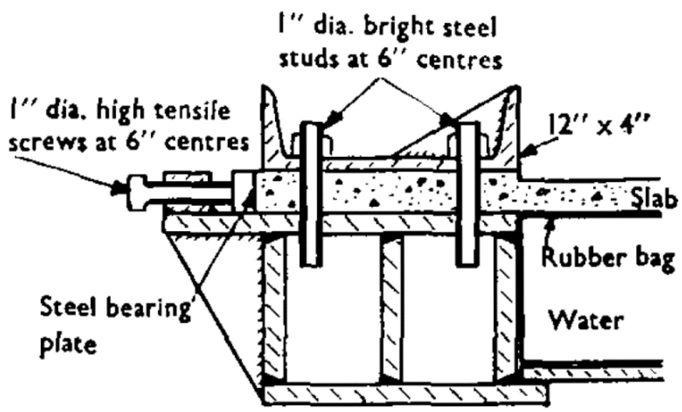

Figure 2.13: Floor slab with fully fixed edge. (From Park et al. (1964))

Table 2.1: Properties of rectangular concrete slab test specimens

Slab	Dimensions $L_y \times L_x \times d$ (inch)	L_y/L_x	L_x/d	Percentage of steel reinforcement	Cube strength of concrete (psi)
D1	60×40×2	1.5	20	0%	6280
D2	60×40×1.5	1.5	26.7	0%	6200

The test specimens listed in Table 2.1 are modeled using nonlinear layered shell elements in SAP2000. Figure 2.14 shows the shell section layer definition used to model test specimen D1, employing the recommendations and guidance given in Section 2.3.1. Because the test specimen has neither reinforcement, nor steel decking, only two concrete layers are defined. As described in the previous section, membrane and plate layers are responsible for simulating the membrane and plate-bending behavior and should be created for nonlinear analysis. After defining the shell section layers, fixed supports were specified around the perimeter of the floor slab as shown in Figure 2.15.

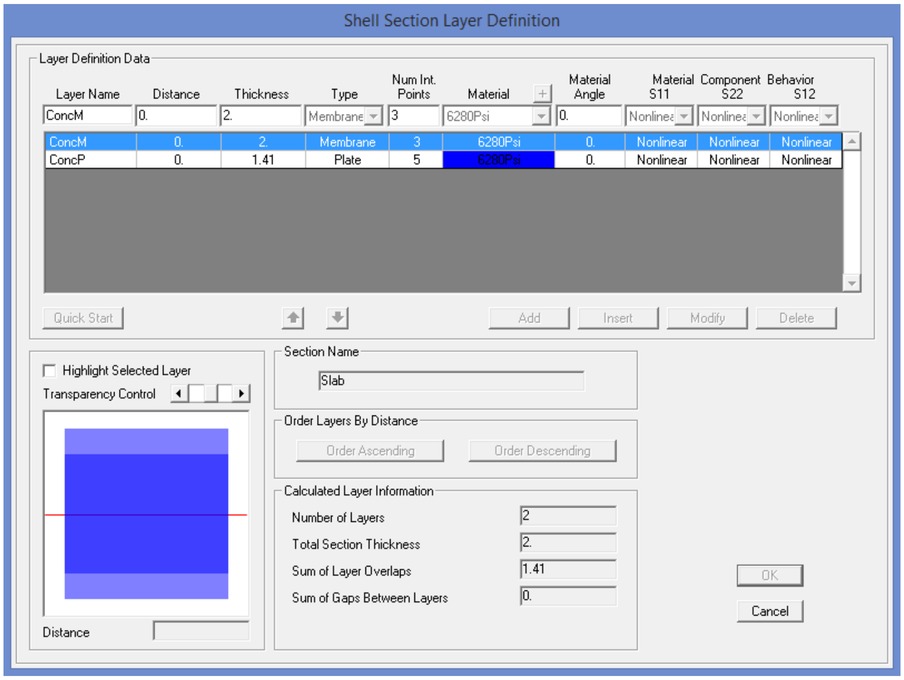

Figure 2.14: SAP2000 software shell section layer definition

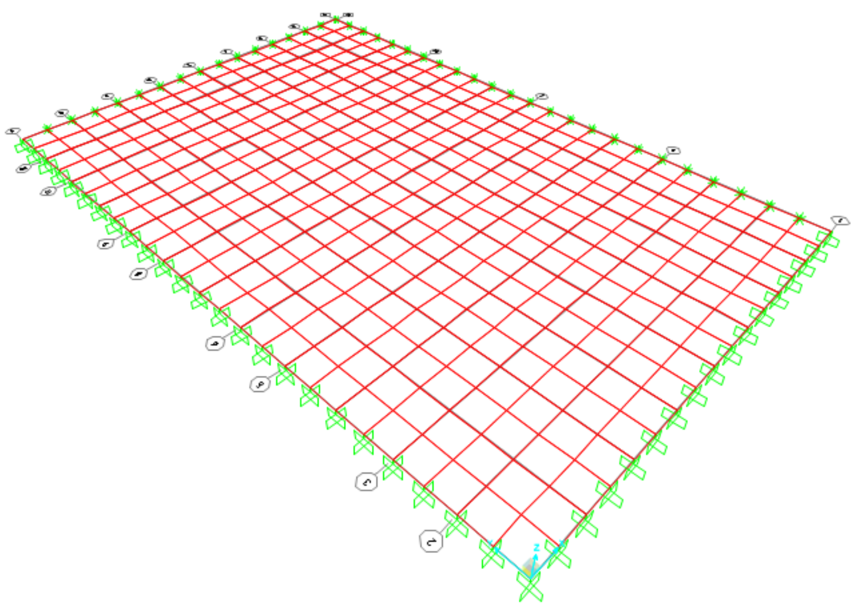

Figure 2.15: SAP2000 concrete slab model for test specimen D1

As specified in the research paper by Park et al. (1964), a uniformly distributed load was applied to the floor slab. Nonlinear analyses were conducted using both P-Delta and P-Delta plus Large Displacement options to determine which approach is best suited for this application. The computed and experimental load-deformation curves for test specimens D1 and D2 are shown in Figure 2.16. Computational analysis results for both P-Delta and P-Delta plus Large Displacement options are also provided for comparison purposes.

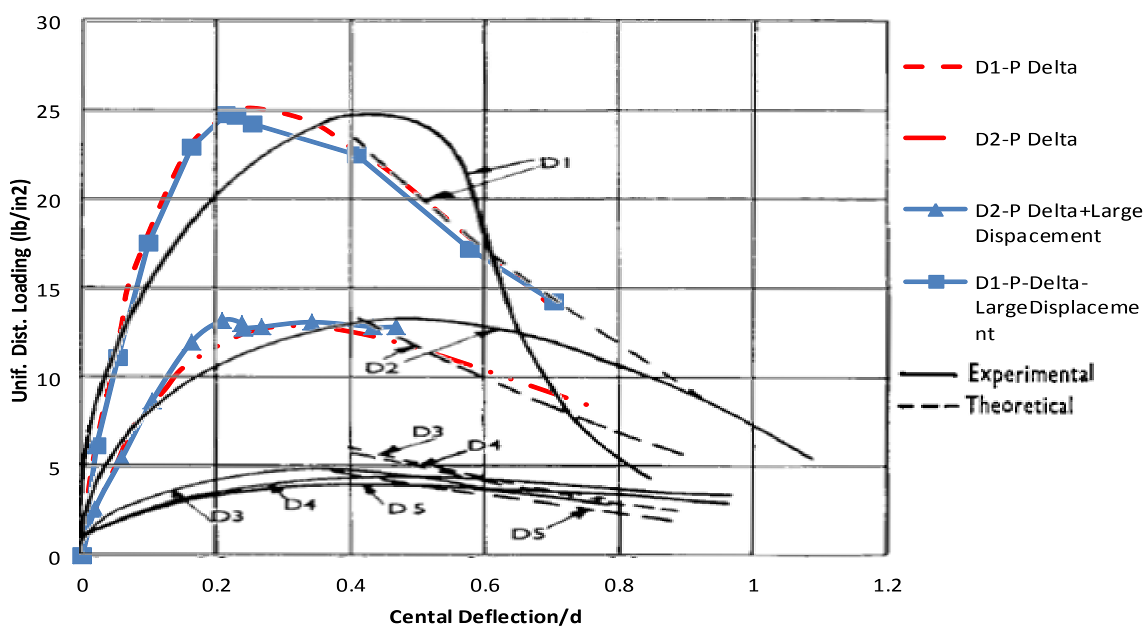

Figure 2.16: Load-deflection curves for concrete slabs

As shown in Figure 2.16, the computed results agree reasonably well with the experimental load-deformation curves. In particular, the initial stiffness and peak load values of the concrete slabs were predicted with good accuracy. The peak deformation, however, is not predicted with similar accuracy. Although, there are minor differences between the P-Delta and P-Delta plus Large Displacement analyses, the results show that both options can be used for floor slab analyses. It should be noted that SAP2000 is not a detailed finite element software and can only provide a close approximation to real response. As indicated in Chapter 1, one of the objectives of this thesis is to provide time efficient and easy-to-use methods to calculate progressive collapse resistance of structures.

The final deformed shape of the floor slab model is shown in Figure 2.17. These results were obtained using a displacement-controlled analysis of the central region of the floor slab. It should be noted that the shell elements at the perimeter of the floor slab do not translate or rotate because fixed supports are specified for all nodes around the perimeter.

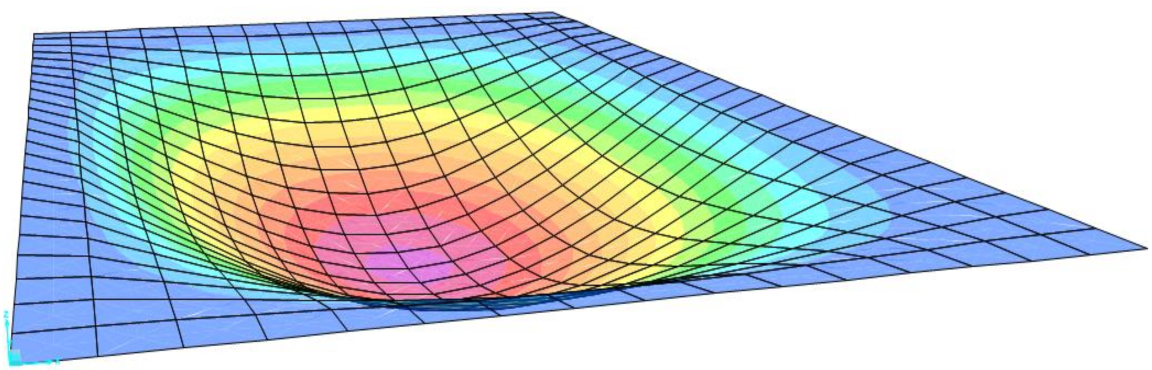

Figure 2.17: Final deflected shape of the floor slab model in SAP2000

2.4 COMPOSITE/PARTIALLY COMPOSITE/NON-COMPOSITE BEAMS

As described previously, the experimental setup used in the current research project includes a composite floor system. Although, Section 2.3 verifies the response of simplified floor slabs, it is still unclear whether the computational models can simulate the behavior of composite, non-composite, and partially composite floor beams. The purpose of this section is to propose a beam model, which can develop composite, non-composite, and partially composite action, and verify the response of the model using available experimental data.

2.4.1 Description of Composite/Partially Composite/Non-composite Beams

Composite, non-composite, and partially composite beams can be simulated using three components: (1) floor slab, (2) steel beam, and (3) shear studs. Because the simplified floor slab model was investigated in Section 2.3, the current section focuses primarily on the shear studs and composite action development. Figure 2.18 shows a sample composite beam model to provide guidance on the configuration of the floor slab, beam, and shear studs.

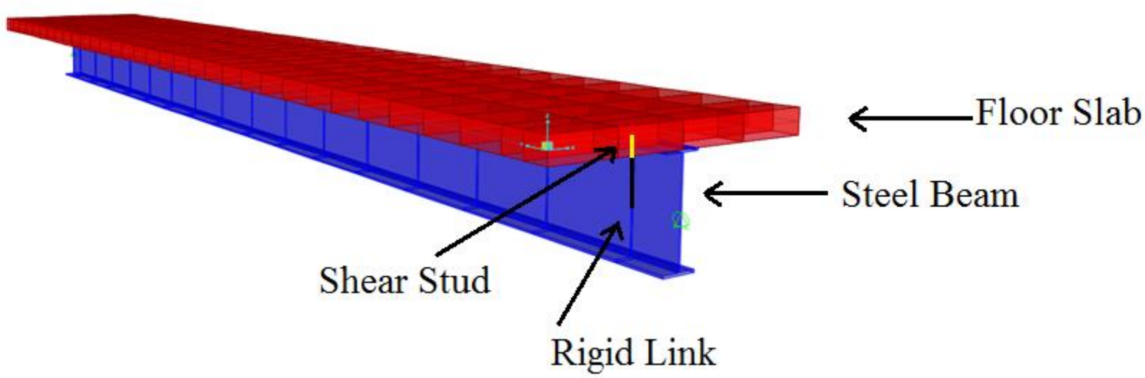

Figure 2.18: Sample composite beam model

As shown in Figure 2.18, the floor slab and the steel beam are connected to each other with shear studs, which are represented computationally using spring elements. To properly define the geometry, rigid links should be placed vertically from the centerline of the beams to the top of the beams because the beam elements are modeled along their centroidal axis. As such, the location of the beam elements is offset from the slab. After placing the rigid links, shear stud springs should connect the rigid links to the nodes of the shell elements that represent the floor slab. Thus, the final composite beam model includes beam elements, rigid links, shear stud springs, and shell elements for the floor slab. It should be noted that the beam nodes have to be aligned directly below the nodes of the floor slab so that these nodes can be connected to each other by shear stud springs.

