SteelConverter & Caltech VirtualShaker: Rapid Nonlinear Cloud-Based Structural Model Conversion and Analysis

Thesis By
Christopher G. Janover, P.E.

In Partial Fulfillment of the Requirements

For the Degree of

Doctor of Philosophy

California Institute of Technology
Pasadena, California

2016
(Defended July 27, 2015)
Acknowledgements

I would like to begin by thanking The California Institute of Technology for providing me with an outstanding education and financial support over the last four years, as well as the opportunity to do research in a field about which I have grown so passionate.

I would like to thank my two advisors, Dr. Tom Heaton and Dr. John Hall, who have guided me through my experience at Caltech. They gave me the freedom to pursue research in a field in which I was immensely interested, and were always there to offer guidance when it was needed.

I wish to thank the MCE staff for their hard work which allowed me to focus on my education and research.

I would also like to express my gratitude to my fellow graduate students who spent the past four years with me. The time we spent together studying for our qualification exams, doing homework, and doing research made the whole process more enjoyable. A special thank you to Arnar Bjornsson, Grant Hollis, Ryan Hurley, and Ramses Mourhatch for their help testing and vetting my work. A huge thank you to Anthony Massari, who spent the past three years working closely with me. Anthony spent a great deal of his time helping me fix my problems and come up with great ideas.

I would like to extend a large thank you to my amazing girlfriend Adriana Carmona, who spent nearly the past three years putting up with me throughout this whole process. During my qualification exams, professional engineer exam, and defense
you were the not-so-calm force who did all of the worrying for me. You were the best and this would have been much more difficult without you. A thank you to Sherlock for being the best corgi around.

Above all I would like to thank my parents. Everything I have accomplished is because of your guidance and unwavering support. You are always looking out for me and always have my best interest in mind. Without you, none of this would have been possible. Thank you.
Abstract

STEEL, the Caltech created nonlinear large displacement analysis software, is currently used by a large number of researchers at Caltech. However, due to its complexity, lack of visualization tools (such as pre- and post-processing capabilities) rapid creation and analysis of models using this software was difficult. SteelConverter was created as a means to facilitate model creation through the use of the industry standard finite element solver ETABS. This software allows users to create models in ETABS and intelligently convert model information such as geometry, loading, releases, fixity, etc., into a format that STEEL understands. Models that would take several days to create and verify now take several hours or less. The productivity of the researcher as well as the level of confidence in the model being analyzed is greatly increased.

It has always been a major goal of Caltech to spread the knowledge created here to other universities. However, due to the complexity of STEEL it was difficult for researchers or engineers from other universities to conduct analyses. While SteelConverter did help researchers at Caltech improve their research, sending SteelConverter and its documentation to other universities was less than ideal. Issues of version control, individual computer requirements, and the difficulty of releasing updates made a more centralized solution preferred. This is where the idea for Caltech VirtualShaker was born. Through the creation of a centralized website where users could log in, submit, analyze, and process models in the cloud, all of the major concerns associated with the utilization of SteelConverter were eliminated. Caltech VirtualShaker
allows users to create profiles where defaults associated with their most commonly run models are saved, and allows them to submit multiple jobs to an online virtual server to be analyzed and post-processed. The creation of this website not only allowed for more rapid distribution of this tool, but also created a means for engineers and researchers with no access to powerful computer clusters to run computationally intensive analyses without the excessive cost of building and maintaining a computer cluster.

In order to increase confidence in the use of STEEL as an analysis system, as well as verify the conversion tools, a series of comparisons were done between STEEL and ETABS. Six models of increasing complexity, ranging from a cantilever column to a twenty-story moment frame, were analyzed to determine the ability of STEEL to accurately calculate basic model properties such as elastic stiffness and damping through a free vibration analysis as well as more complex structural properties such as overall structural capacity through a pushover analysis. These analyses showed a very strong agreement between the two softwares on every aspect of each analysis. However, these analyses also showed the ability of the STEEL analysis algorithm to converge at significantly larger drifts than ETABS when using the more computationally expensive and structurally realistic fiber hinges. Following the ETABS analysis, it was decided to repeat the comparisons in a software more capable of conducting highly nonlinear analysis, called Perform. These analyses again showed a very strong agreement between the two softwares in every aspect of each analysis through instability. However, due to some limitations in Perform, free vibration analyses for the three story one bay chevron brace frame, two bay chevron brace frame, and twenty
story moment frame could not be conducted. With the current trend towards ultimate capacity analysis, the ability to use fiber based models allows engineers to gain a better understanding of a building’s behavior under these extreme load scenarios.

Following this, a final study was done on Hall’s U20 structure [1] where the structure was analyzed in all three softwares and their results compared. The pushover curves from each software were compared and the differences caused by variations in software implementation explained. From this, conclusions can be drawn on the effectiveness of each analysis tool when attempting to analyze structures through the point of geometric instability. The analyses show that while ETABS was capable of accurately determining the elastic stiffness of the model, following the onset of inelastic behavior the analysis tool failed to converge. However, for the small number of time steps the ETABS analysis was converging, its results exactly matched those of STEEL, leading to the conclusion that ETABS is not an appropriate analysis package for analyzing a structure through the point of collapse when using fiber elements throughout the model. The analyses also showed that while Perform was capable of calculating the response of the structure accurately, restrictions in the material model resulted in a pushover curve that did not match that of STEEL exactly, particularly post collapse. However, such problems could be alleviated by choosing a more simplistic material model.
# Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acknowledgements</td>
<td>Acknowledgements</td>
<td>iii</td>
</tr>
<tr>
<td>Abstract</td>
<td>Abstract</td>
<td>v</td>
</tr>
<tr>
<td>Contents</td>
<td>Contents</td>
<td>viii</td>
</tr>
<tr>
<td>List of Figures</td>
<td>List of Figures</td>
<td>xii</td>
</tr>
<tr>
<td>List of Tables</td>
<td>List of Tables</td>
<td>xvi</td>
</tr>
<tr>
<td>Introduction</td>
<td>Introduction</td>
<td>17</td>
</tr>
<tr>
<td>Software Discussion</td>
<td>Software Discussion</td>
<td>20</td>
</tr>
<tr>
<td>1 SteelConverter</td>
<td>SteelConverter</td>
<td>22</td>
</tr>
<tr>
<td>1.1 SteelConverter - Introduction</td>
<td>SteelConverter - Introduction</td>
<td>22</td>
</tr>
<tr>
<td>1.2 ETABS Model Creation</td>
<td>ETABS Model Creation</td>
<td>24</td>
</tr>
<tr>
<td>1.2.1 Grid System</td>
<td>Grid System</td>
<td>25</td>
</tr>
<tr>
<td>1.2.2 Line Elements</td>
<td>Line Elements</td>
<td>26</td>
</tr>
<tr>
<td>1.2.3 Restraints</td>
<td>Restraints</td>
<td>29</td>
</tr>
<tr>
<td>1.2.4 Releases</td>
<td>Releases</td>
<td>30</td>
</tr>
<tr>
<td>1.2.5 Loading / Load Combinations</td>
<td>Loading / Load Combinations</td>
<td>30</td>
</tr>
<tr>
<td>1.2.6 Sections / Custom Sections</td>
<td>Sections / Custom Sections</td>
<td>31</td>
</tr>
<tr>
<td>1.2.7 Springs</td>
<td>Springs</td>
<td>31</td>
</tr>
<tr>
<td>1.2.8 Walls</td>
<td>Walls</td>
<td>31</td>
</tr>
<tr>
<td>1.2.9 Decking</td>
<td>Decking</td>
<td>32</td>
</tr>
<tr>
<td>1.2.10 Materials</td>
<td>Materials</td>
<td>34</td>
</tr>
<tr>
<td>1.3 SteelConverter Configuration File</td>
<td>SteelConverter Configuration File</td>
<td>35</td>
</tr>
<tr>
<td>1.4 Post Processing Tools</td>
<td>Post Processing Tools</td>
<td>46</td>
</tr>
<tr>
<td>1.4.1 LoadData</td>
<td>LoadData</td>
<td>46</td>
</tr>
<tr>
<td>1.4.2 Plot Undeformed Shape</td>
<td>Plot Undeformed Shape</td>
<td>47</td>
</tr>
<tr>
<td>1.4.3 Plot Dynamic Analysis</td>
<td>Plot Dynamic Analysis</td>
<td>51</td>
</tr>
<tr>
<td>1.5 Commentary</td>
<td>Commentary</td>
<td>54</td>
</tr>
<tr>
<td>1.5.1 Diaphragms</td>
<td>Diaphragms</td>
<td>54</td>
</tr>
<tr>
<td>1.5.2 Vertical Connection Elements</td>
<td>Vertical Connection Elements</td>
<td>56</td>
</tr>
<tr>
<td>1.5.3 Modeling of Secondary Frames</td>
<td>Modeling of Secondary Frames</td>
<td>58</td>
</tr>
<tr>
<td>1.5.4 Rayleigh Damping</td>
<td>Rayleigh Damping</td>
<td>58</td>
</tr>
<tr>
<td>1.5.5 Releases</td>
<td>Releases</td>
<td>59</td>
</tr>
<tr>
<td>1.5.6 Damping / Special Columns</td>
<td>Damping / Special Columns</td>
<td>64</td>
</tr>
<tr>
<td>1.5.7 Special Columns</td>
<td>Special Columns</td>
<td>64</td>
</tr>
<tr>
<td>1.5.8 Recommended Values</td>
<td>Recommended Values</td>
<td>67</td>
</tr>
<tr>
<td>1.5.9 Element Strong Axis / Weak Axis Orientation</td>
<td>Element Strong Axis / Weak Axis Orientation</td>
<td>70</td>
</tr>
<tr>
<td>1.5.10 Nodal Mass</td>
<td>Nodal Mass</td>
<td>71</td>
</tr>
<tr>
<td>1.5.11 Decking</td>
<td>Decking</td>
<td>71</td>
</tr>
<tr>
<td>1.5.12 Units</td>
<td>Units</td>
<td>72</td>
</tr>
</tbody>
</table>
1.5.13  Gravity ..................................................................................... 72
1.5.14  Panel Zones ............................................................................. 72
1.5.15  Element Connectivity ................................................................. 73
1.5.16  Axial Load Eccentricity ............................................................... 74
1.5.17  Node Numbering ........................................................................ 74
1.6  STEEL INPUT FILES ................................................................. 76
   1.6.1  for001 ................................................................................. 76
   1.6.2  for002 ................................................................................. 84
   1.6.3  for003 ................................................................................. 84
   1.6.4  for020 ................................................................................. 84
   1.6.5  for021 ................................................................................. 84
   1.6.6  for029 ................................................................................. 85
1.7  RUNNING STEEL ON A PBS SERVER ........................................... 86
   1.7.1  Directory Setup ...................................................................... 86
   1.7.2  Server Scripts ....................................................................... 86
   1.7.3  Submitting a Job .................................................................. 90
   1.7.4  Monitoring Results ................................................................. 90
1.8  SAMPLE 6 STORY MODEL .......................................................... 92
   1.8.1  ETABS Model ..................................................................... 92
   1.8.2  Sample .e2k file .................................................................. 94
   1.8.3  Sample SteelConverter Configuration File ............................ 94
   1.8.4  Sample STEEL Input File ...................................................... 94
   1.8.5  Sample STEEL Output File .................................................... 94
1.9  CHANGE LOG ............................................................................... 95

2  Caltech VirtualShaker ...................................................................... 96

2.1  INTRODUCTION .......................................................................... 96
2.2  GETTING STARTED ................................................................. 97
   2.2.1  Running the U6 – Base sample model .................................... 97
   2.2.2  Creating a Model with the Baseline Default ......................... 106
2.3  VIRTUALSHAKER BACK-END DESCRIPTION ................................. 108
   2.3.1  Amazon Web Services .......................................................... 108
   2.3.2  EC2 Servers ....................................................................... 109
   2.3.3  S3 Cloud Storage ................................................................. 111
   2.3.4  SQS Messaging .................................................................. 112
   2.3.5  SQL Database Design .......................................................... 113
2.4  FEATURES .................................................................................. 116
   2.4.1  Creating an Account ............................................................. 116
   2.4.2  Creating Defaults .................................................................. 118
   2.4.3  Modifying a Default ............................................................. 120
   2.4.4  Uploading Ground Motions .................................................. 121
   2.4.5  Creating a Model ................................................................. 123
   2.4.6  Creating an Analysis ............................................................. 125
   2.4.7  Customizing an Analysis’s Configuration ............................... 127
   2.4.8  Submitting an Analysis ........................................................ 128
   2.4.9  Viewing Analysis Status ....................................................... 129
   2.4.10  Viewing / Downloading Results .......................................... 131
   2.4.11  Downloads ........................................................................ 133
3.3.4 Material Model

2.4.12 Documentation

2.5 RESULTS & POST-PROCESSING

2.5.1 Input Files

2.5.2 Output Files

2.5.3 Post-Processing Files

2.6 CONFIGURATION DESCRIPTION

2.6.1 Model Information

2.6.2 Analysis Options

2.6.3 Damping Options

2.6.4 Diaphragm Options

2.6.5 Convergence Options

2.6.6 Fiber Options

2.6.7 Vertical Constraint Options

2.6.8 Load & Post-Processing Options

2.6.9 Response Time History Options

2.6.10 Material Model Options

2.6.11 Foundation Node Options

2.7 CHANGEOLOG

3 STEEL Verification

3.1 ETABS TO STEEL COMPARISON

3.1.1 Introduction

3.1.2 ETABS Model Description

3.1.3 Material Model Description

3.1.4 Analysis Discussion

3.1.5 Cantilever Column

3.1.6 Three Story Moment Frame

3.1.7 Three Story One Bay Chevron Brace Frame

3.1.8 Two Bay Three Story Moment Frame

3.1.9 Two Bay Chevron Brace Frame

3.1.10 Twenty Story Moment Frame

3.2 PERFORM3D STEEL COMPARISON

3.2.1 Introduction

3.2.2 Perform3D Model Description

3.2.3 Material Model Description

3.2.4 Perform3D Analysis Limitations

3.2.5 Analysis Discussion

3.2.6 Cantilever Column

3.2.7 Three Story Moment Frame

3.2.8 Three Story One Bay Chevron Brace Frame

3.2.9 Two Bay Three Story Moment Frame

3.2.10 Two Bay Chevron Brace Frame

3.2.11 Twenty Story Moment Frame

3.3 TWENTY STORY ANALYSES

3.3.1 Introduction

3.3.2 Software Description

3.3.3 Model Description

3.3.4 Material Model
3.3.5 Software Limitations ........................................................................................................ 231
3.3.6 Pushover Analysis Comparison ...................................................................................... 233
3.3.7 Nepal Time History Analysis .......................................................................................... 239
3.4 Conclusion ......................................................................................................................... 251
3.5 Lessons Learned .................................................................................................................. 254

Works Cited .............................................................................................................................. 256

Appendix ..................................................................................................................................... 259

APPENDIX A – SAMPLE 6-STORY X-Brace Building .e2k File ............................................. 259
APPENDIX B – SAMPLE STEELCONVERTER CONFIGURATION FILE ................................ 282
APPENDIX C – SAMPLE STEEL FOR001 INPUT FILE .......................................................... 286
APPENDIX D – SAMPLE STEEL SECTION CONVERSION FILE .......................................... 302
APPENDIX E – SAMPLE STEEL SLAB CONVERSION FILE ................................................. 310
APPENDIX F – SAMPLE STEEL SEED FILE (FOR029) .......................................................... 311
APPENDIX G – SAMPLE STEEL GRID CONVERSION FILE .................................................. 312
APPENDIX H – SAMPLE STEEL MATERIAL CONVERSION FILE ....................................... 313
APPENDIX I – SAMPLE STEEL SECTION CONVERSION FILE ............................................. 314
List of Figures

Figure 1-1 - Example ETABS Grid Systems ................................................................. 25
Figure 1-2: Example of Non-Acceptable and Acceptable Element Meshing ............... 26
Figure 1-3: Example of Acceptable and Non-Acceptable Column Placement ................ 28
Figure 1-4: ETABS Column Bracing Requirement ..................................................... 29
Figure 1-5: ETABS Decking Input .............................................................................. 32
Figure 1-6: STEEL Decking Input ............................................................................. 33
Figure 1-7: Acceptable ETABS Deck Placement ....................................................... 33
Figure 1-8: STEEL Diaphragm Depiction .................................................................... 54
Figure 1-9: Vertical Connection Elements - Core ..................................................... 56
Figure 1-10: Vertical Connection Elements - Outrigger ............................................ 57
Figure 1-11: Extra Restraints placed on Secondary Frames ....................................... 58
Figure 1-12: STEEL Element Fiber Description ....................................................... 60
Figure 1-13: STEEL Beam/Column Segment Description .......................................... 60
Figure 1-14 - Special Column Force Distribution ...................................................... 66
Figure 1-15: STEEL Beam, Column, and Brace Element Connectivity Information ...... 73
Figure 1-16: STEEL Basement Wall Element Connectivity Information ................. 73
Figure 1-17: Axial Load Eccentricity Factor Variable Description ............................. 74
Figure 1-18: Automatic Node Numbering Technique for SteelConverter .................. 75
Figure 1-19: STEEL Slab and Deck Input Dimensions ............................................. 84
Figure 1-20: 6 Story Example Structure – Isometric View ........................................ 93
Figure 2-1 - Downloads ......................................................................................... 97
Figure 2-2 - Profile Overview ................................................................................ 98
Figure 2-3 - Profile Overview with U6 - Base default active ................................... 99
Figure 2-4 - New Model ......................................................................................... 99
Figure 2-5 - Model Created .................................................................................. 100
Figure 2-6 - New Analysis ................................................................................... 101
Figure 2-7 - Analysis Created for sample model ................................................... 102
Figure 2-8 - Sample Analysis Submitted ............................................................... 103
Figure 2-9 - Sample Model Run Status Tooltip ..................................................... 103
Figure 2-10 - Sample Model Run Status page ........................................................ 104
Figure 2-11 - Sample Model Results Page ............................................................. 104
Figure 2-12 - Sample Model Undefomed Shape ..................................................... 105
Figure 2-13 - Baseline Default Profile Overview ................................................... 107
Figure 2-14 - AWS Workflow ............................................................................. 108
Figure 2-15 - VirtualShaker SQL Database ............................................................ 115
Figure 2-16 - Home Page: Login ........................................................................ 116
Figure 2-17 - New User: Login Screen .................................................................. 117
Figure 2-18 - New User: Create User ..................................................................... 118
Figure 2-19 - Default Overview ................................................................. 119
Figure 2-20 - Typical Default Options Page .............................................. 120
Figure 2-21 - Configuration Tooltip .......................................................... 121
Figure 2-22 - Ground Motions Upload ......................................................... 122
Figure 2-23 - Model Creation ..................................................................... 124
Figure 2-24 - Model Analysis Listing ........................................................... 124
Figure 2-25 - Models Listing ........................................................................ 125
Figure 2-26 - New Analysis .......................................................................... 126
Figure 2-27 - Analysis Options ..................................................................... 126
Figure 2-28 - Configuration Overview .......................................................... 128
Figure 2-29 - Configuration Example ........................................................... 128
Figure 2-30 - Model Submitted ..................................................................... 129
Figure 2-31 - Run Status Popup .................................................................... 130
Figure 2-32 - Detailed Run Status ................................................................. 130
Figure 2-33 - Results Page ........................................................................... 131
Figure 2-34 - Results Dropdown .................................................................... 132
Figure 2-35 - Results Embedded Image ......................................................... 133
Figure 2-36 - Downloads Page ..................................................................... 134
Figure 2-37 - Documentation Page ................................................................. 135
Figure 2-38 - New Post-Process ................................................................... 137
Figure 2-39 - Model Information ................................................................. 141
Figure 2-40 - Analysis Options ..................................................................... 142
Figure 2-41 - Damping Options ................................................................... 144
Figure 2-42 - Diaphragm Options ................................................................ 145
Figure 2-43 - Convergence Options .............................................................. 146
Figure 2-44 - Fiber Options .......................................................................... 148
Figure 2-45 - Vertical Constraint Options ..................................................... 149
Figure 2-46 - Load & Post Processing Options .............................................. 151
Figure 2-47 - Load & Post-Processing Options - Analysis Configuration ...... 152
Figure 2-48 - Response Time History Options ............................................. 153
Figure 2-49 - Material Model Options .......................................................... 154
Figure 2-50 - Foundation Node Options ....................................................... 156
Figure 3-1 - STEEL Material Model Description [1] ................................... 161
Figure 3-2 - Cantilever Column Model Description ....................................... 163
Figure 3-3 - Cantilever Column - Free Vibration Analysis ............................. 164
Figure 3-4 - Cantilever Column – Pushover Analysis .................................... 165
Figure 3-5 – Three Story Moment Frame - Model Description ................. 167
Figure 3-6 - Three Story Moment Frame - Free Vibration Analysis .......... 168
Figure 3-7 - Three Story Moment Frame - Pushover Analysis .................... 169
Figure 3-8 - Three Story One Bay Chevron Brace Frame - Model Description 171
Figure 3-9 - Three Story One Bay Chevron Brace Frame - Free Vibration Analysis 172
Figure 3-10 - Three Story One Bay Chevron Brace Frame - Pushover Analysis 173
Figure 3-11 - Three Story Chevron Brace Frame – Pushover Analysis - Scaled 174
Figure 3-12 - Two Bay Three Story Moment Frame - Section Assignments 175
Figure 3-13 - Two Bay Three Story Moment Frame - Force Assignments .......................... 176
Figure 3-14 - Two Bay Three Story Moment Frame - Free Vibration Analysis ................ 178
Figure 3-15 - Two Bay Three Story Moment Frame - Free Vibration Analysis - Shifted 178
Figure 3-16 - Two Bay Three Story Moment Frame - Pushover Analysis ....................... 179
Figure 3-17 - Two Bay Three Story Moment Frame - Pushover Analysis - Scaled ........... 180
Figure 3-18 - Two Bay Three Story Chevron Brace Frame - Sections ............................. 182
Figure 3-19 - Two Bay Three Story Chevron Brace Frame - Forces ............................... 182
Figure 3-20 - Two Bay Three Story Chevron Brace Frame - Free Vibration Analysis ....... 184
Figure 3-21 - Two Bay Three Story Chevron Brace Frame - Free Vibration Analysis - Shifted ........................................................................................................... 184
Figure 3-22 - Two Bay Three Story Chevron Brace Frame - Pushover Analysis .......... 186
Figure 3-23 - Two Bay Three Story Chevron Brace Frame - Pushover Analysis - Scaled 187
Figure 3-24 - Twenty Story Moment Frame - Section Assignments ............................... 189
Figure 3-25 - Twenty Story Moment Frame - Column and Girder Schedule .................. 189
Figure 3-26 - Twenty Story Moment Frame - Free Vibration Analysis ......................... 191
Figure 3-27 - Twenty Story Moment Frame - Pushover Analysis ................................. 192
Figure 3-28 - Material Model Description - Perform3D vs. STEEL ................................. 196
Figure 3-29 - Cantilever Column - Free Vibration Analysis - Perform ............................ 201
Figure 3-30 - Cantilever Column - Pushover Analysis - Perform .................................. 202
Figure 3-31 - Three Story Moment Frame - Free Vibration Analysis - Perform .......... 205
Figure 3-32 - Three Story Moment Frame - Pushover Analysis - Perform .................... 206
Figure 3-33 - Three Story One Bay Chevron Brace Frame - Free Vibration Analysis - Perform ........................................................................................................... 207
Figure 3-34 - Three Story Chevron Brace Frame - Pushover - Perform ......................... 208
Figure 3-35 - Two Bay Three Story Moment Frame - Free Vibration Analysis - Perform 211
Figure 3-36 - Two Bay Three Story Moment Frame - Pushover Analysis - Perform ..... 212
Figure 3-37 - Two Bay Chevron Brace Frame - Free Vibration Analysis - Perform ........ 214
Figure 3-38 - Two Bay Chevron Brace Frame - Pushover Analysis - Perform .......... 215
Figure 3-39 - Twenty Story Moment Frame - Free Vibration Analysis - Perform ......... 216
Figure 3-40 - Twenty Story Moment Frame - Pushover Analysis - Perform ................ 217
Figure 3-41 - U20 - Model Description ............................................................................. 224
Figure 3-42 - U20 - Structural Details .............................................................................. 225
Figure 3-43 - J20 - Model Description ............................................................................. 226
Figure 3-44 - J20 – Structural Details .............................................................................. 227
Figure 3-45 - Fiber Element Description ......................................................................... 228
Figure 3-46 - U20 Material Model Comparison ............................................................... 231
Figure 3-47 - U20 - Pushover Comparison ..................................................................... 234
Figure 3-48 - U20 Pushover Comparison - STEEL vs ETABS ....................................... 235
Figure 3-49 - U20 STEEL & Perform Collapse Mechanism Comparison .................... 237
Figure 3-50 - Nepal Epicenter Locations [21] ................................................................. 240
Figure 3-51 - Nepal Intensity Plot [21] ........................................................................... 240
Figure 3-52 - Nepal Ground Motion Sensor Readings [21] ........................................... 241
Figure 3-53 - U20 Pushover results for perfect and brittle connections ......................... 244
Figure 3-54 - J20 Pushover results for perfect and brittle connections ......................... 245
Figure 3-55 – Snapshot Time History Response of U20 Structure to Nepal Ground Motion .................................................................................................................................................................................. 247
Figure 3-56 - U20P - Residual Interstory Drift of Structure Subjected to Nepal Ground Motion................................................................................................................................................................................................................................................. 248
Figure 3-57 - Snapshot of Time History Response of J20 Structure to Nepal Ground Motion (J20P and J20B Results are identical) ............................................................................................................................................................................................................................................................................................................. 249
Figure 3-58 - J20 - Residual Interstory Drift of Structure Subjected to Nepal Ground Motion (J20P and J20B results are identical) .............................................................................................................................................................................................................................................................................................................. 249
List of Tables

Table 1-1: STEEL Element Release Definitions ................................................................. 64
Table 2-1 - Post-Processing Options .................................................................................. 138
Table 2-2 - Model Information Options Description ....................................................... 141
Table 2-3 - Analysis Options Descriptions ...................................................................... 143
Table 2-4 - Damping Option Descriptions ...................................................................... 144
Table 2-5 - Diaphragm Option Descriptions .................................................................. 145
Table 2-6 - Convergence Options Descriptions .............................................................. 147
Table 2-7 - Fiber Options Descriptions ......................................................................... 148
Table 2-8 - Vertical Constraint Options Descriptions ................................................... 150
Table 2-9 - Load & Post Processing Options Descriptions .............................................. 151
Table 2-10 - Response Time History Options Descriptions ............................................ 153
Table 2-11 - Material Model Options Descriptions ......................................................... 155
Table 2-12 - Foundation Node Options Descriptions .................................................... 156
Table 3-1 - PFA Calculation Values ................................................................................. 245
**Table 3-2 - PFA and Time History Result Comparison** .................................................. 250
Introduction

In the process of analyzing a structure there are a number of different finite element packages available to an engineer. Very often, particular programs excel in a specific facet of analysis such as concrete structures, post-tensioned slabs, nonlinear analyses etc. At the California Institute of Technology a nonlinear large displacement finite element software, STEEL, was created by Professor John Hall with the goal of providing detailed fiber based analysis of steel structures. Through time, STEEL has developed into an analysis tool used by many researchers at Caltech due to its ability to accurately model highly nonlinear behavior in structures. However, with this increase in ability came an increase in complexity. The process of learning to use STEEL is a difficult one due to the system’s lack of pre- and post-processing abilities, and its text-based input methodology.

SteelConverter was created as a means of simplifying the creation of these nonlinear models through the use of the popular, industry standard, finite element package ETABS. SteelConverter allows engineers and researchers to create models in ETABS and intelligently import them into STEEL. A model creation process that previously took days or weeks to complete can now be accomplished in minutes or hours. SteelConverter not only allows users to import geometric model properties, but also includes information such as loading, load combinations, scale factors, releases, and more. These properties are converted from ETABS format and translated into a format that STEEL understands. The error-prone method of
creating STEEL input files through various model-specific spreadsheets was eliminated, and this new software allows Caltech researchers to use a unified program to create their models.

SteelConverter not only decreases the production time of models but also dramatically decreases the likelihood of errors. For example, allowing researchers to apply loads directly to nodes and combine them with scale factors greatly reduces the chance that values would be inputted incorrectly because the ability to visualize the information was made available.

