University of Cincinnati

Date: 4/29/2011

I, Daniel J. Ruffley, hereby submit this original work as part of the requirements for the degree of Master of Science in Civil Engineering.

It is entitled:
A Finite Element Approach for Modeling Bolted Top-and-Seat Angle Components and Moment Connections

Student’s name: Daniel J. Ruffley

This work and its defense approved by:

Committee chair: Gian Rassati, PhD
Committee member: Thomas Baseheart, PhD
Committee member: James Swanson, PhD
A Finite Element Approach for Modeling Bolted Top-and-Seat Angle Components and Moment Connections

A thesis submitted to the
Graduate School
of the University of Cincinnati
in partial fulfillment of the
requirements for the degree of

Master of Science in Civil Engineering

in the School of Advanced Structures
of the College of Engineering and Applied Science
at the University of Cincinnati

by

Daniel J. Ruffley
Bachelor of Science in Civil Engineering
from the University of Cincinnati, College of Engineering

Committee Chair:
Gian Andrea Rassati, Ph.D.

April 2011
Abstract

A verified procedure for modeling bolted top and seat angle components and connections for potential use in seismic moment frames using finite element analysis software is presented here. The vast usage of top and seat angle connections in moment frames would fall into the range of partially restrained connections. This type of connection is not currently certified to be used by engineers for moment resistance in any major building specification jurisdiction. The main reason for this is the complex analysis required of a structure to take advantage of such a connection effectively, to say nothing of the complex analysis also necessary on the connection level. However, these connections have been demonstrated to provide economic savings should the engineer invest into these analyses. This work is tailored to aid engineers in analysis on the connection level. The modeling procedure was used to recreate two full-scale top and seat angle connection tests as well as multiple component tests. Load – displacement values from FEA model analyses were compared to experimental data and verified. Mechanistic observation also indicated agreement in the modes of failure of the FEA models with the experimental tests. Given the lack of variety in the failure modes of top and seat angle connections and their components, some of the component tests recreated were t-stub component tests to illustrate the accuracy, precision, and confidence of the modeling procedure. The following failure methods were able to be predicted: tension bolt failure, shear bolt failure, and block shear failure. A beam sensitivity analysis was also performed early in the study that accurately demonstrates the ability to predict the formation of plastic hinge phenomena. All models were created using hexahedral solid elements utilizing reduced integration with either eight or twenty nodes. This modeling procedure so verified could be used as a tool by engineers in preliminary analysis and design stages to alleviate the daunting task of building such a model and analysis from scratch.
Acknowledgements

I would like to heartily thank my advisor and committee chair, Dr. Gian Andrea Rassati, for his wisdom, knowledge, patience, and friendship throughout this research. This task would have been impossible without his guidance and support. I would like to thank Dr. James A. Swanson whose input and suggestions were indispensable. I would also like to thank Dr. T. Michael Baseheart for sitting on my thesis committee and being an excellent instructor during my undergrad years.

My weekly audience that embodied the connections research group, Mindy Barber, Elie Hantouche, Wei Zhang, Jared Schippers, Mac Schroeder, and Craig Schrader, was greatly appreciated for their feedback and suggestions. I would not have felt as confident with my research had I not had the opportunity to work with them on a weekly basis.

This research would not have been possible without the experimental data contributions from Dr. Roberto Leon, Dr. James A. Swanson, and Amy Moore.

I would like to personally thank my peers, Ian McElhone and Mindy Barber, for their friendship and camaraderie throughout our graduate tenure. The time spent in graduate school would have been orders of magnitude more difficult without sharing the experience with them.

Finally, I would like to thank my parents, Sue and Tim Ruffley, for their support, motivation, and love in succeeding in this challenging experience.
# Table of Contents

Chapter 1 - Introduction 1
   1.1 Objective 3
   1.2 Organization of Work 3

Chapter 2 – Project Background 4
   2.1 SAC Research Project 4
      2.1.1 SAC Subtask 7.03 4
      2.1.2 Full-Scale Top-and-Seat Angle Cyclic Tests 5
      2.1.3 Component Cyclic Tests 7
   2.2 Recommendations for Strength Determination 8
   2.3 Analytical Modeling 8
   2.4 Finite Element Modeling 9
      2.4.1 Direct Shear 9
      2.4.2 Prediction Models 10
      2.4.3 Finite Element Analysis of SAC Subtask 7.03 10
   2.5 Energy Dissipation 11

Chapter 3 – Methods 12
   3.1 ABAQUS Model Organization 12
      3.1.1 Partitioning 14
   3.2 Elements and Implementation 15
      3.2.1 Reduced Integration 16
      3.2.2 Solid Element Types 16
      3.2.3 Element Sensitivity Analysis 18
      3.2.4 Ramifications of Sensitivity Analysis 20
   3.3 Bolt Modeling 21
   3.4 Constraints, Restraints, and Loading 24
   3.5 Contact Surfaces 25
      3.5.1 Tangential Behavior 26
      3.5.2 Slip Coefficient Recommendations 28
   3.6 Bolt Pretensioning 29
   3.7 Material Properties 31
      3.7.1 Engineering and True Stress and Strain 31
5.2.1 Analysis Events 90
5.2.2 Analysis Results 93

5.3 FS-02 Model Setup 95
   5.3.1 Elements and Meshing 97
   5.3.2 Loading of FS-02 Model 98

5.4 FS-02 Analysis 99
   5.4.1 Analysis Events 100
   5.4.2 Analysis Results 105

Chapter 6 – Conclusions 108
   6.1 Results of Component Models 108
   6.2 Results of Full-Scale Models 109
   6.3 General Recommendations 110
   6.4 Future Research 111

References 113
**List of Tables**

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3-1</td>
<td>Shank void depth calculations</td>
<td>23</td>
</tr>
<tr>
<td>3-2</td>
<td>Effects of temperature gradient pretension method</td>
<td>30</td>
</tr>
<tr>
<td>3-3</td>
<td>A572-50 material property breakdown for the W18x40 section</td>
<td>33</td>
</tr>
<tr>
<td>4-1</td>
<td>Bolt shank void calculations for TA-25 and TA-26 models</td>
<td>51</td>
</tr>
<tr>
<td>4-2</td>
<td>A572-50 material breakdown for the TA-25 t-stub</td>
<td>53</td>
</tr>
<tr>
<td>5-1</td>
<td>A572-50 material breakdown for the L6x8x1 angle</td>
<td>79</td>
</tr>
<tr>
<td>5-2</td>
<td>A572-50 material breakdown for the W14x145 column</td>
<td>80</td>
</tr>
<tr>
<td>5-3</td>
<td>SAC protocol load history applied to the FS-01 experiment (Schrauben, 1999)</td>
<td>88</td>
</tr>
</tbody>
</table>
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-1</td>
<td>WUF-B, Welded unreinforced flange-bolted web connection (Adan and Hamburger 2010)</td>
<td>1</td>
</tr>
<tr>
<td>1-2</td>
<td>Bolted T-stub connection (left) and Top-and-Seat angle connection (right) (Schrauben 1999)</td>
<td>2</td>
</tr>
<tr>
<td>2-1</td>
<td>Load-displacement plot of experimental FS-01 test (Schrauben 1999)</td>
<td>6</td>
</tr>
<tr>
<td>2-2</td>
<td>Load-displacement plot of experimental FS-02 test (Schrauben 1999)</td>
<td>6</td>
</tr>
<tr>
<td>2-3</td>
<td>Typical component test setup (Swanson 1999)</td>
<td>7</td>
</tr>
<tr>
<td>3-1</td>
<td>Partitioned angle</td>
<td>12</td>
</tr>
<tr>
<td>3-2</td>
<td>Partitioned bolt and partitioning planes</td>
<td>13</td>
</tr>
<tr>
<td>3-3</td>
<td>Beam mesh transition using partitions</td>
<td>15</td>
</tr>
<tr>
<td>3-4</td>
<td>Partitioning scheme of model TA-26 (different parts colored)</td>
<td>15</td>
</tr>
<tr>
<td>3-5</td>
<td>Q4 (top) and Q8 (bottom) elements</td>
<td>16</td>
</tr>
<tr>
<td>3-6</td>
<td>Bilinear (top) and quadratic (bottom) rectangular solid (hexahedral) elements</td>
<td>17</td>
</tr>
<tr>
<td>3-7</td>
<td>Load-displacement response from 10 kip load applied to cantilever beam</td>
<td>18</td>
</tr>
<tr>
<td>3-8</td>
<td>Load-displacement response from 25 kip load applied to cantilever beam</td>
<td>19</td>
</tr>
<tr>
<td>3-9</td>
<td>Model f20p deflected and illustrating plastic hinge</td>
<td>20</td>
</tr>
<tr>
<td>3-10</td>
<td>Tension bolt from FS-02 model</td>
<td>22</td>
</tr>
<tr>
<td>3-11</td>
<td>Frame view of a tension bolt from model FS-02</td>
<td>24</td>
</tr>
<tr>
<td>3-12</td>
<td>Monotonic responses of different slip coefficients compared to experimental data</td>
<td>27</td>
</tr>
<tr>
<td>3-13</td>
<td>Normal distribution of slip coefficients constructed from research (Kulak et. al 2001)</td>
<td>28</td>
</tr>
<tr>
<td>3-14</td>
<td>Plot of Engineering vs. True Stress/Strain</td>
<td>32</td>
</tr>
<tr>
<td>3-15</td>
<td>Ultimate bolt strength compared for FS-01 model</td>
<td>34</td>
</tr>
<tr>
<td>4-1</td>
<td>Shear fixture for shear bolt test (Moore 2007)</td>
<td>40</td>
</tr>
<tr>
<td>4-2</td>
<td>Bolt shear model in ABAQUS</td>
<td>40</td>
</tr>
<tr>
<td>4-3</td>
<td>7/8” bolt meshed for basic shear model</td>
<td>42</td>
</tr>
<tr>
<td>4-4</td>
<td>6” x 6” x 1” plate meshed for basic shear model</td>
<td>42</td>
</tr>
<tr>
<td>4-5</td>
<td>Reference points on basic shear model assembly</td>
<td>43</td>
</tr>
<tr>
<td>4-6</td>
<td>Final deflected shape of shear bolt model in ABAQUS</td>
<td>44</td>
</tr>
<tr>
<td>4-7</td>
<td>Shear fixture with modified pins (Moore 2007)</td>
<td>45</td>
</tr>
</tbody>
</table>
4-8  Bolt shear model plate (contours showing PEEQ)  
4-9  Load-displacement plot for the bolt shear model comparing experimental data to FE analysis results  
4-10 TA-25 model assembly in ABAQUS (colors display different parts)  
4-11 Drawing of TA-25 model beam element (modified from Swanson 1999)  
4-12 Drawing of TA-25 model T-stub (Swanson 1999)  
4-13 Bolt placement within holes for TA-25 model and explanation  
4-14 TA-25 model t-stub partitioning scheme  
4-15 TA-25 model t-stub mesh scheme  
4-16 TA-25 model beam element partitioning scheme  
4-17 TA-25 model beam element mesh scheme  
4-18 TA-25 model partitioning scheme for the column flange element  
4-19 TA-25 model mesh scheme for the column flange element  
4-20 TA-25 model partitioning scheme (left) and mesh scheme (right) for the shear bolt  
4-21 TA-25 model partitioning scheme (left) and mesh scheme (right) for the tension bolt  
4-22 View of support constraint for the TA-25 model  
4-23 View of load constraint for the TA-25 model  
4-24 TA-25 model at one inch deformation (contours show Von Mises stresses)  
4-25 Deformed beam element of TA-25 model (contours show PEEQ)  
4-26 Deformed column element from TA-25 model (contours show PEEQ)  
4-27 Deformed shear bolts from TA-25 model (contours show PEEQ)  
4-28 Deformed tension bolts from TA-25 model (contours show PEEQ)  
4-29 Deformed t-stub at three different step times for TA-25 model (contours show PEEQ)  
4-30 TA-25 load vs. displacement comparison  
4-31 TA-26 model assembly in ABAQUS (colors display different parts)  
4-32 Drawing of TA-26 model beam element (modified from Swanson 1999)  
4-33 Drawing of TA-26 model T-stub (Swanson 1999)  
4-34 TA-26 model beam element partitioning scheme  
4-35 TA-26 model beam element mesh scheme  
4-36 TA-26 model T-stub partitioning scheme  
4-37 TA-26 model T-stub mesh scheme  
4-38 Deformed beam element of TA-26 model (contours show PEEQ)  
4-39 Deformed shear bolts from TA-26 model (contours show PEEQ)  
4-40 Deformed t-stub at four different step times for the TA-26 model (contours show PEEQ)
4-41  TA-26 load vs. displacement comparison

5-1  FS-01 model assembly in ABAQUS (colors indicate the different parts)
5-2  Beam model length derivation
5-3  L6x8x1 angle for model FS-01 meshed
5-4  (Left) Partitioning of the cross-section of the FS-01 model beam
5-5  (Above) Partitioning of the FS-01 model beam along its length
5-6  Segments 1 & 2 mesh of the FS-01 model beam
5-7  Segments 3 & 4 mesh of the FS-01 model beam
5-8  Element mesh of the FS-01 column
5-9  Mesh scheme for FS-01 model shear tab
5-10 Mesh scheme for the FS-01 model tension bolts
5-11 Mesh scheme for the FS-01 model shear bolts
5-12 Mesh scheme for the FS-01 model shear tab bolts
5-13 FS-01 model colored by part and illustrating the three reference point constraints
5-14 FS-01 model deflected with Von Mises stress contours
5-15 FS-01 top-and-seat angle prying (contours show PEEQ)
5-16 Clip angle “pinching” and k-zone crippling (Schrauben 1999)
5-17 FS-01 deflected beam at connection end (contours showing PEEQ)
5-18 FS-01 top seat angle tension bolts (contours showing PEEQ)
5-19 FS-01 Load vs. Beam tip displacement comparing experimental data to FE analysis results
5-20 FS-01 (left) and FS-02 (right) angles
5-21 FS-02 model assembly in ABAQUS (colors indicate different parts)
5-22 FS-02 angle meshed
5-23 FS-02 column meshed
5-24 FS-02 model deflected shape (contours displaying Von Mises stress)
5-25 FS-02 top seat angle prying (contours show PEEQ)
5-26 FS-02 angle at step time 1.0 (contours showing PEEQ)
5-27 FS-02 beam at approximately step time 0.37 (contours show PEEQ)
5-28 FS-02 beam at step time 1.0 (contours show PEEQ)
5-29 FS-02 tension bolts (contours show PEEQ)
5-30 FS-02 Load vs. Beam tip displacement comparing experimental data to FE analysis results
5-31 FS-02 Load vs. Beam tip displacement comparison (relevant experimental data portion)
Chapter 1 – Introduction

The Northridge (1994) and Kobe (1995) earthquakes were some of the strongest and most costly (both financially and in terms of loss of life) earthquakes that occurred during the twentieth century. Investigation of the damaged steel structures indicated a surprising number of steel moment connection failures. The majority of these connections were ones in which the beam flange was connected to the column using field-welded, complete penetration groove welds and the beam web was connected to the column using shop welded, field bolted shear plates as shown in Figure 1 (Adan and Hamburger 2010). These connections were both acceptable and typical at the time. The most surprising discovery of these failures was that they were not readily observable without surveying because the resulting inter-story drift was so slight (Shipp 1994).