The shear stud springs are used to develop composite action between the steel beams and the concrete slab. Behavior of the connectors can be simulated by assigning non-linear load-deformation curves to each shear stud spring in all rotational and translational directions. Once these curves are defined, these springs can carry loads under all types of loading conditions by following the same load-deformation data that was entered by the user.

Before calculating directional properties of shear studs, it is important to understand the expected behavior, which can decrease the amount of unnecessary calculations. The shear stud springs of non-composite beams should only have directional fixity in the axial direction, which rigidly connects the floor slab and beam in the vertical direction. The shear stud springs for non-composite beams should have no other prescribed capacity in any of the remaining translational or rotational directions. The shear stud springs of fully composite beams should have directional fixity in all rotational and translational directions. The shear stud springs of partially composite beams should have directional fixities in all rotational and translational directions except the plane perpendicular to the axis of the shear stud. Because shear studs primarily deform in this plane, a non-linear load deformation curve should be assigned to the shear stud springs in this plane. The reason for changing the directional properties of shear studs for different beam types, rather than changing the number of shear studs, is that the number of shear studs is dictated by the mesh size. Additional details regarding the number and directional properties of shear studs for different models will be provided in the following paragraphs.

To be able to assign the non-linear load-deformation data to the shear stud springs in the transverse directions, it is necessary to develop an analytical method that can be used to approximate the load-deformation behavior of a single shear connector. For this purpose, a force-slip relationship from research by Ollgaard et al. (1971) is taken as a reference. Validation of this model is presented in Section 2.4.2. This empirical load-slip relationship, which is given in Eq. (2.8), is based on pushout tests of shear studs without steel decking.

$$\frac{Q}{Q_u}=(1-e^{-0.71\Delta})^{\frac{2}{5}} \tag{2.8}$$

In this equation, Q is the load applied to the shear connector, Q_u is the ultimate strength of the shear connector, and Δ is the slip of the shear connector. The ultimate strength of the shear connectors can be directly calculated using the equations in the AISC Specification (AISC 2010, Section I8.2a).

Before assigning the force-slip data to the shear stud springs based on Eq. (2.8), two major problems concerning shear stud springs should be addressed. First, the number of shear stud springs along the beams and girders depends on the mesh size, and this number will typically be less than the actual number of shear studs physically present. As such, the shear force values of the force-slip data should scaled up accordingly to compensate for the difference in strength. Second, as can be seen in Figure 2.18, the length of the shear stud springs is equal to half of the floor slab thickness, which is typically less than the actual height. The shear force values of the force-slip data should be adjusted by updating the ultimate strength of the shear connector in Eq. (2.8) according to the moment of inertia value obtained in Eq. (2.9) to compensate for the difference in strength.

Equation 2.9 is the combination of two beam deflection equations, and the purpose is to find a relationship between the moment of inertias of actual and modeled shear studs by equating the deflections at the same distance from the ends attached to the beam. Terms on the left side of the equation represent the deflection of the shear stud spring with a concentrated load applied to the top, while terms on the right side of the equation represent the deflection of a uniformly loaded actual shear stud at a distance equal to the length of the shear stud spring from the end attached to the beam. Figure 2.19 illustrates the simplified drawings of the actual shear stud and shear stud spring and summarizes the terms in Eq. (2.9) for convenience.

$$\frac{P_{model}L_{model}^{3}}{3I_{model}}=\frac{w_{actual}L_{model}^{2}}{24I_{actual}}(L_{model}^{2}+6L_{actual}^{2}-4L_{actual}L_{model}) \tag{2.9}$$

In this equation, L_{model} and L_{actual} are the lengths of the shear stud spring and actual shear stud respectively, I_{model} and I_{actual} are the moments of inertia of the shear stud spring and actual shear stud respectively, w_{actual} is the uniformly distributed load on the actual shear stud, and $P_{model}=w_{actual}L_{model}$ is the concentrated load applied to the shear stud spring.

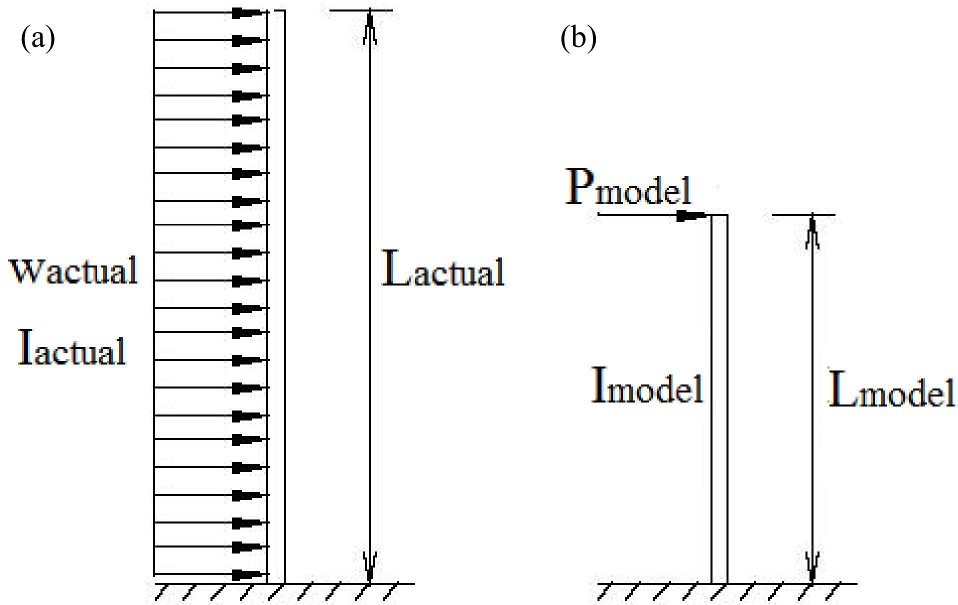

Figure 2.19: Simplified shear stud drawings: (a) actual shear stud; (b) shear stud spring

2.4.2 Verification of Composite/Partially Composite/Non-composite Beams

This section includes the work done to verify the load-deformation behavior of the simplified non-composite, partially-composite, and fully-composite beam models by comparing the results from SAP2000 with available experimental data. As with the other models studied, it is best to isolate non-composite, partially-composite, and fully-composite beams from other structural components to evaluate the advantages and limitations of the proposed modeling approach. For the current validation, data from Kwon et al. (2008) are used.

The research by Kwon et al. (2008) included five full-scale test specimens, which were composed of W30×99 steel girders and reinforced concrete slabs. All specimens were 38-ft. long, simply supported, and loaded statically with a concentrated force at midspan. Figure 2.20 shows the cross-sectional properties of the test specimen and the reinforcement details of the concrete slab. Although, various specimens with different properties were tested, only results from a non-composite and a partially composite specimen are used for validation process. In both of these specimens, shear studs were used as connectors.

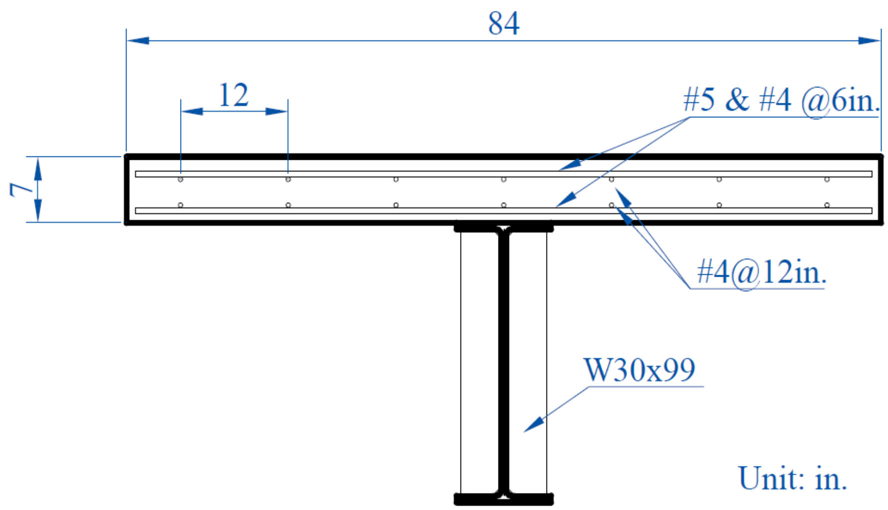

Figure 2.20: Details of the test specimen. (From Kwon et al. (2008))

Table 2.2 shows the material properties of the two full-scale test specimens that are used to validate simplified non-composite and partially composite beam models.

Table 2.2: Properties of rectangular concrete slab test specimens

Specimen	**Girder yield/ultimate stress (ksi)**	**Reinforcement yield/ultimate stress (ksi)**	**Cube strength of concrete (psi)**	**Shear connection ratio (%)**
Non-Composite (NON-00BS)	58.9/78	61.6/103.5	6250	0
Partially Composite (HASAA-30BS)	58.9/78	57.6/99.2	3610	30

The concrete slab portion of the test specimens in Figure 2.20 was modeled using nonlinear layered shell elements. In addition to the concrete layers, steel membrane and plate layers were included in the Shell Section Layer Definition form in Figure 2.15 to account for the reinforcing steel. An easy way to keep track of these layers is to use the Quick Start button, and then manually update the Shell Section Layer Definition form. After defining the shell section layers, a W30×99 steel girder was created by using frame elements. These frame elements were connected to the concrete slab with rigid and shear stud springs as shown in Figure 2.18.

Plastic analysis of frame elements is limited within SAP2000. Although, it is necessary to characterize the yield criteria of a nonlinear frame element, the definition of the nonlinear material behavior in SAP2000 does not directly enable plastic behavior. As a result, users are required to manually assign plastic hinges to the critical locations of the frame elements. The Frame Hinge Property Data form, shown in Figure 2.21, can be used to manually or automatically input the plastic behavior parameters.

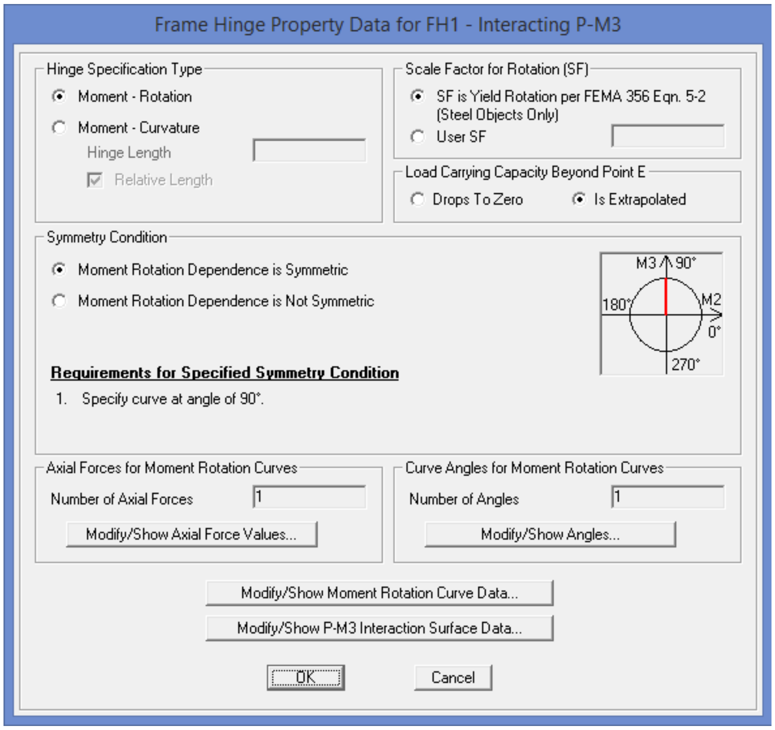

Figure 2.21: The Frame Hinge Property Data form in SAP2000

For consistency with Kwon et al. (2008), the analyses were conducted using displacement-controlled loading at the midspan of the concrete slab. Only nonlinear P-Delta plus Large Displacement analyses were used during the computational assessment of the composite, non-composite, and partially composite beam models because the results given in Sections 2.2 and 2.3 showed that this option gives the most accurate results for both frame and shell elements in a single analysis. The computed and experimental load-deflection curves for non-composite and partially composite beams are shown in Figure 2.22.

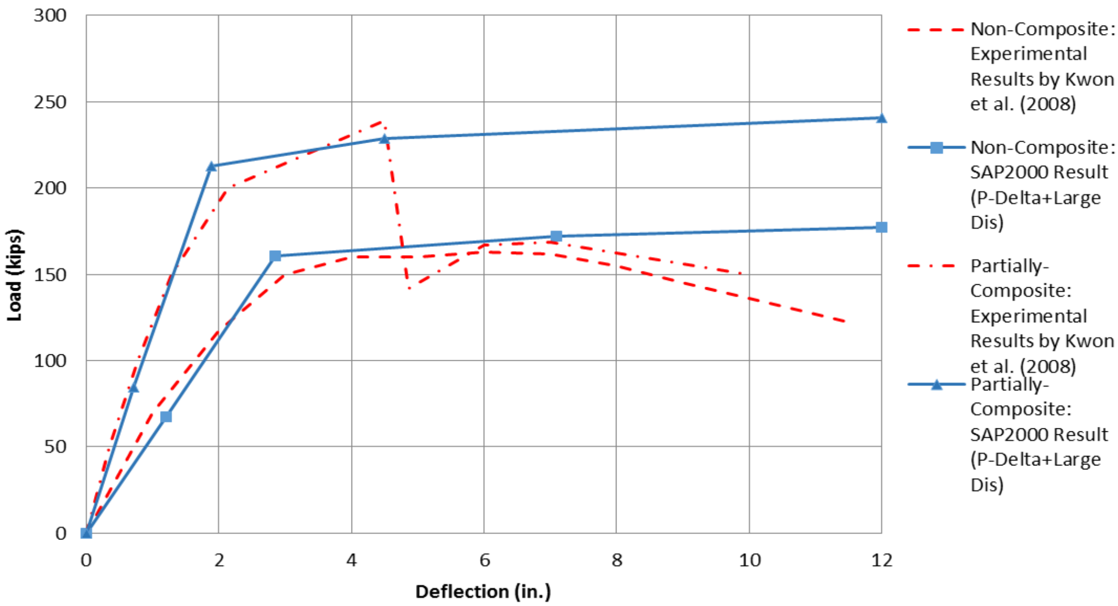

Figure 2.22: Load-deflection curves for non-composite and partially composite beams

As can be seen in Figure 2.22, the computed and experimental load-deformation curves show good agreement. In particular, the initial stiffness and peak load values match closely. The SAP2000 model and experimental results for the non-composite beam start differing after the girder experienced local buckling at large deflections. Because the SAP2000 frame elements cannot simulate local buckling of the beam flange and the web, this difference is expected. The SAP2000 model and experimental results for the partially composite beam match satisfactorily up to the point where the shear stud connectors failed (7-inch). After that point, because the computational model is not able to calculate the failure displacement of the shear studs, the SAP2000 model continues to deform with the same load-deformation slope.