It has always been a major goal of Caltech to spread the knowledge created here to other universities. However, due to the complexity of STEEL it was difficult for researchers or engineers from other universities to conduct analyses. While SteelConverter did help researchers at Caltech improve their research, sending SteelConverter and its documentation to other universities was less than ideal. Issues of version control, individual computer requirements, and the complexity associated with releasing updates made a more centralized solution preferred. This is where the idea for Caltech VirtualShaker was born. Through the creation of a centralized website where users could log in, submit, analyze, and process models in the cloud, all of the major concerns associated with the effective utilization of SteelConverter were eliminated. Caltech VirtualShaker allows users to create profiles in which the defaults of their most commonly run models are saved, and allows them to submit multiple jobs to an online virtual server to be analyzed and post-processed. The creation of this website not only allows for more rapid distribution of this tool, but also creates a means for engineers and researchers with no access to powerful computer clusters to run computationally intensive analyses without the excessive cost of building and maintaining a computer cluster.
The creation of SteelConverter and Caltech VirtualShaker helps extend the ability for researchers to use the tools created by Caltech and helps aid in the distribution of knowledge throughout the field of engineering.
Software Discussion

As the demand to analyze structures in increasingly complicated and sophisticated manners grows, the capabilities of the softwares used for these analyses must as well. Finite element analysis software is used throughout the industry for nearly every aspect of the analysis / design process and countless analysis packages have been developed to meet the individual needs of each engineering task.

The majority of structural analysis falls under the category of linear elastic. This type of analysis is done at most structural engineering firms and is used to analyze structures for loading environments such as gravity, wind, and seismic. For these types of analysis it is beneficial for the software to contain features such as automated design to assist the engineer with the most up-to-date analysis code. The softwares which are by far the most commonly used for this type of analysis is SAP2000 and ETABS developed by Computers and Structures Inc. (CSI) [2]. This software allows engineers to easily construct models, apply loads, and run linear elastic analyses. It also has functionality for more advanced, nonlinear inelastic analyses; however, it is not used primarily for this. There are many other softwares which accomplish the same goals as SAP2000 and ETABS, such as OpenSees [3] and ANSYS [4] but their market share in the private sector is significantly less.

For more complex finite element analyses there is a different set of software researchers will use that are still being developed by companies. Softwares such as Perform3D [5] [6] by CSI or LS DYNA [7], created by the Livermore Software Technology Corporation (LSTC). Perform 3D is widely used among professional engineers to conduct more advanced nonlinear analyses. A
large array of elements with both linear and nonlinear properties as well as numerous pre-built structural engineering elements make it an ideal choice for the analysis of standard building structures to more advanced loading environments. LS DYNA is capable of full 3D nonlinear rigid body dynamics as well as being able to analyze more advanced properties like crack propagation, failure analysis, and fracture.

It is not uncommon for research institutions and analysis firms to develop their own FEA software to meet their individual needs. Software like, STEEL [1], created by Dr. John Hall at the California Institute of Technology, was created to more accurately analyze steel structures at large strains. With features like probabilistic brittle weld failure, a complex material model, and nonlinear damping researchers at Caltech have been able to more accurately predict the ultimate capacity of structures and their behavior during collapse [8] [9] [10]. Dr. Krishnan created the 3D analysis tool FRAME3D based off of STEEL that implements many of the same features including elements such as a plastic hinge element and elastofiber beam element and utilizes a Newton-Raphson iteration strategy applied to an implicit Newmark time-integration scheme [11] [12]. Drain-2D and Drain-2DX, created by Dr. Powell at the University of California, Berkeley [13] [14] is widely used in the research community to study collapse behavior of structures. Post September 11th, 2001 the National Information Service for Earthquake Engineering has done extensive analyses on the necessary steps engineers must take when designing high-profile structures to prevent progressive collapse due to an extraordinary loading environments using this software [15] [16].
1 SteelConverter

1.1 SteelConverter - Introduction

SteelConverter is an automatic model generation tool for the in-house non-linear analysis software STEEL created by Professor John Hall at The California Institute of Technology. SteelConverter allows the user to create models in the widely used analysis and design tool ETABS from Computers and Structures Inc. and import many of the modeled parameters into a text file STEEL understands.

This software was written to aid in the research of graduate students at Caltech as well as allow researchers from other universities to begin utilizing STEEL without the steep learning curve that comes with learning a new piece of software. Additionally, since STEEL has no graphical user interface, creating large models with no errors can be difficult and time consuming. SteelConverter aims to alleviate this by allowing the user to utilize the graphical front end of ETABS.

SteelConverter is custom software written in C++ by Christopher Janover at The California Institute of Technology and works by parsing through the text-based save file created by ETABS (.e2k file) along with supplemental information in the form of a configuration file, reorganizing the data, and then outputting to a format STEEL understands. Additionally, several post-processing tools have been created in Matlab that allow the user to more easily visualize the results from STEEL analyses.
This manual will begin by demonstrating to the user how to properly make models in ETABS by going through every type of element and property available for importing and discussing acceptable modeling techniques. Next, a detailed discussion of the SteelConverter configuration file is done to give the user a thorough understanding of how SteelConverter uses this file to supplement data imported from ETABS. Following this, some of the post-processing tools created in Matlab to assist in visualization of the results are discussed and their source code is given. A commentary section is also included in this manual that discusses the inner workings of both SteelConverter and STEEL and goes through the assumptions made in the current version of the conversion software in addition to methods and techniques to modify the input files to meet the individual needs of the user.

Additionally, this manual gives a detailed description of the format of the STEEL input files so the user may manually modify input to meet their personal needs. Following this, a description of the steps necessary as well as the source code needed to run multiple analyses simultaneously on a PBS server is given, allowing the user to rapidly analyze a model for a series of ground motions. Lastly, an example problem is given for a six story braced frame building developed by Anthony Massari at Caltech. The ETABS .e2k file, SteelConverter configuration file, and STEEL input files are all given to allow the user to verify proper modeling technique. Additionally, ground motions and the results from the analysis can also be made available upon request.

To obtain the most current version of STEEL, SteelConverter, and this manual email cjanover@caltech.edu, and the files and executable will be provided. For any questions, comments, or bug reports email all relevant information to cjanover@caltech.edu.
1.2 ETABS Model Creation

There are several rules and assumptions made by SteelConverter that must be followed when creating an ETABS model. All STEEL models must have an orthogonal Primary and a Secondary direction where the Primary direction is the direction in which earthquake motions will be applied. Since STEEL is a 2D analyses software, to model 3D structures as accurately as possible several specialty elements have been developed which, in conjunction with SteelConverter, transform the full 3D ETABS model into a series of 2D projections in a roughly equivalent STEEL model.

However, since it is impossible to fully capture certain 3D affects in 2D analysis, such as torsion and bi-axial bending, certain restrictions must be made on the ETABS model to yield an accurate 2D representation. First, it is recommended that all ETABS models to be imported be symmetric in the direction the system is being loaded, as this will reduce the amount of torsion in the structure. Second, the lateral system should be designed to avoid the occurrence of biaxial bending. This means avoiding moment connections in two orthogonal directions on a single column. For more information on the 3D to 2D conversion see sections 1.5.2 and 1.5.3

For all STEEL models the direction that is considered Primary will contain the majority of the column elements and is the major focus of the analysis. The choice of Primary vs. Secondary Direction will be specified in the SteelConverter configuration file (which will be discussed later in the manual) allowing for general model construction in both orthogonal directions in ETABS without regard to this constraint. However, the user will be made aware of how their placement of elements affects which grouping the elements are placed into. More information about this 3D to 2D conversion can be found in Section 1.5.3.
1.2.1 Grid System

Grids in ETABS can be created utilizing either the default “Quick Templates” or the grid editor inside the model. All grids must be orthogonal and there are hardly any restrictions on spacing or labeling. Images of example grid system can be seen in Figure 1-1.

![Figure 1-1 - Example ETABS Grid Systems](image)

The grid system shown in part A) of Figure 1-1 is an acceptable grid system because it is symmetric, and has unique grid labels and gridlines which are tangent to either the X or Y directions. The grid system in part B) is not acceptable because the grid system has been rotated away from the X and Y-axis. The grid system in part C) is acceptable; however, it is not recommended, since the gridlines are not symmetric various 3D affects such as torsion will not
be captured properly in the 3D to 2D conversion. Finally, the grid system in part D) is not acceptable because of the non-unique grid labels.

1.2.2 Line Elements

The three types of line elements in ETABS that can be converted to STEEL are columns, beams, and braces. All three have similar restrictions with some additional restrictions placed on columns.

1.2.2.1 General Line Element Restrictions

All line elements must be divided at the intersection of any connecting element or break in floor, while in software such as ETABS it is possible for an “auto-meshing” feature to be enabled. In STEEL if the line elements are not meshed they will behave as though they are not connected. This situation often arises when constructing the lateral system in the model. An example of an acceptable and not acceptable element meshing can be seen in Figure 1-2.
The image on the left shows that the beam spanning over the Chevron brace is not meshed at the intersection of the brace at the midpoint of the beam. This results in the STEEL model treating this configuration as a simply supported beam and a freestanding set of braces. This can be resolved in the model by selecting all elements and using the “divide all frames at intersection” feature located under the Edit->Divide Frame menu. The result of this is the image on the right of Figure 1-3. Here the beam spanning between the columns has been divided in two and now meshes at the intersection point of the chevron brace.

1.2.2.2 Column Element Restrictions

The placement of column elements in ETABS affects how SteelConverter treats these elements. Columns in ETABS must either be placed at the intersection of two grid locations or on a gridline running parallel to either the X or Y-axis. It is not acceptable to place a column in free-space. Examples of acceptable column placements can be seen in Figure 1-3. Placing a column at the intersection of two gridlines results in SteelConverter treating that column as Primary. If a column is placed solely on a gridline that runs parallel to the Primary Direction then the column is treated as Primary. Similarly, if the column is placed solely on a gridline that runs parallel to the Secondary Direction, then the column is treated as Secondary.

While all columns in ETABS default to strong axis bending in the X direction it is possible to import column orientation from ETABS to STEEL. This can be accomplished by selecting a column and assigning a local axis rotation of 90 degrees. This will cause the element to have weak axis bending in the X direction.
1.2.2.3 Beam Element Restrictions

All beam elements placed in ETABS must run parallel to either the Primary or Secondary Directions. It is not acceptable to have a beam that runs diagonally. A beam element will be treated as Primary if it runs parallel to the Primary direction and will be treated as Secondary if it runs parallel to the Secondary direction. Additionally, all columns must be braced on every floor by a beam. It is not acceptable to have a column spanning more than one floor without a beam framing into it, as it will cause instability in the model. It is therefore required that the user place pinned infill beams between all columns even if no lateral system exists at that location. An example of acceptable and unacceptable frame can be seen in Figure 1-4.
1.2.2.4 Brace Element Restrictions

All brace elements must run in a plane parallel to the Z-axis. A brace is considered to be Primary if it lies in a plane parallel to the Primary Direction and is considered to be Secondary if it lies in a plane parallel to the Secondary Direction. Additionally, HSS sections may only be used as brace elements.

1.2.3 Restraints

Any node in the STEEL model can be restrained by assigning restraints in the ETABS model. Nodes can be restrained in horizontal and vertical directions (UX, UY, UZ) as well as rotation about the X and Y-axis (RX, RY). If the Primary Direction of the model is the X direction and the model is restrained in the UX, UZ, RY and the node is Primary, then it will be treated as fixed, while if the node is Secondary then it will be a horizontal roller.

Any combination of restraints can be used in the model, however. It is traditional to only restrain the base nodes.
1.2.4 Releases

Any element in the STEEL model can be given releases by assigning releases in the ETABS model. SteelConverter is capable of importing only moment releases into STEEL. A more detailed discussion on releases can be found in Section 1.5.5.

1.2.5 Loading / Load Combinations

Only point loads can be transferred to STEEL and all loads must be placed in a load combination. STEEL imports two load combinations that are specified in the SteelConverter configuration file by name. Therefore, it is possible to have more combinations created in ETABS and run multiple sets of STEEL analyses by changing the important load combination in the SteelConverter configuration file. Steel uses these two load combinations to apply static loading and mass on the model; therefore it is advised to create load patterns in ETABS for loads such as dead, live, roof, etc. and then combine them with the appropriate load factors into a named combination to be applied to the STEEL nodes. When creating the mass combination assign the loads as a weight in the appropriate unit (i.e., N, lb.) and SteelConverter will apply the mass in both the vertical and horizontal directions on the Primary frames.

It is not acceptable to create combinations of combinations in ETABS. Similarly, all combination and pattern names must be unique. Additionally, only nodal loading is imported into STEEL.

For more information on importing load combinations see Section 1.3.
1.2.6 Sections / Custom Sections

SteelConverter is able to assign the appropriate sections to STEEL given section assigns from ETABS. Both premade and custom sections can be utilized however, only wide-flange and tube shapes are currently implemented. Additionally, any tube section used in the model must have a section name that begins with HSS. To create a custom section use either the I/Wide Flange or the Box/Tube Section tool in the Define->Section Properties->Sections->Add New Property menu and give the section a unique name. It is not acceptable to leave sections with the default section type (FSEC-1). SteelConverter will automatically convert US section properties to Metric when the ETABS .e2k file is exported in a metric unit.

1.2.7 Springs

SteelConverter has the ability to import the location of springs from ETABS into STEEL; however, specific properties about the spring are assigned via the SteelConverter configuration file. To assign a node in STEEL with a specific spring property, define a linear spring type in ETABS with the same name as the non-linear spring definition in the SteelConverter configuration file. For more information how to implement springs see spring input description in Section 1.3.

1.2.8 Walls

Wall elements in ETABS can be used to create basement wall elements in STEEL. SteelConverter only imports the name and location of the wall elements from ETABS. Specific properties of these elements are defined and assigned via the SteelConverter configuration file.
All ETABS wall elements must be rectangular and be drawn vertically. These elements are usually drawn on the bottom floor. Custom sections can be created via the built-in ETABS wall element section definition form. For more information on the function of basement wall elements see Section 1.3.

1.2.9 Decking

SteelConverter has the ability to import both the ETABS deck definitions and locations into STEEL. When defining a deck property in ETABS take note of the different definitions meanings between ETABS and STEEL to ensure the element is defined properly. Figures showing the ETABS deck definition window with visual representation can be seen in Figure 1-5, while a figure showing deck input in STEEL format can be seen in Figure 1-6. Note that SteelConverter automatically converts from ETABS format to STEEL format.

![Figure 1-5: ETABS Decking Input](image-url)
In the current version of SteelConverter only one type of deck may be present on a particular floor. It is not acceptable to draw a floor with multiple deck properties or with decking only a particular location. An example of an acceptable method of placing deck elements in ETABS can be seen in Figure 1-7. For more information on these limitations see Section 1.5.11.
1.2.10 Materials

As the material models used in STEEL are more complex than those used by ETABS, material element assignments or definitions are not directly imported from ETABS. Rather, the user assigns a material to each ETABS element, defines the STEEL material in the SteelConverter configuration file with a lookup between ETABS material name and STEEL material number. This is discussed in more detail in the SteelConverter configuration explanation in Section 1.3.
1.3 SteelConverter Configuration File

In order for SteelConverter to convert the ETABS .e2k file into the STEEL input file several options in a configuration file must be set. Comments can be made in the Configuration file by utilizing a ‘%’ before any text the user wants the parser to ignore. Each configuration option is preceded by a tag inside of brackets (i.e. [ExTH]). The order of the tags does not matter, however, it is recommended that the user does not alter the order. Each input to the configuration file will now be gone through and explained.

• **Program Output Information**
  - [DEBUG] – Toggle to enable or disable debug output (yes or no)
    - Currently not implemented
  - [SECTIONCONVERSION] – Toggle to enable or disable output of section conversion table (yes or no)
  - [MATERIALCONVERSION] – Toggle to enable or disable output of material conversion table (yes or no)

• **Model Information**
  - [TITLE] – Title of the model (Name output data will saved as)
  - [SAVELOC] – Location where input and output files will be saved do (don’t include trailing / in directory)
  - [ETABSTITLE] – Title of ETABS file (Name of .e2k file to be read from)
  - [ETABSLOC] – Location of ETABS input file (don’t include trailing / in directory)
- **[PRIMARYETABS_DIR]** – Direction in the ETABS model to use as the Primary Direction
- **[STEELSECTION]** – Section Database

### Analysis Options

- **[PanelZoneRigidity]** - Rigidity of Panel Zones at non-fixed points (1 = Rigid, 2 = Flexible). Note that weak axis column nodes are always given a flexible panel zone.
- **[MTP]** – Maximum number of turning points in Hysteretic Models (suggested minimum of 20)
- **[NDIM]** – Maximum number of turning point locations (suggested minimum of 100000)
- **[NSS]** – Number of static load steps
- **[BETA]** – Newmark Integration Parameter
  - 0 = Central Difference, 0.25 = Constant Average, 0.166 = Linear Average
- **[GAMA]** – Newmark Integration Parameter (0.5)
- **[DT]** – Time Step for Dynamic Analysis
- **[FOV]** – Multiplier of image stress used to extend linear part of hysterises loop. For BRB’s use 0.3, else use 0.
- **[IRINT]** – Output Interval for response time histories on unit 8
  - 1 means every step
  - 2 means every other
  - Etc.
- **[IROUT]** – Toggle to also output response time histories to unit 4
  - 1 = yes, 0 = no

- **[ISTOP]** – Time step at which current dynamic analysis ends (If empty then uses NDS)

- **Damping Options**
  - **[A0]** – Damping Parameter \( C = A0 \times M + A1 \times K \) (Assumed to be 0 when using special columns to model damping)

- **[FIRSTMODEPERIOD]** – Period of the first mode of the structure. If left blank program assumes \( T = 0.1 \times N \) where \( N \) is the number of stories

- **[DAMPINGRATIO]** – Stiffness Proportional Rayleigh Damping Value. Used to calculate \( A1 \) via \( A1 = 2 \times C_{si \_k} / w_1 \) where \( w_1 \) above (Assumed to be 0.005 when using special columns to model damping)

- **[UnmodeledForceCombo]** – Maximum pushover base shear

- **[SpringYieldDrift]** – ETABS Combination giving axial loads of unmodeled frames for the calculation of p-delta forces

- **[SpringPercent]** – Percent of maximum pushover base shear taken by unmodeled frames as a decimal

- **[SpringPolynomial]** – Polynomial describing change in capping force over the height of the structure. In the form \([SpringPolynomial] a1 a2 a3 ... an\)

- **[DamperYieldVelocity]** – Velocity at which the dampers yield

- **[DamperPercent]** – Percent of maximum pushover base shear the damping forces cap
o **[DamperPolynomial]** – Polynomial Describing change in capping force over the height of the structure. In the form $[\text{DamperPolynomial}] a_1 a_2 a_3 \ldots a_n$

- **Diaphragm Options**
  - **[ALPHACDEF]** - Default diaphragm stiffness
  - **[ALPHAC]** – Override diaphragm stiffness for a particular elevation. Input of the form
    - **[ALPHAC]** $z$ alphac
      - Where $z$ is the ETABS $z$ coordinate of the desired floor and alphac
  - $\text{alphac}$ is the diaphragm stiffness

- **Convergence Options**
  - **[MIG]** – Maximum number of global iterations (default of 20)
  - **[TOL1]** – Force tolerance for global iterations (default of 0.2)
  - **[TOL3]** – Moment tolerance for global iterations (default of 0.2)
  - **[TOL5]** – Force tolerance for local iterations (default of 2.0)
  - **[TOL7]** – Moment tolerance for local iterations (default of 1.0)

- **Vertical Constraint Options**
  - **[ALPHAVC]** – Specific stiffness for vertical connection elements
    - Input of the form $[\text{ALPHAVC}] (x, y, z)$ alphavc
      - Where $(x, y, z)$ are the ETABS coordinates of the node where the property should be applied
      - Alphavc is the vertical connection stiffness to be applied to nodes which occupy the coordinates given
• Fiber Options
  o [ALPHAVCDEF] – Default stiffness for vertical connection elements
    ▪ For more information on recommended values see Section 1.5.2

• Fiber Options
  o [EEC] – Axial Load Eccentricity factor for braces
    ▪ For more information see Section 1.5.16
  o [NSEFBC] – Number of fiber segments for beams or columns (use 8)
  o [NSEFBR] – Number of fiber segments for braces (use 7)
  o [MILF] – Maximum number of element iterations (use 20)

• Load Options
  o [LOADCOMBO] – Name of ETABS load combination to use for loads on STEEL model
    ▪ Do not use combinations of combinations
  o [MASSCOMBO] – Name of ETABS load combination to use for mass on STEEL model
    ▪ Do not use combinations of combinations

• Extra Response Time Histories
  o [PlotAll] – Toggle to automatically output all nodes’ X and Y displacement for all time steps
    ▪ 1 = yes, 0 = no
  o [PlotSecondary] – Toggle to output secondary nodes
    ▪ 1 = yes, 0 = no
If enabled, SteelConverter will search through secondary nodes to find any nodes that occupy same coordinates as any nodal response time history requested.

- **[ExTH]** – Request specific response time history to be given

  For examples see attached sample SteelConverter configuration file in Appendix B.

  Input of the form [ExTH] (x1, y1, z1) (x2, y2, z2) OutputType OutputValue

  - Where:
    - (x1, y1, z1) are the ETABS coordinates of the first node for the time history (required)
    - (x2, y2, z2) are the ETABS coordinates of the second node for the time history (required for element based output)
    - OutputType:
      - 1 = Nodal Response History
        - OutputValue:
          - 1 = STEEL X direction
          - 2 = STEEL Y direction
          - 3 = Beam rotation
          - 4 = Column rotation
      - 2 = Panel Zone History
        - OutputValue:
          - 1 = Panel Zone Moment
• **OutputValue:**
  - 1 = Moment at Node 1
  - 2 = Moment at Node 2
  - 3 = Plastic Rotation at Node 1
  - 4 = Plastic Rotation at Node 2
  - 5 = Axial Force in Element
  - 6 = Plastic Axial Displacement in Element

• **Material Models**
  - **[SteelMat]** – Shear Modulus of Steel and Shear Yield Stress of Steel in the form of
    - [SteelMat] G Tauy
  - **[DefWallShearMod]** – Default shear modulus to use for Basement Wall Elements
  - **[NumMaterial]** – Number of STEEL material models (must be 2)
  - **[MAT]** – STEEL steel Material definition
    - Input of the form: [MAT] E ES SIGY SIGU EPSU PRAT RES
      - E = Young’s modulus for material I for beam/column/brace elements
      - ES = Initial strain hardening modulus for material I for beam/column/brace elements
• SIGY = Yield stress for material I for beam/column/brace elements
• SIGU = Ultimate stress material I for beam/column/brace elements
• EPSS = Strain at onset of strain hardening material I for beam/column/brace elements
• EPSU = Strain at peak stress material I for beam/column/brace elements
• PRAT = Poisson’s ratio material I for beam/column/brace elements
• RES = Residual stress material I for beam/column/brace elements

  ▪ For more information see Section 1.2.10

  o [ConcreteMat] – STEEL concrete material definition

    ▪ Input of the form: [ConcreteMat] MODULUS YieldStrength ConcreteStrPerc

      ▪ MODULATION: Young’s Modulus of the Concrete Material
      ▪ YieldStrength: Yield Strength of the Concrete Material
      ▪ ConcreteStrPerc: Percentage of the concrete strength that leads to tension failure of the concrete

  o [MATERIALCONV] – Conversion information between ETABS materials and STEEL materials

    ▪ Input of the form: [MATERIALCONV] ETABS_Name STEEL_Material_Number
• Conversions must be given for every material used.
• For examples see attached sample SteelConverter configuration file in Section 1.3.

• **Foundation Nodes**
  - [DefFndNode] – Default properties for foundation node springs.
    - Input of the form:
      - [DefFndNode] ALP STRH STRVU STRVD
        - ALP: Post-Yield Stiffness Ratio for Foundation Springs
        - STRH: Yield Strength of Horizontal Spring
        - STRVU: Yield Strength of Vertical Spring in Upward Direction
        - STRVD: Yield Strength of Vertical Spring in Downward Direction
  - [FndNode] – Specific foundation node spring definition.
    - Input of the form:
      - [FndNode] Name ALP STRH STRVU STRVD
        - Name: Name of foundation node type (must match name of spring type in ETABS)
        - STRH: Yield Strength of Horizontal Spring
        - STRVU: Yield Strength of Vertical Spring in Upward Direction
• STRVD: Yield Strength of Vertical Spring in Downward Direction

• **IPC, FRAC segment lengths Beam/Col Elements**
  
  o **[FRAC-BC]** – Segment lengths for Beam and Column element inputs.
    
    ▪ Input of the form:
      
      * [FRAC-BC] val1 len1
      * [FRAC-BC] val2 len2
      
    ▪ Final row must be: [FRAC-BC] 0 0.
    
    ▪ Default input:
      
      * [FRAC-BC] 1 0.03
      * [FRAC-BC] 1 0.06
      * [FRAC-BC] 1 0.16
      * [FRAC-BC] 2 0.25
      * [FRAC-BC] 1 0.16
      * [FRAC-BC] 1 0.06
      * [FRAC-BC] 1 0.03
      * [FRAC-BC] 0 0

  o **[FRAC-BR]** – Segment lengths for Brace elements.
    
    ▪ Input of the form:
      
      * [FRAC-BR] val1 len1
      * [FRAC-BR] val2 len2
      
    ▪ Final row must be: [FRAC-BR] 0 0
Default Input:

- [FRAC-BR] 1 0.25
- [FRAC-BR] 1 0.16
- [FRAC-BR] 1 0.07
- [FRAC-BR] 1 0.04
- [FRAC-BR] 1 0.07
- [FRAC-BR] 1 0.16
- [FRAC-BR] 1 0.25
- [FRAC-BR] 0 0

- **Ground Acceleration Multiplier**
  - [GAMULT] – Scale factor used for ground acceleration input.
1.4 Post Processing Tools

There are currently a limited number of post processing tools that can be used to help visualize the results. As time goes on, more post processing tools will be made and will be updated here.