Figure 1-1: WUF-B, Welded unreinforced flange – bolted web connection (Adan and Hamburger 2010)
These revelations in the wake of these disastrous earthquakes prompted research into repair, retrofit, and design of steel moment frame connections. As is typical when evaluating design, alternate designs to the WUF-B connection were considered. The most promising of these alternate designs took the form of fully bolted connections. The bolted T-stub connection was a favorite for mid to high rise structures, but for low to mid rise structures the bolted top and seat angle (TSA) connection was considered. A variety of research has been performed on TSA connections that has significantly increased the state of knowledge about these connections.

![Figure 1-2: Bolted T-stub connection (left) and Top-and-Seat angle connection (right) (Schrauben 1999)](image)

Even with all this research, there has yet to be a design approach developed for TSA type connections for use in moment frames. The TSA connections being considered are partially restrained and partial strength. Therefore, they cannot, as of yet, be prequalified as per Standard 358-05 of the American Institute of Steel Construction (AISC) (2005). Further research into the behavior of such connections is necessary before a prequalification procedure can be developed.
Such a qualification of these TSA connections would allow for design engineers to employ these connections in various applications.

1.1 Objective

The objective of this thesis and work contained herein is to demonstrate the accuracy to which finite element modeling and analysis using computer software can predict the behavior of TSA connections. Both full-scale connections and component models will be created and analyzed. The results of these analyses will then be compared to existing experimental data. Aims of this work are providing more in-depth understanding of TSA connections in the arena of computer modeling, and aiding to further confidence that these connections could be reliably used by practicing engineers in the future.

1.2 Organization of work

The current chapter serves as an introduction, description of objectives and aims, and an explanation of the organization of this thesis. Chapter 2 serves as the literature review for this research, delving into topics both immediately germane to the work undertaken and more removed. Chapter 3 elucidates the modeling procedure developed to create the models for the subsequent chapters. Chapter 4 details the model setup and analysis results of the component models; bolt shear, TA-25, and TA-26. Chapter 5 details the model setup and the analysis results of the full-scale models, FS-01 and FS-02. Chapter 6 discusses conclusions and recommendations based on the results from chapters 4 and 5.
Chapter 2 – Project Background

A major part of this research consists of the validation of the FE model output against experimental tests. This section will cover in detail the previous work performed by both Schrauben and Swanson in regards to their experimental testing. This chapter will also provide summaries of past and current work being performed and having relevance to this project.

2.1 SAC research project

A comprehensive steel connections project was carried out in a joint venture by the Structural Engineers Association of California (SEAOC), the Applied Technology Council (ATC), and the California Universities for Research in Earthquake Engineering (CUREe) following the aftermath of the Northridge and Kobe earthquakes. It was funded by the Federal Emergency Management Agency (FEMA) and fittingly dubbed the SAC (SEAOC ATC CUREe) research project. The goals of this project were experimental testing of the connection components followed by testing of the full scale connections themselves; data reduction and analysis of the test results; and finally to provide design recommendations based on the tests’ conclusions. Connection types under investigation were as follows: welded connections, end-plate connections, flange-plate connections, bolted T-stub connections, bolted TSA connections, and simple shear connections.

2.1.1 SAC subtask 7.03

Subtask 7.03 of the SAC research project was a portion of the endeavor focused on the testing of T-stub and TSA connection components and their full-scale correspondents. This
subtask was performed at the Georgia Institute of Technology which in turn produced a number of publications on the targeted topics. The goals of the subtask project included the following: to quantify prying and provide recommendations for the strength determination of these connections; and to characterize and develop models to predict connection stiffness and ductility. Key accomplishments of the subtask project included the development of an extensive literature database and the testing of forty-eight T-stub components, ten TSA components, and ten full-scale connections.

2.1.2 Full-scale top-and-seat angle cyclic tests

Schrauben’s thesis (1999) as part of Subtask 7.03 has become the starting point for the research undertaken herein. His research involved four of the ten total full scale tests performed by SAC Subtask 7.03: two T-stub connections and two TSA connections. The latter TSA tests being the primary focus here. His thesis concludes that the full scale TSA connections performed better in tests than their respective components, something that did not happen with any of the other full scale tests. He attributes this to the complexity of interaction among the connection’s elements and the unique hinging phenomena in the angles from interaction with the beam. The results of Schrauben’s TSA tests will be essential to achieving the goals of this research. Figures 2-1 and 2-2 show the load-displacement plots from test FS-01 and FS-02, respectively.
Figure 2-1: Load-displacement plot of experimental FS-01 test (Schrauben 1999)

Figure 2-2: Load-displacement plot of experimental FS-02 test (Schrauben 1999)
2.1.3 Component cyclic tests

Subtask 7.03 also performed a number of cyclic component tests whose results and conclusions appear in Swanson’s dissertation. (Swanson 1999) Swanson’s work included the testing of 48 t-stub components and 10 angle components under cyclic loading. The components were varied and tested to provide a wide range of possible connection configurations. Consequently the results were also used to provide spring parameters used in three finite element analyses of t-stubs to reproduce results and observe the mechanical behavior of these connections. The finite element analyses consisted of a three-dimensional analysis of the t-stub which proved to be too computationally intensive for the existing technology; and two two-dimensional analyses, one of a width of flange and the other of a stem of a t-stub. The data from the component tests was also compared to the full-scale tests performed by Smallidge. This work was able to fill a number of gaps in the vast research database complied by the subtask group, produce a number of firsts in behavioral modeling of t-stubs, and modify two existing t-stub strength models. This work will provide for the component models what Schrauben’s provided for the full-scale models.

Figure 2-3: Typical component test setup (Swanson 1999)
2.2 Recommendations for Strength Determination

Gao (2002) provided a significant contribution to the current knowledge state of TSA connections. The work was intended to provide recommendations for strength determination of TSA connections using the most appropriate and widely used models. These recommendations would be made using numerical component test data from a wide variety of sources. Unfortunately, many of the projects considered to provide such data lacked thorough accounts of necessary test parameters. The only reliable TSA component data originated from the ten tests performed by Swanson for SAC subtask 7.03.

The models considered included the AISC-LRFD model, the Eurocode model, the Modified Kulak model, and the W. F. Chen (1988) model. Some of these models were solely developed for T-stub connections and then were modified for TSA connections by Gao. Even with the models modified for TSA connections, the models were still to be used for less thick components and not the heavier 1” thick angles tested by Swanson (1999). He concludes that only the Modified Kulak model predicted the connection strength accurately for the connection. He further states that a more comprehensive data set of various angle thicknesses would provide more insight into which models would be most appropriate, or could be modified to be most appropriate, for determining TSA connection strength.

2.3 Analytical Modeling

Moment – rotation curve fitting techniques have been sought after using various methods since the late 1930’s. Recently, work has been carried out to form a practical moment-rotation curve fitting methodology by Lee and Moon (2002). They formulated a logarithmic model using
only two parameters that could be obtained very simply from their methods. The model produced excellent results when compared with a number of experimental tests both for bolted web angle and TSA connections. As noted in the study, applying the model to the entire spectrum of possible connection geometries would be difficult, but their model could prove as a useful tool for designers during frame analyses with semi-rigid connections.

2.4 Finite Element Modeling

Academia, as with industry, is continuously trying to move towards more research with less funding. This can readily be seen in this research as a whole. A few full-scale tests were performed to verify the results from many component tests. It follows that a few physical tests will now verify even more FEA models, given the exponential development in computing technologies. Recently, a torrent of research in FEA modeling has been performed on the full-scale and component tests of the last fifteen years.

2.4.1 Direct shear

Danesh et al. (2007) conducted a finite element modeling to experimental test exercise where they evaluated the effect of direct shear on a TSA connection with bolted web angles. Results were compared to Azizinamini’s (1982) experimental work and finite element analyses performed by Citipitioglu et al. (2002) which were also performed on Azizinamini’s work, but to a lesser degree of detail than performed by Danesh et al. It was determined that direct shear on the connection decreased initial stiffness considerably. The extent of this stiffness decrease could be directly correlated to the shear capacity of the web angles.
2.4.2 Prediction models

Pirmoz, et al. (2009) created finite element models of a number of TSA connections in ANSYS to reproduce experimental work performed by Azizinamini (1982) and Komuro et al. (2004). Once they achieved acceptable reproductions, they then used the finite element models to create a “data bank” of various TSA connection geometries. Following this, they extrapolated this “data bank” into a prediction model that could be used to predict initial connection stiffness, plastic moment, yield moment, and nonlinear connection stiffness of a generic TSA connection.

The study goes on to show a step-by-step procedure by where various parameters of the connection can be obtained. They concluded the article with the addition of axial tension on the beam of the connection. They find that for smaller axial loads, their prediction model continues estimating behavior accurately. However, as axially loads increase, the accuracy of the model decreases and they suggest further studies be conducted in this area.

2.4.3 Finite element analysis of SAC subtask 7.03

Other finite element work done on results from the SAC Subtask 7.03 includes the significant contribution by Hu et al. (2010). Their work compared Swanson’s (1999) experimental results on TSA components to finite element models they developed in Abaqus. Their models accurately predicted contact, prying, and slip behaviors. They were also able to provide insight towards event sequence during loading, importance of friction and bolt pretension parameters, and the use of a solution space based on various researches to be potentially used in the design of TSA connections.
2.5 Energy Dissipation

Abolmaali et al. (2009) preformed forty-eight cyclic full-scale connection tests on semi-rigid connections to characterize energy dissipation. Eleven of the forty-eight were TSA connections. Built-up beams were used with depths of fourteen and sixteen inches. It was concluded that hysteresis parameters individually were not directly related to the energy dissipation behavior. A complex nonlinear relationship between these parameters, caused by plastic and pinching behaviors were deduced to cause the energy dissipation characteristics of these connections.
Chapter 3 – Methods

Given the unique nature of this work, the method used to develop these models is as important as the results of the analyses performed. This chapter will elucidate the procedure used to develop the FE models and any rationale necessary to explain the chosen parameters. Specifics unique to each model will be addressed in later chapters.

3.1 ABAQUS Model Organization

It is worth briefly discussing the organization of a model within ABAQUS to clarify how terms will be used throughout this chapter. The most basic part of an ABAQUS model is appropriately named the part. Parts are unique, for example the top and seat angle of the full-shear connections are instances of a single part. Instances are copies of a part used to build the assembly which is the complete geometric arrangement of the physical test. As an example,
model FS-01 has seven parts: angle, tension bolt, shear bolt, shear tab, shear tab bolt, beam, and column. There are two instances of the angle, four of the tension bolt, eight of the shear bolt, and three of the shear tab bolt that are used in conjunction with one of the remaining parts to form the FS-01 assembly, an exact digital replica of the experimental test performed by Schrauben. (1999)

*Partitioning* is the discretization of a part, typically using planes, for the purposes of assigning an element scheme or *meshing*. A mesh can be applied directly to a part or to an instance of a part in the assembly. ABAQUS calls these dependent and independent meshing, respectively. All parts of all models developed in this body of work were meshed dependently.

The intermediary step between partitioning and meshing is called *seeding*. This step assigns the partition or part a “map” onto which the mesh is assigned. It is in seeding that the actual fineness of a mesh is assigned. There is a multitude of ways in which to seed a part or partition. The method used in this work was to assign a local seed size to each partition depending on its interest.

![Figure 3-2: Partitioned bolt and partitioning planes](image-url)
3.1.1 Partitioning

The caveat of exclusively using solid elements is that the fineness of element meshes has to be stringently controlled to keep the number of degrees of freedom within reason. As is typical for structural engineering, each node was assumed to have three degrees of freedom corresponding to translational movement in all three directions. The total number of degrees of freedom directly affects the amount of computation time required to run these analyses.

The fineness of element meshes was majorly controlled via partitioning of parts within the model. Some partitioning was performed because virgin parts could not be meshed given their abrupt geometric changes. The best examples of this are k-zones of rolled shapes and bolts. Other partitioning was performed for meshing convenience purposes. It was desirable in locations of interest to have a mesh of at least two elements deep. Though this could have been accomplished strictly through seeding, using partitions allowed for better control of the more coarse meshes and averted the need to have unnecessarily fine meshes. The locations in which the minimum of two elements deep become apparent is in thinner elements, such as beam flange and t-stub stems. Other locations were partitioned to better control mesh patterns around geometries, such as bolt holes. The liberal use of partitions enabled the models to be quite fine in areas of interest and relatively coarse in areas of little interest, such as the free end of the beam or the entirety of the column. Figures 3-1 through 3-4 demonstrate the usage of partitions described.
3.2 Elements and Implementation

Given the smaller relative size of all models developed, 3-D solid elements were used exclusively. This also allowed for differing element scheme densities and solid element types without the issues associated with transitioning shell or plate elements into 3-D solid elements. Larger FE models, such as bridge spans, will use solid elements in areas of interest and shell or plate elements in areas of less importance. The models developed here vary in element density and number of nodes, rather than element type, to highlight interest.

Figure 3-4: Partitioning scheme of model TA-26 (different parts colored)
3.2.1 Reduced Integration

Reduced integration is a form of numerical integration of a lower-order quadrature rule. Reduced integration is used in finite element analysis to aid in “softening” the response as well as to provide computation savings. The “softening” comes from higher-order polynomial terms becoming negligibly small at sample points of a low-order rule. These negligibly small terms then do not contribute to strain energy, therefore offering less resistance to deformation. These negligible terms are ultimately ignored in the integration process, hence reduced integration. Given the fewer sample points, reduced integration additionally provides for computational savings. ABAQUS will assign an “R” to the element name for reduced integration elements.

3.2.2 Solid Element Types

The two types of 3-D solid elements used were 8-node rectangular solids, called C3D8 by ABAQUS, and 20-node rectangular solids, C3D20. These are the three-dimensional versions of the two-dimensional bilinear rectangle (Q4) and the quadratic rectangle (Q8) elements.

Figure 3-5: Q4 (top) and Q8 (bottom) elements
The 3-D elements share the same advantages and disadvantages as their 2-D counterparts. The bilinear elements’ normal strain fields exhibit constant variation in the direction of strain whereas quadratic elements exhibiting linear variation in the direction of strain. This causes the bilinear elements to undergo a phenomena called *shear locking* which causes unrealistic increases in stiffness to develop. This would inherently make bilinear elements a poor choice for bending, but an equal choice for axial compression or tension. As mentioned in the previous section, reduced integration was used on all elements and provides softening for elements. This dissolves the shear locking issue suffered by the bilinear elements.

![Figure 3-6: Bilinear (top) and quadratic (bottom) rectangular solid (hexahedral) elements](image)
3.2.3 Element Sensitivity Analyses

Early in this work, the question arose as to what type of elements to use where, given that reduced bilinear elements could potentially perform as well as quadratic elements. The fineness of element meshes was also in question at the time. An element sensitivity analysis was undertaken using a cantilever wide flange beam. It was modeled and analyzed in ABAQUS using 8 and 20 node elements, and a generically coarse and fine mesh for a total of four models.