The final deformed shape of the partially composite beam model is shown in Figure 2.23. The deformed shape of the non-composite beam model is not provided since the deformed shapes of the both models are similar. However, it should be noted that the partially composite beam has less slippage at the steel-concrete interface compared to the non-composite beam.

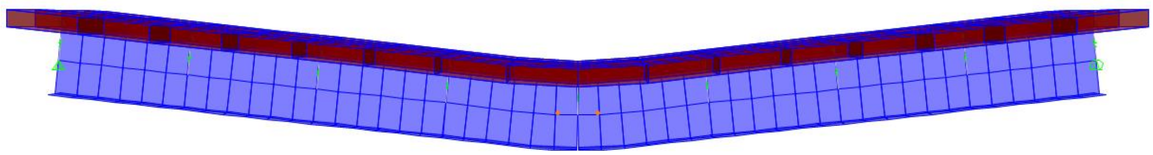

Figure 2.23: Final deflected shape of the partially composite beam in SAP2000

2.5 STEEL GRAVITY FRAME SYSTEMS WITH SINGLE-PLATE SHEAR CONNECTIONS

This section provides step-by-step guidance on how to use the methods described in the previous sections to create a full steel gravity frame model with composite floor system and how to determine the progressive collapse resistance of this structure by using SAP2000. As described in the previous sections, all structural components have been modeled and analyzed separately from the rest of the structure to minimize the interaction among the components. The main purpose of this section is to validate the combined behavior of the structural components before modeling and analyzing the actual test specimen of the project. After creating and analyzing the model, computational analysis results from a detailed finite element program (LS-DYNA) are used for validation.

2.5.1 Description of Steel Gravity Frame Systems with Single-Plate Shear Connections

A steel gravity frame with composite floor system generally consists of five major components: (1) beams, (2) columns, (3) floor slab, (4) shear studs, and (5) steel connections. For the content of this thesis, this section will primarily focus on structures that have similar structural properties with the actual test specimen of the project. As a result, the structural components described in the previous sections are used during the modeling process.

2.5.2 Verification of Steel Gravity Frame Systems with Single-Plate Shear Connections

This section includes the work done to verify the load-deformation behavior of the simplified steel gravity frame models with composite floor systems by comparing the results from SAP2000 with the detailed finite element model results (LS-DYNA) from Main and Sadek (2012). To create a full steel gravity frame model with composite floor system, the connections located at the intersections of beams and girders are modeled with the simplified connection modeling approach from Section 2.2, the composite floor system is modeled with the simplified floor slab modeling approach from Section 2.3, and the shear studs are modeled using the simplified composite/non-composite/partially composite beam approach from Section 2.4. The combined model including all these features is shown in Figure 2.24.

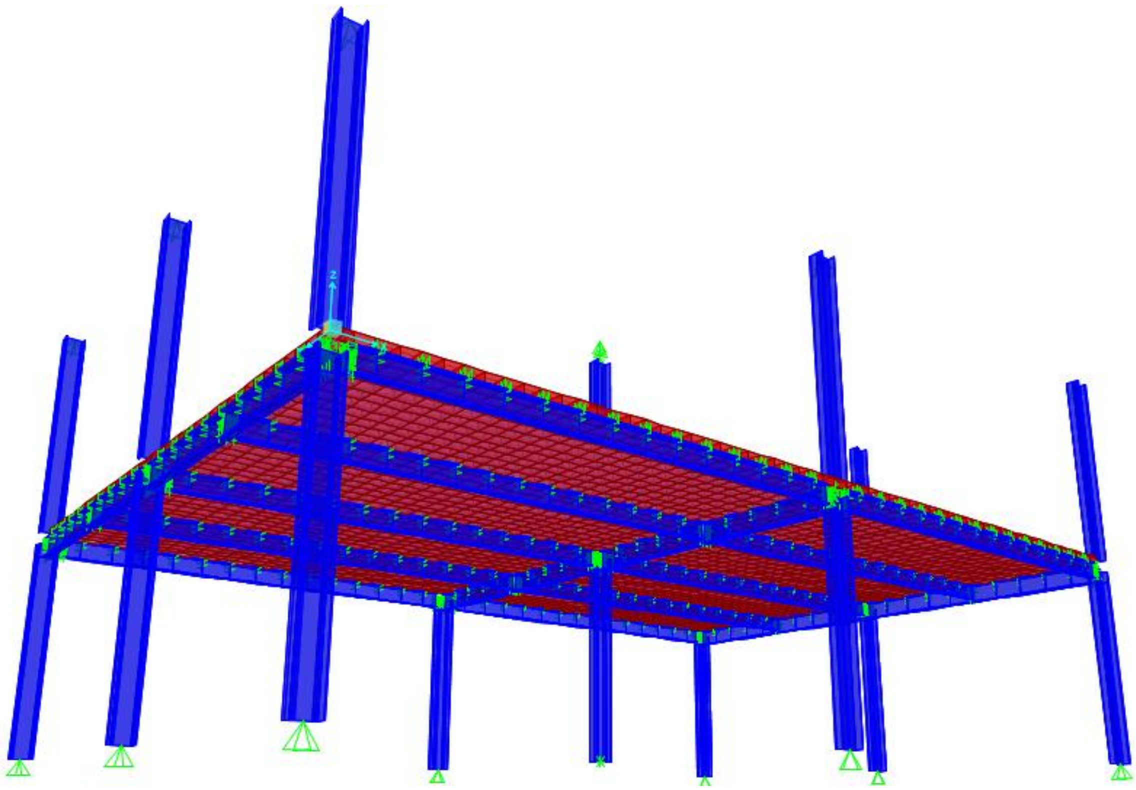

Figure 2.24: Sample steel gravity frame model with composite floor system

The research by Main and Sadek (2012) includes a steel gravity frame model with 2 bay × 2 bay composite floor system. Figure 2.25 shows the plan view of the gravity framing system model. For this structure, the floor system is assumed to be connected to the beams and girders by shear studs only, and the support of the central column in the vertical direction is removed. During the analysis, a displacement controlled load is applied statically to the center of the floor slab to simulate a quasi-static column loss scenario.

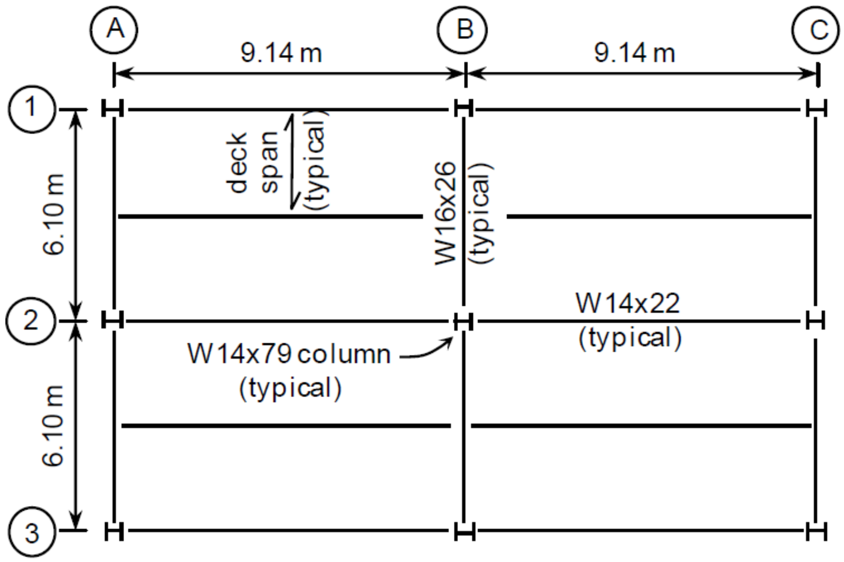

Figure 2.25: 2-bay × 2-bay gravity framing system. (From Main and Sadek (2012))

In the gravity frame of the structure, the steel beams are connected to the columns using simple single-plate shear connections, which are illustrated in Figure 2.26(a). These shear tab connections are modeled using Link/Support elements in SAP2000 and are shown in Figure 2.26(b). Further guidance on connection modeling can be found in Section 2.2. In addition to the steel gravity frame in Figure 2.25, the research by Main and Sadek (2012) also includes a floor slab, which is illustrated in Figure 2.26(c). By using the guidance in Section 2.3, a floor slab model, which is shown in Figure 2.26(d), is created by using nonlinear layered shell elements in SAP2000.

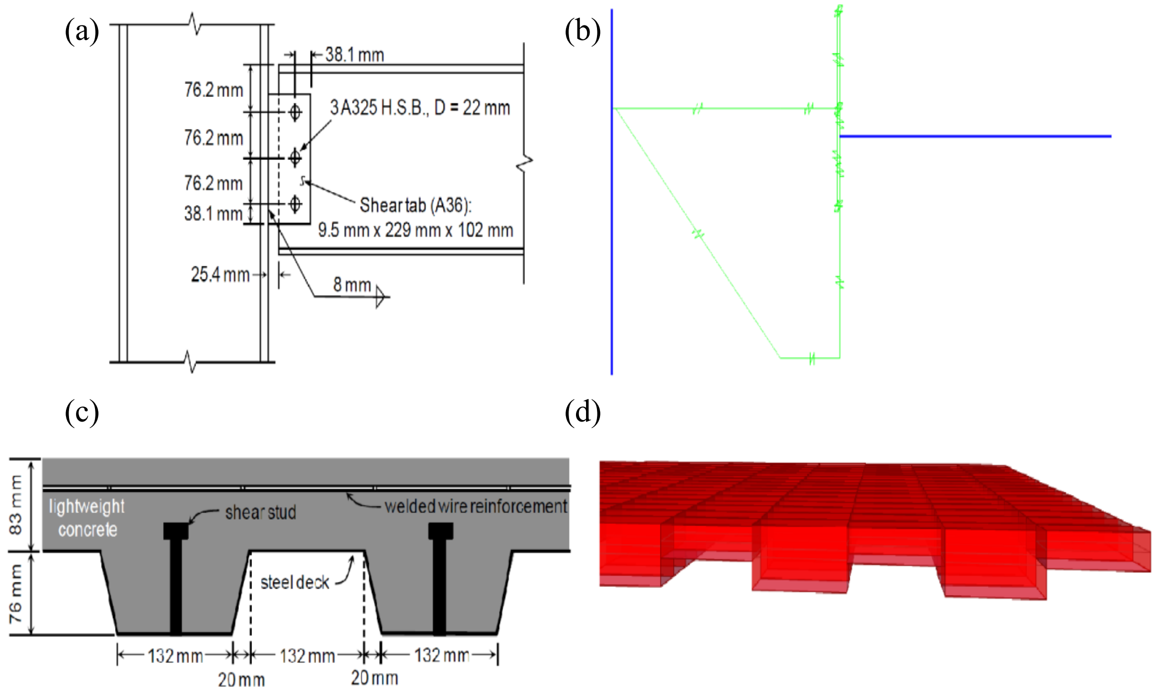

Figure 2.26: Detailed and simplified models: (a) details of single-plate shear connection (From Main and Sadek (2012)); (b) simplified single-plate shear connection model; (c) cross sectional view of the floor slab (From Main and Sadek (2012)); (d) simplified floor slab model.

Figure 2.27 shows the detailed finite element models created by Main and Sadek (2012). Although the analyses that were performed by Main and Sadek (2012) were dynamic, the prescribed displacement and the applied load were increased gradually to maintain quasi-static loading conditions.