1.4.1 LoadData

LoadData is a Matlab script that parses through the primary STEEL output file, for004 which is created during the analysis process. The load data script then stores all relevant data in a saved Matlab workspace so other functions can quickly use the data.

```matlab
% This file must be run first. It loads the for004 file and parses all of % the information. Temporary files are stored in the working directory. warning off clear clc

workdir = '/Users/Chris/Desktop/rwg-shakeout1.2.0-sk0001'; % Path to for004

% Remove existing files
delete([workdir,'/ModelInfo.mat']);
delete([workdir,'/BEAM.mat']);
delete([workdir,'/COORD.mat']);
delete([workdir,'/BEL.mat']);
delete([workdir,'/THINFO.mat']);
delete([workdir,'/FndNode.mat']);

% Parse NNP, NEL, NBEL, NNPFN
unix(['awk ''/NNP =/ {print $NF}'' ', workdir,'/for004 > ',workdir,'/junk']); % Get the last field of which contains 'NEL';
NNP=load([workdir,'/junk']);
unix(['awk ''/ NEL =/ {print $NF}'' ', workdir,'/for004 > ',workdir,'/junk']); % Get the last field of which contains 'NEL';
NEL=load([workdir,'/junk']);
unix(['awk ''/ NBEL =/ {print $NF}'' ', workdir,'/for004 > ',workdir,'/junk']); % Get the last field of which contains 'NEL';
NBEL=load([workdir,'/junk']);
unix(['awk ''/ NNPFN =/ {print $NF}'' ', workdir,'/for004 > ',workdir,'/junk']); % Get the last field of which contains 'NEL';
NNPFN=load([workdir,'/junk']);
```
%Parse original coordinates, elements, basement elements, foundation nodes
unix(['grep -n 'NODE      XCOOR      YCOOR' ','workdir,'/for004 | cut -f1 -d: > ','workdir,'/junk']);
title_line=load([workdir,'/junk']);
unix(['sed -n ','num2str(title_line+1),','num2str(title_line+1+NHP),'p ','workdir,'/for004 > ','workdir,'/COORD']);
COORD=load([workdir,'/COORD']);

unix(['grep -n 'ELEM MEM      MAT     ISS' ','workdir,'/for004 | cut -f1 -d: > ','workdir,'/junk']);
title_line=load([workdir,'/junk']);
unix(['sed -n ','num2str(title_line+1),','num2str(title_line+1+NEL),'p ','workdir,'/for004 > ','workdir,'/BEAM']);
BEAM=load([workdir,'/BEAM']);

unix(['grep -n 'BASEMENT ELEMENT INFORMATION' ','workdir,'/for004 | cut -f1 -d: > ','workdir,'/junk']);
title_line=load([workdir,'/junk']);
unix(['sed -n ','num2str(title_line+2),','num2str(title_line+2+NBEL),'p ','workdir,'/for004 > ','workdir,'/BEL']);
BEL=load([workdir,'/BEL']);

unix(['grep -n ' FOUNDATION ELEMENT INFORMATION' ','workdir,'/for004 | cut -f1 -d: > ','workdir,'/junk']);
title_line=load([workdir,'/junk']);
unix(['sed -n ','num2str(title_line+2),','num2str(title_line+2+NNPFN),'p ','workdir,'/for004 > ','workdir,'/FndNode']);
FndNode=load([workdir,'/FndNode']);

%Clean up directory
delete([workdir,'/BEAM']);
delete([workdir,'/COORD']);
delete([workdir,'/BEL']);
delete([workdir,'/THINFO']);
delete([workdir,'/FndNode']);
delete([workdir,'/junk']);

%Save all information to workdir
save([pwd,'/workdir'],

save([workdir,'/ModelInfo'],

1.4.2 Plot Undeformed Shape

Plot Undeformed Shape is a Matlab script that takes the information from LoadData and plots the undeformed shape from the element connectivity and nodal information. It is possible to limit what coordinates are shown and toggles are available to display node and element numbers as well as restraints, springs, and basement wall elements.
% This file takes the information from Load Model and plots the undeformed configuration. There are options to show node labels, element labels, and restraints. Additionally, a [min max] range can be specified for the x dimension to limit which gridlines get plotted.

clear
clc

% Toggles
nodeLabels = true;
eleLabels = false;
restLabels = true;
springLabels = true;

xLimits = [0 9999999]; % Used to limit what gets plotted

% Load Information
data = load(fullfile(pwd, '/workdir.mat'), 'workdir');
workdir = data.workdir;
% workdir = importdata(fullfile(pwd, '/workdir.mat'));

data = load(fullfile(workdir, '/ModelInfo.mat'));
NNP = data.NNP;
NEL = data.NEL;
NBEL = data.NBEL;
NNPFN = data.NNPFN;
COORD = data.COORD;
BEAM = data.BEAM;
BEL = data.BEL;
FndNode = data.FndNode;

H=figure(1); clf();

% Plot Basement Wall Elements
[m n]=size(BEL);
for i=1:m
    nodes=BEL(i,[6 7 9 8]);
    coordt=COORD(nodes,2:3);
    hold on
    % Check to make sure coordt is within xLimits
    if (coordt(1,1) >= xLimits(1) && coordt(2,1) <= xLimits(2))
        patch(coordt(1,1),coordt(1,2),[1 1 1]*0.8)
    end
end
% Plot Elements
for i=1:NEL;

    beamID = find(BEAM(:,1)==i);
    nodes=BEAM(beamID,7:8);

    coordID(1) = find(COORD(:,1)==nodes(1));
    coordID(2) = find(COORD(:,1)==nodes(2));

    coor=COORD(coordID,2:3);

%Check to make sure coor is within xLimits
if (coor(1,1) >= xLimits(1) && coor(2,1) <= xLimits(2))
    hold on
    plot(coor(:,1),coor(:,2), 'Color','k', 'LineWidth',1.2)
%
%Print Element Number
if (eleLabels)
    avgX = (coor(1,1)+coor(2,1))/2;
    avgY = (coor(1,2)+coor(2,2))/2;

    shiftX = 0;
    shiftY = 0;
    if (coor(1,1) == coor(2,1))
        shiftX = 50;
        shiftY = 20;
    else
        shiftX = -10;
        shiftY = 30;
    end

    text(avgX+shiftX,avgY+shiftY, num2str(i))
end
end
end

% Cycle through all nodes
% Loop through all nodes
[x,y,z] = cylinder(25,50);
for i=1:NNP
% Check to make sure NNP is in the appropriate range
if (COORD(i,2) >= xLimits(1) && COORD(i,2) <= xLimits(2))
  if (nodeLabels == 1)
% Print Node Number
    text(COORD(i,2)+10, COORD(i,3)+20, num2str(COORD(i,1)), 'Color','r')
  end
end
%Check if Restraint Toggle is on
if (restLabels == 1)
    if (COORD(i, 4) == 0 && COORD(i, 5) == 0) %Fixed
        line([COORD(i, 2) - 50 COORD(i, 2) + 50], [COORD(i, 3) COORD(i, 3)])
        line([COORD(i, 2) - 100 COORD(i, 2)], [COORD(i, 3) - 50 COORD(i, 3)])
        line([COORD(i, 2) + 50 COORD(i, 2)], [COORD(i, 3) - 50 COORD(i, 3)])
    elseif (COORD(i, 4) == 0 && COORD(i, 5) == 1) %Vertical Roller
        line([COORD(i, 2) COORD(i, 2)], [COORD(i, 3) + 50 COORD(i, 3) - 50])
        plot((COORD(i, 2) + x - 25)',(COORD(i, 3) + y - 25)', 'b')
        plot((COORD(i, 2) + x - 25)',(COORD(i, 3) + y + 25)', 'b')
    elseif (COORD(i, 4) == 1 && COORD(i, 5) == 0) %Horizontal Roller
        line([COORD(i, 2) + 50 COORD(i, 2) - 50], [COORD(i, 3) COORD(i, 3)])
        plot((COORD(i, 2) + x - 25)',(COORD(i, 3) + y - 25)', 'b')
        plot((COORD(i, 2) + x + 25)',(COORD(i, 3) + y - 25)', 'b')
    end
end
end

%Check if Spring Toggle is on
if (springLabels == 1)
    %Check if Node exists in FndNode
    if (NNPFN ~= 0)
        row = find(FndNode(:,1) == i);
        if (size(row,1) ~= 0) %Then Node has a spring Property
            if (FndNode(row,2) ~= 0) %Then Has Horizontal Spring
                line([COORD(i, 2) COORD(i, 2) - 400], [COORD(i, 3) COORD(i, 3) + 200])
                line([COORD(i, 2) - 800 COORD(i, 2) - 1200], [COORD(i, 3) - 200 COORD(i, 3)])
                line([COORD(i, 2) - 1200 COORD(i, 2) - 1300], [COORD(i, 3) - 500 COORD(i, 3)])
                line([COORD(i, 2) - 1200 COORD(i, 2) - 1300], [COORD(i, 3) + 50 COORD(i, 3)])
            end
            if (FndNode(row,3) ~= 0) %Then Has Vertical Spring
                line([COORD(i, 2) COORD(i, 2) + 200], [COORD(i, 3) COORD(i, 3) - 400])
                line([COORD(i, 2) + 200 COORD(i, 2) - 200], [COORD(i, 3) - 400 COORD(i, 3) - 800])
                line([COORD(i, 2) - 200 COORD(i, 2)], [COORD(i, 3) - 800 COORD(i, 3) - 1200])
                line([COORD(i, 2) - 50 COORD(i, 2) + 50], [COORD(i, 3) - 1200 COORD(i, 3) - 1200])
                line([COORD(i, 2) - 50 COORD(i, 2) - 100], [COORD(i, 3) - 1200 COORD(i, 3) - 1300])
                line([COORD(i, 2) COORD(i, 2) - 50], [COORD(i, 3) - 1200 COORD(i, 3) - 1300])
                line([COORD(i, 2) + 50 COORD(i, 2)], [COORD(i, 3) - 1200 COORD(i, 3) - 1300])
            end
        end
    end
end
end

set(gcf, 'PaperUnits', 'centimeters')
1.4.3 Plot Dynamic Analysis

Plot Dynamic Analysis is a Matlab script that takes the information from LoadData and plots the time history information in the STEEL output file for008. The function then goes through each time step and saves an image file to a specified location that can then be made into a movie file.

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%     Plot Dynamic Analysis     %%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%This file will plot a series of deformations from the dynamic analysis
%This file will plot a series of deformations from the dynamic analysis
clear
clc;
SF = 50.0;
xLimits = [0 99999999];
data = load(pwd, '/workdir.mat', 'workdir');
workdir = data.workdir;
savedir = '/Users/Chris/Desktop/rwg-shakeout1.2.0-sk0001/Movie';

%Load Model Info
data = load([workdir, '/ModelInfo.mat']);
NNP = data.NNP;
NEL = data.NEL;
NBEL = data.NBEL;
NNPFN = data.NNPFN;
COORD = data.COORD;
BEAM = data.BEAM;
BEL = data.BEL;
FndNode = data.FndNode;

%Get the output interval
```
% Get the last field of which contains 'NRTH';
IRINT=load([workdir,'/junk']);

% Get the number of response time histories
unix(['awk''NRTH=/ {print $NF}'' ','workdir,'/for004 > ',workdir,'/junk']); % Get the last field of which contains 'NRTH';
NRTH=load([workdir,'/junk']);

% Parse the Time History Information
unix(['grep-n-RNRTH(1)(2)(3)(4)(5)(6) ','workdir,'/for004 | cut -f1 -d: > ',workdir,'/junk']);
title_line=load([workdir,'/junk']);
unix(['sed-n',num2str(title_line+1),',',num2str(title_line+1+NRTH),',p ','workdir,'/for004 > ',workdir,'/THInfo']);
THInfo=load([workdir,'/THInfo']);

% Loop through and create a lookup table between Time History Number and Node Number
THLookup = zeros(NNP,2);
for (i=1:NRTH)
    THLookup(THInfo(i,2),THInfo(i,3)) = THInfo(i,1);
end

% Read in Timehistory data
TH_Def = load([workdir,'/for008']);

% Go through each ground motion
figure(2); clf();
set(gcf,'PaperUnits','centimeters')
xB = 30; yB = 5;
xBLeft = (21-xB)/2; yBTop = (30-yB)/2;
set(gcf,'PaperPosition',[xBLeft yBTop xB yB]);
set(gcf,'Position',[300 600 xB*50 yB*50])
axis([9.9e4,3e5,-100,1200])

for (i = 1:size(TH_Def,1))
    disp(['Frame: ' num2str(i)])
    DefShape = zeros(NRTH,1);

    for (j=1:NRTH)
        DefShape(j) = TH_Def(i,j+1)*SF + COORD(THInfo(j,2), THInfo(j,3)+1);
    end
```matlab
% Plot the frame
clf();
for j=1:NEL

    nodes = BEAM(j,7:8); % Connectivity of element
    respRow = THLookup(nodes,:); % Which time history response are the nodes

    % Only try to plot if respRow Exists
    if (respRow ~= 0)
        coords = DefShape(respRow); % Get Coordinates

        hold on
        h = plot(coords(:,1),coords(:,2),'Color', 'k','LineWidth',1); % Plot

    end
end
axis equal

% Save Plot
saveas(h, [savedir '/Movie_'.sprintf('%05d',i)].'png');
end
```
1.5 Commentary

In this section the assumptions, reasoning, and mathematics as well as the inner-workings of SteelConverter are discussed to give the user a better understanding of how both the conversion software and STEEL operate thereby allowing for fewer errors and user modification.

1.5.1 Diaphragms

Diaphragms in STEEL behave slightly differently than in ETABS. In ETABS when a rigid diaphragm is assigned to a set of nodes the solver enforces horizontal compatibility between all nodes on the diaphragm. In STEEL, diaphragms act to enforce horizontal compatibility between the nodes on given frames via the penalty element method where the penalty is the inputted diaphragm stiffness. An image showing how diaphragms constrain nodes can be seen in Figure 1-8.

Figure 1-8: STEEL Diaphragm Depiction
This image shows that the first diaphragm will work to enforce compatibility between nodes on the leftmost frame (nodes 7 through 11) and middle frame (nodes 25 through 29) while the second diaphragm will work to enforce compatibility between nodes on the middle frame (nodes 25 through 29) and the rightmost frame (nodes 45 through 49). The stiffness of this diaphragm, defined by ALPHAC, is constant among all diaphragms in the model and should be given a value representative of the shear stiffness of the slab and decking system between connected frames. For an analysis conduct in the units of kN, m a stiffness of 6.9E8 would represent an “infinitely” stiff diaphragm.

The important difference between the behavior of ETABS and STEEL diaphragms is that STEEL diaphragms will allow for strain between nodes in a given frame while ETABS rigid diaphragms will not. The STEEL diaphragm will take the average of the nodal displacements each connected frame and will apply a constraining equation to the stiffness matrix according to the given weighting function. If an extremely large ALPHAC value is given then the average displacement between the two connected frames will be identical.

SteelConverter creates diaphragms automatically by searching for nodes that lie on the intersection of both primary and secondary gridlines. Since the number of nodes on each frame of a diaphragm needs to be constant throughout the model in order for STEEL to run, SteelConverter will search through the model, determine the maximum number of applicable nodes and ensure that all other frames have an equal number of nodes on the diaphragm by repeating the last node until the number is reached. Nodes that land in between gridlines are ignored, allowing for changing in bracing configuration along the height of the building.
It is also of note that diaphragms are only created in the primary direction. For more information on how STEEL parses diaphragms please see the description of the primary steel input file, for001 in Section 1.6.1.

1.5.2 Vertical Connection Elements

SteelConverter has the ability to convert the 3D ETABS models to a “2.5D” Steel model that carries vertical compatibility between nodes on intersecting frames. This is achieved through STEEL’s vertical connection element that acts like a spring between nodes carrying only axial load. A visualization of the way SteelConverter rearranges a 3D ETABS model can be seen in Figure 1-9 and Figure 1-10.
When SteelConverter is creating the secondary brace lines a new set of nodes are created which occupy the same ETABS coordinates as those in the primary direction. SteelConverter finds these secondary nodes and automatically creates a vertical connection element constraining them to their original, primary, node. The arrows in Figure 1-9 and Figure 1-10 represent these connections. These vertical connection elements each can be assigned a stiffness that will adjust how strictly the vertical displacement compatibility between the two attached nodes is enforced.

It important for the user to realize that these elements do not allow the passing of anything other than vertical forces via a linear spring with a given stiffness and as a result will fail to capture 3D affects such as torsion or biaxial bending. However, for symmetric structures loading uniformly the results of these constraints have been shown to provide accurate results.

To provide an example stiffness, for analyses in the units of kN, m a vertical connection element stiffness of 6.9e8 kN/m would adequately represent an “infinitely” stiff vertical connection.
1.5.3 Modeling of Secondary Frames

In order to obtain the proper mass in the STEEL model for purposes of dynamic analysis, the columns are generally only placed in the primary frames. Therefore, to ensure stability in the model, the leftmost node of every floor in every secondary frame is restrained with a vertical roller as shown in Figure 1-11. This prevents the secondary frames from translating horizontally while still allowing them to deflect vertically. Additionally, since this restraint is placed only on one side of the secondary frame, all other nodes will be able to strain horizontally. While this may be a source of computational error, the secondary frames do not strain horizontally significantly when the structure is loaded symmetrically in the primary direction meaning any error associated with this assumption will be minimal.

![Figure 1-11: Extra Restraints placed on Secondary Frames](image)

1.5.4 Rayleigh Damping

The creator of STEEL, Professor John Hall, presented a paper in which he describes some possible unintended consequences of using Rayleigh damping in large displacement non-linear
dynamic models such as excessive energy dissipation during hinge loading and unloading [17]. As a result, Rayleigh damping is used only to ensure entries in the stiffness matrix are non-zero to allow for better computational convergence. It is common practice in STEEL to instead implement damping via special columns, which is discussed in Section 1.5.6.

When editing the SteelConverter configuration file the mass proportional damping multiplier A0 is given a value of 0 and the stiffness proportional damping multiplier A1 is given a value of $\frac{2\xi_1}{\omega_1}$ where $\xi_1$ is a small value such as 0.005 and $\omega_1$ is the fundamental frequency of the building in rad/s.

1.5.5 Releases

Since STEEL uses fiber based elements the creation of pinned connections is not as straightforward as in ETABS. To simulate the release of moments at the end of elements special fiber properties need to be assigned. STEEL accomplishes this through the use of fiber area categories in which specific fibers in specific sections of an element can have their areas increased or decreased by a certain percentage. An image describing the number of fibers per element for beams, columns and braces can be seen in Figure 1-12, while an image describing the default segment breakdown for beams, columns and braces can be seen in Figure 1-13.
Currently, when creating a fixed-fixed connection on a beam element the web of the first two and last two segments are reduced to 30% of their original area to better correspond with empirical data. More information on this can be found in [1].

Creating a pinned connection in STEEL is slightly more complicated. The fiber modifications need to minimize the inertia of the section as much as possible while still allowing the section to generate its full capacity. The inertia is reduced by eliminating the flanges and the top and bottom fibers of the web while the capacity of the section is preserved by increasing the area of the middle two fibers of the web. For example, if a beam had its left end pinned and its right end fixed segments 1 and 2 fibers 1, 2, 7 and 8 would have an area modifier
of 0 to eliminate the flanges, fibers 3 and 6 would have an area modifier of 0 to eliminate the top and bottom fibers of the web, and fibers 4 and 5 would have their area modifier set to a value such that the axial capacity of the section remains roughly constant.

While it would be possible to have exact modifiers for every possible section, the increase to the size of the input file was deemed to be not worthwhile as each section would require 3 premade fiber area modification categories; namely for pinned-pinned elements, pined-fixed elements, and fixed-pinned elements. Instead, only beam sections greater than 18” but less than 36” in depth were chosen as the most common beam sections and an appropriate modifier was chosen which best represented all beams in this range.

To calculate the area modifier an equivalent area was calculated by first determining the height of the web via,

\[ h_{web} = d - 2t_f \]

Where \( d \) is the depth of the beam and \( t_f \) is the thickness of the flange. Since the new modified cross-section has its flanges eliminated with all web area condensed into two equal fibers, each fiber area can be calculated as,

\[ A_{mod\_fiber} = \frac{1}{2} h_{web} t_{web} = \frac{1}{2} (d - 2t_{web}) t_f \]

Therefore, the multiplier to the original fiber area can be found to be,

\[ FAFRAC = \frac{A_{section}}{A_{mod\_fiber}} \]

where FAFRAC is the multiplier for the middle two fibers and \( A_{section} \) is the area of the original section.
Following this calculation for all reasonable beams in the desired range gave a maximum and minimum multiplier of 7.17 and 3.63, an average multiplier of 5.4 with a standard deviation of 0.9. In most sections where the actual multiplier was far from the given average the weight of the section was such that it would be more practical to increase the depth rather than use such a heavy section. Therefore, it was then chosen to assign a fiber area modifier of 6.0 to the middle two fibers of the two segments nearest a pinned connection. Since the multiplier chosen is greater than the minimum there will be a non-conservative area for some sections types, however as drag element failure is generally not a global failure mechanism of interest in lateral analysis the error should not be significant. However, if the user wishes additional area modification categories can be created to achieve a more accurate representation of pinned connections.

It was decided that beams which are fixed-pinned or pinned-fixed would be given no modifications on the fibers of the fixed end since, at this stage in the analysis, this element fixity type only occurs when the beam is meshed at the intersection point of a brace. Since there is continuity of the element over this connection reducing the area of the fibers at this location would be incorrect. However, this does mean that modeling a fixed-pinned or pinned-fixed beam that spanned between a moment frame and a brace frame would result in a non-conservative response, therefore, as of the current version the user should take care to avoid these situations and simply span the space between these types of systems with a pinned-pinned beam.
Element fiber categories for braces are done automatically and can be given a fiber modification category of 0. Similarly, all column elements are given a fiber modification category of 0.

A description of every release type available is shown in Table 1-1. Note that some of the release types are out of date and are unused, namely the column releases as it was determined that pinning columns can result in large computational errors. The user may either create their own release definitions using these as a guide by editing the for001 file or customize the current element definitions utilizing the existing element fiber area modification categories.
### Table 1-1: STEEL Element Release Definitions

<table>
<thead>
<tr>
<th>Type</th>
<th>Orientation</th>
<th>Condition</th>
<th>Category</th>
<th>Segment</th>
<th>Fibers</th>
<th>Area Modifier</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strong</td>
<td>Strong</td>
<td>Fix-Fix</td>
<td>1</td>
<td>8</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Fix</td>
<td>2</td>
<td>8</td>
<td>12 78</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fix-Pin</td>
<td>3</td>
<td>8</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>12 78</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Pin</td>
<td>4</td>
<td>8</td>
<td>12 78</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>12 78</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td>Weak</td>
<td>Weak</td>
<td>Fix-Fix</td>
<td>5</td>
<td>8</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Fix</td>
<td>6</td>
<td>8</td>
<td>123478910</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fix-Pin</td>
<td>7</td>
<td>8</td>
<td>123478910</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>123478910</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Pin</td>
<td>8</td>
<td>8</td>
<td>123478910</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>123478910</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fix-Fix</td>
<td>9</td>
<td>8</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>34 56</td>
<td>0.3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Fix</td>
<td>10</td>
<td>8</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fix-Pin</td>
<td>11</td>
<td>1</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pin-Pin</td>
<td>12</td>
<td>8</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td>Beam</td>
<td>Strong</td>
<td>Fix-Pin</td>
<td>11</td>
<td>1</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>127836</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1</td>
<td>45</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2</td>
<td>45</td>
<td>6</td>
</tr>
</tbody>
</table>

1.5.6 Damping / Special Columns

1.5.7 Special Columns

1.5.7.1 Description

Due to the linear dependence on displacement and velocity the utilization of stiffness and mass proportional damping can yield unrealistically large damping forces at high velocities [1]
and therefore a “capped” damping force was implemented in STEEL utilizing elasto-plastic dashpots with a controllable maximum force value. These elements also allow for the creation of additional springs to model stiffness and p-delta forces obtained from unmodeled columns and framing.

SteelConverter provides several input parameters to allow the user to customize the amount of damping in the structure as well as increase the level of forces to account for unmodeled frames when calculating p-delta forces. The input parameters are UnmodeledForceCombo, SpringYieldDrift, SpringPercent, SpringPolynomial, DamperYieldVelocity, DamperPercent and DamperPolynomial. For each special column element SteelConverter converts the stiffness and strength of any unmodeled framing as well as the “stiffness” and strength of the inter-floor dampers. The strength of the springs and dampers are calculated by first determining the “force” at each floor, as defined by the SpringPolynomial and DamperPolynomial inputs, then dividing the force by the number of columns on a given floor. The spring and damper stiffnesses are is then computed by dividing the spring and damper strength by the inputted SpringYieldDrift and DamperYieldVelocity respectively, namely;

\[
\text{Spring or Damper Stiffness} = \frac{\text{Story Force}}{\text{Number of Columns}} \left( \frac{1}{\text{SpringYieldDrift or DamperYieldVelocity}} \right)