![Load vs. Displacement of Cantilever Beam, Elastic](image)

Figure 3-7: Load-displacement response from 10 kip load applied to cantilever beam

The cantilever beam used was the same as that used for Schrauben’s FS-01 and FS-02 tests (W18x40, 172.6” long). The material definition used was derived from mill reports in Schrauben’s thesis. A bilinear material model was used and will be explained in further detail in
Reference points were used at the extreme ends of the beam to constrain all degrees of freedom of the exposed cross-section to a single point at the center of the section. One reference point was restrained from translation and rotation in all directions and the other was loaded monotonically. The four models were c8, f8, c20, and f20, the “f” representing fine meshing, the “c” representing coarse-meshing, and the 8 or 20 indicating which 3-d solid element used. Two sets of sensitivity tests were performed. Initially, an arbitrary load of 10 kips was applied downward. Figure 3-7 shows these results. Obviously, this is well within the elastic range of the material. As such, the elements experience no inelastic behavior and it should be expected that all four models produced the same result.

Figure 3-8: Load-displacement response from 25 kip load applied to a cantilever beam
The second set of tests intended to produce a plastic hinge in the beam. This would move elements into inelastic regions in which they could behave differently. From beam theory, the load necessary to cause a plastic hinge to form is 23.53 kips. A load of 25 kips was applied to the beam. Figure 3-8 illustrates the results. As this plot suggests, a coarse meshed, 8-node-element schemed beam behaves identically to a fine meshed, 20-node-element schemed beam. It is also worth noting, a plastic hinge was successfully formed in each of the beams. Figure 3-9 shows the f20p model in its deflected shape (“p” represents the modeled was loaded to achieve plastic behavior). The contours show equivalent plastic strain (PEEQ). The dark blue represents material at zero PEEQ, in other words material that is still elastic. Any other color represents yielded material.

![Figure 3-9: Model f20p deflected and illustrating plastic hinge](image)

### 3.2.4 Ramifications of Sensitivity Analyses

Given that the sensitivity analyses suggested using 8-node elements and fairly coarse meshes throughout could yield appropriate results, the question remained; which elements should be used where? It was decided that areas where significant inelastic deformation was expected, except for bolts, 20-node elements would be used in relatively fine mesh schemes. Areas of
elastic deformation or areas of little interest were assigned 8-node elements in meshes which tapered in coarseness from a fine mesh near the 20-node elements to visibly coarse meshes.

Bolts were an exception to the rule. Given their small size, relative to the beam and column, smaller elements had to be used to accurately capture the stress and strain distribution in them. However, because the bolts were mainly subjected to shear or tension, 8-node elements were used. This also provided computational savings.

Coarser meshes were attempted for the bolts, but convergence errors occurred. Saint-Venant effects were assumed to be affecting the bolts, thus finer meshes were assigned.

The last ramification from the sensitivity analyses is that they demonstrated ABAQUS could produce a plastic hinge in a beam. Though this work centers on the idea that the configurations of both top-and-seat angle and t-stub connections would be partial strength, and therefore would not form a plastic hinge in the beam, it is a potential failure mechanism. The fact that this mechanism can be formed by ABAQUS, but was not further noticed, demonstrates the accuracy of this modeling method and the consistency of these connections. This will be explained further in subsequent chapters.

3.3 Bolt Modeling

A490 bolts were used in all models as they were the fastener of choice in all comparable experiments. The head, shank, washer, and nut were all modeled as a single part. The head and nut were simplified to be cylindrical. The washer and nut were modeled as a single geometric entity, using the washer’s nominal outer diameter as the diameter and the nut height plus the maximum washer thickness as the total thickness. These values came directly from AISC Table
Bolts underwent tension, prying, bearing, and shear in these analyses. All bolts in shear were tested excluding the thread from the shear plane, but the thread was not excluded from the grip. All calculations made in originally designing these experiments and made during the analysis work done herein, used the effective area of the shank, not the nominal area suggested by AISC. The following is the equation for effective area provided for reference:

\[ A_{\text{eff}} = \frac{\pi}{4} \left( d - \frac{0.9743}{n} \right)^2 \]  

In the above equation, \( d \) represents the nominal shank diameter and \( n \) is the number of threads per inch. The tension strength of a bolt is weakened by the reduced cross section in the threaded portion of the shank. The above equation accounts for this. The issue with using a diameter based on this effective area for the bolt’s shank is that bearing still occurs on the nominal diameter in both the unthreaded and threaded regions of the shank. Should bearing occur in the threaded region, the ridge of the threads still has the nominal diameter against which the hole can
bear. A method whereby a cylindrical void was created the depth of the threaded region within the grip was used to address both tension and bearing in a bolt.

Table 3-1: Shank Void Depth Calculation

<table>
<thead>
<tr>
<th>Bolt location</th>
<th>1. Grip</th>
<th>2. Bolt length from experiment</th>
<th>3. Bolt diameter</th>
<th>4. Thread Length from AISC</th>
<th>5. Amount of bolt unthreaded (Col. 2 - Col. 4)</th>
<th>6. Threaded region in Grip (Col. 1 - Col. 5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle-Column</td>
<td>2.09</td>
<td>3.5</td>
<td>7/8</td>
<td>1.5</td>
<td>2</td>
<td>0.09</td>
</tr>
<tr>
<td>Angle-Beam</td>
<td>1.525</td>
<td>3</td>
<td>7/8</td>
<td>1.5</td>
<td>1.5</td>
<td>0.025</td>
</tr>
<tr>
<td>Shear tab-Beam</td>
<td>0.6275</td>
<td>2</td>
<td>7/8</td>
<td>1.5</td>
<td>0.5</td>
<td>0.1275</td>
</tr>
</tbody>
</table>

Table 3-1 shows how this void depth was accomplished. Column 6 gives the void depths used for model FS-01 and FS-02 bolts. The diameter of the void is determined by

\[
A_{\text{nominal}} - A_{\text{eff}} = A_{\text{void}} \tag{3-2}
\]

Where

\[
A_{\text{nominal}} = \frac{\pi}{4} d^2 \tag{3-3}
\]

\[
A_{\text{void}} = \frac{\pi}{4} d_{\text{void}}^2 \tag{3-4}
\]

Solving for \( d_{\text{void}} \) gives the following to determine the diameter of the cylindrical void:

\[
d_{\text{void}} = \sqrt{1.9486 \frac{d}{n} - 0.94926 \frac{n}{n^2}} \tag{3-5}
\]

Where \( d \) is the nominal bolt diameter. AISC reports 9 threads per inch for a 7/8” bolt (AISC 2005). Substituting these values into equation 3-5 gives 0.4216” for the diameter of the void. Figure 3-11 shows the same bolt from figure 3-10 in frame view, highlighting the cylindrical void.
3.4 Constraints, Restraints, and Loading

Constraints were used for both restraints and loads as a means to accurately apply them. Both restraints and loads can be applied to surfaces in ABAQUS, but it was deemed more accurate to kinematically constrain a surface to a single point, known as a *reference point*, and then apply the restraint or load to that reference point.

Almost all the constraints used amongst the models developed for this work were kinematic coupling constraints. As mentioned briefly in previous sections, in this type of constraint the degrees of freedom of the locus of nodes that make up a surface are tied to the degrees of freedom of a single point. These reference points were typically placed at the center of the surface to avoid any unforeseen eccentricities.

The only other constraint type used was a *tie*. This was used to simulate the weld between the column and the shear tab in the two full-scale tests modeled (FS-01 and FS-02). Ties are surface to surface constraints where the movement of the nodes on the slave surface follows the movement of the nodes on the master surface at the connection point of the slave to
master. This makes for a seamless entity between column flange and shear tab. This was done as a modeling simplification because the weld was not a point of interest, nor did it experience any excessive inelastic deformation during experimental tests.

Restraints were assigned on a model by model basis depending on how experimental tests were restrained or braced, and in some cases to alleviate unexpected out-of-plane deformation. More detail will be provided in subsequent chapters.

Loads applied to all models were based on loads applied experimentally. Experimental load histories were simplified in all cases from cyclic to monotonic loading. This was done because of the significant complexities prevalent in assigning cyclic loads in FE models. All analyses demonstrated exceptional agreement between their monotonic responses and experimental cyclic responses. Loads were assigned in ABAQUS at reference points as non-zero restraints. This would be akin to assigning settlement in theoretical exercises. For example, models FS-01 and FS-02 were assigned a non-zero restraint of -7.22” in the $y$ direction at the end of the cantilevered beam. Further elaboration shall be provided in subsequent chapters as to how and why loads were chosen for the FE models.

3.5 Contact Surfaces

The definition of contact surfaces is the way in which ABAQUS assigns behavior between instances of a part. Contact surfaces physically correspond to where two entities touch, such as the beam flange and the angle, or where two entities could interact throughout the course of the analysis, such as a bolt and hole. ABAQUS can either automatically find these contact pairs, or they may be manually assigned. Given that a contact pair goes down to part partition scale, it is easy to see how it could be very time consuming to assign all of these manually and
the risk of missing one is extremely high. Therefore, all contact pairs were assigned manually via ABAQUS’s “find contact pairs” module. Once the pairs are found, they must be assigned an interaction property. As far as these models are concerned, the two types of interaction properties considered were normal and tangential behaviors of entities.

The normal behavior interaction property used in all of these models was the default “hard contact”. The physical correspondent to this would be two steel plates meeting face to face. They impose a normal stress on one another that could be constant, linear, parabolic, etc., but any deformation caused is so small it is considered negligible in these models. The case could be different if there was a very small cross-section bearing into a very large cross-section; there might be non-negligible penetration in this case.

3.5.1 Tangential Behavior

The tangential behavior interaction property is more easily explained in physical terms, but far less easily defined analytically. Tangential behavior is referring to the slip, or friction, between two entities. The value to be entered into ABAQUS is Coulomb’s coefficient of static friction for the two interacting surfaces, in this case, steel against steel surfaces. Research shows the average slip coefficient for steel on steel to be 0.34 with a 0.07 standard deviation (Kulak et al. 2001). A slip coefficient of 0.30 was used as a starting point for the models. As the faying surfaces were not prepared according to any AISC procedures for slip surfaces, it was deemed a reasonable starting place.

The determination of the true slip behavior became an iterative process. Given the inherent variability of slip coefficients, there is no guarantee the slip coefficient that matches the experimental results is at all indicative of other experimental tests.
Curve $d$ in Figure 3-12 shows the monotonic response of one of the FS-01 models using a coefficient of friction of 0.30. The plateaus that each of the monotonic responses exhibit after the initial elastic range is the point at which the frictional resistance is overcome and the top angle slips relative to the beam’s top flange. As it can be observed, curve $d$ slips at a much higher load than the experimental data shows. It was decided at this point that various slip coefficients would be investigated to determine the appropriate value. Curve $f$ and $h$ were analyzed with slip coefficients of 0.40 and 0.20, respectively.

Figure 3-10 illustrates that curve $h$ is the best match to the experimental data. Note however that the slip coefficient of 0.20 is lower than both the research recommended 0.34 and the code recommended 0.35, even though the faying surfaces of the experimental test were described as those appropriate to use the 0.35 coefficient for design. This could suggest slip coefficients are far more variable that current research can predict and that a 0.35 coefficient used for design could be a dangerous overestimate. Figure 3-13 shows a normal probability
density function constructed from the research mean slip coefficient 0.34 and its standard deviation 0.07 (Kulak et al. 2001).

The vertical lines indicate the slip coefficients used in Figure 3-10 to bound the true slip coefficient, coincidentally finding 0.20 to be the most accurate. Inspecting Figure 3-11, it can be seen that 0.20 is two standard deviations from the mean.

### 3.5.2 Slip Coefficient Recommendations

The previous observations were based the FS-01 model behavior alone. Consequently, a slip coefficient of 0.20 was used for all models in this work and it provided adequate results in all cases. This similar behavior of faying surfaces for numerous tests from multiple researchers could be attributed to similar storage and lot number. All models analyzed in this work were based on experimental tests conducted at the Georgia Institute of Technology (GT) around the
same time period. It is possible the steel used in these tests was rolled and fabricated around the same time and that the steel was stored at GT under similar conditions.

It is recommended, based on these factors, that if FE models are created to match experimental results that a single model be adjusted such that its slip and the slip of the corresponding experimental test agree. This coefficient of friction could then be used for subsequent models. If the FE models are being created to predict potential experimental results (as this work is intended to help facilitate) further research and investigation must be done to ensure adequate slip coefficients are used, or potential experimental tests must be assumed to have more concern for faying surface preparation, such that design slip coefficients could be used.

3.6 Bolt Pretensioning

All of the experimental tests modeled used pretensioned bolts. It is also common practice to pretension bolts even if a connection is not slip critical even for serviceability. Pretensioning was implemented into the FE models by means of applying a temperature gradient to the bolt shank. Restraints were then applied to the underside of each nut such that the degrees of freedom while the shank compressed were limited for control purposes.

The appropriate temperature gradient was applied in ABAQUS using the following manipulation of thermal expansion equations:

\[ \sigma = E\varepsilon \quad (3-6) \]

\[ \varepsilon_T = \alpha \Delta T \quad (3-7) \]

\[ \sigma = E(\alpha \Delta T) \quad (3-8) \]

\[ \sigma = \frac{P_b}{A_B} \quad (3-9) \]
\[
\frac{P_b}{A_b} = E(\alpha \Delta T) \tag{3-10}
\]

\[
\Delta T = \frac{P_b}{A_b E\alpha} \tag{3-11}
\]

Where \( \epsilon \) is strain, \( \epsilon_T \) is thermal strain, \( \sigma \) is normal stress, \( \Delta T \) is temperature change, \( E \) is Young’s modulus, \( \alpha \) is the coefficient of thermal expansion, \( P_b \) is the minimum bolt pretension force, and \( A_b \) is the nominal area of the bolt shank (Beer et al. 2006). Obviously since steel is the material used, the Young’s modulus is 29,000 ksi and the coefficient of thermal expansion is \( 6.6 \times 10^{-6} \) strain per degree Fahrenheit. The bolt area and minimum pretension force depend on the bolt used and also the type of material, be it A325 or A490. Since 3-11 represents a form of the thermal expansion equation, a negative value of the temperature change output from that equation is used in ABAQUS to cause contraction and therefore induce a pretension into the bolts. Table 3-2 shows the effects of this method of pretensioning.

<table>
<thead>
<tr>
<th>Model</th>
<th>Bolt diameter (inches)</th>
<th>Pb (kips)</th>
<th>Target stress (ksi)</th>
<th>Actual stress (ksi)</th>
<th>Stress difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FS-01</td>
<td>7/8</td>
<td>49</td>
<td>81.48733</td>
<td>93</td>
<td>14.1281706</td>
</tr>
<tr>
<td>FS-02</td>
<td>7/8</td>
<td>49</td>
<td>81.48733</td>
<td>93</td>
<td>14.1281706</td>
</tr>
<tr>
<td>TA-25</td>
<td>1</td>
<td>64</td>
<td>81.48733</td>
<td>100</td>
<td>22.718463</td>
</tr>
<tr>
<td>TA-26</td>
<td>1</td>
<td>64</td>
<td>81.48733</td>
<td>100</td>
<td>22.718463</td>
</tr>
</tbody>
</table>

As stated in a previous section, the area of the shank is not constant and would change where the threads entered the grip for each bolt. The depth of the area change depends on bolt and grip lengths, which changed between the models. The stress difference is attributed to that, but in either case the pretension requirement is a minimum. All actual stress values are higher than the minimum, but not exceedingly higher than, such that bolt strength behavior or slip behavior is affected.
3.7 Material Properties

The method of modeling developed in this chapter was verified using experimental results primarily. It follows that the material properties used in the models would be derived from mill material reports, if available. If material reports were unavailable, a reasonable approximation was made based on the nominal properties of a material. If mill reports were available, an elastic-plastic with hardening model was used. If they were not available, an elastic perfectly plastic model was used. Poisson’s ratio used for all materials in all models was 0.3.