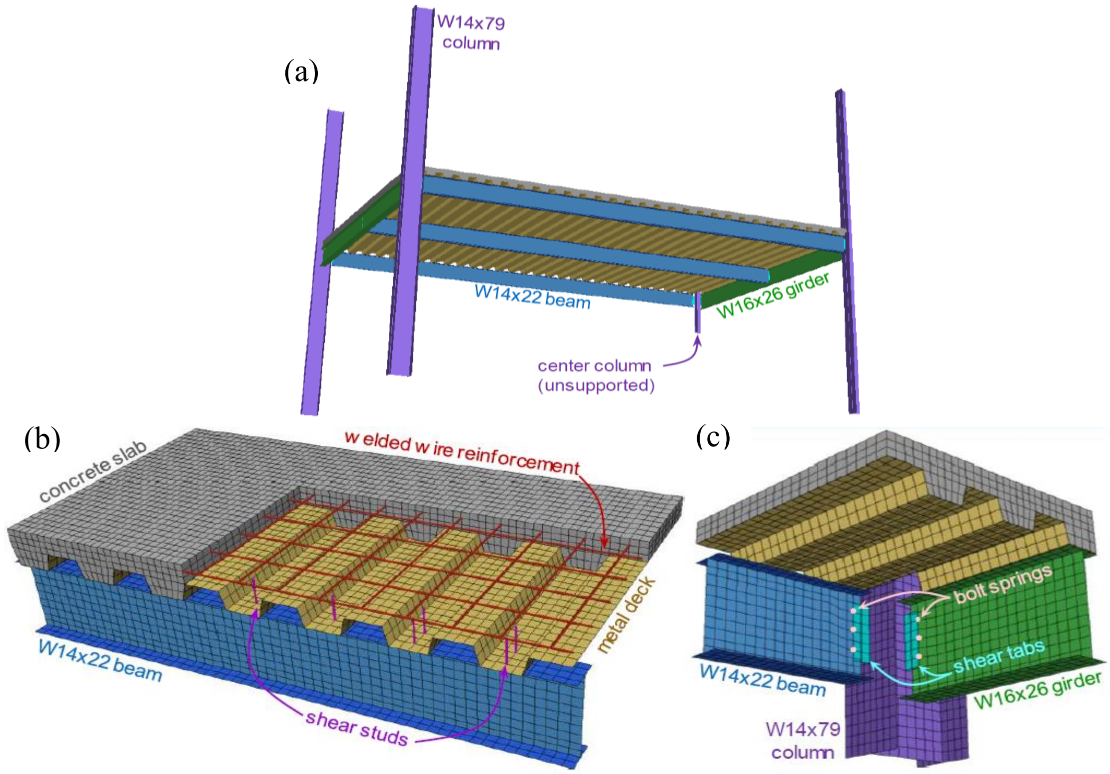

Figure 2.27: Detailed LS-DYNA models: (a) detailed model of composite floor system; (b) composite floor slab; (c) beam-to-column connections. (From Main and Sadek (2012))

As can be seen in Figure 2.24 and Figure 2.27(a), during the modeling in both SAP2000 and LS-DYNA, all columns other than the center column are extended one story above and below the floor slab and pinned at the ends. For the LS-DYNA model, the concrete in the floor slab is modeled with solid elements, the welded wire reinforcement and shear studs are modeled with beam elements, and the profiled steel deck and the wide flange beam and column sections are modeled with shell elements. Table 2.3 summarizes the material properties that are used during the modeling of structural components.

Table 2.3: Material properties of structural components

Component Name	ASTM Designation	Yield Strength, Min. Fy (MPa)	Tensile Strength, Min. Fu (MPa)	Cube strength (MPa)
Steel Plates	A36	250	400	-
Wire Reinforcement	A82	450	515	-
Shear Studs	A108	350	450	-
Steel Deck	A653, Gr 33	230	310	-
Rolled Steel Shapes	A992	345	450	-
Concrete Slab	-	-	-	20.7

SAP2000 cannot directly simulate element failure. Although the nonlinear load-deformation data can be adjusted in a way that the load carrying capacity decreases to zero after reaching a specific deformation value, this is not a practical solution because elements can start carrying load again if deformations decrease. The Staged Construction tool provides a direct way to remove elements permanently from the model. This is a relatively new feature in SAP2000 and only available for Ultimate versions of the software. The Staged Construction tool allows users to remove elements from the model when a specific force or deformation value is reached. Nonetheless, this tool has two major disadvantages. First, users must know the deformation of the system when an element has to be removed. Second, this method is computationally inefficient because it requires many intermediate loading steps, which are described further below. Despite these limitations, the Staged Construction tool is used to remove elements from the SAP2000 models.

A proposed bolt removal procedure using the Staged Construction tool is composed of several steps. First, the analysis should run without having a bolt removal intermediate step. After getting the load deformation curve for the initial conditions, the user should record the amount of displacement at the central column when the first bolt reaches its deformation limit. Second, the user should change the maximum displacement value of the displacement controlled loading to the value which is recorded in the first step. Also, a new Staged Construction load case should be defined to remove the failed bolts from the system. These two steps should be repeated by adding more load cases to the analysis until the targeted displacement is reached.

The analysis is conducted by applying a displacement-controlled loading to the central column as specified in the research paper by Main and Sadek (2012). As described above, only nonlinear, P-Delta plus Large Displacement type analysis is used during the computational assessment of the model. The load-deflection curves from the simplified SAP2000 model and detailed LS-DYNA model are shown in Figure 2.28.

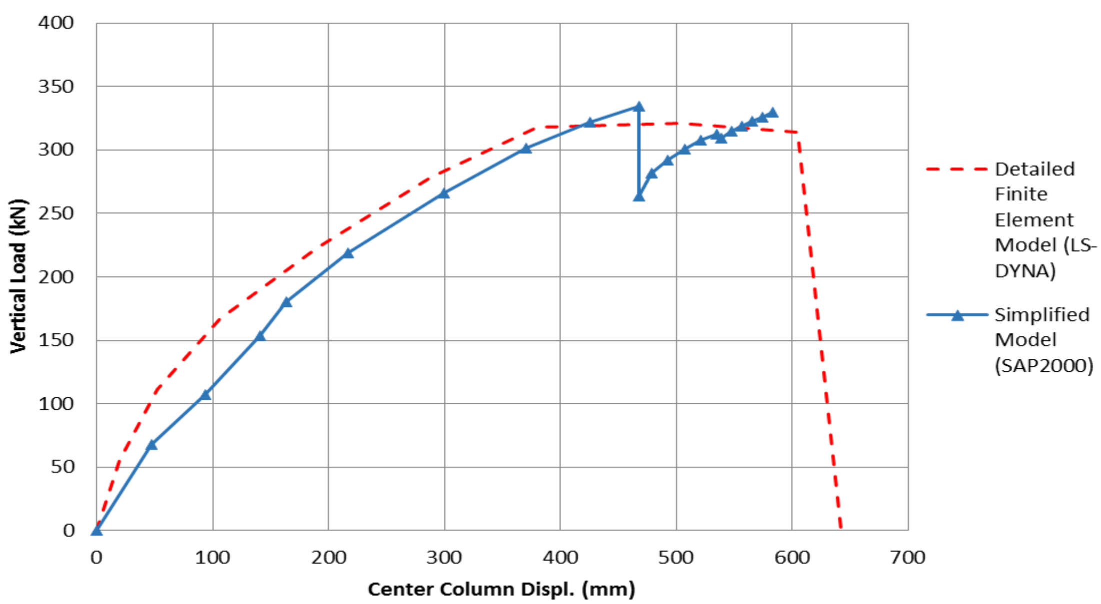

Figure 2.28: Load-deflection curves for steel gravity frame model with composite floor system

Figure 2.28 compares the load-deformation curves for the detailed and simplified models. In general, the two modeling approaches show good agreement, which suggests the simplified modeling approach is suitable for modeling the progressive collapse behavior of composite floor systems. The reason that the detailed modeling approach has slightly larger strength compared to the simplified modeling approach is explained by Main and Sadek (2012) as follows: "The slightly larger strength predicted by the detailed model is partly a consequence of the resistance of the steel deck to extension in the across-rib direction. This resistance is neglected in the reduced model by using weak strips with no contribution from the steel deck". Results from the SAP2000 and LS-DYNA models start differing after a sudden decrement at the load carrying capacity of the SAP2000 model at a central column displacement of 468mm. The reason for this sudden drop is due to the removal of bolt springs, which were oriented in the East-West direction of the central column. Because SAP2000 cannot directly simulate component failure, these bolts were removed manually by using the Staged Construction tool described previously. By effectively using this tool, a user can capture the response of the structure after bolt failure by using SAP2000. At a central column displacement of 583.3mm, the SAP2000 model stopped converging because it was not able to simulate sudden and major strength decrements. Because of the general good agreement between the proposed approach and the detailed analyses of Main and Sadek (2012), and because the simplified model gives a conservative estimate of deformation capacity, this modeling approach is considered acceptable and used for the collapse assessment of the actual test specimen in Chapter 3.

The final deformed shape of the steel gravity frame model with composite floor system is shown in Figure 2.29. As the figure shows, after removing the vertical support of the central column and applying a displacement controlled load, the central column moves downward and pulls the connected beams and floor slab inward. Also, the orientation of the beams and columns after deformation shows that the overall response of the structure under a column removal scenario is primarily controlled by the axial and bending deformations of the connections due to the development of catenary action.

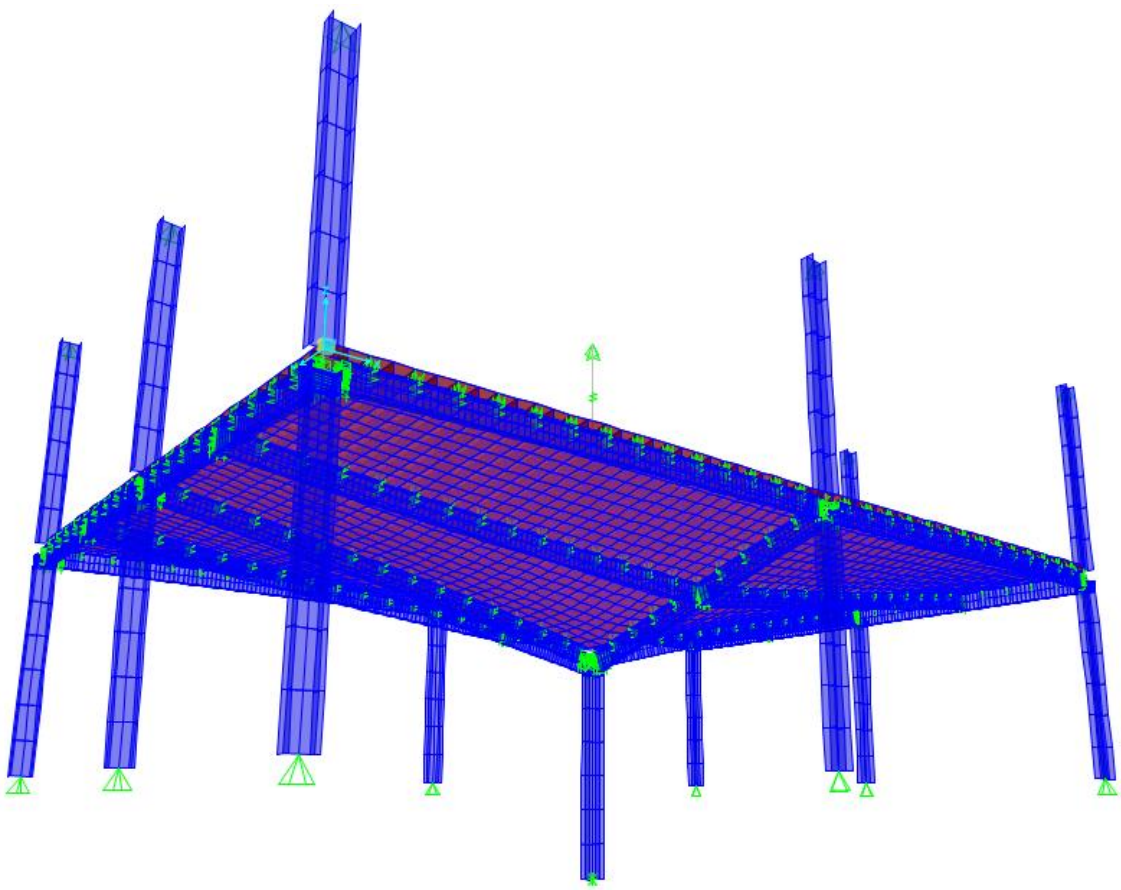

Figure 2.29: Final deflected shape of the steel gravity frame model with composite floor system in SAP2000

2.6 SUMMARY AND CONCLUSIONS

In this chapter, a simplified modeling approach was described for analysis of connection models, floor slabs, composite/non-composite/partially composite floor beams, and steel gravity frames with composite floor systems. Although most of the methods used in this chapter were based on previous research, significant work was done to modify the current approach to fit into simple structural analysis software like SAP2000. Throughout the chapter, the load-deformation behavior of the simplified models was compared with detailed finite element analyses and available experimental data to establish confidence in the simplified modeling approach. After comparing various simplified models, it is clear that steel gravity frames with composite floor systems can be accurately simulated by using simple structural engineering software up to the point of first element failure. Moreover, it was shown that the response of the simplified models are conservative compared to the true behavior of the structure, which is acceptable in terms of structural design.

Consequently, the proposed simplified modeling approach enables complicated systems to be analyzed much more efficiently than the detailed finite element modeling approaches used in previous studies. Moreover, practicing civil/structural engineers can save significant time and money and check the progressive collapse resistance of their preliminary and final designs in a quick and accurate way by following the simple steps provided in this chapter. In Chapter 3, the methods and approaches that were described in Chapter 2 will be used to determine the progressive collapse resistance of the test specimen that was constructed and tested during the current research project.

CHAPTER 3

Description, Modelling and Analysis of the Test Specimen

3.1 GENERAL

This chapter consists of two major sections. Section 3.2 includes a brief description of the test specimen, the experimental setup, and the loading procedure associated with the large-scale testing program conducted at the University of Texas at Austin. Additional information about the test can be found in the thesis by Hull (2013). Section 3.3 shows how the methods and approaches in Chapter 2 are applied to the prototype structure during the computational analysis procedure. The assumptions made during the modeling process are also listed in this section. Moreover, all computational analysis results and experimental data related to the test specimen are provided in Section 3.3.

3.2 DESCRIPTION OF THE TEST SPECIMEN, EXPERIMENTAL SETUP, AND THE LOADING PROCEDURE

A steel gravity frame structure with a composite floor system was designed for this project with the assistance of engineers from Walter P. Moore and constructed at the Ferguson Structural Engineering Laboratory of University of Texas at Austin. Dimensions of the structure were adjusted according to the practical constraints of the experimental setup. Figure 3.1 shows a picture of the test specimen.