\]

\[
\text{Spring or Damper Strength} = \frac{\text{Story Force}}{\text{Number of Columns}}
\]
Note that because non-rigid diaphragms distribute shear according to the relative rigidities of the various columns the assumption of equal distribution of story force among all columns in a given floor breaks down as the floor diaphragm becomes less rigid.

The force at each floor is calculated through the polynomial inputs as seen in Figure 1-14. The floor level capping force at the base is given in terms of the percent of the overall maximum base shear through the SpringPercent and DamperPercent inputs. The floor capping force at any given floor is found by using the given spring and damper polynomials at the mid story height at each floor. The input polynomial coefficients $a_1, a_2, a_3, ..., a_n$ are utilized in the form,

$$Percent \times MaxBaseShear + a_1z + a_2z^2 + a_3z^3 + \cdots a_nz^n$$

where Percent is either SpringPercent or DamperPercent, $z$ is 0 at the base of the structure. This functionality allows the user to have complete customization of the distribution of both the capped damping and spring stiffness added to the model.

![Figure 1-14 - Special Column Force Distribution](image-url)
1.5.8 Recommended Values

1.5.8.1 UnmodeledForceCombo

For this input it is recommended that the user apply ETABS vertical loads on columns to represent the gravity loads of unmodeled columns. These loads should not be applied to the actual analyses and are used solely for determining the additional p-delta force applied due to these unmodeled columns. The ETABS forces should be placed in their own load combination separate from actual analysis loads.

1.5.8.2 SpringYieldDrift

The SpringYieldDrift represents the maximum story drift before the horizontal springs yield. This value is inputted as a percentage of the story height and is a constant value throughout the height of the building. A value representing $\frac{1}{400}$ or 0.0025 is an appropriate starting value for this parameter.

1.5.8.3 SpringPercent

SpringPercent represents the percentage of the overall base shear that is resisted by secondary framing. The higher the ratio of unmodeled frames to modeled frames the larger this value should be over the height of the building. However, for a typical building with all primary framing modeled a value of roughly 0.07, 7% of the overall base shear, would be typical.

1.5.8.4 SpringPolynomial

The polynomial should be chosen such that the distribution is an accurate representation of the quantity of the total base shear anticipated to be taken by secondary framing over the height of the building. For actual structural members the distribution is expected to vary more linearly over the height of the structure while for non-structural members, such as framing or partitions, the distribution can be expected to be more uniform. Care must be taken when selecting this distribution.
1.5.8.5 DamperYieldVelocity

The damper yield velocity should be approximately chosen as the velocity at which the damping forces reach the capping force. Namely when,

\[ C \dot{x} = F_{ss,design} \]

Where \( F_{ss,design} \) is the design story shear force in story \( i \). Substituting for stiffness proportional damping and solving for the velocity yields,

\[ v_{yield} = \frac{F_{ss,design}}{a_n k} = \frac{F_{ss,design}}{2\gamma \omega k} \]

Assuming the stiffness of the structure can be written as \( \frac{2\gamma F_{ss,actual}}{StoryHeight} \), where \( F_{ss,actual} \) is the actual shear strength of story \( i \). The above equation can then be rewritten as,

\[ v_{yield} = \frac{F_{ss,design}}{2\gamma \omega \frac{F_{ss,actual}}{StoryHeight}} = \frac{(StoryHeight)(F_{ss,design})\omega}{1600(F_{ss,actual})(\gamma^2)} \]

Where \( \gamma \) is the desired damping, \( \omega \) is the first mode frequency in rad/s, and \( StoryHeight \) is the average story height of the structure. Note that \( \frac{StoryHeight}{400} \) is an assumed displacement at which a typical wind-controlled building can be expected to achieve nonlinear behavior and is an acceptable starting value. However, the user is encouraged to use different yield displacements to more accurately represent the structure they are attempting to model. For this formulation it is assumed that all dampers yield at the same velocity throughout the height of the building.
1.5.8.6 DamperPercent
The DamperPercent input is used to control the yield velocity of the dampers at the bottom of the building and is inputted as a percentage of the overall building base shear. An approximate value for the yield force of the dampers can be found to be,

\[ DamperYieldFloorCap = 2\gamma (\text{Max Pushover Base Shear}) \]

Where \( \gamma \) is the damping in the model. This can be found by examining the steady-state solution of a damped stiffness-proportional mass-spring system driven at its natural frequency. The damping coefficient for stiffness proportional damping \( a_n \) can be rewritten as,

\[ a_n = \frac{2\gamma}{\omega_n} \]

Where \( \omega_n \) is the frequency of the first mode of the structure in rad/s. Knowing that the steady-state solution will be of the form,

\[ x_{ss} = A \cos(\omega_n t - \varphi) \]

Substitution of the steady-state solution back into the equation of motion will yield the desired result. Therefore, the **DamperPercent** input should be chosen such that,

\[ DamperPercent = \frac{2\gamma (\text{Max Pushover Base Shear})}{\text{Max Pushover Base Shear}} = 2\gamma \]

Where \( \gamma \) will have a value of roughly 0.05, or 5%, for most common structures.

1.5.8.7 DamperPolynomial
It is expected that the distribution of damping throughout the building should approximately follow the distribution of strength. Therefore a linear distribution will often be appropriate.
1.5.9 Element Strong Axis / Weak Axis Orientation

ETABS and STEEL both have the ability to make any element have a strong or weak axis orientation. However, rather than import element orientation from ETABS it was decided instead to make all columns which are pinned on one end and fixed at the other weak axis, all columns which are fixed on both ends strong axis (user beware: utilizing this type of releases on columns can result in excessive vertical displacements in columns and is therefore not recommended), all braces weak axis, and all beams strong axis. This was done to reduce an order of complexity in the ETABS model, as well as provide more accurate results from the STEEL analysis.

If a column is fixed at both ends, it is assumed to be resisting a moment. Therefore, weak axis buckling is the controlling state of the element. On the other hand, if a column is pinned at one end and fixed at the other it is assumed to be functioning in a brace frame where moment capacity is not an issue. Similarly, all braces were chosen to be orientated about their weak axis since weak axis buckling will always be the controlling state of that element. Finally, strong axis orientation for beams was chosen since in practice beams are generally orientated in this manner.

If the user wishes to change the orientation of any element, simply changing the appropriate field in the for001 input file to -1 for weak axis or 1 for strong axis will cause steel to treat the element as such. Information on the for001 STEEL input file can be seen in Section 1.6.1.
1.5.10 Nodal Mass

As discussed earlier, the nodal mass is imported from ETABS via the given load combination defined in the SteelConverter configuration file. The mass in the ETABS file should be given as a vertical downward force on any node where mass is required. SteelConverter will then take the vertical mass on each node and apply it horizontally and vertically to the STEEL nodes in the Primary Direction. No nodes in the secondary frames will be given mass as the ground motions will only be applied in the primary direction and therefore the excitement of the mass in the secondary direction is minimal. Mass can be added manually by editing the for001 file as defined in Section 1.6.1.

1.5.11 Decking

As discussed earlier, SteelConverter has the ability to import slab and deck information from ETABS. However, there are currently limitations on the way the decking must be drawn in ETABS as well as limitations on how the information is imported to STEEL. In the ETABS model only one type of decking can be drawn on any given floor and it must span the entire floor.

As a result, all elements on a given floor will be given the same decking information. It is not possible to assign different types of decking on different beam elements nor is it possible to assign decking to some beam elements on a floor and no decking to other elements on the same floor. It is possible to have different types of decking properties on different floors and it is possible to have some floors with no decking.

When calculating the area for STEEL, SteelConverter uses the ACI 318 code maximum tributary length of 16*Slab Thickness and multiplies this value by Slab Thickness again to obtain
the maximum tributary area for the slabs. To be conservative, the area of the decking is assumed to be zero as, drastically different results are possible depending on the direction the decking is running.

For more information on how STEEL works with composite action see [1].

1.5.12 Units

When exporting an .e2k file from ETABS take care to record the units the file has been exported in and be sure all inputs in the SteelConverter configuration file are in matching units as both STEEL and SteelConverter have little to no automatic conversion information. Properties such as diaphragm stiffness ALPHAC and vertical connection stiffness ALPHAVC need to be scaled to achieve the desired stiffness for the given units. Inputs that are the ratio of values or strains do not need to be scaled, as they are independent of units.

1.5.13 Gravity

The STEEL for001 input parameter AGRAV is assigned automatically based on the exported units of ETABS. The units currently understood by SteelConverter are inches, feet, meters, millimeters, and centimeters, although additional units can be added with little work. Furthermore, it is possible for the user to customize AGRAV manually in the for001 file, but care must be taken to ensure consistency across all inputs.

1.5.14 Panel Zones

In STEEL panel zones exist at the intersection of columns and beams and have the option of being fixed in space, rigid, or flexible. This can be controlled by setting IDJ in the for001 input
file to either 0 for fixed, 1 for rigid, or 2 for flexible. Currently, SteelConverter sets all nodes on the lowest floor to have a fixed panel zone and all other nodes to have a rigid panel zone. There are plans for future versions to contain flexible panel zones if the need arises. Additionally, since a common bracing in the developer’s research is chevron bracing, special code was added for panel zones located where no column is present to shrink the panel zone as small as possible. This was done to assure proper alignment of braces and remove a potential “shear link” behavior that was unintended in the model.

1.5.15 Element Connectivity

STEEL requires element connectivity information to be inputted in a specific order. The required orientation for beam, column and brace elements can be seen in Figure 1-15, while the required orientation for basement wall elements can be seen in Figure 1-16.
1.5.16 Axial Load Eccentricity

In order to encourage buckling in braces STEEL has a global setting whose value offsets all forces in braces by a set number. This value, known as EEC in the SteelConverter configuration file, shifts the axial force in all brace elements away from the centerline of the element by a constant value. This is visualized in Figure 1-17. With no axial load eccentricity factor every brace in the model will be aligned perfectly geometrically resulting in no initial moment in brace elements and therefore no buckling will occur. It is therefore recommended that the user input some reasonable value for this property to accurately represent real-world conditions.

![Axial Load Eccentricity Factor](image)

Figure 1-17: Axial Load Eccentricity Factor Variable Description

1.5.17 Node Numbering

Node numbering in SteelConverter has been done in such a way as to minimize the half bandwidth of slender structures. To accomplish this, SteelConverter sweeps through all nodes
on a given floor starting with primary nodes and numbers them along the primary direction. After all nodes on a given floor have been numbered the process is repeated on each subsequent floor until every node has been numbered. The result of which is a maximum node number differential equal to roughly the maximum number of nodes on a floor. An example of the node numbering scheme for the 6 story braced frame building example can be seen in Figure 1-18. Element numbering in SteelConverter is done on a first-come first-serve basis and is determined by the order the user draws the elements in ETABS.

![Figure 1-18: Automatic Node Numbering Technique for SteelConverter](image)

Figure 1-18: Automatic Node Numbering Technique for SteelConverter
1.6 STEEL Input Files

STEEL requires several input files, some of which are created automatically by SteelConverter. for001 is the main input file. It contains the model and loading information for the analysis. for002 and for003 contain the horizontal and vertical ground motions respectively and are not created by SteelConverter. for020 and for021 contain the section and slab material databases that are created by SteelConverter. Finally, the for029 file contains the random seed that STEEL uses for probabilistic material failures and is not created by SteelConverter.

1.6.1 for001

The input format for the for001 input file will now be discussed allowing the user to make custom changes to the model if required. For an example of a for001 input file please see Section 1.5.1 and additional information about each input line can be seen in Section 1.6.

[1]. Title

- Title: Name of the project file

[2]. NNP NEL NNPFN NBEL NCONEL NNPBF NVCONEL NSS NDS NRTH MIG NC MTP

NDIM

- NNP: Number of nodal points
- NEL: Number of beam/column/brace elements
- NNPFN: Number of brace points per frame
- NBEL: Number of basement wall elements
- NCONEL: Number of connection elements between parallel frames
- NNPBF: Number of nodal points along a floor line for each frame
• NVCONEL: Number of vertical connection elements
• NSS: Number of static load steps
• NDS: Number of dynamic time steps
• NRTH: Number of response time histories
• MIG: Maximum number of global iterations
• NC: Number of special columns
• MTP: Maximum number of turning points in the hysteretic model
• NDIM: Storage parameter for turning point locations

[3]. DT BETA GAMMA A0 A1 AGRAV TOL(1) TOL(3) TOL(5) TOL(7)
• DT: Time step for dynamic analysis
• BETA: Newmark time integration parameter
• GAMMA: Newmark time integration parameter
• A0: Mass proportional damping multiplier
• A1: Stiffness proportional damping multiplier
• AGRAV: Acceleration due to gravity
• TOL(1): Force tolerance for global iterations
• TOL(3): Moment tolerance for global iterations
• TOL(5): Force tolerance for local iterations
• TOL(7): Moment tolerance for local iterations

[4]. EEC NSEFBC NSEFBR MILF
• EEC: Axial load eccentricity factor for braces
• NSEFBC: Number of segments for beams or columns
• NSEFBR: Number of segments for a brace

• MILF: Maximum number of element iterations

[5]. IRINT IROUT ISTOP

• IRINT: Output interval for response time histories on unit 8

• IROUT: Unit 4 response time history output toggle

• ISTOP: Time step at which current dynamic analysis stops

[6]. N C(N,1) C(N,2) ID(N,1) ID(N,2) IDJ(N) IDOUB(N) F(N,1) F(N,2) F(N,3) F(N,4)

[Repeated NNP times]

• N: Node number

• C(N,1): X coordinate of node N

• C(N,2): Y coordinate of node N

• ID(N,1): X dof restraint of node N

• ID(N,2): Y dof restraint of node N

• IDJ(N): Panel zone restraint of node N

• IDOUB(N): Panel zone thickness toggle

• F(N,1): X static force at node N

• F(N,2): Y static force at node N

• F(N,3): X mass at node N in units of force

• F(N,4): Y mass at node N in units of force

[7]. N MT(N) MAT(N) IOR ISS(N) ICS(N) ICT(N) LM(N,1) LM(N,2) WSCALE(N) BRMULT

FOV [Repeated NEL times]

• N: Beam/column/brace element number
• MT(N): Element Type
• MAT(N): Material set number for element N
• IOR: Element axis orientation
• ISS(N): Steel member designator for element N
• ICS(N): Slab designator for element N
• ICT(N): Category of element N for fiber area adjustment and fracture strain specification
• LM(N,1) LM(N,2): Connectivity array for element N
• WSCALE(N): Width multiplier for element N
• BRMULT: Multiplier of EI, which is then added to element stiffness
• FOV: Multiplier of image stress, which is used to extend linear part of the hysteresis loop

[8] NCL(N) PDL(N,1) PDL(N,2) PDL(N,3) PDL(N,7) PDL(N,8) [Repeated NC times]

• NCL(N): Element number of special column N
• PDL(N,1): Positive sum of gravity loads in all columns in non-modeled frames corresponding to special column N for P-Delta calculation
• PDL(N,2): Stiffness of horizontal spring connecting nodal pair
• PDL(N,3): Strength of horizontal spring connecting nodal pair
• PDL(N,7): “Stiffness” of horizontal damper connecting nodal pair
• PDL(N,8): Strength of horizontal damper connecting nodal pair

[9] LMF STFH(I) STFV(I) ALP STRH(I) STRVU(I) STRVD(I) IORB(I) [Repeated NNPFN times]
• LMF: Node number of foundation node I
• STFH(I): Stiffness of horizontal spring attached to foundation node I
• STFV(I): stiffness of vertical spring attached to foundation node I
• ALP: Post-yield stiffness ratio for foundation springs
• STRH(I): Yield strength of horizontal spring
• STRVU(I): Yield strength of vertical spring in upward direction
• STRVD(I): Yield strength of vertical spring in downward direction
• IORB(I): Orientation of wall. 0 for wall in the plane of the framing, 1 for walls perpendicular to the framing.

[10]. \(N \ HB(N) \ WB(N) \ TB(N) \ G(N) \ LMB(N,1) \ LMB(N,2) \ LMB(N,3) \ LMB(N,4)\) [Repeated NBEL times]

• N: Basement wall element number
• HB(N): Height of basement wall element N
• WB(N): Length of basement wall element N
• TB(N): Thickness of basement wall element N
• G(N): Shear modulus for basement wall element N
• LMB(N,1) LMB(N,2) LMB(N,3) LMB(N,4): Connectivity array for basement wall element N.

[11]. \(N \ (MCC1(N,J), J=1, NNPBF) \ (MCC2(N,J), J=1,NNPBF)\) [Repeated NCONEL times]

• N: Connection element number
• MCC1(N,J),J=1,NNPBF: List of NNPBF floor nodes of the first frame to be connected to the second frame by element N
• MCC2(N,J),J=2,NNPBF: List of NNPBF floor nodes of the second frame to be connected to the first frame by element N

[12]. N ALPHAVC MCVC(1) MCVC(2) ALPHAC [Repeated NVCONEL times]

• N: Vertical connection element number
• ALPHAVC: Vertical connection element stiffness
• MCVC(1): Node 1 to be vertically connected
• MCVC(2): Node 2 to be vertically connected
• ALPHAC: Diaphragm stiffness parameter

[13]. IR IDRTH(IR,1) IDRTH(IR,2) IDRTH(IR,3) IDRTH(IR,4) IDRTH(IR,5) IDRTH(IR,6) [Repeated NRTH times]

• IR: Time history number
• IDRTH(IR,1): Node number of history IR
• IDRTH(IR,2): DOF number of history IR
• IDRTH(IR,3): Node number of history IR
• IDRTH(IR,4): Panel zone response type of history IR
• IDRTH(IR,5): Beam/column/brace element of history IR
• IDRTH(IR,6): Beam/column/brace element response type of history IR

[14]. GPZ TAUY

• GPZ: Shear modulus for panel zones
• TAUY: Shear yield stress for panel zones

[15]. E(I) ES(I) SIGY(I) SIGU(I) EPSS(I) EPSU(I) PRAT(I) RES(I) [Repeated 2 times]

• E(I): Young’s modulus for material I for beam/column/brace elements
• ES(I): Initial strain hardening modulus for material I for beam/column/brace elements

• SIGY(I): Yield stress for material I for beam/column/brace elements

• SIGU(I): Ultimate stress for material I for beam/column/brace elements

• SIGX(I): Residual stress for material I for beam/column/brace elements

• EPSS(I): Strain at onset of strain hardening for material I for beam/column/brace elements

• EPSU(I): Strain at peak stress for material I for beam/column/brace elements

• PRAT(I): Poisson’s ratio for material I for beam/column/brace elements

• RES(I): Residual stress for material I for beam/column/brace elements

[16]. E(I) ES(I) SIGY(I) SIGU(I) SIGX(I) EPSU(I) FYFRAC(I)

• E(I): Young’s modulus for concrete material

• ES(I): Initial strain hardening modulus for concrete material

• SIGY(I): Yield stress for concrete material

• SIGU(I): Ultimate stress for concrete material

• SIGX(I): Residual stress for concrete material

• EPSU(I): Ultimate stress for concrete material

• FYFRAC(I): Fiber fracture strain has a fraction of yield strain

[17]. ICAT ISENO IFNO FAFRAC

• ICAT: Element fiber area modification category (shared with fiber strain modification category)
- **ISEN**: Segment affected
- **IFNO**: Fibers in segment affected (filled with 0’s until 10 entries long)
- **FAFRAC**: Percentage of area to adjust given fibers by. (e.g. 0.3 reduces selected fibers’ area to 30% of full value.)

**[18]. ICAT ISEN0 IFNO FYFRAC**

- **ICAT**: Element fiber strain modification category (shared with fiber area modification category)
- **ISENO**: Segment affected
- **IFNO**: Fibers in segment affected (filled with 0’s until 10 entries long)
- **FYFRAC**: Percentage of ultimate strain fiber will probabilistically reach. 10 numbers long, each number in list is given a 10% chance of occurring. (e.g. 1 1 1 10 10 100 100 100 150 150 gives elements assigned a 30% chance of only reaching 1% of ultimate strain, a 20% chance of only reaching 10% of ultimate strain, 30% chance of reaching 100% of ultimate strain and a 20% chance of reaching 150% chance of ultimate strain).

**[19]. IPC FRAC**

- **IPC**: Number of segments for beam/column elements
- **FRAC**: Fraction of segment length for beam/column elements

**[20]. IPC FRAC**

- **IPC**: Number of segments for brace elements
- **FRAC**: Fraction of segment length of brace elements

**[21]. GAMULT**
- GAMULT: Ground motion multiplier

1.6.2 for002

Contains the horizontal ground motion information to be run.

1.6.3 for003

Contains the vertical ground motion information to be run.

1.6.4 for020

Contains the cross-section dimensions for steel members. Input of the form:

- ISSX D TW B TF

Definitions can be seen in Figure 1-19, and where ISSX is the STEEL member designator.

1.6.5 for021

Contains the cross-section for slabs. Input of the form:

- ICSX, ADECK, DDECK, ASLAB, DSLAB

Definitions can be Figure 1-19.
And where ICSX is the STEEL slab designator and the user should always use 111 for columns or for elements with no slab.

1.6.6 for029

Integer seed value in * format for fiber element failure randomization. If using PBS run files, this seed is randomly generated based off of processor clock at the time of analysis.
1.7 Running STEEL on a PBS Server

In the work of many researchers at Caltech it is necessary to subject a model to several thousand ground-motions. Fortunately Caltech has access to several computer clusters on campus that allow STEEL analyses to be run on multiple processors simultaneously thereby greatly decreasing the overall runtime of the analysis. As a result, several scripts were developed to expedite this process and will now be discussed to assist the user in conducting similar types of analyses.

1.7.1 Directory Setup

The server scripts made require that the model directory be made in a specific manner. All analysis directions must contain the two server scripts (run.sh and client.sh), an input folder containing the three input files (for001, for020, and for021), and a file containing the list of ground motions the user wishes to run (ShakeOut_Files). The path to these locations should be noted, as some of them will be needed in modifications to the server scripts. Additionally, the location of the folder containing the ground motions and STEEL executable should be recorded as well.

1.7.2 Server Scripts

Two scripts were created to run the series of ground motions on the computer cluster at Caltech. Run.sh is executed on the head node and its main purpose is to clean up the working directory, setup the locations of the ground motions and version of STEEL to execute, and pass execution of the job to each processor node made available to it. Prior to executing run.sh the
user should modify the name of the ground acceleration folder containing the ground motions to run (GACC) and the version of steel to execute (STEEL_VER). If the user wishes to only run a particular set of motions without deleting existing results, commenting out the line “rm –rf $WORKDIR/output/*” will cause the script to not deleting existing results.

The second script, client.sh gets executed on each node made available to the process and is responsible for actually running the analysis. This script first attempts to get a ground motion to run from the list (it will do nothing if none are left), then it copies the ground motion, input files, and STEEL executable into proper position and executes the analysis. Upon completing the script compressing all the files and copies them back to the original work directory. Prior to executing the analysis the user should modify the scratch folder on the cluster (SCRATCH), the path to the ground acceleration information (GACC), and the path to the STEEL executable (STEEL). It should be noted that client.sh also is setup to append each ground motion run with hundreds of 0 entries to allow the motion to dampen out before the analysis completes.

Explanation for this can be seen in [9].

1.7.2.1 Run.sh