3.7.1 Engineering and True Stress and Strain

The ABAQUS finite element software performs operations assuming true stress and strain as input. However, most engineering applications utilize engineering stress and strain. The difference between true and engineering stress and strain is that true stress/strain uses the actual cross sectional area at every calculated point whereas engineering stress/strain uses the nominal area at all points. It is easy to imagine the Poisson effect altering the dimensions of a body under load, but typically this is ignored by engineers because it is difficult to measure and the dimension change is usually so slight, the effect is negligible. A conversion formula follows:

\[
\varepsilon_{\text{true}} = \ln(1 + \varepsilon_{\text{eng}}) \quad (3-11)
\]

\[
\sigma_{\text{true}} = \sigma_{\text{eng}} (1 + \varepsilon_{\text{true}}) \quad (3-12)
\]

Where \( \varepsilon_{\text{true}} \) is true strain, \( \varepsilon_{\text{eng}} \) is engineering strain, \( \sigma_{\text{true}} \) is true stress, and \( \sigma_{\text{eng}} \) is engineering stress. Given that the Poisson effect only begins to make significant differences between engineering and true stress/strain after yielding, all materials retain the same initial stiffness (Young’s modulus for steel).
3.7.2 Section Material Properties

In addition to mill material reports, the original experimenters would perform further material investigations. They would cut and test coupons for each entity of a section of material tested. For example, the A572-50 material used for the W18x40 section had multiple coupons cut from both the web and the flange. The results from these further tests for both the flange and the web were averaged to give two distinct material property interactions; one for the W18x40 web, the other for the W18x40 flange. Table 3-3 shows the breakdown of the W18x40 material properties.

The W18x40 section was the beam element for the full-scale tests (FS-01 and FS-02). A choice was made for each material section as to which entity property to use for the section in lieu of using two different material properties. Should bending behavior of the section be the major behavior, the flange material properties would be used. This was the case of the W18x40 section as shown in Figure 3-14. Should shear behavior govern, the web material properties would be used. There were a few occurrences where the sections were not used in their usual fashion, i.e. t-stub sections. Subsequent chapters will elaborate more on which material properties were used for each material section.

![Figure 3-14: Plot of Engineering vs. True Stress/Strain](image)

Figure 3-14: Plot of Engineering vs. True Stress/Strain
Table 3-3: A572-50 material properties breakdown for the W18x40 section

### Mechanical Properties:
- **Manufacturer:** Nucor-Yamato
- **Designation:** W18 x 40
- **Grade:** A572-50 (A572GR50-94C / A709-96 GR50)

- **Ave Mill Yield:** 52.0 ksi
- **Ave Mill Ultimate:** 67.5 ksi
- **Ave Mill Elong.:** 28.0 %

### Georgia Tech Tension Tests

<table>
<thead>
<tr>
<th>Cpns</th>
<th>E (ksi)</th>
<th>F_yd (ksi)</th>
<th>ε_s (με)</th>
<th>E_s (ksi)</th>
<th>F_u (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>F-07A</td>
<td>30296</td>
<td>52.6</td>
<td>21812</td>
<td>298</td>
<td>69.0</td>
</tr>
<tr>
<td>F-07B</td>
<td>31119</td>
<td>50.9</td>
<td>19719</td>
<td>353</td>
<td>68.3</td>
</tr>
<tr>
<td>W-07A</td>
<td>30418</td>
<td>62.6</td>
<td>27600</td>
<td>189</td>
<td>75.0</td>
</tr>
<tr>
<td>W-07B</td>
<td>31333</td>
<td>61.7</td>
<td>32913</td>
<td>520</td>
<td>74.9</td>
</tr>
<tr>
<td>Avg F</td>
<td>30708</td>
<td>51.8</td>
<td>20766</td>
<td>326</td>
<td>68.7</td>
</tr>
<tr>
<td>Avg W</td>
<td>30876</td>
<td>62.2</td>
<td>30257</td>
<td>355</td>
<td>75.0</td>
</tr>
</tbody>
</table>

### FLANGE:

#### Engineering:

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.001687</td>
<td>51.8</td>
</tr>
<tr>
<td>0.020766</td>
<td>51.8</td>
</tr>
<tr>
<td>0.21106</td>
<td>68.7</td>
</tr>
</tbody>
</table>

#### True (Log):

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.001685</td>
<td>51.88731</td>
</tr>
<tr>
<td>0.020553</td>
<td>52.81361</td>
</tr>
<tr>
<td>0.191496</td>
<td>81.85577</td>
</tr>
</tbody>
</table>

### Plastic strain from true:

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>51.88731</td>
</tr>
<tr>
<td>0.018867</td>
<td>52.81361</td>
</tr>
<tr>
<td>0.18981</td>
<td>81.85577</td>
</tr>
</tbody>
</table>

### WEB:

#### Engineering:

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.002015</td>
<td>62.2</td>
</tr>
<tr>
<td>0.030257</td>
<td>62.2</td>
</tr>
<tr>
<td>0.211566</td>
<td>75</td>
</tr>
</tbody>
</table>

#### True (Log):

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.002013</td>
<td>62.32518</td>
</tr>
<tr>
<td>0.029808</td>
<td>64.00255</td>
</tr>
<tr>
<td>0.191913</td>
<td>89.39351</td>
</tr>
</tbody>
</table>

### Plastic strain from true:

<table>
<thead>
<tr>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>62.32518</td>
</tr>
<tr>
<td>0.027795</td>
<td>64.00255</td>
</tr>
<tr>
<td>0.189901</td>
<td>89.39351</td>
</tr>
</tbody>
</table>
3.7.3 Bolt material properties

A490 bolts were solely used in all models developed. Elastic-perfectly-plastic models were used for all bolts in all models. High strength bolts have recently been shown to perform at reliably higher strengths than those documented in design manuals (Moore 2007). The expected ultimate strength was used in lieu of the ultimate strength in order to accurately capture the behavior of these bolted connections. This has been shown to be 165 ksi for A490 bolts instead of the typical 150 ksi.

An investigation was undertaken early to reinforce the fact that 165 ksi better represented A490 bolts than 150 ksi. The work producing Figure 3-15 was undertaken at the same time slip was being investigated. This plot shows two models using a slip coefficient of 0.30 and hence the higher plateau, but that is not the focus of this plot with respect to bolts. Attention should be turned to the behavior after the slip plateau. Notice the 150 ksi bolts experience a decrease towards the end, a behavior not reflected by the experimental results. The 165 ksi bolt matches the trend of the experimental data much more effectively and hence the choice to procedure using the effective ultimate strength for these bolts.

Figure 3-15: Ultimate bolt strength comparison for FS-01 model
3.7.4 ABAQUS and Ultimate Values

It should be mentioned how ABAQUS handles material definitions past the last plastic input point. ABAQUS cannot model failure, so after the last entered plastic material point, it assumes another infinite plateau. It is important to recognize this as equivalent plastic strain contours must be adjusted such that strains past the ultimate strain of the material are taken as failure. Given this unique problem, two things are important: the first is to make sure adequate ultimate strengths are known or thorough research is done into what appropriate values could be substituted. The second is to be careful when inspecting stresses in models as the ultimate stress will continue to propagate along the plateau past the ultimate strain.

3.7.5 General Recommendations for Material Properties

Modeling past experiments provides the luxury of using documentation to accurately reproduce material properties. When making models to predict behavior, a different approach must be implemented. Plenty of research has investigated material properties of various steel types. This research should be used when inputting material properties for this modeling method. Similar to how this body of work went about using the expected ultimate strength of bolts; in this way accurate material properties should be obtained. Design material properties provide for a level of conservatism that could skew the true behavior of systems being modeled and thus should be avoided for modeling purposes.

3.8 Analysis Steps

The actual analysis of each model took place using a number of steps. These steps were ordered in a way such that various aspects of the model were created, altered, or deactivated such
that the analysis would replicate a physical test. The first step in every model was the ABAQUS standard *Initial* step. This step is where the interaction pairs identified previously were applied to the model and where initial restraints are applied. Some of these restraints are altered or deactivated in subsequent steps, but they are introduced here. The Initial step could be called step zero. No analysis whatsoever is performed on the inputs of this step. It is merely the foundation for the analysis.

The next step for all models was *Pretension*. This is effectively step one from the analysis standpoint. This is the point at which the temperature gradient is applied to the bolt shanks and the restraint on the underside of the bolt head is released in the longitudinal direction of the bolt, such that the shank is allowed to contract to facilitate pretensioning. As mentioned in a previous section, the two directions orthogonal to the shank were left restrained on the bolt to control the movement to a single direction. The remaining restraints are propagated through this step.

The final step in all models is the loading step (step two), termed various names depending on the loading. Since all models were loaded monotonically, there is a single load step. Had there been more than one, additional steps would be used. The load is created using a non-zero restraint in this step. The magnitude and point of application will be discussed in subsequent chapters.

All restraints are global in nature. This means that the restraints under the bolt head from the pretension step must be deactivated such that the bolts can move during loading. The temperature gradient is propagated to keep the bolts pretensioned. This would cause the cross section of the shank to contract accordingly, but this effect is considered negligible. The
additional restraints orthogonal to the bolt in the pretension step were not needed, but they provided for more controlled pretensioning.

3.8.1 Step Properties

This section details specifics of the analysis steps, so all steps but the initial step. The time period of each step was 1. This could be construed as *one second*, but given these analyses were non-dynamic monotonic loads, 1 was just a convenient method of keeping track of where events occurred in the analysis. For example, time stamp 0.63 would correspond to 63% of the pretension step, 1.74 would correspond to the load step and 74% of the total load applied, etc. Nonlinear geometric effects (referred to in ABAQUS as *Nlgeom*) were enabled in every step to capture proper behavior of the model. This was a simple on/off feature, but represented a huge contribution to overall behavior and was not something to be neglected.

The incrementation options were based solely on the wisdom of more advanced users of ABAQUS. ABAQUS was allowed to automatically increment based on the following parameters:

- Maximum number of increments per step = 100
- Initial increment size = 0.05
- Minimum increment size = 0.00001
- Maximum increment size = 1

These parameters provided ABAQUS with the ability to adjust the increment size from 5% of the time period down to either 0.001% or up to 100%. If too many small increments were needed for the analysis to converge, the analysis halted when it reached 100 increments. This
was done to keep computation time from becoming unreasonable and was based on previous experience.

The remaining properties were left as the default ABAQUS values. These included a direct equation solver method using the default matrix storage, a full Newton solution technique, and a linearly ramping default load variation over the time step.

### 3.8.2 Step Recommendations

It is recommended that unless further research and experience, or greater computational resources are available that the above step properties be used for finite element modeling of bolted connections and their components.
Chapter 4 – Component Models and Results

This chapter will discuss the specific setup of three models; two t-stub component tests from Swanson (1999) and a shear bolt test from Moore (2007). These models were developed to further validate the modeling procedure by verifying failure mechanisms. The modeling procedure is discussed in Chapter 3.

4.1 Basic Shear Model Setup

The basic shear model was developed for the sole purpose of making sure the modeling procedure developed in Chapter 3 could adequately predict shear failure of bolts. None of the full-scale tests failed due to shear in the bolts, as will be discussed in Chapter 5, nor do any of the t-stub component models to be discussed. Realistically, connections and systems are designed with their governing failure mechanism determined. When experimentally tested, these theoretically governing failure mechanisms do not always govern. It was essential that any analysis could give any failure, not just whatever was witnessed in experimental testing or predicted theoretically.

An experimental test performed by Moore (2007) on a 7/8” A490-X bolt was modeled for the basic shear analysis. The specifics of the test setup can be found in Moore’s thesis. Figure 4-1 shows a photograph of the testing apparatus. The model was simplified to the bolt and the two immediate plates it connected as shown in the right photo. Through restraints the test was replicated. Figure 4-2 shows the ABAQUS assembly.
All models were made using the modeling methodology elucidated in Chapter 3. The model assembly was composed of two 6” x 6” x 1” plates and one 7/8” diameter bolt. Holes in
the plate were \(7/8 + 1/16 = 15/16\)" in diameter. Since these bolts would only be experiencing bearing on the shank and shear through the nominal diameter, it was not necessary to use a void in the shank of the bolt as was done for bolts in bearing and tension. The bolt was placed in the center of the hole, as was done for the top-and-seat angle models.

The 7/8” bolt was A490 material and the plates were 1045 steel. 1045 steel was used in the experiment as the immediate steel in contact with the bolt shank because it had the same hardness as A992 steel, but was machineable. This would provide the same bearing behavior as a rolled section of A992. The average yield strength of 1045 steel is 57 ksi and the average tensile strength is 92 ksi. Both the A490 and 1045 material were modeled as elastic-perfectly-plastic in ABAQUS. This was done for the A490 material because it was done this way for all bolt materials in all models as discussed in Chapter 3. An elastic-perfectly-plastic model was used for the 1045 steel because there were no mill reports for the material. No conversions were needed from engineering to true stress/strain for these materials. The standard Young’s modulus for steel, 29000 ksi, was used.

A total of five contact pairs were identified by ABAQUS for this model. All pairs were given the same property that used a coefficient of friction of 0.2 for the tangential behavior and “hard” contact for the normal property.

Steps used for this analysis are the same as those to be described in Chapter 5 for the FS-01 and FS-02 model with the specifics of those steps discussed in detail in Chapter 3.

4.1.1 Elements and Meshing

General methodology about partitioning, seeding, elements, and meshing can be found in Chapter 3. This section will detail the specifics of these aspects for the basic shear parts.
The bolt was partitioned such that its head, shank, and nut/washer were three separate entities, as previous bolts were partitioned. An approximate global seed size of 0.1 inches was used with C3D20R elements. This provided for 5,755 elements and 26,432 nodes for the bolts. This is a massive number of elements and nodes for a single part, but given how small the whole model is relatively, this did not cause any computation issues. Figure 5-3 shows the bolt meshed.

![Figure 5-3: 7/8” bolt meshed for basic shear model](image)

The plates were not partitioned at all. An approximate global seed size of 0.25 inches was used with C3D20R elements. This provided for 2,708 elements and 13,619 nodes for each plate. Figure 5-4 shows a plate meshed.

![Figure 5-4: 6” x 6” x 1” plate meshed for basic shear model](image)
The relatively small size of the entire model made for an opportunity where C3D20R elements could be used in a relatively fine mesh without excessive computation times. As mentioned in chapter 3, much coarser and less precise elements could have been used to produce the same results.