Figure 3.1: Picture of the test specimen

A plan view of the test specimen and the layout of the members are illustrated in Figure 3.2. As shown in the figure, the steel gravity frame was 2-bays long by 2-bays wide, with a span of approximately 15 ft. in each direction. Each beam and girder was connected to the perimeter ring beam. Other than the primary beams and girders, the steel gravity frame also included secondary floor beams that were located at the midspan of each bay in the east-west direction. All frame members were W-shapes made from A992 steel.

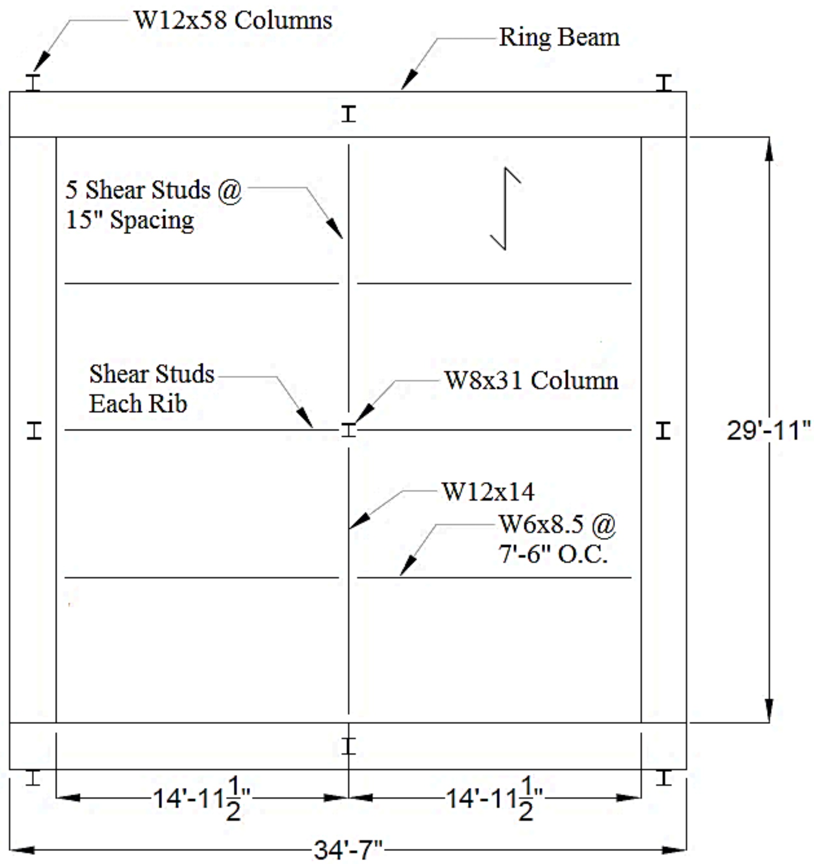

Figure 3.2: Plan view of test specimen

The central column, which was a W8×31, was supported with a long-stroke linear actuator. During the test, the actuator was fully disengaged from the column base to simulate a static column loss scenario. The self-weight of the column can be assumed as a 575 lb. point load, which was applied to the center of the floor system.

An external test frame, which included the ring beam, outer columns, and bracing members, was used to provide lateral restraint and system stability to the structure. The main purpose of the ring beam was to provide boundary conditions at the edges of the test specimen that represent the effects of neighboring bays in an actual steel-framed structure. Figure 3.3 illustrates the cross-sectional properties of the ring beams which were located in the east-west and north-south directions, respectively. Other parts of the external test frame were mainly responsible for supporting the test specimen floor system.

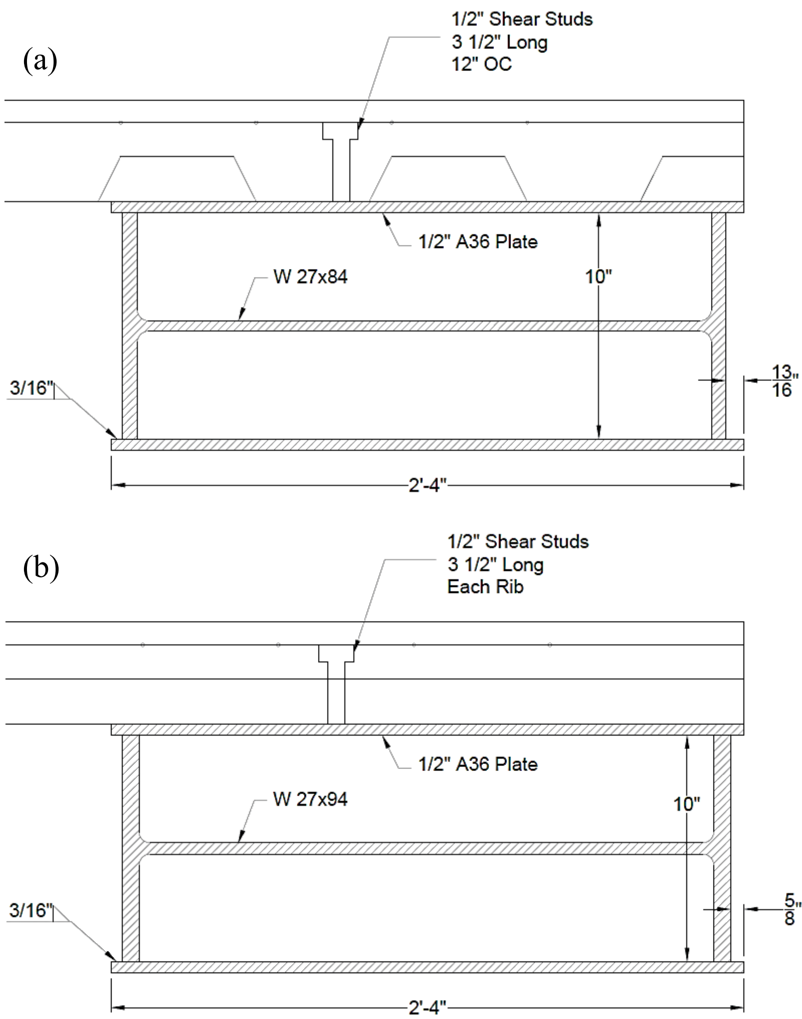

Figure 3.3: Ring beam details: (a) east-west ring beam; (b) north-south ring beam. (From Hull (2013))

The primary beams and girders of the steel gravity frame were connected to the structure with double-angle connections. The secondary beams were connected to both girders and ring beams with simple shear tab connections. Figure 3.4 illustrates the cross-sectional properties both types of connections. The double-angle connections in Figure 3.4(a) were welded to the beams and girders using 3/16-inch fillet welds with E70 electrodes and bolted to the column with two A325 bolts. L3×2-1/2×3/16 angles were used to connect primary beams and girders to the ring beam, while 2L 2×2-1/2×3/16 angles were used to connect primary beams and girders to the central column. Aside from the difference between angle sections, the same connection detail was used for all angle connections. The simple shear tab connection detail in Figure 3.4(b) was used for both secondary beam-to-girder and secondary beam-to-ring beam connections. All angles and plates were specified to be ASTM A36 steel.

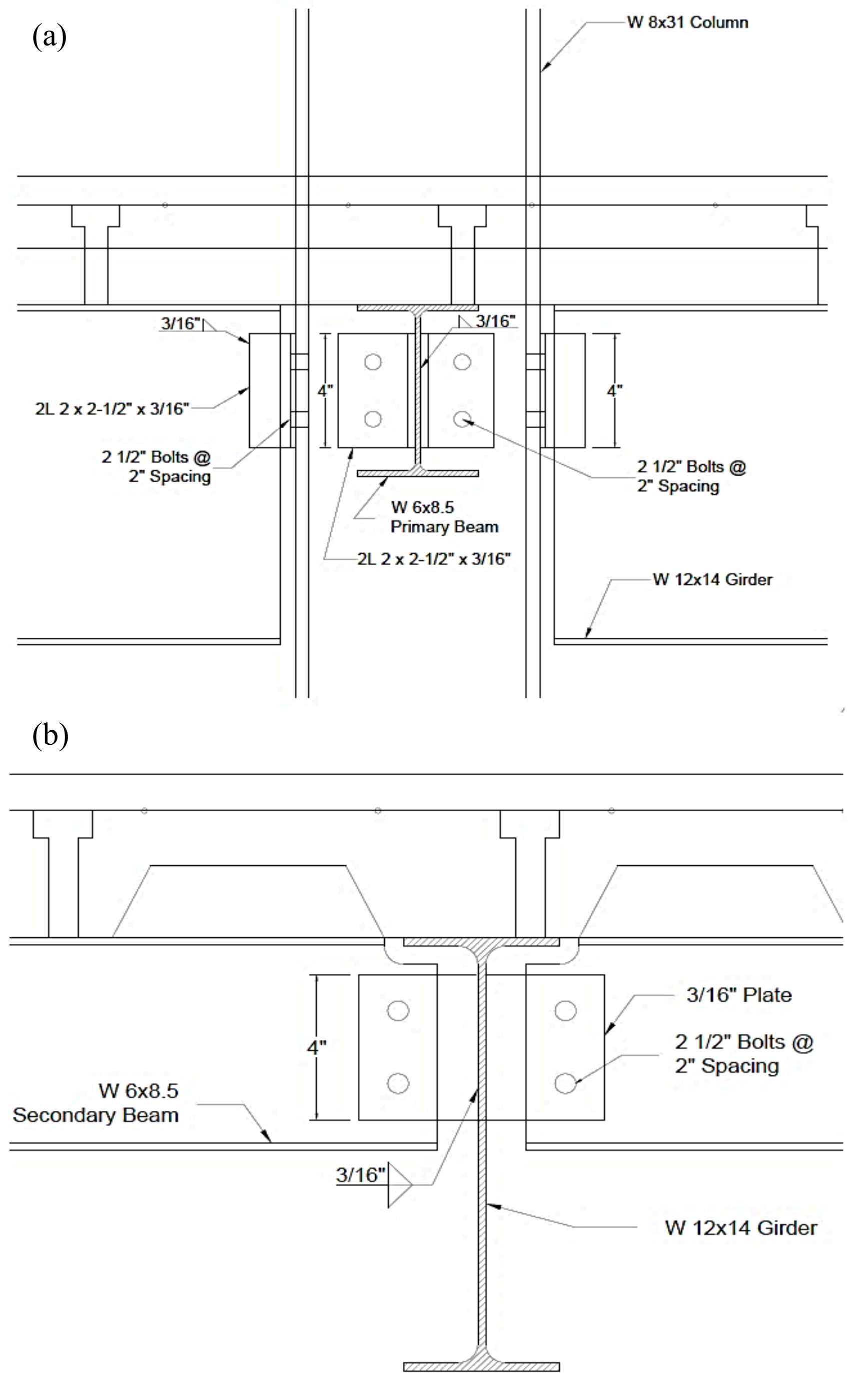

Figure 3.4: Connection details: (a) beam-to-column double angle connection; (b) secondary beam-to-girder shear tab connection. (From Hull (2013))

A 4.5-inch thick cast-in-place concrete floor slab with 22-gage 2-inch corrugated steel decking was placed on top of the steel girders and beams to develop a composite floor system as shown in Figure 3.5. The compressive strength of the concrete was specified as 3.5 ksi. The ribs of the corrugated steel decking were parallel to the W12×14 beam and spanned 7.5 ft. between the W6×8.5 beams. Also, welded wire reinforcement and short #3 extra reinforcing bars were placed into the concrete slab to provide temperature, shrinkage, and cracking resistance. The extra reinforcing bars, which were placed transverse to the girder, were included along the main girder and east and west edges of the floor system with a 12-inch spacing, since these sections were negative moment regions.

As shown in Figure 3.5, shear studs with a ½-inch diameter and 3.5-inch length were used for the floor system and placed with a clear cover of 1 inch. The shear studs were welded at each low flute with a spacing of 15 inches along the girders and 12 inches along the ring beam.

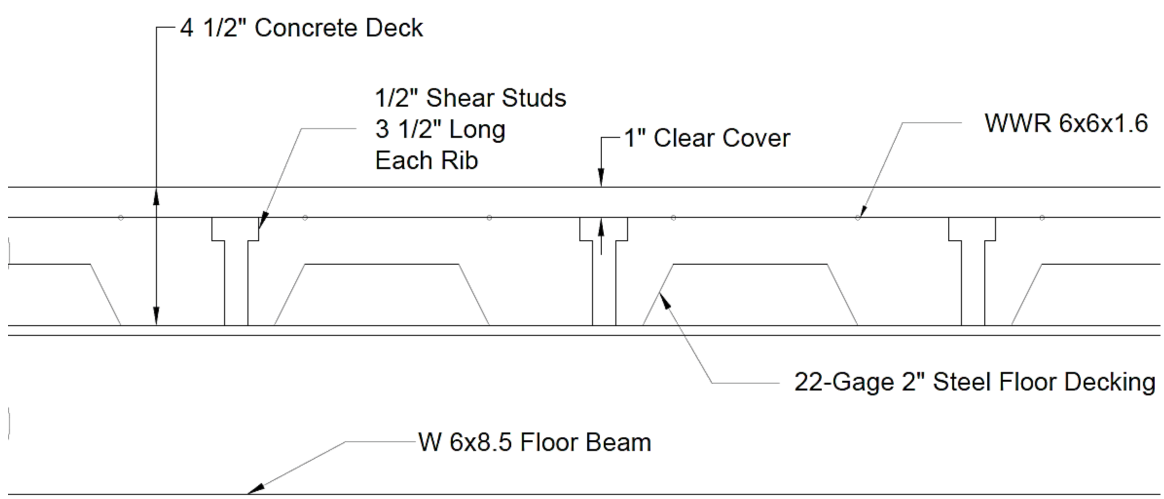

Figure 3.5: Composite floor system (From Hull (2013))

The loads and load combinations in Table 3.1 were considered during design and testing. Even though the structure did not include any special design or detailing provisions to mitigate progressive collapse, the progressive collapse design loads are listed for comparison purposes.