```bash
#!/bin/bash
WORKDIR=$PBS_O_WORKDIR
GACC=ShakeOut_GACC_DT_0.005
BLDG=`basename $WORKDIR`
STEEL_VER=steel-v1_6

echo $BLDG
echo $WORKDIR

declare -a hosts

cat $PBS_NODEFILE > $WORKDIR/hosts
HOSTS=`cat $WORKDIR/hosts`
echo $HOSTS >$WORKDIR/h

cat $WORKDIR/hosts/h

read -a hosts < $WORKDIR/h
element_count=${#hosts[@]}
```
echo $element_count

rm -f $WORKDIR/*.lock

# Uncomment this line to clear all analysis results before starting. client.sh will only run ground motion if the output folder for that ground motion doesn't exist
rm -rf $WORKDIR/output/*
mkdir -p $WORKDIR/output

sleep 20

# If you want to run every ground motion uncomment this line. It will refresh jobfile with every groundmotion in ShakeOut_files. Otherwise fill jobfile with the motions you want to run

cp $WORKDIR/ShakeOut_files $WORKDIR/jobfile

index=0
while [ "$index" -lt $element_count ]
  do
    ssh -XY ${hosts[$index]} $WORKDIR/./client.sh $BLDG $GACC $WORKDIR $STEEL_VER&
    echo running $index on ${hosts[$index]}
    let "index = $index + 1"
    sleep 0.1
  done
wait

1.7.2.2 Client.sh

#!/bin/bash

# Takes three arguments
# 1) Name of the folder containing model information
# 2) Name of the ground acceleration dataset to run
# 3) Path to Working Directory
# 4) Name of the version of steel to run

BLDG=$1
COMMON=$3
STEEL_NAME=$4
SCRATCH=/scratch/cjanover/Models/Pushover/$BLDG
DATADIR=/home/cjanover/Steel/GACC/$2
STEEL=/home/cjanover/Steel/Steel_Software/Compiled/$STEEL_NAME

declare -a jobs

JOBFILE=$COMMON/jobfile
nremjobs=1

while [ "$nremjobs" -gt 0 ]
  do

lockfile -r -l $JOBFILE.lock
jobid=`head -1 $JOBFILE`
T=`wc -l $JOBFILE | awk '{print $1}'`
let "T = $T -1"
tail -$T $JOBFILE > $JOBFILE.new
mv $JOBFILE.new $JOBFILE
rm -f $JOBFILE.lock

if [ ! -d $COMMON/output/$jobid ]; then
  mkdir -p $SCRATCH/$jobid
  cp -r $COMMON/input/* $SCRATCH/$jobid
  cp $DATADIR/$jobid.bz2.tar $SCRATCH/$jobid
  cp $STEEL $SCRATCH/$jobid

cd $SCRATCH/$jobid
tar -xj $jobid.bz2.tar
rm $jobid.bz2.tar

NP1=`wc for090 | awk '{print $2}'`
NP3=`wc zeros | awk '{print $2}'`
let "NP2 = $NP1 + $NP3"

sed 's/$NP1/'"$NP2"'/g' <for090 > junk
cat junk zeros > for002

sed 's/$NP1/'"$NP2"'/g' <for092 > junk
cat junk zeros > for003
rm for09*

sed 's/ASNI4/'"$NP2"'/ < for001 > hestur
mv hestur for001

sed 's/ASNI3/'"$NP2"'/ < for001 > hestur
mv hestur for001

echo $NP2 > DSTPSTOT

RAN=`date +%N | cut -c1-2`
ISEED=1000$RAN
echo $ISEED > for029

./$STEEL_NAME

rm -f $STEEL_NAME junk zeros
gzip *
cp -r $SCRATCH/$jobid $COMMON/output/$jobid
sleep 10

cd
rm -rf $SCRATCH/$jobid
fi

lockfile -x -l $JOBFILE.lock
nremjobs=`wc -l $JOBFILE | awk '{print $1}'`
rm -f $JOBFILE.lock
echo $nremjobs
}
done

1.7.3 Submitting a Job

To submit a job to the PBS cluster move the current directory to the model directory and type,

    qsub –l nodes = # ./run.sh

Where # is the number of nodes the user wishes to utilize during analysis. The user should take care to use an appropriate number of nodes in an appropriate interval when running the analysis to allow maximum usage of the cluster.

It is also possible to receive email alerts about the jobs being run by adding the flags –m and –M to the submit job as follows,

    qsub –l nodes=## -m ake –M somebody@something.com -M ... ./run.sh

With this, the email addresses given will be sent alerts upon job start, job error, and job completion.

1.7.4 Monitoring Results

It is possible to view progress on analysis being run by first determining what nodes are currently running the job. This can be done by opening the hosts file created by run.sh and
secure shelling (ssh) into that node. Moving to the scratch directory specified in client.sh will allow the user to open the for004 file and determine the state of the current analysis.
1.8 Sample 6 Story Model

1.8.1 ETABS Model

The building presented here as an example to represent the functionality of SteelConverter was designed and developed by Anthony Massari at the California Institute of Technology for a future research endeavor. The building is an office structure located in downtown Los Angeles, and is designed per the latest codes and standards in the region. This includes, but is not limited to, ASCE 7-10, IBC 2012, AISC 360-10, and AISC 341-10.

The buildings lateral system is a special concentrically braced frame developed using capacity based design procedures recommended in AISC 341-10. Since the system is assumed to be of the “special” type, an underlying assumption in the modeling presented is ductile performance of all connections in the model. Therefor the computational model focuses of “element failure” and not “connection based failure.”

The building uses a peripheral inter-story x-brace lateral system to resolve all lateral forces in the structure. The braces used are HSS shapes, and are compliant with AISC standards for high seismicity local slenderness to prevent local buckling from occurring in the sections prior to overall element non-linear response. As such, our focus on “element level” failure is appropriate, and STEEL provides a good platform for analyzing this type of structure. More information will be made available about the building designs in future work presented by Massari and an isometric view of the model can be seen in Figure 1-20.
Figure 1-20: 6 Story Example Structure – Isometric View
1.8.2 Sample .e2k file

The sample .e2k file for the 6-Story X-Brace Building can be found in Appendix A. Additionally, the file can be made available by emailing cjanover@caltech.edu.

1.8.3 Sample SteelConverter Configuration File

The sample SteelConverter configuration file for the 6-Story X-Brace Building can be found in Appendix B. Additionally, the file can be made available by emailing cjanover@caltech.edu.

1.8.4 Sample STEEL Input File

The sample STEEL input file for the 6-Story X-Brace Building can be found in Appendices C through I and can be made available by emailing cjanover@caltech.edu.

1.8.5 Sample STEEL Output File

The results from STEEL for the 6-Story X-Brace Building can be made available by emailing cjanover@caltech.edu.
1.9 Change Log

- V1.0 – Base version of STEEL and SteelConverter as described in original SteelConverter manual.
2 Caltech VirtualShaker

2.1 Introduction

VirtualShaker was created by Christopher Janover, P.E. as partial fulfillment of the Ph.D. requirement at the California Institute of Technology. The aim of this website is to facilitate and streamline the process of conducting advanced non-linear models by allowing users to create, upload, and analyze these models in the cloud. All analyses are conducted using STEEL, an advanced non-linear large-displacement finite element analysis tool created by Professor Hall at Caltech. This software is used widely in the Civil engineering department and VirtualShaker aims to make this software more widely available.

VirtualShaker utilizes the SteelConverter tool created by Christopher Janover, P.E. to convert ETABS models to a format STEEL is capable of understanding. With this software, models that used to take days to construct in STEEL now take minutes thereby eliminating a large amount of the overhead cost that comes with creating a new model. Additionally, this conversion tool helps professors at other universities as well as professional engineers to conduct non-linear analyses using steel. As ETABS is software many Civil Engineering professors and engineers understand well the learning curve that comes with using STEEL is greatly diminished, reducing the amount of time it takes a user to begin using STEEL.

The goal of this section is to provide examples and explanations for all features VirtualShaker is capable of providing. However, as this is an ongoing project, revisions will be made. For the most up-to-date version of this manual please visit the VirtualShaker website or email cjanover@caltech.edu.
2.2 Getting Started

2.2.1 Running the U6 – Base sample model

When a user creates a new account they are automatically provided with a sample ETABS e2k file, U6 – Base.e2k, several ground motions, and two sample defaults, described fully in Section 2.4.2. The first default, named “U6 - Base - Sample Default” is built to allow the user to run the U6 – Base model immediately with no extra work required. To do this first visit the Downloads page in the website by clicking the Downloads button in the navigation bar on the top of the screen. An image of this can be seen in Figure 2-1.

![Figure 2-1 - Downloads](image)

Clicking the U6 ETABS Model link under the Sample U6 Model section in the navigation bar on the left of the screen will allow the user to download the U6 – Base ETABS e2k file that the user will use to create a new model.

From here, visit the user’s profile by clicking the Profile button on the top of the screen followed by View. This will redirect the user to the profile overview screen shown in Figure 2-2.
From here, activate the “U6 – Base – Sample Default” default by clicking the Load button to the right of this default. This will cause the U6 – Base default to become active. The user can tell if this was done successfully if the light-blue highlighting shifts to cover the desired default. An image of this can be seen in Figure 2-3.
Next, the user can create a new model using the download e2k file by first clicking on the **Models** dropdown in the navigation bar on the top of the screen followed by the **Create** button. This will redirect the user to the new model page, shown in Figure 2-4.
The user must first enter the name “U6 – Base” in the name field and upload the downloaded e2k file by clicking the **Choose File** button and selected the desired file. The model can then be created by clicking the **Create Model** button. This will then redirect the user to the analysis listing page for the newly created model, as shown in Figure 2-5.

![Figure 2-5 - Model Created](image)

The user can then create a new analysis for this model by clicking the **Create Analysis** button. This will redirect the user to the new analysis page as shown in Figure 2-6.
Before creating the analysis, the user must first enter a name. For this example the name “Base Analysis” was chosen. Entering this name in the text-field and clicking the **Create Analysis** button will redirect the user back to the analysis listing for the U6 – Base model, as seen in Figure 2-7.
The user should note that the newly created analysis now appears in the analysis listing for the U6 – Base model. It would now be possible to customize the configuration of this analysis, however, the sample configuration comes pre-built to run. The user is encouraged to look through the sample configuration to help gain a better understanding of reasonable values for some of the options. To submit the analysis, click the **Submit** button. Clicking the button will cause a popup alerting the user to the fact that, upon accepting the message, any existing analysis results for this analysis will be deleted. After clicking **ok** the website will show a popup saying the model has been submitted. This can be seen in Figure 2-8.
Mousing over or clicking the Run Status result in either a popup or a redirection to a page allowing the user to view information on the current status of the analysis's jobs. Images of this can be seen in Figure 2-9 and Figure 2-10.
Once all runs have been completed the user can view the results by clicking on the **View Results** button in the analysis listing page. This will redirect the user to the results page, shown in Figure 2-11.
This figure shows that 4 out of 4 runs have been completed. To view a result use the Select Ground Motion dropdown and select “elCentro_dt.tar.gz”. This will then cause the Input Files, Output Files, and Post-Processing Files dropdowns to populate with options for the user to view or download. Select the Post-Process Files dropdown and select Undefomed Shape: Grid A.png this will cause an image of the undeformed shape to appear on the screen as well as a download button. This is shown in Figure 2-12.

This concludes the U6 – Base tutorial. The user is encouraged to explore the other files in the result screen as well as examine the configuration used to run this model. Reading other portions of this manual in addition to the SteelConverter manual will help the user gain a better understanding of the capabilities of VirtualShaker.
2.2.2 Creating a Model with the Baseline Default

In addition to the U6 – Base default provided to new users, discussed in Section 2.2.1, VirtualShaker also provides users with a baseline default filled with recommended values. Users can view this default by first navigating to their profile overview by first clicking the Profile dropdown on the top of the screen followed by the View button. From here, activate the “Sample Default” default by clicking the Load button to its right. This will cause the light-blue highlight, indicating the default is active to move to the baseline default. An image of this can be seen in Figure 2-13.

This default is constructed such that any of the options with recommended values come prefilled, meaning the user only needs to add a small number of options in order to create running models. It is recommended that users start with this default until learning more about some of the advanced options VirtualShaker provides to the users.

Before creating an analysis users must go through the sample default and modify the following values:

- Dynamic Time Step (DT) from the Analysis Options page
- LoadCombo, MassCombo, Ground Acceleration Multiplier (GAMULT) from the Load Options page
- ETABS Name, Initial Strain Hardening Modulus (ES), Yield Stress (SIGY), Ultimate Stress (SIGU) for materials 1 and 2 as well as ETABS name, Young’s Modulus, and Crushing Stress for the concrete material in the Material Options section

The default values on any default page can be modified by pressing the edit link at the bottom of each page.
After modifying the baseline default the user can create a new model by going to the **Models** dropdown on the top of the screen and clicking **Create**. The user would then fill in the desired model name and direct the website to the location of the ETABS e2k file to be uploaded. Pressing the **Create** button will redirect the user to the analysis listing page. From here, the user should create a new analysis by pressing the **Create Analysis** button followed by typing in the desired analysis name and pressing **Create**. The user at this point could run the model by pressing **Submit**.

For more information on the model or analysis creation process see the U6 – Base analysis guide in Section 2.2.1

![Figure 2-13 - Baseline Default Profile Overview](image-url)
2.3 VirtualShaker Back-end Description

VirtualShaker utilizes several services and is a cumulation of several years of work. When a user interacts with the website a number of things are happening behind the scenes to allow the user to create, store, submit, analyze, and view results of models. VirtualShaker uses a web development framework on top of a backend database to display and store information locally for the user. The cloud storage and computing is handled by Amazon Web Services where additional code is run to convert the ETABS models into STEEL format through the use of SteelConverter, analyze models through STEEL, and run custom post-processing software.

2.3.1 Amazon Web Services

VirtualShaker relies heavily on Amazon Web Services (AWS) for cloud computing, messaging, storage, and scalability. AWS is a service provided by Amazon as an inexpensive alternative to building and maintaining a personal analysis server. The workflow of VirtualShaker can be seen in Figure 2-14.

![Figure 2-14 - AWS Workflow](image-url)
When a User interacts with the VirtualShaker website they are actually interacting directly with AWS. The webserver that responds to user requests is running on an instance of a virtual server known as an EC2 Server. This is where the framework, stylesheets, JavaScript, HTML, and database all operate. The compute nodes, which are responsible for running SteelConverter, STEEL, and post-processing are also instances of EC2 Servers. When the user creates or analyzes a model the files are all uploaded to the Amazon cloud storage known as S3. Additionally, there is a messaging service Amazon provides, known as SQS, which is used to alert the various components of the system when new information requires their attention, such as when a model is created, submitted, or analyzed. These various AWS components will now be discussed in more detail.

2.3.2 EC2 Servers

As mentioned earlier, when a user interacts with the webserver or when any of the ComputeNodes carry out the conversion, analysis, or post-processing operations all the work is being handled by one of Amazons virtual EC2 Servers. The number of virtual servers running at any given moment for analysis can change depending on the instantaneous workload. When the system determines it no longer needs the additional ComputeNodes to reasonably deal with the required user demand it would be possible to shut excess servers down to save on cost. Amazon offers a wide range of EC2 servers of varying size and speed in a number of different platforms, making it easy to meet any demands placed on the analysis package. The two types of EC2 servers used in VirtualShaker, the Webserver and the ComputeNodes will now be discussed.
2.3.2.1 **Webserver**

VirtualShaker was built using the Ruby on Rails (RoR) framework for the webserver and a SQL backend for storing user and model data. The RoR framework allows for rapid creation of the various website pages through the “convention over configuration” mindset. The framework interfaces with the website HTML and provides an abstraction layer to the SQL backend allowing for automatic generation of HTML code. When the user interacts with a page by submitting forms or selecting links the HTML code sends requests to the webserver’s backend through either static actions, for normal website navigation, or through AJAX, for dynamic website navigation. For both backend interaction techniques, new information is displayed to the user depending on the request of the user.

When the user submits a job for analysis the Webserver sends a message to the “to-do” messaging queue where the job is placed in a line with all the other pending jobs. The jobs remain in the queue until a ComputeNode pulls the message from the “to-do” queue. In order for the Webserver to read messages from the “done” queue, a special daemon needed to be set up to poll for analysis complete messages. This was done via the Redis and Sidekiq services. When these two services are used together and placed as a daemon background process they enable the Webserver to poll for new messages while still allowing the Webserver to rapidly respond to user requests.

When a new message is received, Sidekiq pulls the message from the “done” queue, parses the message, and then modifies the SQL backend database. The webserver is then able to view these modifications and display them to the user real-time, thereby allowing live updates on the current state of every analysis the user submitted.
2.3.2.2 ComputeNodes

ComputeNodes are responsible for the model conversion, analysis, and post-processing. The conversion is carried out by SteelConverter, an application written in C++ which is capable of converting ETABS model into the California Institute of Technology in-house non-linear large displacement plastic analysis tool STEEL. More information on these tools can be found in [1] and [18]. After running the converter and analysis package the Compute node will then run any requested post-processing that was requested by the user. Some of the options for post-processing include nodal and element based response, a time history video, and plots of the inputted ground motions.

If at any point the ComputeNode encounters an issue, such as an incorrectly formatted ground motion file or a convergence error the programming will attempt to recover and return a message to the user explaining what went wrong via the “done” messaging queue. It will also attempt to run some of the requested post-processing. An image of the undeformed shape of the model showing element and node numbering, for example, could be helpful in determining if an improperly built ETABS model is the cause of the analysis error. Upon completion of the post-processing the ComputeNode will then upload all results to the S3 cloud storage and respond to the “done” queue stating the complete status of the analysis. Additionally, the ComputeNode sends messages back to the Webserver at various points in its run cycle. Status updates can be seen on the website and are discussed in Section 2.4.9.

2.3.3 S3 Cloud Storage

As mentioned earlier, the cloud storage for VirtualShaker is handled by the AWS S3 system. The information for each user is stored separately and in an organized fashion allowing the Webserver and ComputeNodes to know exactly where the request piece of information is
located. Amazon has a Software Development Kit (SDK) for a number of different languages allowing the various VirtualShaker components to push and pull files from storage.

At no point are users able to directly access the S3 file system, rather when a specific file is requested for viewing or downloading on the website a temporary secure link is created for a short period of time. If the user attempts to use that link after the time has expired the link will no longer function and a new request must be made. This was done to ensure security in the cloud and to prohibit users from accessing other users’ models, ground motions, and results.

2.3.4 SQS Messaging

The messaging service provided by Amazon is used to alert the various components of VirtualShaker when a new job was submitted or when a job was finished. VirtualShaker utilizes two SQS queues, one for when jobs are submitted, known as the “to-do” queue, and a second for when jobs are complete (or when a message needs to be seen by the user when an error occurs) known as the “done” queue.

When a job is submitted by the user the Webserver places a message in the “to-do” queue with information such as the user id, model name, ground motion to run, and any post-processing requested. The ComputeNodes are constantly polling for new messages, essentially asking the “to-do” queue repeatedly if there are any jobs pending. When a job is found the ComputeNode removes the message from the queue and processes the message.

After the ComputeNode finishes work on the job or when it needs to send a status update to the Webserver a message is created in the “done” queue with the user id, model name, ground motion and status. The various statuses include: Post-Processed, Analyzed, and Converted.
As discussed earlier, the Webserver uses the Reddis and Sidekiq servers running as a background daemon to read and respond to messages in the “done” queue. The daemon runs for 45 seconds every minute and will repeatedly poll the “done” queue while it is running. When a message is received, the message is removed from the queue and the Sidekiq process parses the message and modifies the appropriate SQL database entry containing the current status of that job.

2.3.5 SQL Database Design

The database used by VirtualShaker was built in SQL. The database stores all the information about users, models, defaults, configurations, and job statuses. An image showing the SQL database used can be seen in Figure 2-15.

As the figure shows, there is a hierarchy to the database stemming from the User. Working down the bottom part of the tree every User has Models and each model has a set of Grids, Floors, Points, and Nodes. Additionally, every Model has a set of Analyses which each have a default (configuration) and results. Each User also has a set of Ground Motions that are currently active for a given analysis. When the analysis is submitted a run is created for each active ground motion and a result is created for each run. The default (configuration) for a given analysis is based on the active default in the User’s profile, which is described in the upper portion of the diagram.

A user can have any number of defaults, which can be activated at any time to rapidly create the configuration for a new model’s analysis. Each default fills the role of the conversion configuration file needed for SteelConverter to properly convert the ETABS model to an input STEEL understands. Each default has options for model information, analysis options,
convergence properties, fiber options, vertical constraint properties, load options, foundation node properties, post processes, response time histories, and material models. Each default can be customized to best fit the model it is meant to represent. With multiple defaults a user can utilize the configurations of multiple models simultaneously thereby eliminating the need for multiple SteelConverter configuration files to be organized and maintained. More information on each configuration/default option can be found in the SteelConverter Manual [18].
Figure 2.15 - VirtualShaker SQL Database
2.4 Features

VirtualShaker has a number of features aimed at streamlining the model creation, analysis, and result generation processes. A number of these features will now be discussed. For further information on individual options and configurations view Section 2.6 or view the SteelConverter manual found on the downloads or documentation section of the website [18].

2.4.1 Creating an Account

To create an account, click on the **Login** button on the top right corner of the home page, shown in Figure 2-16.

![Figure 2-16](#)

Figure 2-16 - Home Page: Login

Clicking that link will take the user to the user login page. To create a new account press the **Create New User** link located below the **Login** button, as shown in Figure 2-17. If the user already has an account with VirtualShaker simply enter the login credentials and press **Login**.
Pressing the **Create New User** link will navigate the user to the new user page where they are required to enter a valid email address (used as the user’s login name) and a password. An image of this page can be seen in Figure 2-18. After filling in the required information pressing the **Create User** button will create the new account and redirect the user back to the home page. The user may then login by navigating back to the **Login** button on the top right corner of the screen and filling the required information.

![Figure 2-17 - New User: Login Screen](image_url)
After logging in the user will have access to their own defaults, models, analyses, and results separate from other user’s data. In order to create models or run analyses the user must be logged in. If the user attempts to access restricted portions of the website while not logged in, the website will redirect the user to the login page before continuing.

2.4.2 Creating Defaults

Defaults are one of the most powerful features VirtualShaker implements to help facilitate creation and organization of a user’s models. Defaults allow a user to create a set of custom configuration and post-processing options that, upon creation of an analysis, will be used as a template for the new analysis’s configuration. Defaults can be created, copied, and deleted by first clicking the Profile link on the top right corner of the page and selecting View. Doing so will redirect the user to the profile overview page, shown in Figure 2-19.
From the profile overview page users can create, edit, delete, copy, and activate a default. Pressing the **New Default** button allows users to create a new default by entering a name. The **Load** button to the right of each default will activate that default, allowing the user to edit its configuration using the various options in the navigation button on the left of the screen. The **Edit** button allows the user to change the name of a default, while the **Delete** button deletes the default. Pressing the **Copy Active Default** button in the middle of the screen creates a new default with the exact configuration options as whatever default is currently active.

At any given time only one default can be active. The user can tell which default is active by the light blue highlighting appearing over the default row (as seen by the “2D Single Bay” default in Figure 2-19). Additionally, the active default’s name is displayed in the navigation bar to the left of the screen. Keeping track of which default is currently active is important as it allows users to create new analyses with the same settings as defined in the active default. It is
expected that a user will have at least one default for every type of model they are working with so information such as convergence properties, foundation node properties, and time history options only need to be entered once.

2.4.3 Modifying a Default

As shown in Figure 2-19, when a default is activated a series of options appear in the left navigation bar. These options are Model Information, Analysis Options, Convergence Options, Fiber Options, Vertical Constraint Options, Load & Post Processing Options, Response Time History Options, Material Model Options, and Foundation Node Options. Selecting any of these links will redirect the user to a screen similar to Figure 2-20.

![Figure 2-20 - Typical Default Options Page](image)

This figure shows the Convergence Options page where there are several options. In this state the user can only view the inputted values for each option. Mousing over some of the options results in a popup tooltip giving the user more information about the specific option. An
example of this can be seen in Figure 2-21. For specific information on each option that make up a default see Sections 2.6.1 through 2.6.11.

Figure 2-21 - Configuration Tooltip

2.4.4 Uploading Ground Motions

From the Profile Overview page selecting the **Ground Motions** link in the left navigation pane, as shown in Figure 2-19, redirects users to the ground motions page, shown in Figure 2-22.
Every new account comes preloaded with several ground motions for the user to use. However, VirtualShaker allows users to upload custom ground motions by pressing the **Import** button. There are several rules regarding the format of the ground motion files and compression format, which are necessary to ensure STEEL, executes with no errors. First, the horizontal and ground motion files must be compressed in a `.tar.gz` format. Second, the two ground motion files must be named `for002` (for horizontal ground motions) and `for003` (for vertical ground motions). These files must be in plain text format. Finally, the data in each file must follow a strict formatting rules. The first line of the ground motion files is assumed to be a header and is ignored by STEEL. Every line after this must consist of six ground acceleration entries in the following format.