4.1.2 Constraints, Restraints, and Loads

Only two constraints were used in this model; one for loading and one for restraining. The edge of one plate was constrained to a reference point at its center and the same was done for the other plate on the opposite edge. One reference point was applied a fully fixed restraint (no translation or rotation in any direction) and the other was loaded with a single displacement of 0.24 inches in the direction that would cause the plates to slide with respect to one another and would put the bolt into pure shear. Figure 4-5 is to provide clarity to this description. RP-1 was the restrained reference point and RP-2 was loaded. An applied displacement of 0.24 inches was chosen because it was slightly past the failure point of the bolt in shear according to the experimental results.

![Figure 4-5: Reference points on basic shear model assembly](image-url)
The only other restraint used was the one on the bolt head for the purposes of pretensioning explained in detail in Chapter 3. The experimental test did not use pretensioning, but attached the bolt by finger tightening. The pretension force provided by finger tightening was estimated. Five percent of the tensile strength of the bolt was chosen. That is 8.25 ksi. Solving for the temperature gradient using equation 3-11 gives a temperature needed of -43 degrees Fahrenheit. Even though these bolts were not pretensioned, the same method used for pretensioning described in Chapter 3 was used here to induce a stress for finger tightening.

4.2 Basic Shear Analysis

This section will discuss both the analysis events and results. The only event that occurred prior to the bolt failing in shear was the slip of the plate. However, since the behavior of the plates is not the real goal of the model, it is secondary to the behavior to the bolt. Figure 4-6 shows the deflected shape of the model assembly with contours indicating Von Mises stresses.

Figure 4-6: Final deflected shape of bolt shear model in ABAQUS
The plates in Figure 4-6 show some out-of-plane motion. This was not restrained against. The shear apparatus was restrained from out-of-plane moment, but not indefinitely. Figure 4-7 shows a photo of the means by which the shear apparatus was kept in plane. However, out of plane motion was still seen experimentally within the tolerance of those machined pins (Moore, 2007).

![Figure 4-7: Shear fixture with modified pins (Moore 2007)](image)

The value of ultimate plastic strain was unavailable for the 1045 material, but even at the lowest strain bounds, the plates showed no sign of shear rupture, shear yielding, or bearing failure. There was some inelastic behavior in the bearing region, but it did not propagate to the edge of the plate in any fashion. Figure 4-8 shows the PEEQ for the more heavily loaded plate. The PEEQ bounds are from 0 to 0.02. The grey represents PEEQ outside of those bounds. As illustrated, no plastic strain propagates to the edge of the plate.
Figure 4-8 adequately justifies that the plate did not govern the behavior of the analysis and therefore that the bolt did. This is in agreement with the experimental observations. Figure 4-9 shows the analysis results compared to the experimental data provided by Moore.

**Load vs. Displacement**

- Experimental data (C11-X-01)
- Analysis results (Dec. 6)

Figure 4-9: Load-displacement plot for the bolt shear model comparing experimental data to analysis results
Comparing the initial behavior is difficult as material specifics were unknown about the two plates being joined by the bolt in the experimental test. The 1045 steel was the material of the *inserts* used in the test apparatus in Figure 4-1. The inserts were the material in direct contact with the bolt shank, which was why their bearing behavior was intently monitored. However, the larger shear apparatus was made intentionally tougher, so that excessive deformation would not occur during the testing of a massive number of bolts (Moore 2007). The material details, such as slip resistance for instance, were unknown for those plates. Initial behavior for this exercise should therefore be ignored.

It is important to note that both the curves begin overturning at the same point, regardless of initial behavior. This is the point at which the bolt begins to experience excessive deformation and to govern the behavior of the system. After the overturning point, around 0.115 inches, the curve of the experimental data and the analysis results follow one another closely, until the bolt fractures in shear in the experimental test. The max load in the experimental data was 60.28 kips and occurred at 0.1972 inches of displacement. The closest displacement in the analysis results was 0.1978 inches and the load there was 59.93 kips. This is a 0.6% difference which is by far the best corroboration of experimental data and analysis results, but it is by far the simplest model. This does not trivialize the fact that bolt failure from shear can indeed be accurately predicted in models developed by the modeling procedure in Chapter 3.

### 4.3 TA-25 Model Setup

The final two models created were two of the many t-stub component tests performed by Swanson (1999). These were modeled because they were similar in concept to the top-and-seat angles used in the full-scale tests. They were chosen among the many t-stub component tests
because both exhibited failure mechanisms that neither the top-and-seat angles in the full-scale tests nor Swanson’s angle component tests experienced. This section covers the specific setup of model TA-25, a t-stub component test that failed from net section fracture. This model was composed of the following:

- Flange of a stiffened W14x257 column stub
- Built-up beam with 6” x 1” flange
- 15-1/2” t-stub cut from a W16x100
- Eight 1” diameter, 4-1/2” long tension bolt
- Eight 1” diameter, 3-1/2” long shear bolts (A490-X)

All bolts were A490 steel and the column flange, beam element, and t-stub were all A572-50 steel. Figure 4-10 shows the ABAQUS assembly. The specifics of the experimental test setup may be found in Swanson’s dissertation (1999). The analysis steps for this model follow the method outlined in Chapter 3 and are the same as the specific steps to be described for the FS-01 and FS-02 models in Chapter 5.

Figure 4-10: TA-25 model assembly in ABAQUS (colors display different parts)
4.3.1 Model Geometry Liberties

A number of modeling liberties were taken with the dimensions of the physical components, none of which will affect the analysis, but are documented nonetheless. The column stub in the experimental test was designed to be a rigid element and therefore saw negligible deformation. This was made such that a single column stub could be used for multiple component tests. Since the column was designed to not impact the response of the system, it was modeled as a single plate with a perfectly fixed restraint encompassing the side opposite the t-stub. The width of the flange was the same direction as the length of the t-stub, leaving for 1/4" extension of the column flange on either side of the t-stub. The length of the column flange was decided to be 1.5 times the width, for a total of 24 inches. This dimension needed to be only larger than the t-stub flange width. The column flange thickness was 1.89 inches, taken directly from the 13th edition of the AISC Steel Construction Manual (AISC 2005).

The experimental beam elements were designed to have roughly the same size flange a beam used with that t-stub would have, but be strong enough to be used in multiple component tests. A much larger flange was tapered down to a 6” x 1” flange, and a web was welded to this flange to increase stiffness. The beam element was a much larger built up t-stub. The flange at the end not connected to the t-stub was bolted to the actuator and was loaded axially. The end of the beam element connected to the actuator was not modeled for simplicity. The length of the beam element was shortened to three times the length of the 6” x 1” flange, totaling 36”. Figure 4-11 shows a drawing of the beam element modeled. The beam setback was 2 inches, as in the experiment.

The t-stub was modeled exactly after the experimental drawings. Figure 4-12 shows the t-stub drawing. All bolt holes were modeled to be 1/16” larger than the nominal bolt diameter.
The shear bolts were placed at distances linearly varying within the holes, as recommended by Oltman (2007). The outer bolts, in the direction of load, were placed against the exterior side of the hole, while the inner bolts varied linearly in placement. Figure 4-13 displays how the shear bolts were oriented. Table 4-1 shows the void calculations for modeling the bolts, following the methods discussed in section 3.3. Note that the void depth calculation for the tension bolts is negative. This is a remnant from bolt manufacturing tolerances. Table 7-15 in the 13th edition of the AISC Steel Construction Manual states that the thread length stated is in accordance with ASTM A325 and A490 tolerances. ASTM further references an ASME publication (ASME 2006). This publication verifies that thread length is variable within a tolerance. It was therefore assumed that the void depth of the tension bolts in the TA-25 and TA-26 models was zero for simplicity. The actual thread lengths were unknown and could have varied between bolts.

![Figure 4-11: Drawing of TA-25 model beam element (modified from Swanson 1999)](image)
Table 4-1: Bolt shank void calculations for TA-25 & TA-26 models

<table>
<thead>
<tr>
<th>Bolt</th>
<th>1. Grip</th>
<th>2. Bolt length from experiment</th>
<th>3. Bolt diameter</th>
<th>4. Thread Length from AISC</th>
<th>5. Amount of bolt unthreaded (Col. 2 - Col. 4)</th>
<th>6. Threaded region in Grip, void depth (Col. 1 - Col. 5)</th>
<th>7. Void diameter (using Eq. 3-5)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>inches</td>
<td>inches</td>
<td>inches</td>
<td>inches</td>
<td>inches</td>
<td>inches</td>
<td>inches</td>
</tr>
<tr>
<td>Tension</td>
<td>2.875</td>
<td>4.5</td>
<td>1</td>
<td>1.75</td>
<td>2.75</td>
<td>0.125</td>
<td>0.4783</td>
</tr>
<tr>
<td>Shear</td>
<td>1.585</td>
<td>3.5</td>
<td>1</td>
<td>1.72</td>
<td>1.78</td>
<td>-0.195</td>
<td>0.4783</td>
</tr>
</tbody>
</table>

Figure 4-12: Drawing of TA-25 model T-stub (Swanson 1999)
4.3.2 Material Properties

This section will address the material properties unique to the TA-25 model. General material property information can be found in Chapter 3. Table 4-2 illustrates the material property derivations for the material of the t-stub. Mill report details have been left out of this table. They may be found in Swanson’s dissertation (1999).

The web material of the t-stub was chosen as the governing material because in the TA-25 experiment (and TA-26), the failure mechanism occurred in the t-stub stem, or the web of the W16x100 section from which it was cut. The bolt material was A490 and was defined as was discussed in Chapter 3 for bolts.
The A572-50 steel used for the beam and column was modeled as elastic-perfectly-plastic, with a slight modification to the yield stress. The whole point in doing the experimental tests on t-stub and top-and-seat angle components and connections was to begin the process to prequalify to develop a process to prequalify them for usage as seismic moment connections. It was suggested for the TA-25 and TA-26 models that the modified yield stress for high seismic areas be used. That would be

$$F_{y,\text{mod}} = 1.1R_yF_y$$  \hspace{1cm} (4-1)
Where $F_y$ is the nominal yield strength and $R_y$ is a factor depending on the grade of steel used. Fifty grade steel has a $R_y = 1.1$. Plugging in $R_y$ and $F_y = 50$ ksi into equation 4-1 gives 60.5 ksi for the modified yield strength. This was the yield value used in the elastic-perfectly-plastic model for both the beam and column. An elastic-perfectly-plastic material model was used since no inelastic deformation was expected or observed in either the beam or column.

### 4.3.3 Elements and Meshing

General methodology about partitioning, seeding, elements, and meshing can be found in Chapter 3. This section will detail the specifics of these aspects for the TA-25 model parts.

The t-stub part was thoroughly partitioned not only because of its odd geometric characteristics, but also to allow for careful mesh control. C3D20R elements were used exclusively for this part because it was the major area of interest in this model, as will be the case with the angles in the FS-01 and FS-02 models. An approximate global mesh size of 0.3 inches was used which produced 16,036 elements and 78,906 nodes for the t-stub. Figure 4-14 displays the partitioning scheme and Figure 4-15 shows the mesh scheme for the TA-25 model t-stub part.
The beam element for the TA-25 model had as unique geometries as the t-stub. C3D8R elements were used for the beam element with an approximate seed size of 0.75 inches. The beam and column elements were not observed to experience much deformation and were designed to do so, as mentioned previously, so less accurate elements and a less fine mesh scheme were deemed appropriate. This part had 3,356 elements and 5,355 nodes. Figure 4-16 displays the partitioning scheme and Figure 4-17 displays the mesh scheme for the TA-25 model beam element part.
The column flange element did not need to be partitioned at all, as it is just a plate, but given the proximity to the t-stub part, it needed to be meshed thoroughly for continuity between the t-stub and column. C3D8R elements were used with an approximate global seed size of one inch, since as with the beam element, little to no deformation was observed or expected for the column stub. This produced 1,348 elements and 2,217 nodes. Figure 4-18 displays the partitioning scheme and Figure 4-19 displays the mesh scheme for the TA-25 model column flange element part.
The tension and shear bolt parts used in TA-25 were partitioned with greater care than bolts in previous models. This was done in attempt to control the mesh a bit better, but ultimately the same method of partitioning used in previous models could have been applied here with the same results. Both bolt parts were made using C3D8R elements and an approximate global seed size of 0.25 inches. This produced 722 elements and 1,008 nodes for each shear bolt and 872 elements and 1,207 nodes for each tension bolt. Figures 4-20 and 4-21 display the partitioning scheme and mesh scheme for the shear bolt part and tension bolt part, respectively.
4.3.4 Constraints, Restraints, and Loads

General information about constraints, restraints, and loading can be found in Chapter 3. This section deals with the constraints and restraints specific to the TA-25 model. There were only two constraints used in this model; one for the support and the other for the load. The support constraint was the surface of the column flange element opposite the surface bearing against the t-stub being constrained to a reference point at the center of the surface. This reference point is RP-1 in Figure 4-22. This reference point was then a perfectly fixed restraint that functioned as the support provided by the column stub in the experiment.

Figure 4-21: TA-25 model partitioning scheme (left) and mesh scheme (right) for the tension bolt

Figure 4-22: View of support constraint for the TA-25 model
The other constraint was the load on the opposite side of the model. The end surface of the beam element was constrained to a reference point in the center of the surface. This is RP-2 in Figure 4-23. The reference point was then applied a translation of one inch in the direction that would cause axial displacement of the beam and a translational restraint in the vertical direction to keep the beam from displacing upward during loading. This functioned as the load induced to the beam element by the actuator. The one inch displacement was decided on from inspection of the experimental data.

There were two other restraints not made by constraints. These were the restraints on the bolt heads during pretensioning. These are discussed in detail in Chapter 3. An initial temperature for the pretensioning field was obtained from equation 3-11. This produced a prestress in the bolts that was below the minimum prescribed by AISC. A trial and error method was then used to obtain a temperature that would cause a prestress greater than the minimum. This temperature turned out to be -585 degrees Fahrenheit.
4.4 TA-25 Analysis

This section details the analysis results as compared to the experimental data. The TA-25 experiment was designed to fail via net section fracture and did so when tested. Figure 5-24 shows the deflected shape of the model at the end of the analysis with contours showing Von Mises stresses.

Figure 4-24: TA-25 model at one inch deformation (contours show Von Mises stresses)

Swanson’s t-stub component tests could be grouped by geometry. The group containing the TA-25 test contained two other t-stub component tests (one of which was TA-26 and is discussed in subsequent sections). The only difference between the three was the spacing of their shear bolts. The spacings were 3d, 2.67d, and 2.5d corresponding to TA-09, TA-25, and TA-26 tests, respectively. TA-09 and TA-25 failed in net section fracture and TA-26 failed due
to block shear. TA-25 was selected between the two net section fracture failures because it had bolt spacing closer to that of the block shear failure. It was desired to see if the modeling procedure could produce two separate models with such close geometries that failed in two very distinct ways. The theoretical calculations show that the TA-25 and TA-26 net section fracture and block shear strengths are so close that they are within the realm of error of the equations used. The point of this is that very close post-processing of the FE analysis results must be done to not overlook the nuances between the two modes of failure.