Table 3.1: Test Specimen Design Load

Load Type	**Load Combination**	**Load (psf)**
Dead	DL	75
Live	LL	50
Ultimate Gravity Load	1.6LL + 1.2DL	170
Progressive Collapse Design Load	0.5LL + 1.2DL	115
Amplified PC Design Load	1.33(0.5LL + 1.2DL)	155

During testing, the progressive collapse design load was applied to the structure before removing the column. The main purpose of applying this load in advance was to force the structure to redistribute the forces when the vertical support of the central column was removed. After removing the support of the central column, the floor system remained intact. To determine the collapse load, water was added to plywood buckets, which were located on the top of the floor slab. The experimental load-displacement data of the test specimen is provided in Section 3.3.

3.3 COMPUTATIONAL ANALYSIS OF THE TEST SPECIMEN

This section includes the work done to computationally simulate the load-deformation behavior of the test specimen by using SAP2000. In addition to the verification process in Chapter 2, the load-deformation behavior of the different parts of the simplified SAP2000 model are compared with sophisticated finite element models that successfully predict the behavior observed in physical testing to establish further confidence in the simplified models. These detailed finite element models of the test specimen were developed by Imperial College of London, who were important contributors to the research team. After the analysis of the full structural model in SAP2000, computational analysis results are also compared with the experimental data.

One of the biggest challenges of modeling and analyzing steel gravity frame structures with composite floor systems is numerical convergence. During each step of the analysis, SAP2000 tries to establish equilibrium. Models with large numbers of shell, link, and frame elements require significant time and calculation steps to reach equilibrium, and in many cases the solution stops converging after only a partial application of the desired load or deformation level is applied. To reduce computational demand, symmetry was utilized by modeling only one-fourth of the test structure and enforcing appropriate boundary conditions. Figure 3.6 illustrates the plan view of the computational model and summarizes the essential boundary conditions. The boundary conditions are assigned to the edge nodes of both the shell and frame elements.

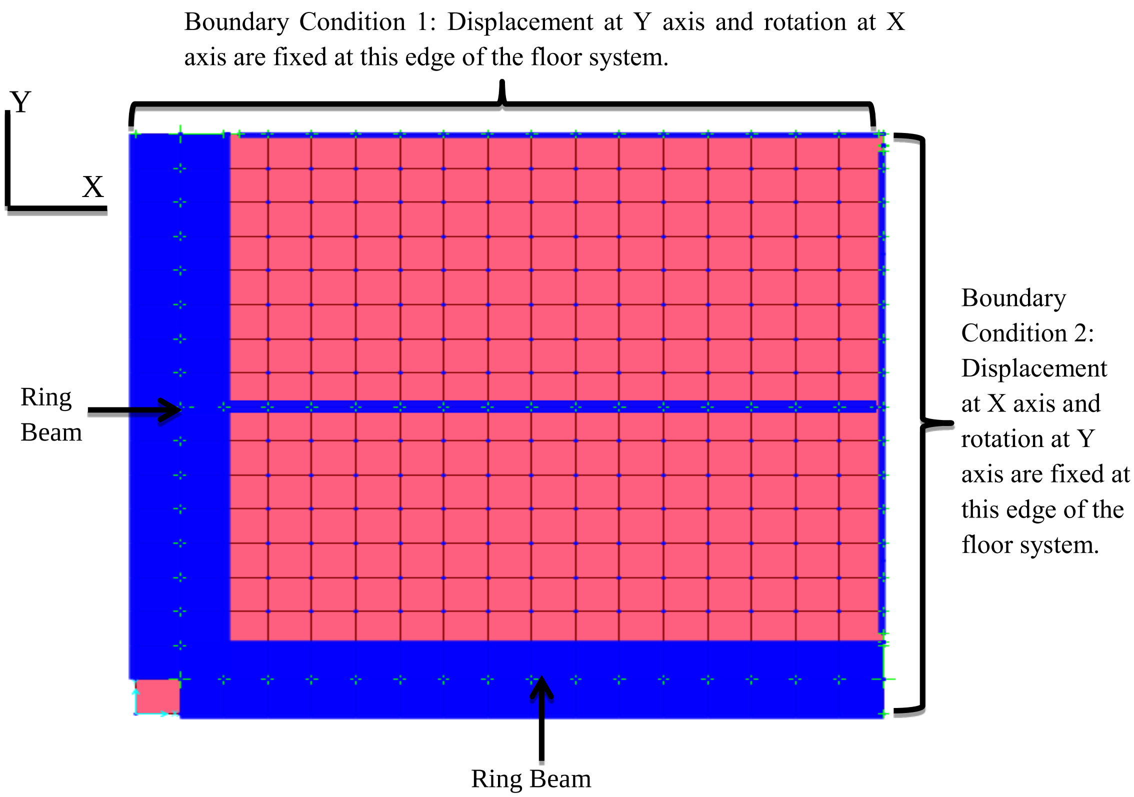

Figure 3.6: Plan view of the SAP2000 model and summary of essential boundary conditions

The steel gravity frame model with composite floor system in Figure 3.6 is composed of a single bay and has a width and length of approximately one-half of the test specimen. Only half of the frame sections, which are located at the north and east edges, are modeled to exactly simulate one-fourth of the structure. The ring beam and secondary beam sections are modeled without any section modifications.

As mentioned in Chapter 2, the adopted simplified modeling approach allows the use of frame elements to simulate beams, girders, and columns. Consequently, all primary and secondary beams, girders, and ring beams are modeled by using the frame element tool within the SAP2000 software. Moreover, the original cross-section details and material properties are used to be consistent with the test specimen.

The exterior columns of the test specimen are modeled with fixed supports, although these connections can be more appropriately represented by pinned conditions. This decision was made with the purpose of reducing the number of degrees of freedom at the supports and increasing the computational efficiency. Furthermore, the choice to use fixed column supports was made after conducting several analyses and observing that the column boundary conditions did not significantly affect the load-deformation response of the simplified model of the test specimen. The interior column is modeled with a support which is free to displace only in the vertical direction. During analysis, displacement control is used at the support of the interior column in the vertical direction to simulate a static column loss scenario.

As indicated previously, the test specimen included double-angle and simple shear tab connections. For the double-angle connections, which were used to connect the primary beams and girders of the steel gravity frame to the structure, the simplified connection model in Figure 2.2(b) is used. It should be noted that no force-deformation relationship was provided in Chapter 2 for bolt rows of double angle connections. The behavior of these connections under tension loading was obtained by creating and analyzing a detailed finite element model using ADAPTIC (Izzuddin (1990)). Instead of modeling the full steel connection, a single angle and a single bolt were modeled, and the response of the model was extrapolated to predict the response of the full connection. Figure 3.7 shows the detailed finite element model created by Imperial College researchers. Additional detailed information about modeling and analyzing angle connections can be found from previous research by Vlassis et al. (2007).

For the simple shear tab connections, which were used to connect the secondary beams to both girders and the ring beams, the simplified connection model in Figure 2.1 is used. Because the expected failure mode of the simple shear tab connections was found to be bearing/tearout failure, the nonlinear force-deformation relation proposed by Rex and Easterling (2003) is used to predict the stiffness of bolt row components. All other properties of both types of connections are calculated and assigned by following the steps described in Section 2.2.

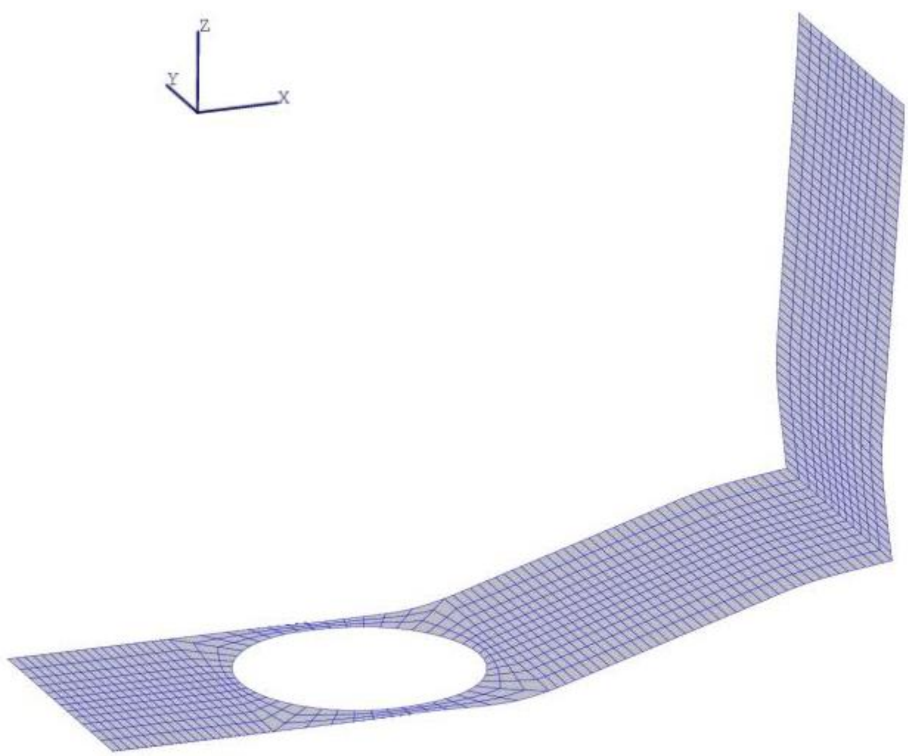

Figure 3.7: Detailed ADAPTIC model for the double angle connection. (From Imperial College of London researchers (Unpublished))

The corrugated steel decking, concrete floor slab, and welded wire reinforcement are modeled by following the steps described in Section 2.3. As described previously, the nonlinear layered shell element tool of the SAP2000 software is used for this process.

Figure 3.8 shows the shell section layer definitions for the strong and weak strips. The extra reinforcing bars, which are described in Section 3.2, are not included in the model because these bars are only located at the edges of the specimen and have negligible structural effect. Actual material properties are assigned to the layers to be consistent with the test specimen.

(a)

Layer Definition Data

Layer Name	Distance	Thickness	Type	Num Int. Points	Material	Material Angle	Material Component Behavior S11	S22	S12
ConcM	0.	3.552	Membrane	3	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
ConcM	0.	3.552	Membrane	3	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
Bar1M	0.651	0.002045	Membrane	1	Reinforcement	0.	Nonlinear	Inactive	Nonlinear
Bar2M	0.651	0.002045	Membrane	1	Reinforcement	90.	Nonlinear	Inactive	Nonlinear
ConcP	0.	2.503	Plate	5	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
Bar1P	0.651	0.002045	Plate	1	Reinforcement	0.	Nonlinear	Inactive	Nonlinear
Bar2P	0.651	0.002045	Plate	1	Reinforcement	90.	Nonlinear	Inactive	Nonlinear
DeckM	-1.7905	0.029	Membrane	3	Steel Deck	0.	Nonlinear	Nonlinear	Nonlinear
DeckP	-1.7905	0.029	Plate	5	Steel Deck	0.	Nonlinear	Nonlinear	Nonlinear

(b)

Layer Definition Data

Layer Name	Distance	Thickness	Type	Num Int. Points	Material	Material Angle	Material Component Behavior S11	S22	S12
ConcM	0.	1.973	Membrane	5	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
ConcM	0.	1.973	Membrane	5	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
Bar1M	-0.1385	0.002045	Membrane	1	Reinforcement	0.	Nonlinear	Inactive	Nonlinear
Bar2M	-0.1385	0.002045	Membrane	1	Reinforcement	90.	Nonlinear	Inactive	Nonlinear
ConcP	0.	1.3904	Plate	10	5700Psi	0.	Nonlinear	Nonlinear	Nonlinear
Bar1P	-0.1385	0.002045	Plate	1	Reinforcement	0.	Nonlinear	Inactive	Nonlinear
Bar2P	-0.1385	0.002045	Plate	1	Reinforcement	90.	Nonlinear	Inactive	Nonlinear

Figure 3.8: Shell section layer definition: (a) strong strip; (b) weak strip

First, as can be seen in Figure 3.8, the concrete membrane and plate thicknesses of both weak and strong strips are adjusted according to the recommendations given in Section 2.3. Second, steel layers, which are named as DeckM and DeckP, are added to the strong strip shell section layer definition to include corrugated decking in the modeling. Third, all material component behaviors are marked as Nonlinear, and the number of integration points is increased for deck and concrete slab layers for improved accuracy. Finally, reinforcing bar layers, which are named as Bar1M, Bar2M, Bar1P, and Bar2P, are added to both the weak and strong strip shell section layer definition forms to model welded wire reinforcement. Because SAP2000 does not have a specific tool to model welded wire reinforcement, two perpendicular reinforcing bar layers are defined to simulate this component.

The test specimen included shear studs with ½-inch diameter and 3.5-inch length, which are embedded in the concrete slab with a clear cover of 1 inch. Shear stud spacing along the girders and the ring beam are given in Section 3.2. By following the calculation steps in Section 2.4.1, and plugging in the parameters given in Section 3.2 and Section 3.3 into the equation proposed by Ollgaard et al. (1971) (Eq. (2.7)), the force-slip relationship of the test specimen shear studs is obtained.

The material properties of all steel and concrete building components, which are used for the simplified SAP2000 model, are specified in Table 3.2. Instead of using the pre-specified values, measured material properties are used when available.