```
0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
```

Each of the six ground acceleration values can have no more than five significant figures.
After the ground motion tar.gz file is uploaded, a new entry is created in the ground motions list table and will become available to any analysis the user creates. Note, it is also possible to delete ground motions by pressing the Delete button next to the desired ground motion.

2.4.5 Creating a Model

Models form the basis of a user’s analysis. A model can have numerous analyses for the different type of ground motions and configurations a user may want to run. Creating a model requires a valid ETABS .e2k file. For more information on proper model building techniques view the SteelConverter manual in the downloads or the documentation portion of the website [18].

To create a new model first click the Models dropdown at the top of the screen followed by the Create button. Doing so will redirect the user to the create model page, shown in Figure 2-23. Once on this page, enter a valid model name (no HTML characters), press the Choose File button, navigate to the .e2k file and press Create Model. Following this, the website will redirect the user to the model’s analyses page, shown in Figure 2-24. From here the user can view every analysis for the newly created model. Users can also view a list of every created model by either pressing the Models link under the Navigation tag near the top of the screen or by first pressing the Models dropdown followed by View in the navigation bar on the top of the screen. An image of the models view page can be seen in Figure 2-25.
Figure 2-23 - Model Creation

This page allows the user to view all analyses for the selected model.
- Pressing the “Create Analysis” button will allow the user to create a new analysis. The link will send the user to the configuration page.
- The “Configure” button allows the user to customize the currently active default configuration for the specific analysis.
- The current status of the analysis is seen under the “Run Status” column. The different statuses are: Not Run, In Queue, Pending Approval, Running, and Complete.
- The “View Results” button allows the user to view and download the analysis results.
- Pressing the “Delete” button causes the selected selected analyses and results to be permanently deleted.

Figure 2-24 - Model Analysis Listing
2.4.6 Creating an Analysis

After a model is created users can create an analysis for the model. When created, the analysis’s configuration will be based on whatever the current active default is. For information on defaults see Sections 2.4.2 and 2.4.3. To create a new analysis, from the analysis view screen press the **Create Analysis** button as seen in Figure 2-24. Clicking this link will redirect the user to the create analysis screen, shown in Figure 2-26.
After entering the desired analysis name pressing the **Create Analysis** button will create the analysis and apply the current active default to the analysis’s configuration. The website will then redirect the user back to the model listing page.

Several buttons are located next to each model in the models list. A close-up of these buttons can be seen in Figure 2-27.

![Figure 2-27 - Analysis Options](image)

The **Edit** button is used to change the analysis name. The **Customize** button will redirect the user to the analysis’s personal configuration settings where the user can make modifications to the default for this analysis only. More information on this can be seen in Section 2.4.7. The **Submit** button will submit the analysis to the ComputeNodes to be converted, analyzed, and
post-processed. More information on this can be seen in Section 2.4.8. The **Run Status** button will give the user information on the current status of every ground motion being analyzed for the analysis. More information on this can be seen in Section 2.4.9. The **View Results** button will redirect the user to the results page where an analysis results can be viewed and downloaded. Information on this can be seen in Section 2.4.10. Lastly, the **Delete** button will delete the analysis along with all results and other related files from the cloud.

### 2.4.7 Customizing an Analysis’s Configuration

As discussed in Section 2.4.6 it is possible for users to customize an analysis’s configuration from the default used to create it. To do this press the **Customize** button as seen in Figure 2-27. Pressing this will bring the user to the analysis configuration screen as seen in Figure 2-28. The options on the left navigation bar are almost identical to those seen in the defaults discussion in Section 2.4.3, the only difference being the lack of profile options such as Overview and Ground Motions. Pressing any of the links will redirect the user to pages similar to those discussed earlier as seen in Figure 2-29. For more information on each option see Sections 2.6.1 through 2.6.11.
2.4.8 Submitting an Analysis

As discussed in Section 2.4.6, Figure 2-27, pressing the Submit button will tell the website to submit the analysis and all of its ground motions to the ComputeNode to be run.
After pressing the submit button the website will alert the user via a popup that submitting the analysis will delete any existing results on the server. After clicking okay the request will be sent out and a notification will be sent to the user as seen in Figure 2-30.

![Figure 2-30 - Model Submitted](image)

After this, the analysis will be placed in the “todo” queue.

2.4.9 Viewing Analysis Status

After submitting an analysis, users can view the current status of their jobs in two ways. For a quick overview of the status of every job mouse over the Run Status button as shown in Figure 2-27 in Section 2.4.6. An image of this can be seen in Figure 2-31. The popup shows the total number of jobs, the number of jobs in-queue, the number currently being run, the number who are in the post-processing stage, and the number that are complete.

For a more detailed view of the status of every job press the Run Status button. The website will then redirect the user to the run status page shown in Figure 2-32. This page shows, for
each job, the current status of the job, the time in the current stage, and the overall runtime. Jobs which are complete will display N.A. for the time in current stage, and N.A. for the Runtime if the job is in queue.

![Figure 2-31 - Run Status Popup](image1)

---

![Figure 2-32 - Detailed Run Status](image2)
2.4.10 Viewing / Downloading Results

Selecting the **View Results** button from Figure 2-27 in Section 0 will redirect the user to the results overview page seen in Figure 2-33. The top of this page displays the current analysis being viewed as well as the overall status of all the analysis’s jobs. Clicking the **Select Ground Motion** dropdown will allow the user to select which job to view results for. Once a job is selected via the dropdown the **Input Files**, **Output Files**, and **Post-Processing Files** dropdowns become populated with files users can either download or view depending on the file type. An example of the dropdown can be seen in Figure 2-34.

![Results Page](image.png)
The dropdown shows the name of each file the user has the ability to download. For image and gif files the website is able to generate an image or video of the file and display it to the user. An example of this can be seen in Figure 2-35. For all file types selected a Download link appears which creates a temporary secure link to the S3 storage in the cloud allowing the user to download the file to their local machine. For more information on the different types of results and post-processing files refer to Section 2.5.3.
2.4.11 Downloads

Clicking the Downloads button in the navigation bar on the top of the screen will redirect the user to the downloads page of the website. Here, users can download the various software, manuals, papers, models, and verification models used throughout the VirtualShaker toolkit. An image of the downloads page can be seen in Figure 2-36.
2.4.12 Documentation

Clicking the **Documentation** button on the top of the screen will redirect the user to the documentation page. Here, users can view the SteelConverter and VirtualShaker manuals online as well as the model verification paper. An image of this can be seen in Figure 2-37.
Documentation

To view the manuals for VirtualShaker and SteelConverter or a guide on how to use VirtualShaker, use the links on the left of the screen under the Manuals section. Additionally, papers on the validity of this software can be found beneath the Papers section on the left.

To download aspects of any of the papers visit the Downloads page by clicking on the link on the top of the page.

Figure 2-37 - Documentation Page
2.5 Results & Post-Processing

After running an analysis the user is provided with a multitude of files for them to view and download. These files are broken down into three categories, Input Files, Output Files, and Post-Processing Files.

2.5.1 Input Files

Input Files comprise all files that went into running SteelConverter and STEEL and include files such as the STEEL primary input file (for001), the STEEL horizontal and vertical ground motion files (for002 and for003), the SteelConverter configuration file, and the STEEL seed file (for029). These files can be viewed by first going to the results page for a particular analysis, selecting a run to view, and then utilizing the Input Files dropdown. After selecting a file an option will appear allowing the user to download the selected file from the cloud. For more information on the various STEEL or SteelConverter input files see the SteelConverter manual [18].

2.5.2 Output Files

Output Files are files which are direct outputs from the STEEL analysis package. These files include the primary STEEL output file (for004) and the STEEL dynamic output file (for008). These files can be viewed by first going to the results page for a particular analysis, selecting a run to view, and then utilizing the Input Files dropdown. After selecting a file an option will appear allowing the user to download the selected file from the cloud. For more information on the various STEEL or SteelConverter input files see the SteelConverter manual [18].
2.5.3 Post-Processing Files

Post-Processing files comprise the bulk of what users will be using after running analyses. These files include any post-processing requested during the configuration process as well as placing the major input and output files in a standardized form to facilitate custom post-processing by the user.

As discussed in the configuration section of this manual, post-processing can be requested by navigating to the Results & Post-Processing page while editing a default configuration. From here, clicking the **New Post-Process** button will navigate the user to the post-process creation screen, seen in Figure 2-38. From here, the user can select what type of post-processing they would like VirtualShaker to conduct. A table summarizing the different types of post-processing as well as the required inputs for each type can be seen in Table 2-1.

![Figure 2-38 - New Post-Process](image)
<table>
<thead>
<tr>
<th>Output Name</th>
<th>Description</th>
<th>Data Required if in Profile Default</th>
<th>Data Required if in Analysis Configuration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undeformed Shape Plot</td>
<td>Creates a png of the undeformed shape of a particular grid showing node numbers, element numbers, wall elements, springs, and fixity.</td>
<td>Grid Name - manually entered</td>
<td>Grid Name - dropdown</td>
</tr>
<tr>
<td>Ground Motion Plot</td>
<td>Creates a png of the provided horizontal and vertical ground motions.</td>
<td>None</td>
<td>None</td>
</tr>
<tr>
<td>Simple Time History Video</td>
<td>Creates a gif of the motion of a particular frame to the applied ground motions. This video contains only line elements.</td>
<td>Grid Name - manually entered Frame Skip - Number of frames to skip between renders</td>
<td>Grid Name - dropdown Frame Skip - Number of frames to skip between renders</td>
</tr>
<tr>
<td>Detailed Time History Video</td>
<td>Creates a gif of the motion of a particular frame to the applied ground motions. This video contains information such as plastic hinge generation, stresses in members, buckling information among other advanced properties.</td>
<td>Grid Name - manually entered Frame Skip - Number of frames to skip between renders</td>
<td>Grid Name - dropdown Frame Skip - Number of frames to skip between renders</td>
</tr>
<tr>
<td>Nodal Response</td>
<td>Creates a png of the requested nodal output information. Options include nodal X &amp; Y translation, beam rotation, column rotation as well as panel-zone rotation and moment</td>
<td>ETABS node - Coordinates entered manually</td>
<td>ETABS node - selected from dropdown</td>
</tr>
<tr>
<td>Element Response</td>
<td>Creates a png of the requested element response. Options include moment and plastic rotation at either side of the element, axial force, and plastic axial displacement</td>
<td>ETABS nodal coordinates for the left and right node entered manually</td>
<td>Element selected from dropdown</td>
</tr>
<tr>
<td>Pushover Plot</td>
<td>Creates a png of the pushover curve for the modal</td>
<td>None</td>
<td>None</td>
</tr>
<tr>
<td>Story Drift Plot</td>
<td>Creates a png of the story drift plot for a particular column</td>
<td>ETABS point coordinates entered manually</td>
<td>ETABS point selected from a dropdown</td>
</tr>
</tbody>
</table>
In addition to post-processing requests, VirtualShaker also generates a standardized form of the STEEL primary input file (for001), the STEEL static analysis results (for004), the STEEL dynamic analysis results (for008), and the SteelConverter ETABS to STEEL grid conversion file. These files are all organized in JSON format and allow users to access all data in an array using Key => Value pairs. Information regarding the structure of each of these output files can be seen in the Downloads portion of the website by clicking on the JSON Format button in the navigation bar on the left of the screen.
2.6 Configuration Description

Additional information will now be given for each option in an analysis’s configuration. For a more detailed description of the workings of SteelConverter view the manual in the download or documentation portion of the website [18].

For many of these options, additional information may be available by mousing over a given option in the form of a tooltip popup. Additionally, for any option that requires model specific information, such as a load case to be imported or the location of a node for time-history output, if the option is being modified in the user’s profile the information will need to be hard-coded, meaning the specific load combination or coordinates will be entered by hand. However, if the option is being edited in the analysis portion of the website (when customizing a default for a particular analysis) the same options will be populated with dropdowns allowing the user to select the desired load combination or node from a list created by parsing the ETABS e2k file. The following description for each option will alert the user when such a change is available.

2.6.1 Model Information

An image of the Model Information configuration options can be seen in Figure 2-39. This page has a few options pertaining to general information about the model. A table summarizing these options can be seen in Table 2-2. Pressing the edit button at the bottom of Figure 2-39 will redirect the user to the edit screen where modification of the option values can be made.
2.6.2 Analysis Options

An image of the Analysis Options default / configuration page can be seen in Figure 2-40. These options pertain to overall properties about the model. Information describing each option can be seen in Table 2-3. Pressing the **edit** link at the bottom of the page will allow the user to modify these values.
Figure 2-40 - Analysis Options
Table 2-3 - Analysis Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Panel Zone Rigidity</td>
<td>Toggle for how to treat non-base nodes. 1 = rigid, 2 = flexible</td>
<td></td>
</tr>
<tr>
<td>Maximum Number of Turning Points in Hysteretic Model (MTP)</td>
<td>Maximum number of turning points in Hysteretic Models (suggested minimum of 20)</td>
<td></td>
</tr>
<tr>
<td>Maximum Number of Turning Point Locations (NDIM)</td>
<td>Maximum number of turning point locations (suggested minimum of 100000)</td>
<td></td>
</tr>
<tr>
<td>Number of Static Steps (NSS)</td>
<td>Number of static load steps</td>
<td></td>
</tr>
<tr>
<td>Newmark Integration Parameter (Beta)</td>
<td>Newmark Integration Parameter</td>
<td>0 = Central Difference, 0.25 = Constant Average, 0.166 = Linear Average</td>
</tr>
<tr>
<td>Newmark Integration Parameter (Gamma)</td>
<td>Newmark Integration Parameter (0.5)</td>
<td></td>
</tr>
<tr>
<td>Dynamic Time Step (DT)</td>
<td>Timestep used in dynamic analysis</td>
<td></td>
</tr>
<tr>
<td>Output Interval for Time Histories (PRINT)</td>
<td>Output interval for response time histories on unit 8</td>
<td>1 means every step, 2 means every other, etc.</td>
</tr>
<tr>
<td>Additional Time History Option Toggle (IROUT)</td>
<td>Toggle to also output response time histories to unit 4</td>
<td>1 = yes, 0 = no</td>
</tr>
<tr>
<td>Time Step where Dynamic Analysis Ends (ISTOP)</td>
<td>Timestep to stop outputting response time history information</td>
<td>If left blank, program assumes value equal to number of dynamic time steps</td>
</tr>
<tr>
<td>Debug Level (DEBUGLEVEL)</td>
<td>Toggle to enable or disable debug output</td>
<td>0 = Errors only through 4 = all information</td>
</tr>
<tr>
<td>Generate Section Conversion File (SECTIONCONVERSION)</td>
<td>Toggle to enable or disable output of section conversion table</td>
<td>yes or no</td>
</tr>
<tr>
<td>Generate Material Conversion File (MATCONVERSION)</td>
<td>Toggle to enable or disable output of material conversion table</td>
<td>yes or no</td>
</tr>
</tbody>
</table>

2.6.3 Damping Options

An image of the Damping Options default / configuration page can be seen in Figure 2-41.

These options pertain to the application of damping throughout the model. Information describing each option can be seen in Table 2-4. Pressing the edit link at the bottom of the page will allow the user to modify these values.
## Table 2-4 - Damping Option Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rayleigh Damping Parameter (A0)</td>
<td>Damping Parameter for Rayleigh Damping (C = A0<em>M + A1</em>K)</td>
<td>Assumed to be 0 when modeling damping with special columns</td>
</tr>
<tr>
<td>Assumed Period of First Mode</td>
<td>Period of the first mode of the structure</td>
<td>When left blank, program assumes T = 0.1*N where N is the number of stories</td>
</tr>
<tr>
<td>Damping Ratio</td>
<td>Damping ratio of columns for calculating A2</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>Maximum Pushover Base Shear</td>
<td>Pushover base shear used in calculating damping</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>ETABS Unmodeled Force Combo</td>
<td>Name of ETABS Load Combination containing vertical loads representing unmodeled frames</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>Spring Yield Drift</td>
<td>Drift percent as a decimal at which the springs yield</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>Fraction of max Pushover Base Shear taken by Unmodeled Frames at Base</td>
<td>Percentage as decimal</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>Damper Yield Velocity</td>
<td>Velocity at which column dampers yield</td>
<td>View SteelConverter manual for more information</td>
</tr>
<tr>
<td>Capping Force of base Column Dampers as Fraction of max Pushover Base Shear</td>
<td>Percentage as decimal</td>
<td>View SteelConverter manual for more information</td>
</tr>
</tbody>
</table>
2.6.4 Diaphragm Options

An image of the Diaphragm Options default / configuration page can be seen in Figure 2-42.

These options pertain to the application of diaphragms throughout the model. Information describing each option can be seen in Table 2-5. Pressing the edit link at the bottom of the page will allow the user to modify these values.

![Diaphragm Options Page](image)

Figure 2-42 - Diaphragm Options

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default Diaphragm Stiffness (alphacdef)</td>
<td>Default diaphragm Stiffness</td>
<td>Applied to all diaphragms</td>
</tr>
<tr>
<td>Override Diaphragm Stiffness (alphac)</td>
<td>Diaphragm Stiffness used for a particular floor</td>
<td></td>
</tr>
</tbody>
</table>
2.6.5 Convergence Options

An image of the Convergence Options default / configuration page can be seen in Figure 2-43. These options pertain to the limits of the analysis module. Information describing each option can be seen in Table 2-6. Pressing the edit link at the bottom of the page will allow the user to modify these values.

Figure 2-43 - Convergence Options
Table 2.6 - Convergence Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum number of global iterations (MIG)</td>
<td>Number of global iterations</td>
<td>Default of 20</td>
</tr>
<tr>
<td>Force tolerance for global iterations (TOL1)</td>
<td>Global force tolerance</td>
<td>Default of 0.2</td>
</tr>
<tr>
<td>Moment tolerance for global iterations (TOL3)</td>
<td>Global moment tolerance</td>
<td>Default of 0.2</td>
</tr>
<tr>
<td>Force tolerance for local iterations (TOL5)</td>
<td>Local force tolerance</td>
<td>Default of 2</td>
</tr>
<tr>
<td>Moment tolerance for local iterations (TOL7)</td>
<td>Local moment tolerance</td>
<td>Default of 1.0</td>
</tr>
</tbody>
</table>

2.6.6 Fiber Options

An image of the Fiber Options default / configuration page can be seen in Figure 2-44. These options effect the distribution of fibers STEEL uses in its elements. A table summarizing these options can be seen in Table 2-7. It is recommended that the user use the recommended values, which can be found in the SteelConverter manual. Pressing the edit link at the bottom of the page will allow the user to modify these values.
2.6.7 Vertical Constraint Options

An image of the Vertical Constraint Options default / configuration page can be seen in Figure 2-45. These options effect the stiffness STEEL uses when assigning vertical constraints to nodes to enforce vertical compatibility. The default vertical constraint stiffness will be applied
to all vertical connection elements unless a specific override is found for that node. A table summarizing these options can be seen in Table 2-8. It is recommended that the user use the values provided in the SteelConverter manual. Pressing the edit link at the bottom of the page will allow the user to modify these values. Note that adding a specific vertical constraint in the profile default will require the user to enter the ETABS coordinates of the node manually while adding a constraint in the analysis configuration, which allows the user to select the desired ETABS node from a dropdown.

![Figure 2-45 - Vertical Constraint Options](image)

Figure 2-45 - Vertical Constraint Options
Table 2-8 - Vertical Constraint Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default vertical connection stiffness</td>
<td>Default stiffness of vertical connection</td>
<td>See the SteelConverter manual for more information</td>
</tr>
<tr>
<td>(alphavcdef)</td>
<td>elements</td>
<td></td>
</tr>
<tr>
<td>Specific vertical constraint</td>
<td>Override stiffness for vertical connection</td>
<td>Searches for ETABS nodes at this location (if in the profile default) or allows for a dropdown of all ETABS nodes (if in the analysis configuration) and applies the given stiffness to the primary and secondary nodes who are derived from the supplied ETABS node. For more information see the SteelConverter manual</td>
</tr>
<tr>
<td>connection</td>
<td>elements</td>
<td></td>
</tr>
</tbody>
</table>

2.6.8 Load & Post-Processing Options

An image of the Load and Post-Processing Options default / configuration page can be seen in Figure 2-46. These options allow the user to send custom post-processing requests to the ComputeNodes and have these requests automatically be applied to all new analyses. A table summarizing these options can be seen in Table 2-9. For more information on the various post-processing options see section 2.5.3. Pressing the edit link at the bottom of the page will allow the user to modify these values. Note that adding a post-processing request in the profile default will require the user to enter the ETABS model information manually, such as nodal coordinates or grid names, while adding a request in the analysis configuration allows the user to select the desired ETABS information from a dropdown.

If the Load & Post-Processing options page is viewed during the specific analysis configuration an additional option of which ground motions to run will be provided. An image of this can be seen in Figure 2-47. Ground motions can be removed or added by pressing the edit button and checking or unchecking the ground motion of interest. Additionally, pressing the Customize Post-Processing button next to an earthquake will allow the user to set what
post-processing to run for that specific ground motion. Pressing the **Apply to All** button just above the ground motion listing will apply the general post-processing requests to all ground motions, replacing any customization done.

![Figure 2-46 - Load & Post Processing Options](image)

**Table 2-9 - Load & Post Processing Options Descriptions**

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load combination (LOADCOMBO)</td>
<td>Name of the ETABS load combination STEEL will use as the source for loads</td>
<td>If edited while in the profile default the user will need to enter the load combinations by hand. If edited while in the analysis configuration the user can select the combination from a dropdown.</td>
</tr>
<tr>
<td>Mass Combination (MASSCOMBO)</td>
<td>Name of the ETABS load combination STEEL will use as the source for masses</td>
<td></td>
</tr>
<tr>
<td>Ground acceleration multiplier (GAMULT)</td>
<td>Multiplier to apply to ground motions</td>
<td></td>
</tr>
<tr>
<td>Default post procession options</td>
<td>Series of options allowing users to create post processing to automatically be applied to new analyses</td>
<td>If edited while in the profile default the user will need to enter the load combinations by hand. If edited while in the analysis configuration the user can select the combination from a dropdown.</td>
</tr>
</tbody>
</table>
2.6.9 Response Time History Options

An image of the Response Time History Options default / configuration page can be seen in Figure 2-48. These options allow the user to send custom requests for a particular node, panel-zone, or element time history output and have these requests automatically be applied to all new analyses. A table summarizing these options can be seen in Table 2-10. For more information on the various post-processing options see the SteelConverter manual. Pressing the edit link at the bottom of the page will allow the user to modify these values. Note that adding a time history requests in the profile default will require the user to enter the ETABS model information manually while adding a request in the analysis configuration allows the user to select the desired ETABS information from a dropdown.
### Table 2-10 - Response Time History Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plot response of all primary nodes</td>
<td>Toggle to automatically output all primary nodes’s X and Y displacement for all timesteps.</td>
<td>1 = yes, 0 = no. For more information on primary and secondary nodes see the SteelConverter manual</td>
</tr>
<tr>
<td>(PlotAll)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plot response of all secondary node</td>
<td>Toggle to automatically output all secondary nodes’s X and Y displacement for all timesteps.</td>
<td></td>
</tr>
<tr>
<td>se (PlotSecondary)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Extra response time histories</td>
<td>Options to create customized response time histories that will automatically apply to all new analyses.</td>
<td>If edited while in the profile default the user will need to enter the load combinations by hand. If edited while in the analysis configuration the user can select the combination from a dropdown. For more information see the SteelConverter website</td>
</tr>
</tbody>
</table>
2.6.10 Material Model Options

An image of the Material Model Options default / configuration page can be seen in Figure 2-49. These options allow the user to define the material properties STEEL uses during its analysis. STEEL is capable of using a maximum of two STEEL materials and one concrete material. For more information on the conversion process see the SteelConverter manual. A table summarizing these options can be seen in Table 2-11. Pressing the edit link at the bottom of the page will allow the user to modify these values.

![Material Model Options Image](image_url)
Table 2-11 - Material Model Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel material shear modulus</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel material yield stress modulus</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Default shear modulus for basement wall elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ETABS Material name for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus (E) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Initial strain hardening modulus (ES) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ultimate stress (SIGU) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strain at onset of strain hardening (EPSS) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strain at peak stress (EPSU) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Poisson’s ratio (PRAT) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Residual stress (RES) for STEEL material 1 and 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ETABS material name for STEEL concrete material</td>
<td>Name used to match ETABS materials to STEEL materials. All ETABS elements with the assigned material will be given the corresponding STEEL material</td>
<td>For more information see the SteelConverter manual</td>
</tr>
<tr>
<td>Young’s modulus (E) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Initial Strain Hardening Modulus (ES) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yield Stress (SIGY) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ultimate Stress (SIGU) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Residual Stress (SIGX) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ultimate Strain (EPSS) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fiber Fracture strain as fraction of yield strain (FYFRAC) for STEEL concrete material</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

2.6.11 Foundation Node Options

An image of the Foundation Node Options default / configuration page can be seen in Figure 2-50. These options affect how STEEL converts ETABS foundation springs. The default nonlinear spring properties will be applied to foundation node elements unless a specific override is found for that spring type. A table summarizing these options can be seen in Table 2-12. Pressing the **edit** link at the bottom of the page will allow the user to modify these values.
Figure 2-50 - Foundation Node Options

Table 2-12 - Foundation Node Options Descriptions

<table>
<thead>
<tr>
<th>Option Name</th>
<th>Description</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default post-yield stiffness ratio for</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Default yield strength for horizontal spring (STRH)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Default yield strength of vertical spring in upward direction (STRVU)</td>
<td></td>
<td>For more information see the SteelConverter manual</td>
</tr>
<tr>
<td>Default yield strength of vertical spring in downward direction (STRVD)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Specific foundation node option</td>
<td>Searches for ETABS springs with the corresponding name. Matching springs will give the given nonlinear properties (ALP, STRH, STRVU, STRVD) instead of the default nonlinear properties</td>
<td>If edited while in the profile default the user will need to enter the load combinations by hand. If edited while in the analysis configuration the user can select the combination from a dropdown. For more information see the SteelConverter website</td>
</tr>
</tbody>
</table>
2.7 Changelog

3/8/15 - 1.00 – Base Version
3 STEEL Verification

3.1 ETABS to STEEL Comparison

3.1.1 Introduction

Following the creation of SteelConverter and Caltech VirtualShaker it was required to determine the accuracy of Caltech’s in-house nonlinear large displacement finite element analysis package STEEL. To accomplish this several comparison models were created, analyzed, and compared to results obtained from the industry standard finite element tool ETABS. For each of these models two analyses were conducted.

First, a linear elastic free vibration analysis was done to form a baseline comparison of basic properties of the models, namely the elastic stiffness, mass, and damping. For these analyses a static horizontal force was first applied to the model followed by a slow, linearly increasing horizontal acceleration to every mass resulting in a non-dynamic displacement. Finally, the horizontal acceleration was removed causing the structure to begin oscillation about its equilibrium point, eventually coming to rest due to damping. For each software the initial elastic displacement, displacement just before removal of the horizontal acceleration, approximate period, and amplitude of oscillation after several cycles were compared.

Following the free vibration analysis, a nonlinear pushover analysis was conducted. In this analysis a slow, linearly increasing horizontal acceleration was applied to each mass until the model was no longer able to converge. For each model the horizontal base shear was plotted against the top node displacement and the results of the two software systems are compared. Pushover analyses provide the user with an ultimate capacity of a building and is an excellent method of predicting the demands that will be placed on a structure during an earthquake.
Pushover analyses are also one of the most popular method of categorizing the strength of a building. Additionally, as this type of analysis can be highly nonlinear it is a good metric to determine the nonlinear accuracy of STEEL.

3.1.2 ETABS Model Description

In order to make a legitimate comparison between ETABS and STEEL the fiber element hinge property of ETABS was used to allow for nonlinear behavior in the pushover analysis. These hinges allow the user to define a fiber cross-section for each structural section, provide locations and areas for fibers, and designate a material for each fiber. While it would be possible to use the ETABS P-M2-M3 hinge for this analysis, it was decided to use the fiber hinges due to the similarity to STEEL fiber elements, thereby allowing for a more direct comparison, and, as stated in the CSI Analysis Reference, “The Fiber PMM hinge is more “natural” than the coupled PMM hinge described above, since it automatically accounts for interaction, changing moment-rotation curve, and plastic axial strain.” [2]. The fiber hinges allow for a more realistic analysis at the cost of computational intensity. For more information on the use of ETABS fiber hinges view the CSI Analysis Reference. The ETABS fiber hinges were created using the same fiber layout as described in Section 1.5.5 and was created for each section in the model. These hinges were then given a length corresponding to the element distribution discussed in Section 1.2.4, and were then applied to each ETABS element. Additionally, after applying the fiber hinges to the ETABS model it was then necessary to add large property modifiers to the area and moments of inertias in order to compensate for the elastic softening induced through the addition of fiber hinges. More information on this can be
seen in [19]. When running the model the ETABS nonlinear-static displacement controlled analysis was used.

Every model created for this comparison is available for download from the website by visiting the Downloads section of the website.

3.1.3 Material Model Description

The material model used in STEEL can be seen in Figure 3-1. This symmetric material model consists of a linear region of slope $E$ which ends at a stress of $\sigma_y$. The curve then contains a region of constant stress until the onset of strain hardening at $\varepsilon_{SH}$. The material model then uses a cubic ellipse with initial slope $E_{SH}$ to define the strain hardening behavior, culminating in an ultimate stress of $\sigma_u$ at a strain of $\varepsilon_u$. The curve then continues on the cubic ellipse until a strain of $2\varepsilon_u - \varepsilon_{SH}$ at which point the stress drops to a residual value of $\sigma_R$ (not shown in Figure 3-1).

For these analyses a Young’s modulus ($E$) of 199.948 GPa (29000 ksi) was chosen with a yield stress ($\sigma_y$) of 344.74 MPa (50 ksi). The onset of strain hardening ($\varepsilon_{SH}$) begins at 0.015 and with an initial cubic ellipse slope of 199.948 GPa) (29000 ksi). The ultimate stress of the material model ($\sigma_u$) is 448.16 MPa (65 ksi) occurring at a strain ($\varepsilon_u$) of 0.11. After a strain of 0.205 the material drops to its residual value ($\sigma_R$) of 172.37 MPa (25 ksi). Since rupture is not available in ETABS, no rupture strain was included in the STEEL material model. As ETABS allows the user to define the material model via a series of stress-strain coordinates, the ETABS material model was chosen to be exactly that of STEEL.