4.4.1 Analysis Events

The experimental design and analysis model both assumed that little to no deformation would occur in the beam and column elements. Figure 4-25 shows the beam element at the full one inch displacement with contours showing PEEQ ranging from zero to 2%. The areas in grey are showing plastic strain greater than 2%, but that is an inaccuracy resulting from a large change in strain happening over a single element. The element tries to extrapolate the strain producing much higher strain values than are actually present, as can be verified by the immediate area around the very high points of strain. The non-zero plastic strain around the holes can be attributed to some bearing deformation occurring. This was observed in the experimental test. It was so unimportant however that the experimenters would reuse the same beam and would zero-out the deformation, so subsequent tests would be accurate with respect to their own bearing (Swanson 1999).
The column element shows even less plastic strain. This was expected, as no observations were made during the experimental test that the column stub showed even the slightest deformation. Figure 4-26 shows the column flange element with contours showing PEEQ ranging from zero to 0.005% at the full one inch displacement.
It is important to verify that bolt failure due to shear or tension did not occur as it has been proven that these modes of failure can occur in models produced using this modeling procedure. Figure 4-27 shows the eight shear bolts with contours showing PEEQ between zero and 1% at the full one inch displacement. The bolt with the most plastic strain can be seen in that view and the most plastic strain it is experiencing is roughly 0.833%. It was discussed in previous sections that bolt failure is assumed to be around 6% strain, corresponding to roughly 5.5% plastic strain, but given the tolerance of the assumption, it could be assumed that bolt failure occurs at 6% plastic strain. 0.833% plastic strain is obviously less than 6% plastic strain, so by this logic, bolt failure due to shear does not occur.

The tension bolts show even less plastic strain than the shear bolts. Figure 4-28 shows the tension bolts with contours showing PEEQ ranging between zero and 0.01% at the full one
inch displacement. These bolts were obviously nowhere near fracture from tension long past when the t-stub failed to net section fracture.

Careful observation had to be made about the t-stub with respect to the propagation of plastic strain, given how close net section fracture and block shear failure were to one another. Figure 4-29 shows the deformed t-stub with contours showing PEEQ ranging between zero and 0.066959 plastic strain (the ultimate plastic strain). Any sections showing grey have strains higher than the ultimate plastic strain. This causes issues in that past the ultimate plastic strain represents fracture, but to ABAQUS it is just a higher strain and redistribution still occurs. The figures chosen attempted to highlight the propagation of the plastic strain and to illustrate that net section fracture, not block shear failure, occurred in this t-stub.
The first figure illustrates the closeness between the two prevalent failure modes. The diagonals from the bolts closest to the flange to the cut t-stub edge are clearly yielding, but the right three bolts in both rows are also clearly showing yielding occurring between them. The primary argument that net section fracture governs here is that the area between the left-most and second to left-most bolts (area in the red oval) in each row shows little to no yielding occurring. This will not be the case for the TA-26 model which will be discussed in subsequent sections and was observed to fail in block shear. The second figure highlights the beginnings of failure and shows much higher values of plastic strain along the diagonal sections than between the two bolts per row closest to the t-stub flange. The last figure is to illustrate what was meant previously about careful observation needing to be made of these models to verify them. The last figure shows strains above the ultimate plastic strain along the diagonal sections and between all bolts, showing no indication whatsoever as to which failure mode governed.
4.4.2 Analysis Results

This section details the numerical comparison between the experimental data and the finite element analysis results for the TA-25 model. Figure 4-30 shows the load-displacement plot that compares the two.

The previous section demonstrates how little inelastic deformation any part of the model except for the t-stub showed. It follows that the behavior shown in the plot is therefore dominated by the t-stub as was true for the experimental test. Comparing the initial stiffness shows agreement between experiment and analysis. The slip plateau falls at a value of load that seems to average two of the lower cycles’ slip plateau load values. The length of the slip plateau...
also seems to average the two earlier cycles’ slip plateau lengths. The post-slip elastic slope climbs slightly faster than the experimental cycles’ slopes. The inelastic behavior of the analysis shows to almost envelope the later experimental cycles. Once again these comparisons, as before, are being made between a monotonic analysis and a cyclic experimental test. Given the variability present in the fundamental differences between these two; the analysis results predict the cyclic experimental behavior very well. The maximum load carried by the experimental test was 414.0 kips at an applied displacement of 0.638 inches. The maximum load carried by the FE model was 401.5 kips at an applied displacement of 0.571 inches. The maximum deformation occurred in the experimental test at 0.649 inches and the apparent failure of the FE model analysis occurred at 0.636 inches.

4.5 TA-26 Model Setup

This section covers the specific setup of model TA-26, a t-stub component tested by Swanson (1999) that failed from block shear, and that had a near identical setup to the TA-25 model. The TA-26 model consisted of all the same parts as the TA-25 model except that the shear bolt spacing per row was changed from $2.67d$ to $2.5d$, where $d$ is the nominal bolt diameter. All introductory commentary provided for the TA-25 model applies to the TA-26 model. Figure 4-31 shows the TA-26 model assembly in ABAQUS.
All modeling geometry liberties taken in the TA-25 model apply to the TA-26 model. The t-stub and beam element dimensions are obviously different because different bolt spacings were used. The void calculations and bolt placement within the holes are the same as described TA-25 model. Figure 4-32 shows a drawing of the modified beam element and Figure 4-33 shows a drawing of the TA-26 t-stub.

Figure 4-32: Drawing of TA-26 model beam element (modified from Swanson 1999)

Figure 4-33: Drawing of TA-26 model T-stub (Swanson 1999)
The material properties described for the TA-25 model are the exact same as those used for the TA-26 model. The only mesh schemes that changed for the TA-26 model were the beam element and t-stub. The same partitioning scheme, approximate global seed size and element type as the TA-25 model were used in TA-26. This resulted in 3,138 elements with 5,019 nodes for the beam element and 11,888 elements with 61,292 nodes for the t-stub for TA-26. Figures 4-34 through 4-37 show the partitioning and mesh schemes of the beam element and t-stub, respectively.
All constraints and restraints for the TA-25 model apply to the TA-26 model. The same applied displacement of one inch was used. As mentioned previously, all aspects of the model that could remain the same were left the same as the TA-25 model.
4.6 TA-26 Analysis

This section will cover the analysis events and results of the TA-26 model. Recall again that this model was observed experimentally to have failed in block shear.

4.6.1 Analysis Events

The only difference between the TA-25 model and the TA-26 model is the shear bolt spacing per line. It follows that many of the behaviors of the TA-25 model will be shared by the TA-26 model. Thus, little to no inelastic deformation is expected in the column flange element, beam element, or tension or shear bolts. Figure 4-38 shows the deformed beam element with contours showing PEEQ ranging between zero and 0.5%. It appears some bearing deformation has taken place, but this was witnessed during the experiment, as mentioned in the TA-25 analysis section. Plastic strains in the 5% range resulted from the coarse mesh used around the holes and ABAQUS attempting to extrapolate strains, again as discussed in the TA-25 analysis section.

Figure 4-38: Deformed beam element of TA-26 model (contours show PEEQ)
The column and tension bolts show no inelastic deformation. This corroborates the experimental observations as well as the results from the TA-25 model. These will not be displayed in figures as they were for the TA-25 model because there is no difference. The shear bolts did experience some inelastic deformation, but at very low levels of plastic strain. Figure 4-39 shows the deformed bolts at the full displacement with contours showing PEEQ ranging from zero to 1.2%, a value decided by ABAQUS. The view shown highlights the area having the largest plastic strains, and as can be seen, the value of plastic strain is around 1%. Previous sections have discussed why 6% plastic strain is assumed the ultimate strain for bolts. Since the shear bolts barely approached 1% inelastic strain, it can be said that the bolts did not fail in shear.

![Figure 4-39: Deformed shear bolts from TA-26 model (contours show PEEQ)](image)

The plastic strain development of the t-stub must be carefully handled, as discussed in the TA-25 analysis section. The governing failure mode between net section fracture and block shear failure are so close in this test setup that they appear to almost be occurring simultaneously, with one mode reaching its ultimate strength slightly before the other. Figure 4-
40 shows the plastic strain propagation, beginning with the top left figure, moving right, and then down. The primary difference between the TA-25 model and the TA-26 model can be seen most clearly in the top left figure.

Figure 4-40: Deformed t-stub at four different step times for the TA-26 model (contours show PEEQ)
The contours show PEEQ ranging between zero and the ultimate plastic strain of 0.066959. The area between the first bolt and second bolt of each row, moving away from the flange, shows a very visible line of plastic strain that was not present in Figure 4-29 at around the same step time for the TA-25 model. The rate of development of plastic strain in this area is the key difference between the TA-25 model and the TA-26 model. This demonstrates why TA-26 failed from block shear and TA-25 failed from net section fracture.

The top right figure shows the dual mechanism formation more clearly. Yielding is occurring along the diagonal section lines as well as between all bolts in a row and between the first two bolts in each row. The bottom two figures illustrate further how block shear governs over net section fracture by showing plastic strains at or above ultimate occurring in the block region while the diagonal section lines still only indicate yielding.

A comparison can be made between the bottom right figure of Figure 4-40 and the right-most figure of Figure 4-29. Both figures occur at step times at roughly the same time and when apparent failure occurs experimentally. Figure 4-29 shows much higher plastic strains along the diagonal section lines whereas Figure 4-40 shows much higher plastic strains between the bolts in the block shear region. It is by these comparisons and observations that the TA-26 model is said to have failed in block shear analytically and that the TA-25 model is said to have failed from net section fracture analytically.

### 4.6.2 Analysis Results

This section details the comparison between the FE analysis results with the experimental data for the TA-26 model. Figure 4-41 shows the load-displacement plot that illustrates this comparison.
The initial stiffness of the analysis matches very well with the experimental data. The slip plateau occurs at a level slightly below the second cycle group with a slip plateau length that seems to average the first and second cycle group of the experimental data. The post-slip elastic stiffness matches fairly well with the later cycle groups of the experimental data. The post-yield behavior of the analysis seems to envelope the post-yield behavior of the cycle groups.

Comparisons made between monotonic and cyclic responses can only be taken so far. The analysis results for the TA-26 model match very well overall with the experimental data and are within the expectations of comparing a monotonic analysis to a cyclic test. The maximum experimental load was 399.2 kips occurring at 0.5837 inches of applied displacement and the maximum experimental deformation occurred at 0.6190 inches at a load of 377.0 kips. The maximum analytical load was 387.0 kips occurring at 0.4371 inches of applied displacement and the apparent failure occurred at 0.6164 inches at a load of 380.0 kips.

Figure 4-41: TA-26 load vs. displacement comparison
Chapter 5 – Full-Scale Models and Results

This chapter will discuss the specific setup of the two full-scale tests modeled using finite elements and the results of their analyses. All models will follow the methodology elucidated in Chapter 3.

5.1 FS-01 Model Setup

FS-01 was one of the two full-scale top-and-seat angle tests performed by Schrauben (1999). It was composed of the following:

- 10’-9” W14x145 column
- 15’ W18x40 beam
- Two 8” long L6x8 angles cut from L8x8 sections with 2-1/2” upstanding leg gage distance (top and seat)
- PL9x3-1/8x5/16 shear tab
- Two 7/8” diameter, 3-1/4” long tension bolts for each angle
- Four 7/8” diameter, 3” long shear bolts for each angle
- Three 7/8” diameter, 2” long bolts for the shear tab

All steel material was designated as A572-50 and all bolts were A490 material. Figure 5-1 shows the ABAQUS assembly.

5.1.1 Model Geometry Liberties

A number of modeling liberties were taken with the dimensions of the physical components, none of which will affect the analysis, but are documented nonetheless. The 10’-9” column dimension in the experimental setup included two 1-1/2” thick end plates welded to the W14x145 to mount the column to the apparatus that would allow the column to have a pinned-pinned end condition. The end plates are not included in the model because they are not needed, but the entire 10’-9” dimension is used for the column.

The beam span was documented as being 15’-1/2” and being the distance from the column centerline to beam tip. It is assumed beam tip refers to the cross section of beam at the centerline of the actuator used to apply the load. Any beam length past this point of load is not needed for modeling, so only 172.6” of the beam was modeled. Figure 5-2 illustrates how that dimension was derived. P represents the point load of the actuator.
Chapter 3 explains how the tension, shear, and shear tab bolts were modeled. Table 3-1 shows the void depth calculations for the FS-01 bolts.

Bolt holes are nominally 1/16” larger than the nominal bolt diameter. When designing, it is common to take bolt holes as an additional 1/16” to take into account hole manufacturing, summing to a total of 1/8” larger than the nominal bolt diameter. Bolt holes were modeled as only 1/16” larger than the nominal bolt diameter given there is no need for the additional 1/16” from manufacturing.

Bolts are typically installed without much care as to where the bolt is within the hole, and it is assumed this occurred in the experimental setup. The bolts were modeled at the center of the holes because research has shown that for two rows of bolts transverse to the force, the location of the bolts within the holes does not create a noticeable difference between experimental data and analysis results with respect to slip and bearing behavior (Oltman 2007). The tension bolts were obviously unaffected by such placement.
5.1.2 Specific Material Properties

This section will address the material properties specific to model FS-01. General method information about how material properties were decided, derived, and assigned can be found in Chapter 3. Tables 3-3, 5-1, and 5-3 illustrate the material property derivations for the FS-01 beam, angle, and column, respectively.

Table 5-1: A572-50 material breakdown for the L6x8x1 angle

<table>
<thead>
<tr>
<th>Mechanical Properties:</th>
<th>Nucor-Yamato</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manufacturer:</td>
<td>L8 x 8 x 1</td>
</tr>
<tr>
<td>Grade:</td>
<td>A572-50</td>
</tr>
<tr>
<td>(A572GR50-94C / A709-96 GR50)</td>
<td></td>
</tr>
</tbody>
</table>

| Ave Mill Yield:        | 55.0 ksi     |
| Ave Mill Ultimate:     | 76.5 ksi     |
| Ave Mill Elong.:       | 21.0 %       |

Georgia Tech Tension Tests

<table>
<thead>
<tr>
<th>Cpn</th>
<th>$E$ (ksi)</th>
<th>$F_{yd}$ (ksi)</th>
<th>$\varepsilon_s$ ($\mu$)</th>
<th>$E_s$ (ksi)</th>
<th>$F_{ud}$ (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>F-06A</td>
<td>31476</td>
<td>52.5</td>
<td>7087</td>
<td>624</td>
<td>76.8</td>
</tr>
<tr>
<td>F-06B</td>
<td>29819</td>
<td>52</td>
<td>6950</td>
<td>576</td>
<td>76.6</td>
</tr>
<tr>
<td>Average</td>
<td>30647.5</td>
<td>52.25</td>
<td>7018.5</td>
<td>600</td>
<td>76.7</td>
</tr>
</tbody>
</table>

Engineering: True (Log):

<table>
<thead>
<tr>
<th>origin:</th>
<th>strain</th>
<th>stress</th>
<th>True (Log): strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>yield:</td>
<td>0.001705</td>
<td>52.25</td>
<td>0.001703</td>
<td>52.339</td>
</tr>
<tr>
<td>begin hardening:</td>
<td>0.007019</td>
<td>52.25</td>
<td>0.006994</td>
<td>52.61544</td>
</tr>
<tr>
<td>ultimate:</td>
<td>0.127833</td>
<td>76.7</td>
<td>0.120298</td>
<td>85.92689</td>
</tr>
</tbody>
</table>

Plastic strain from true:

<table>
<thead>
<tr>
<th>yield:</th>
<th>strain</th>
<th>stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>begin hardening:</td>
<td>0.005291</td>
<td>52.61544</td>
</tr>
<tr>
<td>ultimate:</td>
<td>0.118595</td>
<td>85.92689</td>
</tr>
</tbody>
</table>
The flange material properties were used for both the beam and column as it was decided they would influence the target behavior of the model more than the corresponding web materials. The column experienced no inelastic behavior and even very little elastic behavior. For these reasons, the element of the column being connected (its flange) was chosen as the
material property for the entire section. The flange material property was chosen for the beam because the flanges govern the bending behavior of a beam and because the beam flange is the element connected to the moment resisting elements (angles). The shear tab used the angle material properties. Bolt material properties are discussed in Chapter 3.