Table 3.2: Material properties of the test specimen components

Building component	Yield strength, Fy (ksi)	Tensile strength, Fu (ksi)	Cube strength of concrete after 28 days (psi)
Concrete Slab	-	-	5700
Ring Beam	55	71.5	-
W12×14	56.1	71.286	-
W6×9	54.062	70.379	-
Double Angle (Connection)	58.015	78.175	-
Shear Tab Plate (Ring Beam)	43.192	54.824	-
Shear Tab Plate (Perimeter Beam)	51.706	83.4	-
2VLI11 (Steel Corrugated Deck)	47.064	57.435	-
Welded Wire Reinforcement	36	42	-

For the loading, a displacement-controlled load is applied to the support of the interior column in the vertical direction to simulate a static column loss scenario. Only nonlinear static analysis is conducted using "P-Delta plus Large Displacement" because results given in Chapter 2 demonstrated that this analysis type provides accurate results for both frame and shell elements. Figures of the simplified bare steel frame, ribbed slab, and fully composite steel-reinforced concrete floor system models are provided in Figure 3.9.

(a)

(b)

(c)

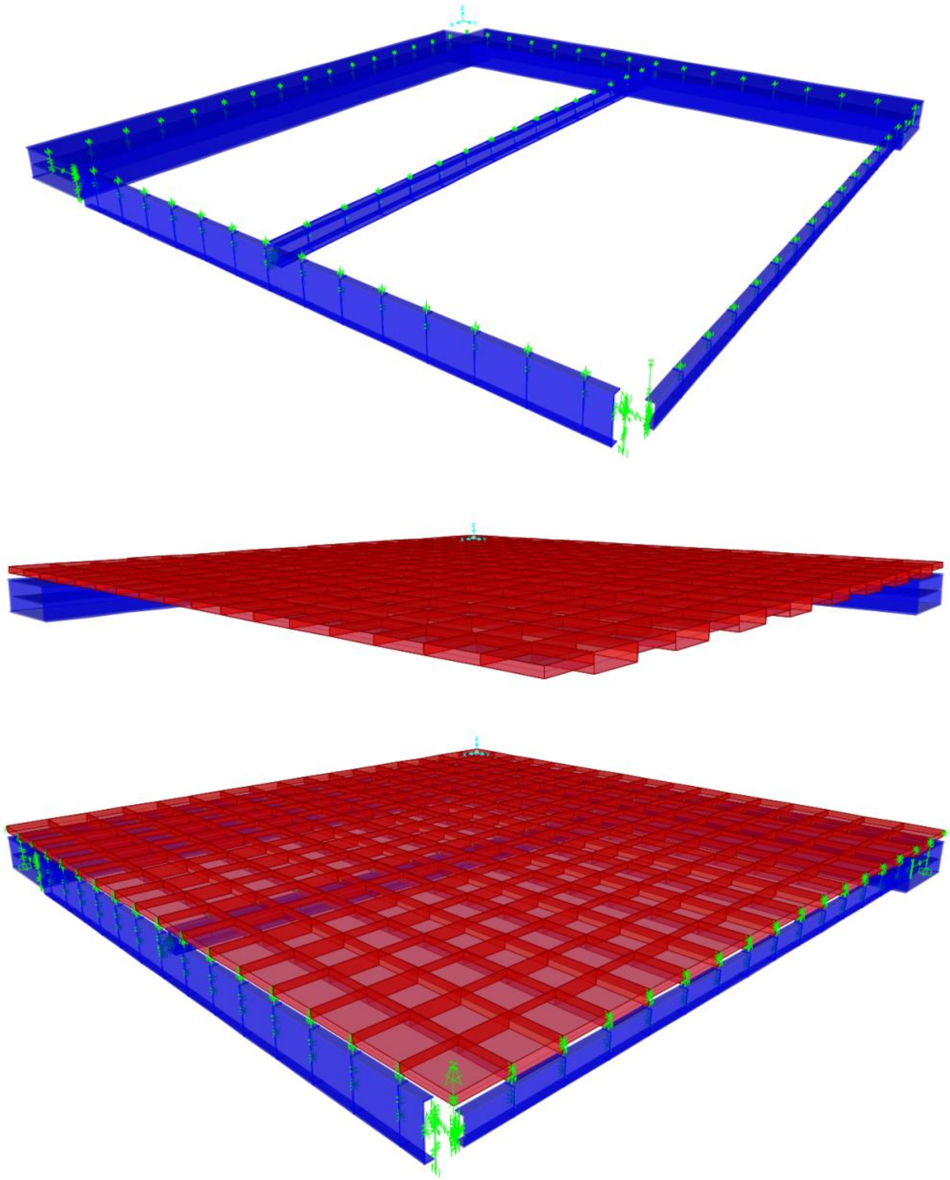

Figure 3.9: Simplified SAP2000 models of different floor systems: (a) bare steel frame model; (b) ribbed slab model with surrounding edge beam; (c) fully composite steel-reinforced concrete floor system model

The computational (SAP2000 model) load-deformation curves of the bare steel frame, ribbed slab, and fully composite steel-reinforced concrete floor system models are given in Figure 3.10. As mentioned previously, the detailed models of the bare steel frame, ribbed slab, and fully composite steel-reinforced concrete floor system were created by researchers at Imperial College of London. For comparison purposes, results from the detailed ADAPTIC models are also provided. Finally, experimental data for the full test specimen are plotted in Figure 3.10 to evaluate the performance of the simplified SAP2000 model.

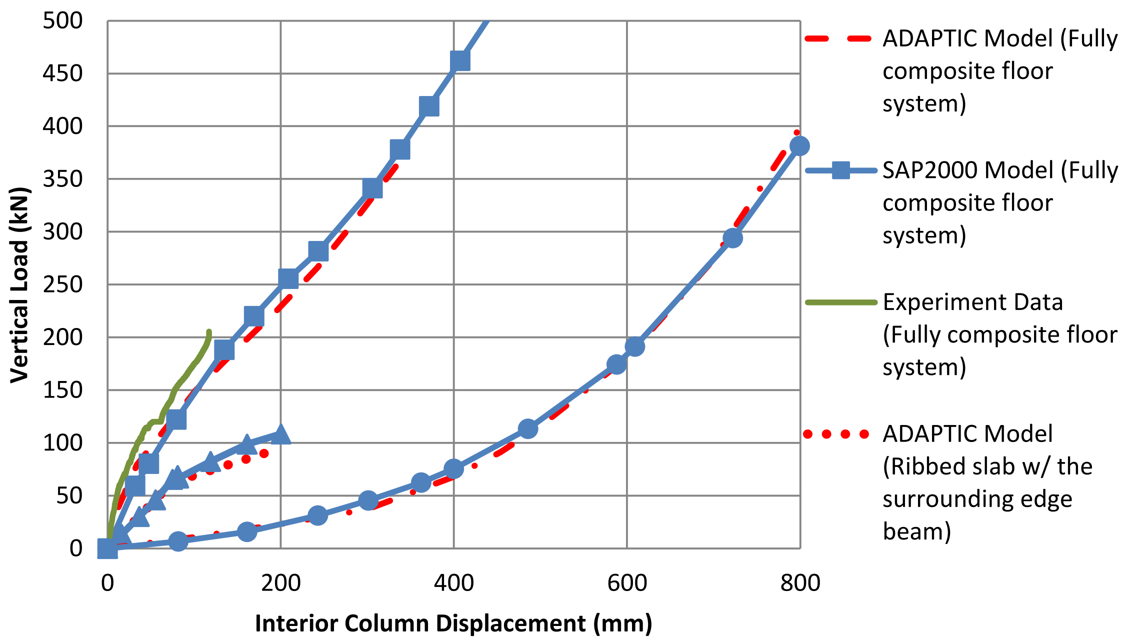

Figure 3.10: Load-displacement curves of the bare steel frame, ribbed slab, and fully composite steel-reinforced concrete floor system

As shown in Figure 3.10, the load-deformation curves of the detailed ADAPTIC models and simplified SAP2000 models show satisfactory matches for all three cases. This agreement between simplified and detailed modeling approaches verifies the simplified modeling approach used to model the response of steel-framed structures subjected to an interior column loss scenario. It should be noted that the stiffness and peak load of the fully composite steel-reinforced concrete floor system is higher than the bare steel frame model and the ribbed slab model, which is expected. Because SAP2000 cannot directly simulate component failure, the load-deformation curves continue to increase for loads and displacements well beyond those tested. An easy way to identify and remove the failed members is to check the amount of elongation at each member after an analysis and remove the failed members by using the Staged Construction feature within SAP2000, which is described in Section 2.5.2.

The experimental (Test Specimen) and computational (SAP2000 model) load-deformation curves for the fully composite steel-reinforced concrete floor system show good agreement. It should be noted that the test specimen did not fail under the ultimate load and deformation values given in Figure 3.10. The experimental data plotted in Figure 3.10 shows the response of the specimen up to the point of maximum load that the loading system could superimpose on top of the floor slab. Because the experiment was divided into four sub-experiments and the test specimen failed after some modification to the connections, the failure and the post-ultimate response of the specimen is not included in Figure 3.10. As a result, no element was removed from the model by using the Staged Construction tool to simulate failure. For the provided portion of the experimental results, the initial stiffness of the test specimen is predicted with good accuracy by the SAP2000 model. Because there are no major differences between the experimental and computed load-deformation curves, this modeling approach is considered acceptable to use for the collapse resistance assessment of steel gravity frame structures with composite floor systems.

The final deformed shape of the bare steel frame and the fully composite steel-reinforced concrete floor system models are shown in Figure 3.11. As mentioned in Section 2.5.2, the orientation of the beams and columns after deformation shows that the overall response of the structure under a column removal scenario is primarily controlled by the axial and bending deformations of the connections due to the development of catenary action.

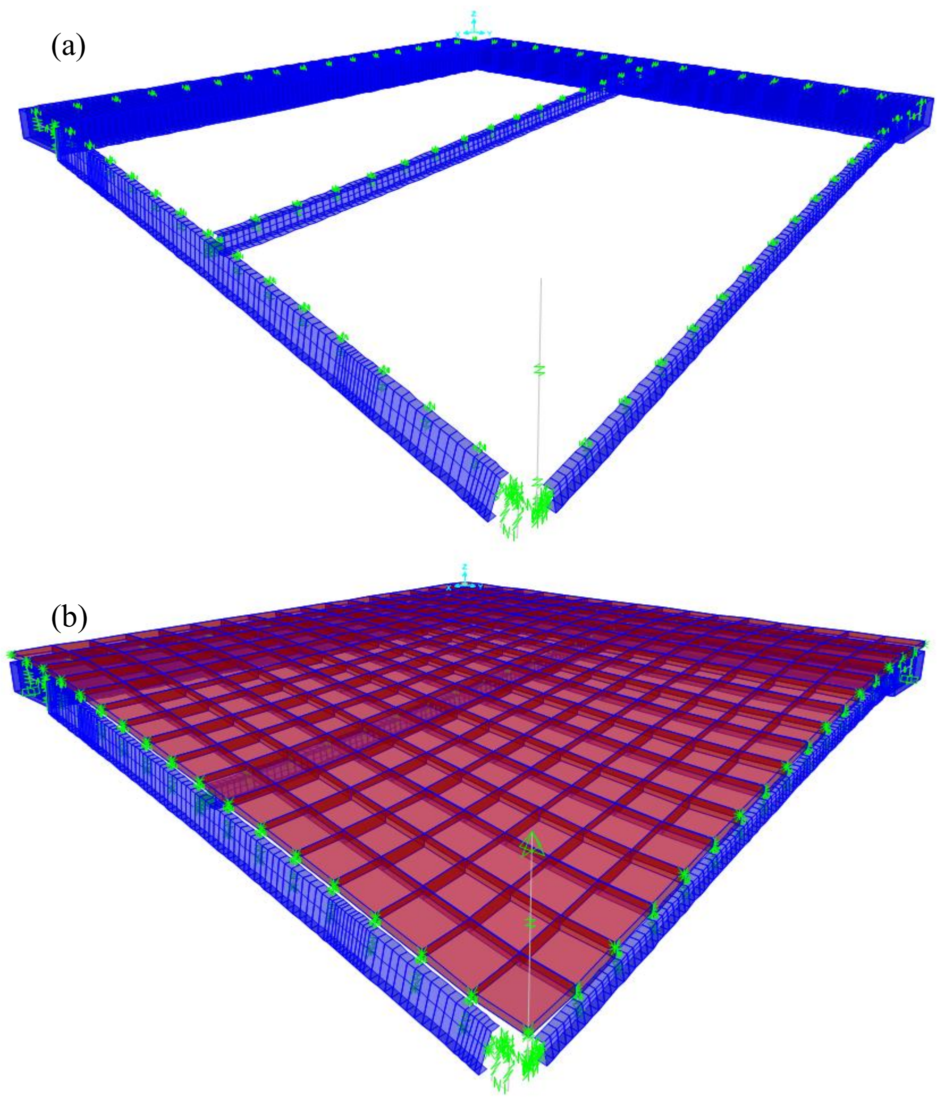

Figure 3.11: Deflected shapes of simplified SAP2000 models: (a) bare steel frame model; (b) fully composite steel-reinforced concrete floor system model

In this chapter, a brief description of the large-scale testing program conducted at the University of Texas at Austin was given. Moreover, step-by-step guidance was provided to show how the test specimen was computationally analyzed by using the methods that were described previously in Chapter 2. All computational analysis results and experimental data related to the test specimen were presented to show the accuracy of the proposed method. The next chapter will conclude this thesis by providing the summary, conclusions, and recommendations of the research.