3.1.4 Analysis Discussion

A total of six models were analyzed in both STEEL and ETABS and their results compared. Each model was chosen to test a specific feature of STEEL to determine how the assumptions compare to those of ETABS. The models begin with a simple three element cantilever column followed by a three story single bay moment frame and a three story single bay chevron brace frame. From here, the models increase in complexity to a two bay three story moment frame and then a two bay three story chevron brace frame. Lastly, a twenty story moment frame based on Hall’s U20 building [1] was be analyzed.

For some of these models properties such as the section sizes and masses may be considered nonrealistic, however, as the goal of this study was to compare the results of ETABS and STEEL the actual specifics of the models are not as important as the comparative results from the two software systems. Often, particular masses or section sizes were chosen to

![Figure 3-1 - STEEL Material Model Description](image)

<table>
<thead>
<tr>
<th>Material</th>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STEEL</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ETABS</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

---

1. Hall, U20 building [1].
encourage a specific type of nonlinear behavior or to simplify the model so as to reduce the number of variables for comparison.

3.1.5 Cantilever Column

The goal of this model was to introduce the fewest complexities into the analyses. This simple cantilever column is meant to test the basic ability of STEEL to calculate properties such as linear stiffness, damping, and yielding. The column is divided into three elements, rather than a single element, to allow for the interaction of fixed connections to be tested.

3.1.5.1 Model Description

The cantilever column model is comprised of three elements 1.2192 m (4 ft.) tall. The bottom member is given a fixed end connection and there are no releases along the length of the member. Rather than use element self-weight to determine the mass in the ETABS model, a vertical force is assigned to every node and ETABS is told to use these forces for mass in all dynamic analyses. The top node is given a vertical force of 0.94 kN (0.212 kip) while the middle and bottom node are given a vertical force of 1.89 kN (0.424 kip). For the static load case, the top node is given a horizontal force of 2.2241 kN (0.5 kip). All three elements in this model were given a size of W12x106. Images showing the model, section assignments, and force assignments can be seen in Figure 3-2. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.
3.1.5.2 Free Vibration Analysis

The results from the free vibration analysis for both STEEL and ETABS can be seen in Figure 3-3. This plot shows a very strong correlation between the two softwares. The peak static displacement of the STEEL model due to the 2.2241 kN load is 0.49484 mm while the peak static displacement of the ETABS model is 0.48808 mm, or a difference of 1.384%. Following the slow horizontal acceleration the STEEL model had an amplitude of -4.95305 mm while the ETABS model had an amplitude of -4.891634 mm. Taking the difference about the static equilibrium point results in a difference of 1.256%. After releasing the mass by removing the horizontal acceleration, it was found that the STEEL model had a first peak amplitude of 4.75889 mm while the ETABS model had a peak amplitude of 4.69837 mm, or a difference of 1.288% about their respective equilibriums. Next, after 27 oscillations it was found that the STEEL model had an amplitude of 0.32065 mm while the ETABS model had an amplitude of
0.31046 mm, or a difference of 3.281%. Finally, by taking the average peak-to-peak time of the oscillations for both the STEEL and ETABS model resulted in periods of 0.0375 s and 0.037 s respectively.

![1 - Cantilever Column - Free Vibration](image)

**Figure 3-3 - Cantilever Column - Free Vibration Analysis**

3.1.5.3 Pushover Analysis

The results from the pushover analysis for the cantilever column can be seen in Figure 3-4. This plot shows identical linear stiffness, as was shown in the free-vibration analysis, and yield drifts of roughly 3.4% for STEEL and 3.5% for ETABS, or a difference of 2.3%. Additionally, post-yield the two models follow similar slopes however, the ETABS model failed to converge after a drift of roughly 6.86% while the STEEL model was capable of converging all the way through p-delta instability.
3.1.5.4 Discussion

Overall, the results from the cantilever column model showed a strong correlation between the STEEL and ETABS models for both linear and nonlinear, static and dynamic. The results from the free vibration analysis showed that both models had nearly identical damping characteristics and the difference in period seen in those results was due in part by the small difference in elastic stiffness. Additionally, the results from the pushover curve post-yield showed a similar stiffness despite the difference in initial yield displacement.
3.1.6 Three Story Moment Frame

The goal of this model was to determine STEEL’s capability at analyzing a simple moment frame. Here, the assumptions made by STEEL regarding the reduction in web area near moment connections for beams were tested and its results compared to the assumptions made by ETABS. For this model an identical section for both beams and columns was chosen to help reduce the complexity of the model. In subsequent models more realistic sections were be chosen to allow testing of more complex and real-world behaviors.

3.1.6.1 Model Description

This model is comprised of single bay of moment frame with a 3.6576 m (12 ft) story height and a 7.3152 m (24 ft) bay width over three stories for a total height of 10.9728 m (36 ft). The model uses W12x106 sections for both the columns and beams and uses an applied weight of 0.94 kN (0.212 kip) for the top nodes and a weight of 1.89 kN (0.424 kip) for all other nodes. For the static analysis horizontal forces of 2.2241 kN (0.5 kip) were applied to both top nodes. An image detailing this model can be seen in Figure 3-5. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.
3.1.6.2 Free Vibration Analysis

The results from the free vibration analysis can be seen in Figure 3-6. This figure once again shows a very strong correlation between the two softwares. The elastic displacement of the STEEL and ETABS models is 1.1932 mm and 1.1911 mm respectively, or a difference of 0.175%. Following the horizontal acceleration the peak displacement of the STEEL model was -14.099 mm while the peak displacement of the ETABS model was -13.952 mm, or a difference of 1.057%. After the removal of the horizontal acceleration the STEEL and ETABS models reach a first peak displacement of 13.527 mm and 12.904 mm respectively for a difference about their respective equilibrium points of 4.827% and after 27 oscillations it is found that the peak displacement of the two models is 3.904 mm and 3.747 mm for an equilibrium difference of 4.198%. Averaging the peak-to-peak times for the two models found approximate periods of 0.0547 s for STEEL and 0.0553 s for ETABS.
3.1.6.3 Pushover Analysis

An image of the results of the pushover analysis can be seen in Figure 3-7. As with the cantilever column pushover analysis, the two softwares give pushover curves that have nearly identical linear stiffness. The two softwares have slightly different initial yield forces, roughly 764 kN for STEEL and 660 kN for ETABS, however, the two curves converge following this initial discrepancy. Once again, the ETABS model fails to converge after a drift of roughly 1.8% while the STEEL model is capable of carrying the analysis through p-delta instability.
3.1.6.4 Discussion

Again, this model demonstrates the strong correlation between STEEL and ETABS for linear, nonlinear, static, and dynamic properties. The behavior from the free vibration analysis demonstrates that the linear properties of the models are nearly identical while the small differences in the pushover analysis are likely due to the fixed beam approximates made by STEEL as discussed in Section 1.5.5. Despite these differences, however, the two softwares show nearly identical load paths and have a very similar post-yield stiffness.
3.1.7 Three Story One Bay Chevron Brace Frame

The results from the three story one bay chevron brace frame will now be discussed. The goal of this modal was to determine the affectiveness of STEEL at modeling pinned connections. The chevron brace configuration was chosen due to its popularity among design engineers.

3.1.7.1 Model Description

This model is comprised of single bay of chevron braces with a 3.6576 m (12 ft) story height and a 7.3152 m (24 ft) bay width over three stories for a total height of 10.9728 m (36 ft). The model uses W12x136 sections for both the columns and braces and uses W21x83 sections for the beams. An applied weight of 0.94 kN (0.212 kip) for the top nodes and a weight of 1.89 kN (0.424 kip) for all other nodes is used to calculate the mass of the structure and, for the static analysis, horizontal forces of 444.82 kN (100 kip) were applied to both top nodes. An image detailing this model can be seen in Figure 3-8. Note that in accordance with the rules regarding the modeling of beams discussed in Section 1.2.2.3, the beam spanning the chevron brace is meshed at midpoint to accommodate that connection with the braces. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.
3.1.7.2 Free Vibration Analysis

The results of the free vibration analysis can be seen in Figure 3-9. The plot shows a static displacement of 5.574 mm and 5.786 mm for STEEL and ETABS respectively, or a difference of 3.673%. Following the application of the horizontal acceleration it was found that the STEEL model had an amplitude of -9.61 mm while the ETABS model had an amplitude of -9.81 mm, a difference of 2.07%. After removing the horizontal acceleration it was found that the STEEL model reached a peak amplitude of 9.75 mm while the ETABS model reached a peak amplitude of 10.06 mm, a difference of 3.07%. After completing 30 oscillations it was found that the STEEL model was oscillating with an amplitude of 0.2969 mm while the ETABS model was oscillating with an amplitude of 0.3125 mm, a difference of 4.97%. Finally, averaging the peak-to-peak oscillation time resulted in approximate periods of 0.0267 s and 0.0273 s for STEEL and ETABS respectively.
The results of this analysis demonstrate excellent agreement between the two softwares despite the difference in assumptions. As discussed in Section 1.5.5 the STEEL model assumes a non-zero moment capacity in the pinned beams while ETABS assumes a perfect pinned connection. However, as the beam sections are small, this difference becomes more negligible. Removing this assumption, however, would result in the STEEL model becoming more flexible, bringing the two results closer together. Additionally, while a difference in period between these two softwares may seem large in the plot, the difference in stiffness is nearly completely at fault for the difference in period. Assuming the brace frame is oscillating as a single degree of freedom system, a period differential of 0.0006 s is a difference of roughly 4.8% in stiffness, reasonably close to the 3.67% difference in elastic stiffness.

![Figure 3-9 - Three Story One Bay Chevron Brace Frame - Free Vibration Analysis](image-url)
3.1.7.3 Pushover Analysis

The results from the pushover analysis can be seen in Figure 3-10. As with the previous analyses, this plot shows a strong correlation between the results from the two softwares through the initiation of yielding. Following this, the ETABS model fails to converge, after a drift of approximately 0.3% while the STEEL model continues to converge until a drift of approximately 1%. The difference in linear stiffness seen in these results is due to the 3.67% linear stiffness differential between the two models shown in the previous section. Increasing the stiffness of the ETABS results by this percentage, shown in Figure 3-11, shows the similarity in the initial yield location between the two softwares.
3.1.7.4 Discussion

These models demonstrated STEEL’s ability to properly analyze a simple brace frame structure. The free vibration analysis showed nearly identical results in terms of stiffness and damping while the pushover analysis showed very similar yield locations. STEEL’s analysis algorithm allowed for convergence at a drift more than three times that of ETABS, allowing engineers to gain a better understanding of the ultimate capacity of a structure.
3.1.8 Two Bay Three Story Moment Frame

Next, the results of the two bay, three story moment frame structure will be discussed. The goal of this model was to test the capability of the two softwares to deal with the pass-through forces generated by the moment frames separated by a pinned connected beam. This model will also help evaluate STEELs ability to conduct nonlinear analyses on more complex moment frame structures.

3.1.8.1 Model Description

This model consists of two bays of moment frame, three stories tall, connected by pinned beams. An image showing the section assignments for this model can be seen in Figure 3-12. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.

For this model more realistic sizes of W24x94’s and W21x111’s were chosen for the beams in the moment frames and pass-through frame respectively while W12x136’s were chosen for the columns. As before the floors are 3.6576 m (12 ft) tall, yielding a total structure height of 10.9728 m (36 ft), with a bay width of 7.3152 m (24 ft).
Weight and force assignments for this model can be seen in Figure 3-13. This figure shows weight assignments of 0.94 kN (0.212 kip) for all nodes on the top floor and 1.89 kN (0.424 kip) for all other nodes. Additionally, the top nodes were given a horizontal force of 22.241 kN (5 kip) for the linear stiffness comparison.

![Figure 3-13 - Two Bay Three Story Moment Frame - Force Assignments](image)

3.1.8.2 Free Vibration Analysis

The results from the free vibration analysis can be seen in Figure 3-14. The free vibration analysis shows an initial elastic deformation of 5.3 mm and 5.5 mm for STEEL and ETABS respectively, or a difference of 0.84%. The application of the horizontal acceleration resulted in a pre-release amplitude of -6.55 mm and -6.46 mm for STEEL and ETABS respectively, or a baseline difference of 1.815%. Following the removal of the horizontal acceleration, the first peak for the two softwares was 6.158 mm and 6.179 mm, or a difference of 0.351% and after 30 oscillations it was found that STEEL had an amplitude of 0.581 mm while ETABS had an amplitude of 0.591 mm, a difference of 1.67%. Finally, averaging the peak-to-peak oscillation
time for STEEL and ETABS resulted in an approximate period of 0.0474 s and 0.0462 s respectively.

Again, the results from this analysis demonstrate agreement between the two softwares. The majority of the difference is found in the initial elastic displacement. Shifting the ETABS results down so the two results share a common static equilibrium displacement helps demonstrate this point. An image of this can be seen in Figure 3-15. This plot shows that the period of oscillation and damping between the two softwares is nearly identical. The marginal difference in elastic stiffness can be explained by the difference in pinned connections used by the two softwares. While ETABS assumes a perfect pinned connection, meaning zero moment capacity; STEEL assumes a small, but non-zero, capacity. As discussed in Section 1.5.5, STEEL assumes the middle two fibers of the web are given an area modifier such that the total area of the section is roughly preserved, this results in a small, but not insignificant, moment capacity resulting in an increase in stiffness. As these results show, the STEEL results are consistently stiffer than the ETABS results which, in part, is due to the extra capacity in the pass through elements. As true pinned connections actually contain a small amount of moment capacity, it is expected that the true result would be somewhere in-between these two results.
Figure 3-14 - Two Bay Three Story Moment Frame - Free Vibration Analysis

Figure 3-15 - Two Bay Three Story Moment Frame - Free Vibration - Shifted
3.1.8.3 Pushover Analysis

The results from the pushover analysis can be seen in Figure 3-16. This plot again shows an overall agreement in the elastic stiffness of the model, the small difference is due to the assumptions that pinned connections have a non-zero moment capacity. However, the yield paths of the two models is nearly identical. In fact, if the ETABS curve is scaled back by the 5.191% elastic stiffness differential the similarity between the two curves becomes apparent. An image of this can be seen in Figure 3-17. This plot shows that the two softwares produce nearly identical initial yield paths when the difference in stiffness is taken into account. This plot also shows the difference in convergence characteristics between the two softwares. While ETABS fails to converge after a drift of roughly 1%, the STEEL model is capable of converging through p-delta instability.

![Two Bay Three Story Moment Frame - Pushover](image)

*Figure 3-16 - Two Bay Three Story Moment Frame - Pushover Analysis*
3.1.8.4 **Discussion**

These models demonstrated the ability of STEEL to analyze pass-through forces in beams as well as properly determine the interaction between a moment frame and pinned connections. The differences found in the elastic stiffness between the two softwares is due largely in part to the assumptions made in STEEL. Allowing pinned connections to have a non-zero moment capacity results in additional stiffness that is not modeled in ETABS. While neither software is incorrect, a limited moment capacity in pinned connections is more realistic. Additionally, the similarities in the pushover curves between the two models, especially after the adjustment for discrepancies in elastic stiffness, demonstrates the ability of STEEL to properly determine the ultimate capacity of more complex moment frame structures.
3.1.9 Two Bay Chevron Brace Frame

Now the results from the two bay chevron brace frame will be discussed. The goal of this model was to test the STEEL pinned connection capabilities in a more complex setting. The two chevron brace frames separated by pass-through beams will be a good test of STEEL’s ability to correctly model the interaction of a multi-frame system. Additionally, the pinned pass-through beams will help determine what affect assuming a non-zero moment capacity pinned connection has on the overall stiffness and strength of the frame.

3.1.9.1 Model Description

This model consists of two bays of chevron bracing, three stories tall, connected by pinned beams. Images showing the section assignments and force assignments for this model can be seen in Figure 3-18 and Figure 3-19. These figures show the use of W12x136’s for the columns and braces, W21x83’s for the beams spanning the chevron brace, and W21x111’s for the pass-through beams in-between the two chevron brace frames. Additionally, a weight of 0.94 kN (0.212 kip) was assigned to all nodes on the top floor of the structure, except those at the brace connection point in the middle of the frame, while a weight of 1.89 kN (0.424 kip) was assigned to all other non-chevron connection point nodes. A horizontal force of 22.241 kN (100 kip) was assigned to all column nodes on the top floor of this frame to provide the baseline linear displacement used to calculate the elastic stiffness. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.
3.1.9.2 Free Vibration Analysis

The results of the free vibration analysis can be seen in Figure 3-20. As with all previous models, this plot shows a strong agreement between the two softwares. An initial elastic displacement of 0.5469 mm and 0.57862 mm was found for STEEL and ETABS respectively. Additionally, following the application of the horizontal acceleration it was found a displacement amplitude of -0.9467 mm for STEEL and -0.9813 mm for ETABS, or a difference of 3.52%. After removing the horizontal acceleration it was found that STEEL reached an
amplitude of 0.95842 mm on its first peak while ETABS reached an amplitude of 1.006 for a difference of 4.7%. After 25 oscillations the STEEL model had an amplitude of oscillation of 0.05189 mm while ETABS had an amplitude of 0.05652 mm; a difference of 8.19%. Finally, averaging the period of oscillation over several cycles resulted in approximate periods of 0.03225 s for STEEL and 0.325 s for ETABS; a difference of 1.53%.

The results of this analysis showed both STEEL and ETABS are calculating the stiffness and damping properties of the frame in a similar manner despite the difference in approaches. Shifting the ETABS result down such that the static equilibrium point of the two models coincide, as shown in Figure 3-21, reveals more clearly that the two softwares yield very similar damping characteristics. The difference in period between the two models is due in part to the difference in stiffness. Assuming the system is oscillating as a single degree of freedom system it is easy to show that a period differential of 0.00005 seconds correlates to an approximate stiffness discrepancy of 1.53%, nearly a quarter of the total elastic stiffness difference. The resulting stiffness difference between the two softwares is primarily caused by the difference in assumptions regarding the moment capacity of pinned connections. As STEEL assumes a small amount of moment resistance at pinned connections it is expected that the results of the analysis would be slightly stiffer than that of ETABS.
**Figure 3-20** - Two Bay Three Story Chevron Brace Frame - Free Vibration Analysis

**Figure 3-21** - Two Bay Three Story Chevron Brace Frame - Free Vibration - Shifted
3.1.9.3 Pushover Analysis

The results from the pushover analysis can be seen in Figure 3-22. This plot shows a strong agreement in the elastic stiffness of the two models as well as an agreement on the location of initial nonlinearity. As was shown in the previous section, the difference in elastic stiffness is due largely in part by the assumptions that pinned connections have a non-zero moment capacity. To help illustrate this point the stiffness of the ETABS results was artificially increased by the calculated 5.484% and is shown in Figure 3-23.

As with all previous models, this analysis demonstrates the ability of STEEL to converge well beyond that of ETABS. For this pushover analysis it was found that the ETABS analysis failed to converge after the initializing of nonlinearity, at a drift of approximately 0.3%, while the STEEL analysis was able to converge until a drift of approximately 1%, roughly three times as far.
Figure 3-22 - Two Bay Three Story Chevron Brace Frame - Pushover Analysis
**Figure 3-23 - Two Bay Three Story Chevron Brace Frame - Pushover Analysis - Scaled**

### 3.1.9.4 Discussion

These results showed STEEL’s ability to properly analyze a more complex braced frame system. The differences found in the elastic stiffness between these two models was largely due to a difference assumptions between STEEL and ETABS. STEEL assuming a non-zero moment capacity for pinned connections acts to increase the stiffness of the analysis over that of ETABS thereby affecting the results. STEEL was able to bring the structure to p-delta instability; a drift more than three times that of ETABS.
3.1.10 Twenty Story Moment Frame

Now the results from the twenty story moment frame will be discussed. The goal of this model was to test STEEL’s ability to analyze a complex moment frame. In taller structures geometric instability plays a more significant role than in the shorter structures analyzed up until this point. Testing a tall frame will help determine the accuracy of STEEL’s large displacement algorithm. The sizes used for this model are based on the U20 structure proposed by Hall. More information on this structure can be seen in [1].

3.1.10.1 Model Description

This model consists of three bays of moment frame with a bay with of 7.3152 m (24 ft) and a story height of 3.6576 m (12 ft), bringing the total height of the structure to 73.152 m (240 ft). At every node a weight of 44.482 kN (10 kip) was assigned in addition to a horizontal force of 111.2 kN (25 kip) assigned to the nodes on the top floor. An image showing the overall shape of the structure can be seen in Figure 3-24 while a table of the column and girder schedule can be seen in Figure 3-25. This model used a Rayleigh damping model with a mass proportional coefficient of 0 and a stiffness proportional coefficient of 0.00196.
Figure 3-24 - Twenty Story Moment Frame - Section Assignments

<table>
<thead>
<tr>
<th>Label</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>C10</td>
<td>W14x109</td>
</tr>
<tr>
<td>C9</td>
<td>W14x132</td>
</tr>
<tr>
<td>C8</td>
<td>W14x159</td>
</tr>
<tr>
<td>C7</td>
<td>W14x176</td>
</tr>
<tr>
<td>C6</td>
<td>W14x211</td>
</tr>
<tr>
<td>C5</td>
<td>W14x257</td>
</tr>
<tr>
<td>C4</td>
<td>W14x283</td>
</tr>
<tr>
<td>C3</td>
<td>W14x311</td>
</tr>
<tr>
<td>C2</td>
<td>W14x342</td>
</tr>
<tr>
<td>C1</td>
<td>W14x370</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Label</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>C20</td>
<td>W21x122</td>
</tr>
<tr>
<td>C19</td>
<td>W14x146</td>
</tr>
<tr>
<td>C18</td>
<td>W14x146</td>
</tr>
<tr>
<td>C17</td>
<td>W24x162</td>
</tr>
<tr>
<td>C16</td>
<td>W24x176</td>
</tr>
<tr>
<td>C15</td>
<td>W24x176</td>
</tr>
<tr>
<td>C14</td>
<td>W27x178</td>
</tr>
<tr>
<td>C13</td>
<td>W27x178</td>
</tr>
<tr>
<td>C12</td>
<td>W30x191</td>
</tr>
</tbody>
</table>

G10  W27x83
G9   W27x94
G8   W30x99
G7   W30x108
G6   W30x116
G5   W30x116
G4   W30x116
G3   W30x116
G2   W30x116

Figure 3-25 - Twenty Story Moment Frame - Column and Girder Schedule
3.1.10.2 Free Vibration Analysis

The results from the free vibration analysis can be seen in Figure 3-26. As with all previous analyses, the agreement between the STEEL and ETABS models is very strong. These analyses found a static displacement due to the initial horizontal load of 98.63 mm for STEEL and 99.94 mm for ETABS; a difference of 1.311%. Following the application of the horizontal acceleration an amplitude of -105.91 mm was found for STEEL and an amplitude of -106.02 mm was found for ETABS; a difference of 0.108%. After the horizontal acceleration was removed, the amplitude of the first peak was found to be 105.13 mm for STEEL and 105.25 mm for ETABS; a difference of 0.116%. After 27 oscillations an oscillation amplitude of 48.07 mm was found for ETABS while an amplitude of 48.07 mm was found for STEEL; a difference of 0%. Finally, averaging several peak-to-peak oscillation times found both models had an approximate period of 1.04 s.

As this analysis shows, STEEL is capable of accurately calculating basic model properties, such as stiffness and damping, for a significantly more complex system. The assumptions made by STEEL regarding the fiber properties near a fixed connection work well to simulate a taller system and the geometric nonlinearity algorithms used in STEEL agree closely with those used in ETABS.
3.1.10.3 Pushover Analysis

The results from the pushover analysis can be seen in Figure 3-27. This plot shows the same strong agreement in the linear region as with the free vibration analysis. The location of the onset of nonlinearity is nearly identical between the two softwares indicating that STEEL's geometric nonlinear algorithms do well in simulating the stresses in the various element fibers. As with the previous analyses, the ETABS model failed to converge shortly after the onset of nonlinearity, a drift of approximately 0.46%, while the STEEL model was able to converge through the onset of p-delta instability.
3.1.10.4 Discussion

These results demonstrated STEEL’s ability to properly analyze a significantly more complex system. This twenty story moment frame requires accurate implementation of a number of nonlinear algorithms, and STEEL was capable of matching the results from ETABS for both the free vibration and pushover analyses. Additionally, as with all previous analyses STEEL was capable of allowing the structure to drift through p-delta instability while the ETABS analyses failed to converge shortly following the onset of nonlinearity.
3.2 Perform3D STEEL Comparison

3.2.1 Introduction

While the results from the ETABS to STEEL comparison demonstrated a very strong correlation between the two softwares for static stiffness, yielding, and damping, the limitations of the ETABS fiber analysis prevented proper verification of the nonlinear analysis capabilities of STEEL. Therefore, it was decided to repeat the comparison with a software more capable of performing highly nonlinear analysis. Perform3D is another industry standard software used by many engineers for advanced analysis.

As with the previous comparisons each analysis will first consist of a linear elastic free vibration analysis to generate a baseline of basic properties such as elastic stiffness, mass, and damping. The same static horizontal force used in previous analyses is again applied to the model followed by the same slow, linearly increasing horizontal acceleration to the mass of the model. Following this, the acceleration is removed and the model oscillates about its new equilibrium position before eventually damping out. For each model the initial elastic displacement, displacement just before removal of the horizontal acceleration, approximate period, and amplitude of oscillation after several cycles were compared.

Following the free vibration analysis, the same nonlinear pushover analysis was conducted for each model. As with before, a slow, linearly increasing horizontal acceleration was applied to each mass until the model was no longer able to converge. Then the horizontal base shear was plotted against the top node displacement.
3.2.2 Perform3D Model Description

The Perform3D model was constructed to replicate STEEL models as closely as possible. All elements were given a fiber cross section with fiber areas matching those in STEEL as discussed in Section 1.5.5. Each fiber was designated with the same material model and the appropriate fiber area reductions were applied for fixed end connections and pinned connections. For brace elements an actual moment release element was applied to either end of the compound element to simulate the manner with which STEEL treats braces. For each Perform compound element a rigid end offset was applied to either end with the appropriate length matching that of STEEL. Furthermore, shear deformation in beams and columns was enabled using shear areas defined by,

\[ A_{12} = dt_f \]
\[ A_{21} = 2bt_w \]

Where \( A_{12} \) is the shear area along the web and \( A_{21} \) is the shear area along the flanges. The shear modulus was found using,

\[ G = \frac{E}{2(1 + \nu)} \]

Where \( E \) is the Young’s Modulus and \( \nu \) is the Poisson’s ratio taken to be 0.3.

As STEEL is a 2D analysis software, special restraints needed to be placed on the Perform3D model to ensure that out-of-plane behavior was eliminated. Therefore, all non-base nodes were given fixed conditions for the H2, RH1, RZ directions. All base nodes were given full fixed conditions.
3.2.3 Material Model Description

Due to limitations in both STEEL and Perform3D, it is not possible to get an exact match in the material model. As discussed previously, the STEEL material model is symmetric and consists of a linear elastic region, followed by a region of constant stress, which is then preceded by a strain hardening region represented by a cubic ellipse. Following this the curve will then drop to a defined residual value. In Perform3D, the nonlinear, non-buckling material model consists of a linear elastic region, followed by a secondary linear region, leading to a region of constant stress before reaching a region of negative slope before ultimately dropping to a residual value.

In an attempt to match these material models as closely as possible, the material used for the ETABS to STEEL comparison described in Section 3.1.3 was modified. The steel material model begins with a linear slope of 199.95 GPa (29000 ksi), yielding at 344.737 MPa (50 ksi). The region of constant stress continues until a strain of 0.004 where a cubic ellipse with an intersection slope of 3467257 MPa (502883 ksi) and ultimate stress of 448.1 MPa (65 ksi) at a strain of 0.11. At a strain of 0.205 the STEEL material model drops to a residual stress of 0 MPa (0 ksi).

The Perform3D model also has a linear slope of 199.95 GPa (29000 ksi) with a yield stress of 344.747 MPa (50 ksi). The secondary linear region ends at a stress of 441.26 (64 ksi) and a strain of 0.05. The region of constant stress ends at a strain of 0.16 and then decreases to a stress of 344.747 MPa (50 ksi) at a strain of 0.207272. The Perform3D material will then remain at this stress until a strain of 0.218181 after which it drops to 0 stress. An image of this can be seen in Figure 3-28.
Figure 3-28 - Material Model Description - Perform3D vs. STEEL
3.2.4 Perform3D Analysis Limitations

While conducting several of the linear elastic time history analyses in Perform3D it was noticed that for the specific models, namely the two brace frame and twenty story moment frame, while STEEL and Perform3D yielded similar results for both the elastic stiffness, initial release point, mass, and period; following the removal of the horizontal acceleration the Perform3D model exhibited practically no damping. While the exact method is not perfectly clear it was revealed by CSI customer support that for steel fiber elements 0% of the axial stiffness for Rayleigh damping is used [20]. This was done by the Perform3D programmers to combat excessive energy dissipation found in their analyses when these axial fibers yield, resulting in an axial expansion. The issue is said to be caused by coupling between the bending and axial affects that is generally not present before yielding or cracking. Through their own analyses it was found that this affect resulted in excessive energy dissipation when using stiffness proportional Rayleigh damping [20]. The decision to remove axial stiffness from steel fiber elements was done to yield a more conservative answer in users’ analyses.

For most users this affect will not cause a large error in their simulation because, in general, fiber elements are only used in small, localized regions. However, in order to achieve a valid comparison between these two softwares the usage of fiber elements throughout the Perform3D model results in a practically no damping being applied in specific models. As a result, the free vibration analysis for both chevron brace frame models as well as the twenty story model resulted in a solution which could not be properly compared to the results of STEEL. However, for each of these models the static stiffness, initial release point from the analysis can still be compared as well as the results from the pushover analysis. For the
cantilever column, three story moment frame, and two bay three story moment frame models
this approximation of no axial damping done by Perform3D is small enough that a relatively
accurate comparison can still be made from the free vibration analysis as these models act
primarily in bending.

Additionally, Perform3D does not include geometric updating. While for most typical
analyses this will not result in a considerable difference, when calculating the ultimate capacity
of the building as well as determine at what drift the structure experiences geometric instability
the lack of geometric updating can begin to play a large difference. It is therefore expected that
at very large strains the results of the STEEL and Perform analyses will diverge and it is expected
for some of the more realistically weighted buildings that the STEEL analyses will become
unstable at a lower drift. Furthermore, for extremely high drifts it was seen that the results
from the STEEL pushover analysis can show a rapid increase in base shear following the collapse
of the structure. It is believed this is caused by the manner in which STEEL conducts pushover
analyses. Due to the monotonically increasing applied accelerations the amount of force being
applied to the structure increases regardless of whether the structure is capable of resisting this
force. However, since this force must be resolved in the system the base shear of the structure
will continue to increase. Through examining the deformed shape of the structure it is clear
that this increase in base shear is not indicative of an increase in structural capacity post-
collapse but rather an artifact of conducting a force controlled analysis.
3.2.5 Analysis Discussion

A total of six models were analyzed in both STEEL and Perform and their results compared. Each model was chosen to test a specific feature of STEEL to determine how the assumptions compare to those of Perform. The models begin with a simple three element cantilever column followed by a three story single bay moment frame and a three story single bay chevron brace frame. From here, the models increase in complexity to a two bay three story moment frame and then a two bay three story chevron brace frame. Lastly, a twenty story moment frame based on Hall’s U20 building [1] was analyzed.