5.1.3 Elements and Meshing

General methodology about partitioning, seeding, elements, and meshing can be found in Chapter 3. This section will detail the specifics of these aspects for the FS-01 parts.

The angles were only partitioned around the rolled edge as shown in previous figures. Given the importance of this part, C3D20R elements were used exclusively and were meshed with an approximate global mesh size of 0.3 inches. This resulted in 3,936 elements and 20,782 nodes in each angle. Figure 5-3 shows the angle meshed.

![Figure 5-3: L6x8x1 angle for model FS-01 meshed](image-url)
The beam had to be more thoroughly partitioned to adequately control the mesh. The cross-section of the beam was partitioned around the k-zone and the flanges were split in half to guarantee at least two elements in the flange thickness. Along the length of the beam, a partition was made at the end of the angle on the connection side, three beam depths from the edge of the beam on the connection side, and one beam depth from the free end of the beam. These were made such that the beam mesh under the angle could match with the mesh of the angle better; the elements within three beam depths (where a potential plastic hinge could occur) were 20 node elements; and the free end of the beam was decided to be made rigid. In addition to all of those reasons, these partitions allowed for the tapering of the mesh from fine to coarse moving from the connection end to the free end. Figures 5-4 and 5-5 illustrate the partitioning just described.

Figure 5-4: (Left) Partitioning of the cross-section of the FS-01 model beam

Figure 5-5: (Above) Partitioning of the FS-01 model beam along its length
The beam segments in Figure 5-5 will be assigned numbers for the purposes of explaining the mesh scheme used for this beam. Starting from left to right, they shall be segments 1, 2, 3, and 4. Segment 1 utilized C3D20R elements given the proximity to both the angles and the shear tab. An approximate global seed size of 0.3 inches was used for Segment 1. Segment 2 utilized C3D20R elements because if a plastic hinge would form, not that it was expected to, it would most likely form in this region. Segment 2 had an approximate global seed size of 1 inch. Segments 3 and 4 both used C3D8R elements as their influence on the behavior of interest was minor and utilized an approximate global seed size of 3 inches. Altogether the beam had 7,829 elements and 36,144 nodes. Figure 5-6 and 5-7 show the mesh schemes of segments 1 & 2 and 3 & 4, respectively.

A single partition was used on the column to separate the flange being bolted from the remainder of the column. As mentioned before, since the column saw very little force, minimal effort was used to mesh it. It should be noted that in the experimental test, so little force was seen in the column in a single full-scale test that only one member was used for both tests, it was simply reversed so one flange was used per test (Schrauben 1999). An approximate global seed
size of 4 inches was used for the column. Since so few elements were being used with such a large seed size, it was deemed appropriate to use C3D20R elements. There were 848 elements total and 5,601 nodes. Figure 5-8 shows the column mesh scheme. The oddly shaped elements in the center are occurring around the holes for the tension bolts.

No partitions were made on the shear tab given the simple geometry. The shear tab was designed to carry only the shear of the connection, but given that this connection is tested to failure, it is highly possible that at later points in the analysis when the angles begin to see inelastic strain that redistribution may occur. This would cause the shear tab to begin to see some bending. For these reasons, C3D20R elements were used for the shear tab. An approximate global seed size of 0.35 inches was used to produce 285 elements and 2,239 nodes. Figure 5-9 shows the mesh scheme of the shear tab.

Three different bolt parts were made for this model depending on their grip. There were bolts from the beam flange to the angle (shear bolts), bolts from the column flange to the angle
(tension bolts), and bolts from the beam web to the shear tab (shear tab bolts). The head, shank, and nut/washer were all partitioned in each case and then the void depth was partitioned depending on whatever the void depth was calculated to be. For the bolts used in the FS-01 and FS-02 models, Table 3-1 shows these dimensions.

Given the relative size of bolts, the approximate global seed sizes were fairly small. The tension and shear bolts were 0.18 inches and the shear tab bolts were 0.22 inches. Bolts were modeled using C3D8R elements to reduce degrees of freedom. The element and node count for each bolt type is as follows: each tension bolt had 1,208 elements and 1,643 nodes, each shear bolt had 1,231 elements and 1,656 nodes, and each shear tab bolt had 522 elements and 766 nodes.
nodes. As can be seen, the number of nodes from the bolts alone is substantial with respect to the entire model. This was why the use of C3D8R elements was considered justified simply for computational time. Figures 5-10 through 5-12 show the mesh schemes for the tension, shear, and shear tab bolts respectively.

5.1.4 Specific Contact Surfaces and Properties

The methodology outline for contact surfaces and their properties from Chapter 3 was followed here apart from a few minor changes. Using the automatic contact pair locating function in ABAQUS gave 99 interactions for the entire FS-01 model. 95 of those were assigned the tangential and normal behaviors discussed in Chapter 3 (coefficient of friction of 0.20 and “hard” contact). 4 of the 99 were assigned a much higher coefficient of friction and the same normal behavior.

The reason for this involves an anecdote from the working stages of this research. The FS-01 model had been analyzed and completed, and the FS-02 model was near completion. A newer version of ABAQUS was going to be used to analyze FS-02 than what had been used to create the model for FS-02. FS-02 had been modeled in the same version FS-01 had been modeled and analyzed. Almost 95% of the FS-02 model is the same as FS-01, and it was assumed that the newer version of ABAQUS would have no issue in running an older version’s model. FS-02 diverged consistently for reasons unknown. After numerous investigations, it was determined that the angles connected to the column were slipping excessively during analysis using the 0.20 coefficient of friction. The coefficient was increased to 0.80 and the model completed with outstanding results when compared to the experimental data. It was decided that the FS-01 should be analyzed on the new version of ABAQUS using the increased coefficient in
the same location to determine what impact, if any, this change would have on the analysis results. FS-01 showed almost no change with the 0.80 coefficient suggesting there was some disconnect between modeling in one version in ABAQUS and analyzing in another. Future research should avoid switching between versions of modeling and analyzing to avoid these issues.

5.1.5 Specific Constraints, Restraints, and Loads

General information about constraints, restraints, and loading can be found in Chapter 3. This section deals with the constraints and restraints specific to the FS-01 model. The FS-01 model utilized four total constraints: one for the load, two for the ends of the column, and one for the shear tab weld. The constraints for the load and column ends were kinematic coupling constraint whereas the shear tab constraint was a tie.

The FS-01 experimental test was designed with a column that had a pinned-pinned end condition. These restraints were applied in ABAQUS to the constraints at the ends of the column. Translation was restrained in all three. Given the lack of movement in the column, no out-of-plane moment was expected and therefore all rotation was left unrestrained. This made for perfectly pinned column restraints. These can be seen in Figure 5-13 as RP-1 and RP-2.

The loading of the FS-01 model was done through the constraint at the end of the beam. The load history applied in the experiment came from the SAC protocol. Table 5-3 details the loading used for the experiment derived from the SAC load history. Again note that the experiment used cyclic loading whereas the model was monotonic loading.
Schrauben notes in his thesis that final loading was reached in the 8th step when tension bolt fracture occurred at 7.16 inches of beam tip displacement. The load was then reversed in an attempt to reach this in the opposite direction when the same failure occurred, but at a beam tip displacement of 6.33 inches. (Schrauben 1999) Due to these experimental occurrences, a downward displacement was applied of 7.22 inches, corresponding to the 8th load step. During initial analysis runs, out-of-plane motion of the beam occurred, both torsion and lateral movement. These were restrained in the experiment with use of lateral bracing, but this occurring during analysis provides another form of verification. The final analysis runs had the point of load restrained from lateral translation and rotation along the axis of the beam. The point of application of the displacement load is RP-3 on Figure 5-13.

There were three other restraint cases that did not use constraints for their points of assignment. These restraints were used on the bolts for the purposes of pretension. More detail about the function of these restraints can be found in the Bolt Pretensioning section of Chapter 3. A different restraint set was needed for each type of bolt because the bolt shanks were all oriented in different directions. The temperature used for the temperature gradient applied to the

<table>
<thead>
<tr>
<th>Load Step #</th>
<th>Peak Deformation, θ (rad)</th>
<th>Number of Cycles</th>
<th>Beam Tip Displacement (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00375</td>
<td>6</td>
<td>0.677</td>
</tr>
<tr>
<td>2</td>
<td>0.005</td>
<td>6</td>
<td>0.903</td>
</tr>
<tr>
<td>3</td>
<td>0.0075</td>
<td>6</td>
<td>1.354</td>
</tr>
<tr>
<td>4</td>
<td>0.01</td>
<td>4</td>
<td>1.805</td>
</tr>
<tr>
<td>5</td>
<td>0.015</td>
<td>2</td>
<td>2.708</td>
</tr>
<tr>
<td>3a</td>
<td>0.0075</td>
<td>1</td>
<td>1.354</td>
</tr>
<tr>
<td>6</td>
<td>0.02</td>
<td>2</td>
<td>3.610</td>
</tr>
<tr>
<td>7</td>
<td>0.03</td>
<td>2</td>
<td>5.415</td>
</tr>
<tr>
<td>8</td>
<td>0.04</td>
<td>2</td>
<td>7.220</td>
</tr>
<tr>
<td>9</td>
<td>0.05</td>
<td>2</td>
<td>9.025</td>
</tr>
</tbody>
</table>

(a) Beam span (beam tip to column centerline) was 15'-1/2".
(b) Repeated after every major load step.
(c) Continue with increments of 0.01 radians and perform two cycles at each major step.
bolts was -686 degrees Fahrenheit. The Bolt Pretensioning section of Chapter 3 discusses how this was calculated.

5.1.6 Specific Steps

Two steps aside from the initial step were used in the FS-01; one for pretension and one for loading. Interactions, the two column restraints, and the three bolt restraint sets are applied in the initial step. The temperature gradient is applied to the bolt shanks in the first analysis step and the bolt restraint sets are modified to allow for the bolts to deform. The load is created and applied in the second analysis step (third step total) and the bolt restraint sets are deactivated to allow for complete global movement of the bolts. Step property specifics were the same as those outlined in detail in Chapter 3.
5.2 FS-01 Analysis

This section details the analysis results as compared to the experimental data. Figure 5-14 shows the deflected shape of the model with the contours showing Von Mises stresses.

![Figure 5-14: FS-01 model deflected with Von Mises stress contours](image)

5.2.1 Analysis Events

A number of observations were made during the experimental tests and were documented (Schrauben, 1999). Some of these are more easily compared to the analysis results than others. The first event that will be compared is prying of the top seat angle. Schrauben documented this visually occurring during the 7th load step. From table 5-3 taking the beam tip displacement for the 7th step divided by the applied 7.22 inches, the analysis step time can be obtained. This value calculates to step time 0.75. Figure 5-15 shows the angle with contours showing plastic equivalent strain, ranging from zero (blue) to half of the ultimate strain (red), at one step before...
the closest step time to 0.75, at the closest step time to 0.75, and one step after the closest step time to 0.75.

As can be seen, the angle is definitely experiencing inelastic deformation. The beginnings of plastic hinges can be seen in the stem just outside the k-zone and in the upstanding leg at about the gage distance. This would obviously induce prying forces into the tension bolts.

Another observation was yielding of the beam web in the k-zone under the top seat angle and the beam flange. Figure 5-16 shows a figure from Schrauben’s thesis documenting the web crippling experienced under the angles during the experimental test. His thesis also mentions flange yielding around the angles. Figure 5-17 shows the beam with contours showing equivalent plastic strain ranging from 0 to half of the ultimate strain. The figure demonstrates that these behaviors were all sufficiently reproduced by the model. Also to note, the very small grey contour showing PEEQ above one half of the ultimate strain of the beam is still below the ultimate strain.
Figure 5-16: Clip angle “pinching” and k-zone crippling (Schrauben 1999)

Figure 5-17: FS-01 deflected beam at connection end (contours showing PEEQ)
The final observation was the failure mechanism. As mentioned in Chapter 3, ABAQUS cannot model failure easily, so it is the modeler’s responsibility to monitor plastic strains to know when a material exceeds its ultimate strain. This theoretically can represent failure and from a modeling perspective, is the only way to represent failure. The only part of model FS-01 that came remotely close to having strains at or exceed the ultimate strain of their respective materials were the top seat angle tension bolts. Figure 5-18 shows the top seat tension bolts with contours showing PEEQ ranging from zero to the ultimate plastic strain. The ultimate plastic strain of these bolts was assumed 6%. This is obviously a rough value, so it is well within reason that these bolts could have failed within the material specified tolerance for their ultimate strain.

5.2.2 Analysis Results

This section covers the quantitative results obtained through analysis of the FS-01 model. The experimental data was arranged in a number of different plots, such as load vs. displacement, moment vs. total rotation, moment vs. plastic rotation, etc. Seeing as all of these plots were derived from load vs. displacement data, the load vs. displacement comparison would be the most appropriate to make. Figure 5-19 shows the FS-01 load vs. beam tip displacement plot.
Given that the experimental test was loaded cyclically and the analysis was loaded monotonically, a number of realizations must be explained before comparisons are made. For clarity sake, the plot only shows the first quadrant of the experimental data as that is the only quadrant where the analysis monotonic loading is present. This quadrant was chosen over the third quadrant because it was the first part of the cycle loaded. The FE analysis results will be compared to the general behavior of the cycles or those cycles that are most relevant to the behavior in question. The last thing to note is that the analysis does not model failure; the monotonic loading was simply stopped at 7.22 inches of tip displacement. As such, the load at that displacement can be compared to the ultimate load of the experimental test, but to say that the actual analysis failed at the same point as the experiment would not be accurate.
The plot shows the initial stiffness of the analysis results matching well the initial stiffness of the earlier experimental cycles. This is important as these are the cycles in which all parts of the system are still free of plastic deformation. Moving further along the curve, the slip plateaus are the same length and approximately the same location in terms of tip displacement. Again the earlier cycles of the experimental test are compared with the analysis because as the faying surface is slipped over repeatedly, the resistance to slip decreases. The post-slip behavior of the analysis matches the trend of the experimental data. Since earlier cycles began reverse loading and the later cycles already had noticeable plastic deformations, an approximate envelope of those cycles could be formed. It would be this envelope that the analysis results would follow in the post-slip behavioral region. The maximum load from analysis was 20.61 kips and the maximum load from the experiment was 20.26 kips. This is a 1.7% difference. These values occurred at 6.94 inches and 6.90 inches, respectively, of beam tip displacement.

The slip comparison brings verification to a number of modeling choices. The analysis slip is very close to the experimental slip, but it is slightly lower than the experimental slip. This again agrees with the idea what slip on these faying surfaces is highly variable and affected by many aspects of modeling. Coefficient of friction, bolt pretension, location of the bolts within the holes, material properties, and contact interactions are the major parameters affecting the slip behavior.