CHAPTER 4

Summary, Conclusions, and Recommendations

4.1 INTRODUCTION

This chapter summarizes the most significant findings from the computational assessment of a 2-bay × 2-bay steel gravity frame structure with a composite floor system under an interior column removal scenario using the SAP2000 software. Also, recommendations for future research are provided.

To provide guidance to practicing civil/structural engineers, a simplified modeling approach that is compatible with SAP2000 was developed, and the performance of the modeling approach was evaluated under various cases. For this purpose, four simple models were created and analyzed in Chapter 2. Moreover, the current simplified/reduced modeling approach, which was developed by previous researchers, was improved by proposing alternatives for improving accuracy and increasing computational efficiency. These methods included reducing the 2-bay × 2-bay composite floor system to be represented by a single bay, not modeling structural components with negligible structural effect, and modeling stiff columns as supports. All methods and assumptions were verified multiple times by using available test data and results from detailed finite element models. No major differences were observed between the load-deformation curves of the simplified models, detailed models, and test specimens. By using the methods provided in this thesis, engineers can save time and effort in assessing the collapse resistance of steel-framed structures.

After establishing confidence in the proposed simplified modeling approach, a SAP2000 model was created to represent the specimen tested during the current research program. By using the methods and approaches provided in this thesis, the simplified SAP2000 model predicted the load-deformation behavior of the test specimen within 10-15% of the test data.

This thesis has provided a useful guide for practicing civil/structural engineers to investigate their structure's progressive collapse resistance by using simple structural analysis software. This chapter further summarizes and explains the results of the research performed.

4.2 RESEARCH LIMITATIONS

It should be noted that the modeling approaches for progressive collapse analysis proposed in this thesis were developed based on the recommendations of previous researchers. Nonetheless, because available test data are limited and because composite floor systems in steel-framed structures include a wide range of parameters, caution must be exercised when applying the recommended modeling approach to general cases. Additional test data are needed to validate the proposed modeling methodology.

During the modeling and analysis process, many assumptions were made due to limitations of the SAP2000 software. First, as it can be seen from Section 2.2.2 and Section 2.3.2, frame member and shell element models give better results with the P-Delta plus Large Displacement analysis compared to the P-Delta analysis. As such, only P-Delta plus Large Displacement was analyses were used for computing the response of the test specimen. Second, as mentioned previously, SAP2000 cannot simulate component failure. Although, the Staged Construction tool of SAP2000 partially solves this problem for link and frame elements, the results are not very realistic for the post-failure region, which can be seen from Figure 2.26. Third, convergence problems are one of the major limitations of using the SAP2000 software. During the modeling process, users should always keep in mind that models with large numbers of shell, link, and frame elements need more time and calculation steps for analysis, which can cause convergence problems and slow down/stop the progress of analysis. Moreover, sudden decreases (failure) and increases (hardening) in the stiffness of complex structures can also cause convergence problems. For this reason, users should try to keep their analysis models as simple as possible.

Attempts were made to validate all methods and approaches proposed in this thesis. At the end of this process, it was concluded that all models developed with the proposed methods and approaches had either a very close or acceptable and conservative load-displacement response compared to the experimental data and results from detailed finite element models. Moreover, experimental data from the test specimen showed that the load-deformation response of the simplified model was reasonably accurate. However, it should be noted that the simplified modeling approach can provide less accurate results for structures with different properties. Additional structures should be modeled and analyzed using SAP2000 (or similar software) to decrease uncertainty concerning the computational assessment of the collapse behavior of steel-framed structures under column removal scenarios. Future researchers and civil/structural engineers should be aware of the reasons for the differences between the experimental and computational responses before using any portion of this research.

4.3 SUMMARY AND CONCLUSIONS

This thesis attempted to provide a new method for practicing civil/structural engineers to investigate the progressive collapse resistance of steel-framed structures by using simple structural analysis software. For this reason, past research, modeling approaches, and experimental data were studied in great depth to find a computational modeling method that can be used for a wide variety of steel-framed structures. All formulations, modeling approaches, simplifications, assumptions, and limitations were described in detail. Many useful results and observations were made during the research. This section briefly summarizes the computational analysis and modeling work that were carried out during the research, and conclusions from the study are provided at the end of this section.

In Section 2.2, a simplified shear tab connection was investigated under pure shear and pure tension loads. The load-deformation curves showed that the simplified connection models can accurately represent the actual tensile and shear behavior of single plate shear connections under nonlinear static analysis using the "P-Delta plus Large Displacement" option. Conversely, the shear response of the connection model under nonlinear static analysis with P-Delta effect was overly conservative compared to the test data, which showed that this analysis type is not appropriate for connection models. In addition, the research on simplified connection models showed that these models lose accuracy as the complexity of the failure mode increases.

A floor slab model, which was composed of nonlinear layered shell elements, was investigated in Section 2.3. The computational and experimental load-deformation curves showed a satisfactory match, and the initial stiffness and peak load values of the concrete slabs were predicted satisfactorily by the simplified SAP2000 model. It was concluded in this section that models with only shell elements give better results with the P-Delta plus Large Displacement analysis compared to P-Delta analysis. As a result, P-Delta plus Large Displacement analyses were used for remainder of the research.

In Section 2.4, composite, non-composite, and partially composite beams were investigated. The load-deformation curves showed that the simplified SAP2000 models can predict the behavior of such beams accurately up to the point of local buckling initiation and shear stud failure. After that point, the load-deformation curves from the computational models overestimate the strength of the structure because SAP2000 cannot readily capture local buckling and post-failure response.

A full steel gravity frame model with composite floor system was investigated in Section 2.5 to establish confidence in the simplified models before modeling the actual test specimen. The load-deformation curve of the simplified SAP2000 model showed good general agreement with the detailed LS-DYNA model up to the point of first connection bolt failure. After the first failure, results from the SAP2000 and LS-DYNA models started to deviate, but the failure displacement of the structure was predicted accurately. Because there was no major difference between the detailed and simplified model load-deformation curves, and the simplified model gave a conservative response, the simplified modeling approach was considered acceptable and used for the remainder of the research.

Finally, the actual test specimen, which was a steel gravity frame structure with composite floor system, was modeled and analyzed in Chapter 3. The load-deformation behavior of the SAP2000 model showed good agreement with the detailed ADAPTIC model and the experiment results. This agreement between the simplified SAP2000 model, detailed ADAPTIC model, and experimental data verifies the accuracy and capability of the simplified modeling approach. As stated above, because SAP2000 cannot simulate component failure and no element removal load case was added to the analysis, the failure point of the structure was not determined precisely. Nevertheless, it was concluded that the simplified modeling approach is acceptable because no major differences between the experimental and computed load-deformation curves were observed, and the simplified model gave a slightly conservative response.

The results of this research showed that steel gravity frame structures with composite floor systems can be accurately simulated using simple structural engineering programs up to the point of first component failure. Moreover, it was shown that the response of the simplified models are conservative compared to the true behavior of the structure, which is acceptable in terms of structural design. It is clear that practicing civil/structural engineers can save significant time and expense by checking the progressive collapse resistance of their preliminary and final designs in a quick and accurate way by following the simple steps provided in this thesis.

The ultimate goal of this thesis was to provide step-by-step guidance on modeling and analyzing full-size structures to help practicing civil/structural engineers mitigate progressive collapse. The information provided in this research, which includes methods of creating and simplifying computer models for analysis, showed that the first critical steps to this goal have been achieved.

4.4 FUTURE RESEARCH AND RECOMMENDATION

Future research and development on simplified modeling approaches are encouraged. Further experimental research on simple connections, floor systems, and steel gravity frame structures with composite floor systems should be carried out and compared with the proposed methods to enhance confidence in the simplified modeling approach and to improve the assumptions and simplifications. Moreover, new experiments should include test specimens with different cross-sectional properties, material properties, and loading conditions to understand the effects of these variables on the overall structural behavior. The new experimental data will help to confirm the accuracy of the methods, models, assumptions, and simplifications proposed in this thesis and to detect the sources of error or problems in the development of the models.

As mentioned previously, one of the major limitations of the research was the lack of erosion and element failure options in the SAP2000 software. It is recommended that consideration be given to adding these features to the software, which can greatly increase the effectiveness of the methods and approaches recommended in this thesis.

Results provided in this research demonstrate that the simplified modeling approach is promising. Conducting additional research on this subject will enable engineers to mitigate progressive collapse in cost-effective ways.

References

ACI Committee 318. 2011. Building Code Requirements for Structural Concrete (ACI 318-11) American Concrete Institute, Famington Hills, MI.

AISC. (2010). AISC, Steel Construction Manual, American Institute of Steel Construction, Chicago, IL.

Alashker, Y., El-Tawil, S. and Sadek, F. (2010). "Progressive Collapse Resistance of Steel-Concrete Composite Floors." Journal of Structural Engineering, 136(10), 1187-1196.

Alashker, Y., Li, H. and El-Tawil, S. (2011). "Approximations in progressive collapse modeling." Journal of Structural Engineering, 137(9), 914-924.

ANSI. (1972). ANSI A58.1, Minimum Design Loads for Buildings and Other Structures, American National Standards Institute, New York, NY.

ASCE. (2010). ASCE 7-10, Minimum Design Loads for Buildings and Other Structures, American Society of Civil Engineers, Reston, VA.

Breen, J.E., ed. (Sept. 1976). "Progressive collapse of building structures (summary report of a workshop held at the University of Texas at Austin, Oct. 1975)." U.S.

Federal Emergency Management Agency (FEMA). (2000). "State of the art report on connection performance." FEMA 355D, SAC Joint Venture and FEMA, Washington, D.C.

Guravich, S. 2002. Standard beam connections in combined shear and tension. Ph.D. thesis, Department of Civil Engineering, University of New Brunswick, Fredericton, N.B.

Hull, L. (2013). Experimental Testing of a Steel Gravity Frame with a Composite Floor under Interior Column Loss. M.S. Thesis, Department of Civil, Architectural and Environmental Engineering, University of Texas at Austin.

Izzuddin, BA. Nonlinear dynamic analysis of framed structures. Ph.D. thesis. Department of Civil Engineering, Imperial College, University of London; 1991.

Izzuddin, B.A., Vlassis, A.G., Elghazouli, A.Y., and Nethercot, D.A. (2008). "Progressive collapse of multi-storey buildings due to sudden column loss – Part I: Simplified assessment framework." Engineering Structures, 30, 1308-1318.

Kwon, G. (2008). "Strengthening existing steel bridge girders by the use of post-installed shear connectors." Ph.D. dissertation, Dept. of Civil, Architectural and Environmental Engineering, Univ. of Texas, Austin, TX.

LS-DYNA (2014). Livermore Software Technology Corporation, Livermore, CA. LS-DYNA. Retrieved from http://www.lstc.com/lsdyna.htm

Main, J.A., Bao, Y., Sadek, F., Lew, H.S. (2010). "Experimental and computational assessment of robustness of steel and reinforced concrete framed buildings." Applications of Statistics and Probability in Civil Engineering: Proceedings of the 11th International Conference, Held in Zürich, Switzerland, August 1-4, 2011, Taylor & Francis Group, London, 2184-2192.

Main, J.A.; Sadek, F. (2012). Robustness of steel gravity frame systems with single-plate shear connectors, National Institute of Standards and Technology (NIST) Technical Note 1749. Gaithersburg, MD.

Marjanishvili, S., Agnew, E. 2006. Comparison of Various Procedures for Progressive Collapse Analysis. Journal of Performance of Constructed Facilities 20, (11): 365-374.

Ollgaard, J.G., Slutter, R.G., and Fisher, J.W. (1971). "Shear strength of stud connectors in lightweight and normal weight concrete." Engineering Journal, AISC, 8(2), 55-64.

Park, R., 'Ultimate strength of rectangular concrete slabs under short-term uniform loading with edges restrained against lateral movement', Proceedings Instn. Civ. Engrs, Vol.28, June 1964, pp. 125 – 150.

Rex, C.O., and Easterling, S.W. (2003). "Behavior and modeling of a bolt bearing on a single plate." Journal of Structural Engineering, 129(6), 792-800.

Sadek, F., El-Tawil, S., and Lew, H.S. (2008). "Robustness of composite floor systems with shear connections: modeling, simulation, and evaluation." Journal of Structural Engineering, 134(11), 1717-1725.

Sadek, F., Main, J.A., Lew, H.S., Robert, S.D., Chiarito, V.P., El-Tawil, S. (2010). "An experimental and computational study of steel moment connections under a column removal scenario." NIST Technical Note 1669, National Institute of Standards and Technology, Gaithersburg, MD.

SAP2000. Ultimate Version 15.1.0. Computers & Structures, Berkeley, CA.

Thompson, S.L. (2009) "Axial, shear and moment interaction of single plate 'shear tab' connections." Master's Thesis, Milwaukee School of Engineering, Milwaukee, WI.

Vlassis AG, Izzuddin BA, Elghazouli AY, Nethercot DA. Progressive collapse of multi-storey buildings due to sudden column loss — Part II: Application. 2007. Companion paper

Weigand, J.M., Meissner, J.E., Francisco, T., Berman, J.W., Fahnestock, L.A., and Liu, J. (2012). "Overview of AISC/NSF structural integrity research and preliminary results." Proceedings, 2012 Structures Congress, American Society of Civil Engineers, Reston, VA.

Vita

Umit Can Oksuz was born on July 5th, 1990, in Ankara, Turkey. He earned two Bachelor's degrees in Civil Engineering from the State University of New York at Buffalo, United States of America and Istanbul Technical University, Republic of Turkey in 2012. He graduated as valedictorian of his program. In the fall of 2012, he entered the MS. Structural Engineering program in Department of Civil, Architectural and Environmental Engineering at the University of Texas at Austin. He obtained his Master of Science in Engineering in May of 2014.

Permanent email: umitcanoksuz@gmail.com

This thesis was typed by the author.