5.3 FS-02 Model Setup

The FS-02 model was the second of two full-scale top-and-seat angle experimental tests conducted by Schrauben (1999). The exact same components were used except that the gage distance of the upstanding leg of the angles was changed from 2-1/2” to 4”. Figure 5-20
illustrates this difference, showing the angles used in FS-01 on the left and the angles used in FS-02 on the right.

Figure 5-20: FS-01 (left) and FS-02 (right) angles

Figure 5-21: FS-02 model assembly in ABAQUS (colors indicate the different parts)
All geometric modeling liberties, specific material properties, specific contact surfaces and properties, constraints, restraints, and steps in the FS-01 model were exactly the same as in the FS-02 model. Figure 5-21 shows the assembly colored by part of the FS-02 model.

5.3.1 Elements and Meshing

General methodology about partitioning, seeding, elements, and meshing can be found in Chapter 3. This section will detail the specifics of these aspects for the FS-02 parts.

All parts were partitioned, seeded, and meshed exactly the same as their corresponding parts were in the FS-01 model except the angles and column. The angles were meshed using the same methodology and rationale, but the total number of elements and nodes created were different given the different gage distance. This resulted in 4,002 elements and 21,112 nodes in each element. Figure 5-22 shows the FS-02 angle meshed.

Figure 5-22: FS-02 angle meshed
The column was meshed using the same method and rationale outlined for the FS-01 model column. Since the gage distance on the angles changed, the holes on the column also changed. This resulted in there being 857 elements and 5,664 nodes in the column mesh for FS-02. Figure 5-23 illustrates this mesh.

5.3.2 Loading of FS-02 Model

FS-02 was designed to be the more ductile of the two full-scale tests, and experimentally performed as such. The maximum displacement in both directions of loading surpassed the 8th load step, or 7.22 inches of beam tip displacement. The failure of the test came not from simultaneous failure of both tension bolts, but rather one failing after the other, creating a very interesting load vs. displacement plot. Even after the two tension bolts failed, the connection continued to be loaded to attain a maximum tip displacement of 9.03 inches at a load of around 6 kips by entirely relying upon the shear tab. The loading of the shear tab to attain a “maximum” tip displacement does not fit the goal of the test, as after the two tension bolts failed, the angle
failed to provide any resistance. This is useful to know from a practical standpoint that the shear tab can support some load should both bolts connecting the angle to the column fail, but from a modeling standpoint, this is near impossible to replicate. Instead, the tip displacements at which the tension bolts failed were averaged. The first tension bolt failed at 6.8 inches and the second at 7.44 inches. This average is 7.12 inches tip displacement. It was decided that to keep uniformity between the two full-scale models that the tip displacement from FS-01 be used as the applied tip displacement for FS-02 (7.22 inches). This was slightly greater than the average of the bolt failures, but should facilitate for the verification of the modeling methodology.

5.4 FS-02 Analysis

This section details the analysis results of the FS-02 model. Figure 5-24 shows the deflected shape of the model with contours showing Von Mises stresses.

![Figure 5-24: FS-02 model deflected shape (contours displaying Von Mises stress)](image)
5.4.1 Analysis Events

As mentioned in the corresponding FS-01 section, these events are far more difficult to compare to experimental observations than it is to compare quantitative analysis results to experimental data. Since the FS-02 setup was designed to be more ductile, more cases of “yield” are observed experimentally. Quotations are used with yield here as what the experimenters are viewing and documenting as yield is actually whitewash spalling, which can easily occur anytime between initial yield and strain at eleven times yield strain. The observation is still useful for comparison with the analysis in two ways: the first is that regardless of the level of yielding, the area has yielded. The second is that the observers noted with more emphasis that yielding was occurring with these angles at this point as opposed to the previous test (FS-01). This suggests that greater plastic equivalent strain should be occurring when the contours are set the same bounds and the same step time is used.

The first incident of the experimental test was noticeable yielding near the k-line of the shear leg of the angle during the 7th step. The step time corresponding to this is the same as that noted for the FS-01 test, 0.75. Using the closest step time before 0.75, at 0.75, and after 0.75 with contours ranging from 0 PEEQ to half of the ultimate strain PEEQ produces Figure 5-25.

![Figure 5-25: FS-02 top seat angle prying (contours show PEEQ)](image)
The strain shown in Figure 5-25 is much higher than that found in Figure 5-15, as is evident by the orange, red, and grey areas. Note that the grey values are still below the ultimate strain, so the angle still does not fracture, but it does experience a much higher degree of inelastic strain than the FS-01 angle did. Appropriately the plastic hinge formation experiencing the greater plastic strain is in the vicinity of the k-line zone in the shear leg of the angle as observed experimentally as opposed to the hinge formation in the upstanding leg at the gage distance.

Figure 5-26: FS-02 angle at step time 1.0 (contours showing PEEQ)

The loading scheme mentioned previously affects this event briefly. Since the connection was loaded further experimentally than it was analytically, it was noted that around the 8th step, plastic hinges had fully formed in the shear leg of the angle. The 8th step
corresponds to step time 1.0, the end of the analysis. Figure 5-26 shows the angle at the end of the analysis with contours showing PEEQ from 0 to ultimate strain.

The hinge formation is difficult to see from a single viewing angle and a single contour scheme, but it has indeed formed. The line of strain under the angle reaches above half of the ultimate plastic strain. The yielding across the entire length of the angle also further suggest that a hinge has formed, whereas if the hinge attempting to form in the upstanding leg is observed, yielding has barely started to occur between the two bolt holes.

Another event occurring as it was in the FS-01 experiment was the k-line yielding of the web of the beam and yielding in parts of the beam flange. Both of these began occurring at the 5th load step, corresponding to step time 0.37, and continued to the 8th load step, step time 1.0.

Figure 5-27: FS-02 beam at approximate step time 0.37 (contours show PEEQ)
Figure 5-27 shows the beam at the closest step time to 0.37. The contours show PEEQ ranging from 0 to the automatically calculated value by ABAQUS, roughly 0.0283. This corresponds to roughly 15% of the ultimate plastic strain. This shows that yielding is indeed occurring at the k-line of the web, but at a very low value of strain. The model shows no yielding in the flange just outside of the connection at this point, but a very tiny bit of yielding around the holes of the beam. This corresponds with experimental observations. Figure 5-28 shows the beam at step time 1.0.

![Figure 5-28: FS-02 beam at step time 1.0 (contours show PEEQ)](image)

The PEEQ range on Figure 5-28 goes from 0 to half of the ultimate plastic strain. It is evident that yielding in the k-line of the web is occurring, but not very evident that yielding is occurring in the flange just outside of the connection, as was observed. What is actually
happening follows that if there is more inelastic deformation of the angles that there would be less in the beam, but this was not observed experimentally. It is a possibility that spalling of the whitewash just outside of the connection could have been mistaken as yielding and was actually slip in the angle.

The last event of note was the failure mechanism, as before. No other part showed PEEQ near the ultimate plastic strain aside from the tension bolts. Figure 5-29 shows the tension bolts with contours showing PEEQ ranging from 0 to 0.06 (6% strain).

Figure 5-29: FS-02 tension bolts (contours show PEEQ)

The values of plastic strain here are obviously less than those seen in FS-01. This could be attributed to the load used, but regardless the bolts are the most heavily loaded parts of the FS-02 model. It would be within reason to assume if the analysis was all that was looked that the bolts would fail first.
5.4.2 Analysis Results

This section covers the qualitative comparison between the experimental data and finite element analysis results of the FS-02 model. The initial commentary provided in the corresponding FS-01 model section applies to FS-02 as well. Figure 5-30 shows the load-displacement comparison between the experimental data and analysis results from the FS-02 model.

![FS-02 P/Δ](image)

Figure 5-30: FS-02 Load vs. Beam tip displacement comparing experimental data to FE analysis results

A number of irregularities occurred with this experimental test that will be discussed prior to the comparison. As mentioned in the loading section for FS-02, this experimental setup continued to be loaded past the failure of the angle. There are two visible discontinuities in
Figure 5-30 where the tension bolts from the angle to the column fail. The green and black arrows in Figure 5-30 indicate the failures of the first and second tension bolt failures, respectively. After the failure of the second bolt, the load drops abruptly and begins to be picked up again indicated in the black oval. At that point, all forces and moments in the connection in this direction of load are being resisted by the shear tab. Conventional shear tabs are not designed to resist moment, so this connection effectively failed after the second tension bolt failed. Figure 5-31 shows the same plot, but only the relevant area of the experimental data.

![FS-02 P/Δ](image)

Figure 5-31: FS-02 load vs. beam tip displacement comparison (relevant experimental data portion)

This comparison highlights the variability in experimental testing. The initial stiffness of the analysis matches relatively well with the experimental data. The analysis shows slip occurring roughly 1.2 kips higher than it occurred experimentally. A brief sensitivity analysis was run with regard to slip resistance. The slip from an analysis using a coefficient of friction of
0.175 was found to match the experimental slip much better, but it was decided to keep the 0.2 coefficient of friction for the final analysis. It would not only discredit the modeling procedure if the coefficient of friction needed to be changed for every model, but it was also believed that the 1.2 kip difference between slip plateaus was within reason.

The strength of the model was compared to the first bolt failure experimentally. The first bolt failed experimentally at 6.8 inches of displacement at a load of 16.8 kips. The corresponding analysis load at 6.8 inches of displacement was 16.1 kips. This is a 4.1% difference, which is obviously non-negligible, but still fairly accurate for the number of assumptions made in the modeling process.

It is interesting to note the slopes of the curves as they approach ultimate. The FS-01 model and experimental test both reach their maximum force prior to their maximum displacement, effectively ending with a zero or very small negative slope when the test failed or when the analysis ended. The FS-02 model and experimental test, being designed to be more ductile, both show a very obvious positive slope at the analysis end or at failure, respectively. It is possible that this modeling procedure is more appropriate for modeling less ductile connections, but further models will need to be created and verified to know definitely.
Chapter 6 – Conclusions

The main objective of this research was to develop and verify a modeling procedure that could be used to create finite element models for top-and-seat angle connections. The procedure is developed in Chapter 3, in which a number of recommendations are provided. The procedure developed was used to create a number of finite element models that were then analyzed and compared to experimental data. Additional models were developed for the purpose of verifying the procedure both quantitatively and mechanistically.

A total of five models were developed for comparison with the results of experimental tests; two full-scale top-and-seat angle tests, a shear bolt test, and two t-stub component tests. The full-scale models were developed to verify the procedure quantitatively and mechanistically, whereas the remaining three tests were primarily developed to verify the procedure mechanistically. Section 6.1 will draw conclusions from the results of the component models from Chapter 4. Section 6.2 will draw conclusions from the results of the full-scale models from Chapter 5. Section 6.3 will draw general recommendations about the modeling procedure. Specific recommendations for aspects of the procedure can be found in their corresponding sections in Chapter 3. Section 6.4 will discuss a few areas of potential future research.

6.1 Results of Component Models

Experimental component tests were modeled using the proposed procedure to verify its capability of reproducing the following failure modes: bolt shear, net section fracture, and block shear failure. The conclusions are as follows:

- The procedure is capable of producing finite element models that allow the analyst to identify failure in the bolts due to tension or shear, net section fracture (and by extension

108
gross section yielding), block shear failure, and beam plastic hinge formation. The ability of recognizing these failure mechanisms is paramount to finite element models created to investigate structures that could experience seismic events.

- The progression by which an analysis produces its results is as important as the final results themselves. This has to do with ABAQUS not being able to easily model failure. The burden is on the analyst to carefully observe plastic strain propagation and the corresponding values of plastic strain. Although bounds can be established between zero (yield) and ultimate plastic strain (rupture), ABAQUS will continue redistributing strains and stresses well beyond what could be considered the ultimate strain. Therefore, a modicum of judgment has to be exercised by the analyst to identify the occurrence of failure, by carefully examining plastic strain contour plots.

- With this in mind, it can be concluded that the proposed procedure is capable of producing models that allow the analyst to identify failure modes within a few percentage points of the experimental results, without the need for further tweaking or calibration.

### 6.2 Results of Full-Scale Models

The results from the full-scale top-and-seat angle connection models produced the following conclusions:

- Slip resistance and pretension force are two pieces of the same puzzle and both were demonstrated to be extremely difficult to model accurately. The magnitude of pretension directly affects the applied force corresponding to the onset of slip in the connection. For the proposed procedure, pretension was applied at a level corresponding to the minimum required force. The value of friction coefficient to be used in the procedure was then
calibrated so that the modeled system matched one of the experimental tests. It is important to note that slip is affected by the combination of pretension forces and friction coefficients: the procedure presented herein establishes the pretension force and the corresponding value of friction coefficient to be used is calibrated with respect to that force alone. Therefore, the friction coefficient used in the procedure does not purport to be representative of an actual friction coefficient between faying surfaces, but rather is the appropriate coefficient to be used with the pretension force that is applied.

- The apparent accuracy of the FS-01 model compared to the FS-02 model is in part due to the fact that the friction coefficient was calibrated based on FS-01 results, leading to minor differences with respect to FS-02. The more complex failure sequence of FS-02, associated to its greater ductility, also may have contributed to a decreased prediction capability of the model.

- Although small discrepancies between analytical and experimental results existed, the modeling procedure has proven to be sufficiently accurate to be used as a prediction tool for top-and-seat angle connections to assess load-displacement behavior, or derived moment-rotation behavior.

### 6.3 General Recommendations

This section will make a number of general recommendations about the modeling procedure. Specific recommendations for aspects of the procedure can be found in the corresponding sections in Chapter 3.
• It is recommended that connections modeled to be moment resisting contain a shear resisting element(s) in addition to the moment resisting element(s), and be a form of top-and-seat angle or t-stub connection.

• Nominal values for dimensions, material properties, tolerances, etc. should be avoided in areas of expected inelastic deformation. There were plenty of assumptions made during the creation of the procedure that were based on having more accurate values. This is not to say an experimental replica of any model created is needed, but that at the very least, values for dimensions, material properties, and tolerances should be taken from the latest accurate research, or should be using expected values.

• The approach developed here does not need to be adjusted on a model-by-model basis to achieve accurate results. FS-01 was the model on which the procedure was calibrated. All subsequent models in this work used the exact procedure as calibrated on FS-01, outside of the obvious geometry and boundary condition differences.

6.4 Future Research

The following are a number of aspects from this research that could be delved into further:

• A more accurate means to incorporate slip resistance for surfaces and pretension in bolts must be developed if bolted connections are going to be modeled further and for more complicated configurations. Oltman’s work provided invaluable information about bolt placement in holes for modeling. Similar work is needed with regard to slip resistance and bolt pretension.
• The analyses performed on the models created were done so using monotonic loading and non-dynamic testing. Cyclic loading is very difficult to model, but given the demand for seismically acceptable connections, it is essential that cyclic loadings be applied analytically. Dynamic testing would also be beneficial, but would fall behind applying cyclic loading in priority.

• Validation of the modeling procedure against the current limited test base is good, but further models should be created and verified, and the modeling procedure should be further honed for more robust connection configurations.
References


steelf structures: Behaviour, strength, and design. Faculty of Civil Engineering and Geosciences of the Delft University of Technology.