For some of these models, properties such as the section sizes and masses may be considered nonrealistic, however, as the goal of this study was to compare the results of Perform and STEEL the actual specifics of the models are not as important as the comparative results from the two software systems. Often, particular masses or section sizes were chosen to encourage a specific type of nonlinear behavior or to simplify the model so as to reduce the number of variables for comparison.
3.2.6 Cantilever Column

The goal of this model was to introduce the fewest complexities into the analyses. This simple cantilever column is meant to test the basic ability of STEEL to calculate properties such as linear stiffness, damping, and yielding. The column is divided into three elements, rather than a single element, to allow for the interaction of fixed connections to be tested.

3.2.6.1 Model Description
A detailed description of this model is presented in Section 3.1.5.1

3.2.6.2 Free Vibration Analysis
The results from the free vibration analysis for the cantilever column can be seen in Figure 3-29. This plot shows nearly identical results from the two softwares. Under the 2.2241 kN load it was found that the STEEL model deflected 0.4948 mm while the Perform model deflected 0.4966 mm, a difference of 0.35%. Following the slow horizontal acceleration the STEEL model had an amplitude of -4.9531 mm while the Perform model had an amplitude of -4.9747 mm, or a difference of 0.44%. After removal of the horizontal acceleration, it was found that the STEEL model had a first peak amplitude of 4.7589 mm while the Perform model had an amplitude of 4.7862 mm, a difference of 0.57%. After 27 oscillations it was found that the STEEL model had an amplitude of 0.3205 mm while the Perform model had an amplitude of 0.3214 mm, a difference of 0.27%. Finally, by taking the average peak-to-peak time of the oscillations for both STEEL and ETABS models resulted in approximate periods of 0.0375 s and 0.037 s, a difference of 0.35%.
3.2.6.3 Pushover Analysis

The results from the pushover analysis can be seen in Figure 3-30. This plot shows nearly identical linear stiffness, as was shown in the free-vibration analysis, and shows both structures yielding at around 4.3% with maximum capacities of approximately 549 kN for STEEL and 537 kN for Perform. Additionally, both structures reach instability at nearly the same strain. Both analyses demonstrate nearly identical nonlinear behavior and post-yield slopes and both softwares were also capable of converging at large strains.
3.2.6.4 Discussion

The free vibration and pushover analyses demonstrated a very strong correlation between the STEEL and Perform models for both linear and nonlinear, static and dynamic analyses. The results from the free vibration showed both models were able to calculate basic properties of the models, i.e. stiffness and damping, while the pushover curve demonstrated the ability of both softwares to capture nonlinear behavior.

Furthermore, these analyses helped to verify the choice of the STEEL and Perform material models. The small differences in post-yield behavior are caused by the differences in the stress strain curves between the two softwares. The cubic ellipse strain hardening region of STEEL results in a pushover curve that has smoother curves as opposed to the more straight line
pushover curve developed by Performs tri-linear material model. The difference in ultimate capacity is caused by the slight difference in ultimate stress chosen between the two softwares. This was done to allow for a closer mat in the strain hardening region of the curve. Finally, STEEL reaching a larger drift before instability is caused by Perform requiring a non-infinite slope between the post-ultimate linear region and the residual region.
3.2.7 Three Story Moment Frame

The goal of this model was to determine STEEL’s capability at analyzing a simple moment frame. Here, the assumptions made by STEEL regarding the reduction in web area near moment connections for beams were tested and its results compared to the assumptions made by Perform. For this model an identical section for both beams and columns was chosen to help reduce the complexity of the model. In subsequent models more realistic sections were be chosen to allow testing of more complex and real-world behaviors.

3.2.7.1 Model Description
A Detailed description of this model is presented in Section 3.1.6.1.

3.2.7.2 Free Vibration Analysis
The results from the free vibration analysis can be seen in Figure 3-31. This figure again shows a very strong correlation between the two softwares. The elastic displacement of the STEEL and Perform models are 1.1932 mm and 1.2394 mm respectively, a difference of 3.73%. Following the horizontal acceleration the peak amplitude of the STEEL model was -14.0994 mm while that of the Perform model was -14.4923 mm, a difference of 2.71%. After removing the horizontal acceleration it was found that the first peak had an amplitude of 13.569 mm for STEEL and 13.5798 mm for Perform, a difference of 0.39%. After 27 oscillations it was found that the amplitude of oscillation for STEEL was 3.0964 mm while for Perform it was 3.2397 mm, a difference of 4.42%. Finally, averaging the peak-to-peak times for the two models found an approximate period of 0.066 s for STEEL and 0.06599 s for Perform.
3.2.7.3 Pushover Analysis

The results of the pushover analysis can be seen in Figure 3-32. This plot again shows the agreement between these two softwares when analyzing nonlinear behavior in a structure. Both analyses yield identical linear elastic stiffnesses with yield happening at a drift of approximately 1.2%. The ultimate capacity of both analysis is approximately 1185 kN and both analysis show instability at similar strains. Additionally the STEEL and Perform analyses demonstrate nearly identical nonlinear behavior and post-yield slopes and both softwares were also capable of converging at large strains.
3.2.7.4 Discussion
These analyses demonstrate STEEL’s ability to accurately calculate linear, nonlinear, static, and dynamic properties. The small differences in post-yield behavior seen in the pushover analysis is caused by the differences in material models. However, despite these differences the two softwares show nearly identical load paths and have very similar post-yield stiffness.
3.2.8 Three Story One Bay Chevron Brace Frame

The results from the three story one bay chevron brace frame will now be discussed. The goal of this modal was to determine the affectiveness of STEEL at modeling pinned connections. The chevron brace configuration was chosen due to its popularity among design engineers.

3.2.8.1 Model Description
A Detailed description of this model is presented in Section 3.1.7.1

3.2.8.2 Free Vibration Analysis
As discussed in Section 3.2.4 limitations in the implementation of stiffness proportional damping make a direct comparison of the free vibration analysis between these two softwares impossible. However, the linear stiffness and amplitude just prior to the removal of the horizontal acceleration can still be compared. A table summarizing this can be seen in Figure 3-33. This table showed a static displacement for STEEL of 5.5738 mm and a static displacement of 5.4645 mm for Perform, or a difference of 1.96%. After applying the horizontal acceleration the amplitude of the STEEL analysis was -9.6103 mm while for Perform the amplitude was -9.3579 mm, a difference of 2.63%.

<table>
<thead>
<tr>
<th>Static Displacement</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Displacement (mm)</strong></td>
</tr>
<tr>
<td>STEEL</td>
</tr>
<tr>
<td>Perform</td>
</tr>
<tr>
<td><strong>Diff</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Release Point</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Displacement (mm)</strong></td>
</tr>
<tr>
<td>STEEL</td>
</tr>
<tr>
<td>Perform</td>
</tr>
<tr>
<td><strong>Diff</strong></td>
</tr>
</tbody>
</table>

Figure 3-33 - Three Story One Bay Chevron Brace Frame - Free Vibration Analysis -Perform
3.2.8.3 Pushover Analysis

An image of the results of the pushover analysis can be seen in Figure 3-34. This plot again shows a strong correlation between the two softwares. Both analyses have the same linear stiffness and yield at a drift of approximately 0.3%. STEEL has an ultimate capacity of 12688 kN while Perform has an ultimate capacity of 13232 kN, a difference of 4.1%. Both analysis then experience instability due to buckling of the first floor brace at approximately the same drift.

![Three Story Chevron Brace Frame - Pushover](image)

**Figure 3-34 - Three Story Chevron Brace Frame - Pushover - Perform**

3.2.8.4 Discussion

The results of this analysis demonstrate STEEL’s ability to properly calculate the properties of a chevron brace frame. While it was not possible to check STEEL’s free vibration properties against Perform, from the ETABS comparison in Section 3.1.7 it is known that the
results compare well. From the Perform analyses however, it is clear that the linear stiffness as well as nonlinear stiffness of the two models are very similar. The difference in ultimate capacity of the two softwares can be attributed to the different material models used by both. However, up until the onset of instability both softwares provide nearly identical results as well as similar post-buckling behavior up until the lack of large deformation begins to play a factor in the calculated capacity of the structure.
3.2.9 Two Bay Three Story Moment Frame

Next, the results of the two bay, three story moment frame structure will be discussed. The goal of this model was to test the capability of the two softwares to deal with the pass-through forces generated by the moment frames separated by a pinned connected beam. This model will also help evaluate STEELs.

3.2.9.1 Model Description
A Detailed description of this model is presented in Section 3.1.8.1.

3.2.9.2 Free Vibration Analysis
The results from the free vibration analysis can be seen in Figure 3-35. This analysis shows an initial elastic deformation of 5.0696 mm for STEEL and 5.2402 mm for Perform, a difference of 3.26%. The application of the horizontal acceleration resulted in a pre-release amplitude of -6.2893 mm for STEEL and -6.4344 mm for Perform, or a difference of 2.25%. Following the removal of the horizontal acceleration, the first peak for the two softwares was 5.9378 mm for STEEL and 5.8545 mm for Perform, or a difference 1.42%. After 27 oscillations it was found that STEEL had an amplitude of 0.7033 mm while Perform had an amplitude of 0.7182 mm, or a difference of 2.07%. Finally, averaging the peak-to-peak oscillation time for STEEL and Perform resulted in an approximate period of 0.046 s for both STEEL and Perform.
3.2.9.3 Pushover Analysis
The results from the pushover analysis can be seen in Figure 3-36. This plot again shows an overall agreement in the elastic stiffness of the model as well as the post-yield behavior. Both softwares begin yielding at a drift of approximately 0.71% and have ultimate capacities of approximately 3317 kN. The two softwares also show very similar instability strains and nonlinear stiffness.
3.2.9.4 Discussion

The results from these models demonstrated STEEL’s ability to analyze pass-through forces in beams as well as properly determine the interaction between a moment frame and pinned connections. Unlike the ETABS to STEEL comparison for this model done in Section 3.1.8, Perform was capable of closely replicating the linear stiffness observed in STEEL due to giving a non-zero moment capacity in pinned connections. This also resulted in a more similar pushover curve from the elastic region all the way through to instability.
3.2.10 Two Bay Chevron Brace Frame

Now the results from the two bay chevron brace frame will be discussed. The goal of this model was to test the STEEL pinned connection capabilities in a more complex setting. The two chevron brace frames separated by pass-through beams will be a good test of STEEL’s ability to correctly model the interaction of a multi-frame system. Additionally, the pinned pass-through beams will help determine what affect assuming a non-zero moment capacity pinned connection has on the overall stiffness and strength of the frame.

3.2.10.1 Model Description
A Detailed description of this model is presented in Section 3.1.9.1.

3.2.10.2 Free Vibration Analysis
As discussed in Section 3.2.4 limitations in the implementation of stiffness proportional damping make a direct comparison of the free vibration analysis between these two softwares impossible. However, the linear stiffness and amplitude just prior to the removal of the horizontal acceleration can still be compared. A table summarizing this can be seen in Figure 3-37. This table showed a static displacement for STEEL of 0.54691 mm and a static displacement of 0.5394 mm for Perform, or a difference of 1.393%. After applying the horizontal acceleration the amplitude of the STEEL analysis was -0.94676 mm while for Perform the amplitude was -0.933840 mm, a difference of 1.383%.
\[ W = 18.88 \text{ kN} \]

<table>
<thead>
<tr>
<th>Static (mm)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>STEEL</td>
<td>0.54691</td>
</tr>
<tr>
<td>Perform</td>
<td>0.5394</td>
</tr>
<tr>
<td>Diff</td>
<td>1.393%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Release Point (mm)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>STEEL</td>
<td>-0.94676</td>
</tr>
<tr>
<td>Perform</td>
<td>-0.933840</td>
</tr>
<tr>
<td>Diff</td>
<td>1.383%</td>
</tr>
</tbody>
</table>

Figure 3-37 - Two Bay Chevron Brace Frame - Free Vibration Analysis - Perform

3.2.10.3 Pushover Analysis

An image of the results of the pushover analysis can be seen in Figure 3-38. This plot again shows a strong correlation between the two softwares. Both analyses have the same linear stiffness and yield at a drift of approximately 0.3\%. STEEL has an ultimate capacity of 26380 kN while Perform has an ultimate capacity of 26614 kN, a difference of 0.88\%. Both analyses then experience instability due to buckling of the first floor brace at approximately the same drift.
3.2.10.4 Discussion

The results of this analysis demonstrate STEEL’s ability to properly calculate the properties of a combined chevron brace frames with pass-through forces. While it was not possible to check STEEL’s free vibration properties against Perform, from the ETABS comparison in Section 3.1.9 it is known that the results compare well. From the Perform analyses however, it is clear that the linear stiffness as well as nonlinear stiffness of the two models are very similar. The small difference in ultimate capacity of the two softwares can be attributed to the different material models used by both. However, up until the onset of instability both softwares provide nearly identical results as well as similar post-buckling behavior up until the lack of large deformation begins to play a factor in the calculated capacity of the structure.
3.2.11 Twenty Story Moment Frame

Now the results from the twenty story moment frame will be discussed. The goal of this model was to test STEEL’s ability to analyze a complex moment frame. In taller structures geometric instability plays a more significant role than in the shorter structures analyzed up until this point. Testing a tall frame will help determine the accuracy of STEEL’s large displacement algorithm. The sizes used for this model are based on the U20 structure proposed.

Model Description

A Detailed description of this model is presented in Section 3.1.10.1.

3.2.11.1 Free Vibration Analysis

As discussed in Section 3.2.4 limitations in the implementation of stiffness proportional damping make a direct comparison of the free vibration analysis between these two softwares impossible. However, the linear stiffness and amplitude just prior to the removal of the horizontal acceleration can still be compared. A table summarizing this can be seen in Figure 3-39. This table showed a static displacement for STEEL of 98.63 mm and a static displacement of 101.5 mm for Perform, or a difference of 2.82%. After applying the horizontal acceleration the amplitude of the STEEL analysis was -105.9053 mm while for Perform the amplitude was -105.3934 mm, a difference of 0.49%.

<table>
<thead>
<tr>
<th>Static Displacement</th>
<th>Displacement (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>STEEL</td>
<td>98.6333</td>
</tr>
<tr>
<td>Perform</td>
<td>101.5000</td>
</tr>
<tr>
<td>Diff</td>
<td>2.82%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Release Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement (mm)</td>
</tr>
<tr>
<td>STEEL</td>
</tr>
<tr>
<td>Perform</td>
</tr>
<tr>
<td>Diff</td>
</tr>
</tbody>
</table>

Figure 3-39 - Twenty Story Moment Frame - Free Vibration Analysis - Perform
3.2.11.2 Pushover Analysis

An image of the results of the pushover analysis can be seen in Figure 3-40. This plot again shows a strong correlation between the two softwares. Both analyses have the same linear stiffness and yield at a drift of approximately 0.4%. Both softwares have an ultimate capacity of approximately 5000 kN. The pushover plot also shows the similarities in the displacement at which instability occurs.

![Twenty Story Moment Frame - Pushover](image)

**Figure 3-40 - Twenty Story Moment Frame - Pushover Analysis - Perform**

3.2.11.3 Discussion

These results demonstrated STEEL’s ability to properly analyze a significantly more complex system. This twenty story moment frame requires accurate implementation of a number of nonlinear algorithms, and STEEL was capable of matching the results from ETABS for both the free vibration and pushover analyses. While it was not possible to check STEEL’s free
vibration properties against Perform, from the ETABS comparison in Section 3.1.10 it is known that the results compare well. From the Perform analyses however, it is clear that the linear stiffness as well as nonlinear stiffness of the two models are very similar. The small difference in ultimate capacity of the two softwares can be attributed to the different material models used by both. However, up until the onset of instability both softwares provide nearly identical results as well as similar post-instability behavior up until the lack of large deformation begins to play a factor in the calculated capacity of the structure.
3.3 Twenty Story Analyses

3.3.1 Introduction

Structural collapse has been a major research interest at Caltech for some time. As a result, Dr. John Hall created the analysis tool STEEL as a means to allow Caltech researchers to conduct analyses of structures through the inelastic range and past the point of geometric instability. However, due to the complexity of STEEL it has been difficult to both teach new Caltech researchers as well as researchers and engineers from other universities how to build and analyze models with this software. Therefore, Caltech Virtual Shaker was created by Christopher Janover, P.E. to reduce the learning curve associated with STEEL in addition to providing a platform for researchers to create, analyze, and conduct post-processing on structures without the need of expensive computer clusters.

The goal of this study is to first verify the results of the STEEL and Caltech Virtual Shaker analysis tools against industry standard solvers ETABS and Perform by demonstrating to other researchers and engineers the ability of STEEL to accurately and reliably analyze structures in collapse scenarios. These analyses utilize nearly every feature STEEL and Caltech Virtual Shaker are capable of providing. For more information on the use of these softwares see the Caltech Virtual Shaker and SteelConverter manuals. Following this comparison two Hall designed twenty story structures, U20 and J20 [1], will be subjected to the ground motion from the recent Nepal earthquake. Their results will first be estimated using Song’s new ground intensity measure, Peak Filtered Acceleration (PFA) [21], and then be analyzed in STEEL via a time history analysis. These models will each be broken down into two separate cases, one with perfect welds, designated by a “P”, and one with brittle welds, designated by a “B”, to demonstrate the
affect the inclusion of fiber fracture has on the results of a model with respect to structural collapse.

3.3.2 Software Description

3.3.2.1 STEEL & Caltech Virtual Shaker

STEEL is a 2D, nonlinear, large displacement, fiber based finite element tool created by Dr. Hall of the California Institute of Technology for the analysis of STEEL structures. STEEL is a text based tool that allows users a large amount of freedom in the type of models they run and includes features such as foundation springs, foundation walls, diaphragms, and fiber fracture with a complex material model. The capability of STEEL to run large displacement analyses makes it ideal for simulation of structural collapse through pushover or ground accelerations. More information on STEEL can be seen in [1] with manuals for the software found in [22].

The aim of this Caltech VirtualShaker is to facilitate and streamline the process of conducting advanced non-linear models by allowing users to create, upload, and analyze these models in the cloud. All analyses are conducted using STEEL, software is used widely in the Civil engineering department, and VirtualShaker aims to make this software more widely available.

VirtualShaker utilizes the SteelConverter tool created by Christopher Janover, P.E. to convert ETABS models to a format STEEL is capable of understanding. With this software, models that used to take days to construct in STEEL now take minutes thereby eliminating a large amount of the overhead cost that comes with creating a new model. Additionally, this conversion tool helps professors at other universities as well as professional engineers to conduct non-linear analyses using STEEL. As ETABS is software many Civil Engineering professors and engineers understand well, the learning curve that comes with using STEEL is greatly diminished, reducing the amount of time it takes a user to begin using STEEL.
3.3.2.2 ETABS
ETABS is software created by Computers & Structures Inc. (CSI) for the analysis of steel and concrete structures and can be regarded as the industry standard analysis package for structural engineers. This software is a graphical analysis tool that allows users to create finite element models of varying degrees of complexity ranging from simple elements to custom fiber elements. ETABS is capable of running both large displacement and nonlinear analyses. The ease of use and familiarity is what made ETABS the ideal software to base Caltech Virtual Shaker from. More information on ETABS can be found on CSI’s website or in their manual found in [2].

3.3.2.3 Perform3D
Perform3D is also created by Computer & Structures Inc. for the analysis of steel and concrete structures, however, Perform is often seen as a more advanced solver; capable of doing nonlinear inelastic analyses that ETABS tends to struggle with. As a result, it is not as widely used in the structural engineering community but is regarded as one of the best packages available to run capacity based analyses. Perform is also capable of creating fiber based elements, however, there is no ability for large displacement analyses. More information on Perform can be found on CSI’s website.

3.3.3 Model Description
3.3.3.1 U20 Model Geometry
The first building used for this study is the U20 structure designed by Dr. John Hall of the California Institute of Technology in the paper “Seismic Response of Buildings to Near Source Ground Motions” [1]. The structure is 77.88 m tall (255.512 ft.) with a 5.49 m (18.0118 ft.) basement. The structure consists of 5 lines of framing; the exterior (Frame A) containing 3 bays
of moment frame while the interior frames (Frame B) have pinned connections. The structure rests on top of foundation springs with foundation walls placed in the three bays of each exterior frame, additionally, the exterior columns in the three interior frames are orientated for weak axis bending. Column splices are located between floors G and 2 and then occur every other story throughout the structure. Images describing the building’s geometry and structural details can be seen in Figure 3-41 and Figure 3-42. As the seismic response of the structure in the narrow direction is of most interest, only framing parallel to this direction was modeled and the nodes were restrained to only move and rotate in this plane. Rigid diaphragms were used throughout the model restricting the movement of each frame to enforce compatibility. Images describing the building’s geometry and structural details can be seen in Figure 3-43 and Figure 3-44.

3.3.3.2 J20 Model Geometry

The second structure used in this study is the J20 structure, also designed by Dr. John Hall [1]. This structure is also 77.88 m tall (255.512 ft) with a 5.49 m (18.0118 ft) basement. The structure consists of 5 lines of framing; the exterior (Frame A) containing 3 bays of moment frame. However, unlike U20, the J20 structure has an additional line of moment frames in the middle frame (Frame C), while other the interior frames (Frame B) have pinned connections. The structure also rests on top of foundation springs with foundation walls of identical strengths and stiffnesses to the U20 structure. All columns in intermediate frames (Frame B and Frame C) are orientated for weak axis bending and all columns contain column splices starting between floors G and 2 and then occurring every other floor. As with the U20 structure, only framing parallel to the applied earthquake direction were modeled and all nodes were
restrained to only move and rotate in this place. Additionally, rigid diaphragms were used throughout the model enforcing compatibility between adjacent frames.

3.3.3.3 Model Loading

The loading on the structure consists of design dead loads of $0.391 \text{ tons/m}^2$ (80 psf) for the roof, $0.464 \text{ tons/m}^2$ (95 psf) for the floors, and $0.171 \text{ tons/m}^2$ (35 psf) for the cladding. The design live load for the floors are $0.244 \text{ tons/m}^2$ (50 psf). Gravity loads and seismic mass were computed with the full dead load and $0.073 \text{ tons/m}^2$ (15 psf) of floor live load. Using these loads it was found that the total seismic weight of the structure, including the basement) was 642.46 tons (1416.395 kips). For more information on the design of the U20 structure see the Description of Buildings section of Dr. Hall’s paper [1].
<table>
<thead>
<tr>
<th>Frame A</th>
<th>Frame B</th>
</tr>
</thead>
<tbody>
<tr>
<td>C10</td>
<td>C10</td>
</tr>
<tr>
<td>20</td>
<td>C20*</td>
</tr>
<tr>
<td>19</td>
<td>C20*</td>
</tr>
<tr>
<td>18</td>
<td>C19*</td>
</tr>
<tr>
<td>17</td>
<td>C19*</td>
</tr>
<tr>
<td>16</td>
<td>C18</td>
</tr>
<tr>
<td>15</td>
<td>C18</td>
</tr>
<tr>
<td>14</td>
<td>C17*</td>
</tr>
<tr>
<td>13</td>
<td>C17*</td>
</tr>
<tr>
<td>12</td>
<td>C16</td>
</tr>
<tr>
<td>11</td>
<td>C16</td>
</tr>
<tr>
<td>10</td>
<td>C15</td>
</tr>
<tr>
<td>9</td>
<td>C15</td>
</tr>
<tr>
<td>8</td>
<td>C14</td>
</tr>
<tr>
<td>7</td>
<td>C14</td>
</tr>
<tr>
<td>6</td>
<td>C13</td>
</tr>
<tr>
<td>5</td>
<td>C13</td>
</tr>
<tr>
<td>4</td>
<td>C12</td>
</tr>
<tr>
<td>3</td>
<td>C12</td>
</tr>
<tr>
<td>2</td>
<td>C11</td>
</tr>
<tr>
<td>1</td>
<td>C11</td>
</tr>
<tr>
<td>0</td>
<td>C11</td>
</tr>
</tbody>
</table>

Figure 3-41 - U20 - Model Description [1]
Columns

<table>
<thead>
<tr>
<th>C10</th>
<th>W14X109</th>
<th>C20</th>
<th>W21X122</th>
<th>C30</th>
<th>W14X74</th>
</tr>
</thead>
<tbody>
<tr>
<td>C9</td>
<td>W14X132</td>
<td>C19</td>
<td>W24X146</td>
<td>C29</td>
<td>W14X74</td>
</tr>
<tr>
<td>C8</td>
<td>W14X159</td>
<td>C18</td>
<td>W24X146</td>
<td>C28</td>
<td>W14X82</td>
</tr>
<tr>
<td>C7</td>
<td>W14X176</td>
<td>C17</td>
<td>W24X162</td>
<td>C27</td>
<td>W14X109</td>
</tr>
<tr>
<td>C6</td>
<td>W14X211</td>
<td>C16</td>
<td>W24X176</td>
<td>C26</td>
<td>W14X132</td>
</tr>
<tr>
<td>C5</td>
<td>W14X257</td>
<td>C15</td>
<td>W24X176</td>
<td>C25</td>
<td>W14X159</td>
</tr>
<tr>
<td>C4</td>
<td>W14X283</td>
<td>C14</td>
<td>W27X178</td>
<td>C24</td>
<td>W14X193</td>
</tr>
<tr>
<td>C3</td>
<td>W14X311</td>
<td>C13</td>
<td>W27X178</td>
<td>C23</td>
<td>W14X211</td>
</tr>
<tr>
<td>C2</td>
<td>W14X342</td>
<td>C12</td>
<td>W27X178</td>
<td>C22</td>
<td>W14X233</td>
</tr>
<tr>
<td>C1</td>
<td>W14X370</td>
<td>C11</td>
<td>W30X191</td>
<td>C21</td>
<td>W14X283</td>
</tr>
</tbody>
</table>

Girders | Beams | Foundations | Slabs

| G10 | W27X84 | B1 | W21X50 | F1 | 468-2.5 | S1 | 1160-7.6 |
| G9  | W27X94 |   |        | F2 | 336-2.5 | S2 | 2090-7.6 |
| G8  | W30X99 |   |        | F3 | 353-2.5 |
| G7  | W30X108|   | Walls  | F4 | 534-2.5 |
| G6  | W30X116|   |        |    |          |
| G5  | W30X116|   | W1 61 cm thk |
| G4  | W30X116|   |        |    |          |
| G3  | W30X116|   |        |    |          |
| G2  | W30X116|   |        |    |          |
| G1  | W30X116|   |        |    |          |

foundations: $K_H - D_{YH}$ where $K_H$ = horizontal stiffness (tons/cm) and $D_{YH}$ = yield displacement for horizontal (cm).
For vertical: $K_V = K_H$, $D_{YD} = D_{YH}$ (down) and $D_{YU} = D_{YH}/2$ (up).
slabs: $A_{10} - h_{10}$ where $A_{10}$ = effective area ($cm^2$) and $h_{10}$ = distance from top of girder/beam to centroid of slab (cm).

Figure 3-42 - U20 - Structural Details [1]
<table>
<thead>
<tr>
<th>Frame A</th>
<th>Frame B</th>
<th>Frame C</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>G10/S1</td>
<td>G10/S1</td>
</tr>
<tr>
<td>C10</td>
<td>C20</td>
<td>C20</td>
</tr>
<tr>
<td>20</td>
<td>G10/S1</td>
<td>G10/S1</td>
</tr>
<tr>
<td>C10</td>
<td>C20</td>
<td>C20</td>
</tr>
<tr>
<td>19</td>
<td>G9/S1</td>
<td>G9/S1</td>
</tr>
<tr>
<td>C9</td>
<td>C19</td>
<td>C19</td>
</tr>
<tr>
<td>18</td>
<td>G9/S1</td>
<td>G9/S1</td>
</tr>
<tr>
<td>C9</td>
<td>C19</td>
<td>C19</td>
</tr>
<tr>
<td>17</td>
<td>G8/S1</td>
<td>G8/S1</td>
</tr>
<tr>
<td>C8</td>
<td>C18</td>
<td>C18</td>
</tr>
<tr>
<td>16</td>
<td>G8/S1</td>
<td>G8/S1</td>
</tr>
<tr>
<td>C8</td>
<td>C18</td>
<td>C18</td>
</tr>
<tr>
<td>15</td>
<td>G7/S1</td>
<td>G7/S1</td>
</tr>
<tr>
<td>C7</td>
<td>C17</td>
<td>C17</td>
</tr>
<tr>
<td>14</td>
<td>G7/S1</td>
<td>G7/S1</td>
</tr>
<tr>
<td>C7</td>
<td>C17</td>
<td>C17</td>
</tr>
<tr>
<td>13</td>
<td>G6/S1</td>
<td>G6/S1</td>
</tr>
<tr>
<td>C6</td>
<td>C16</td>
<td>C16</td>
</tr>
<tr>
<td>12</td>
<td>G6/S1</td>
<td>G6/S1</td>
</tr>
<tr>
<td>C6</td>
<td>C16</td>
<td>C16</td>
</tr>
<tr>
<td>11</td>
<td>G5/S1</td>
<td>G5/S1</td>
</tr>
<tr>
<td>C5</td>
<td>C15</td>
<td>C15</td>
</tr>
<tr>
<td>10</td>
<td>G5/S1</td>
<td>G5/S1</td>
</tr>
<tr>
<td>C5</td>
<td>C15</td>
<td>C15</td>
</tr>
<tr>
<td>9</td>
<td>G4/S1</td>
<td>G4/S1</td>
</tr>
<tr>
<td>C4</td>
<td>C14</td>
<td>C14</td>
</tr>
<tr>
<td>8</td>
<td>G4/S1</td>
<td>G4/S1</td>
</tr>
<tr>
<td>C4</td>
<td>C14</td>
<td>C14</td>
</tr>
<tr>
<td>7</td>
<td>G3/S1</td>
<td>G3/S1</td>
</tr>
<tr>
<td>C3</td>
<td>C13</td>
<td>C13</td>
</tr>
<tr>
<td>6</td>
<td>G3/S1</td>
<td>G3/S1</td>
</tr>
<tr>
<td>C3</td>
<td>C13</td>
<td>C13</td>
</tr>
<tr>
<td>5</td>
<td>G2/S1</td>
<td>G2/S1</td>
</tr>
<tr>
<td>C2</td>
<td>C12</td>
<td>C12</td>
</tr>
<tr>
<td>4</td>
<td>G2/S1</td>
<td>G2/S1</td>
</tr>
<tr>
<td>C2</td>
<td>C12</td>
<td>C12</td>
</tr>
<tr>
<td>3</td>
<td>G1/S1</td>
<td>G1/S1</td>
</tr>
<tr>
<td>C1</td>
<td>C11</td>
<td>C11</td>
</tr>
<tr>
<td>2</td>
<td>G1/S1</td>
<td>G1/S1</td>
</tr>
<tr>
<td>C1</td>
<td>C11</td>
<td>C11</td>
</tr>
<tr>
<td>0</td>
<td>G1/S1</td>
<td>G1/S1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>FLOOR PLAN</th>
</tr>
</thead>
</table>
| Frame C...

<table>
<thead>
<tr>
<th>Model Description</th>
</tr>
</thead>
</table>

| Figure 3-43 - J20 - Model Description [1] |
### Columns

<table>
<thead>
<tr>
<th>C10</th>
<th>W14X109</th>
<th>C20</th>
<th>W21X122</th>
<th>C30</th>
<th>W21X122</th>
</tr>
</thead>
<tbody>
<tr>
<td>C9</td>
<td>W14X132</td>
<td>C19</td>
<td>W24X146</td>
<td>C29</td>
<td>W24X146</td>
</tr>
<tr>
<td>C8</td>
<td>W14X159</td>
<td>C18</td>
<td>W24X162</td>
<td>C28</td>
<td>W24X162</td>
</tr>
<tr>
<td>C7</td>
<td>W14X176</td>
<td>C17</td>
<td>W24X176</td>
<td>C27</td>
<td>W27X178</td>
</tr>
<tr>
<td>C6</td>
<td>W14X211</td>
<td>C16</td>
<td>W27X178</td>
<td>C26</td>
<td>W30X191</td>
</tr>
<tr>
<td>C5</td>
<td>W14X257</td>
<td>C15</td>
<td>W27X178</td>
<td>C25</td>
<td>W30X211</td>
</tr>
<tr>
<td>C4</td>
<td>W14X283</td>
<td>C14</td>
<td>W30X191</td>
<td>C24</td>
<td>W30X235</td>
</tr>
<tr>
<td>C3</td>
<td>W14X311</td>
<td>C13</td>
<td>W30X191</td>
<td>C23</td>
<td>W30X261</td>
</tr>
<tr>
<td>C2</td>
<td>W14X342</td>
<td>C12</td>
<td>W30X211</td>
<td>C22</td>
<td>W30X292</td>
</tr>
<tr>
<td>C1</td>
<td>W14X370</td>
<td>C11</td>
<td>W30X235</td>
<td>C21</td>
<td>W30X292</td>
</tr>
</tbody>
</table>

### Girders | Beams | Foundations | Slabs

| G10 | W27X84 | B1 | W21X50 | F1 | 468-2.5 | S1 | 1160-7.6 |
| G9  | W27X102 |    |        | F2 | 336-2.5 | S2 | 2090-7.6 |
| G8  | W30X108 | Walls |        | F3 | 353-2.5 |    |          |
| G7  | W30X116 |    |        | F4 | 534-2.5 |    |          |
| G6  | W30X124 |    |        |    |          |    |          |
| G5  | W30X132 | W1 | 61 cm thk |    |          |    |          |
| G4  | W30X132 |    |        |    |          |    |          |
| G3  | W30X132 |    |        |    |          |    |          |
| G2  | W30X132 |    |        |    |          |    |          |
| G1  | W30X132 |    |        |    |          |    |          |

- foundations: \( K_H - D_{YH} \) where \( K_H \) = horizontal stiffness (tons/cm) and \( D_{YH} \) = yield displacement for horizontal (cm).
- For vertical: \( K_V = K_H \), \( D_{RD} = D_{YH} \) (down) and \( D_{RU} = D_{YH}/2 \) (up).
- slabs: \( A_{10} - h_{10} \) where \( A_{10} \) = effective area (cm\(^2\)) and \( h_{10} \) = distance from top of girderbeam to centroid of slab (cm).

3.3.3.4 Fiber Element Description

For all three softwares fiber elements were utilized throughout the model. While this is not often necessary for typical models, when conducting capacity analysis the level of detail
provided by fiber elements yields a far more accurate description of a structure when it is nearing its ultimate capacity.

Each element was subdivided into 8 segments of lengths 0.03L, 0.06L, 0.16L, 0.25L, 0.25L, 0.16L, 0.06L, 0.03L while each segment consists of 8 or 10 fibers. 8 for columns, 10 for beams with slab elements. Images of this can be seen in Figure 3-45 [1] while more information in [1]. Caltech Virtual Shaker automatically creates these elements in the conversion process from ETABS while ETABS and Perform require the user to create each element manually and assign them to the proper element with the proper location.

![Figure 3-45 - Fiber Element Description](image-url)
3.3.3.5 Fiber Fracture

STEEL models have the ability to simulate brittle connections via the use of fiber fracture strain modification categories. Here, the user can input for what fiber and in what segment to modify the fracture strain then specify a percent reduction for that specific fiber. If left blank the software will assume STEEL fibers will not fracture; what was done for the pushover comparison analyses as well as the U20P and J20P models. For the U20B and J20B brittle models, the same fiber fracture strain modification categories as used by Bjornsson [9] and Song [21]. These categories were designed to simulate the susceptibility of pre-Northridge beam-to-column connections, column base plate connections, and column splices and were found by Hall to be consistent with empirical results from the 1994 Northridge earthquake [1].

The fracture criteria consists of two distributions:

\[ D_1: \frac{\varepsilon_f}{\varepsilon_y} = 0.7, 1, 10, 50, 100 \text{ with likelihoods of } 20\%, 40\%, 20\%, 10\%, 10\%, \text{ respectively.} \]

\[ D_2: \frac{\varepsilon_f}{\varepsilon_y} = 1, 10, 100 \text{ with likelihoods of } 40\%, 30\%, 30\%, \text{ respectively.} \]

For modeling pre-Northridge beam-to-column connections, distribution D1 was applied to beam top fibers (fibers 1 through 4) and distribution D2 was applied to beam bottom fibers (fibers 5 through 8) of the segment closest to the column element. For column base plate connections, distribution D1 was applied to all fibers of the bottom most segment. Lastly, column splices were modeled with distribution D1 applied to all fibers in the fourth segment of the element.

3.3.4 Material Model

The material model used in STEEL can be seen in Figure 3-46. This symmetric material model consists of a linear region of slope \( E \) which ends at a stress of \( \sigma_y \). The curve then contains
a region of constant stress until the onset of strain hardening at $\varepsilon_{SH}$. The material model then uses a cubic ellipse with initial slope $E_{SH}$ to define the strain hardening behavior, culminating in an ultimate stress of $\sigma_u$ at a strain of $\varepsilon_u$. The curve then continues on the cubic ellipse until a strain of $2\varepsilon_u - \varepsilon_{SH}$ at which point the stress drops to a residual value of $\sigma_R$.

For these analyses a Young's modulus (E) of 199.948 GPa (29000 ksi) was chosen with a yield stress ($\sigma_y$) of 344.74 MPa (50 ksi). The onset of strain hardening ($\varepsilon_{SH}$) begins at 0.004 and with an initial cubic ellipse slope of 3.4673 GPa (502.883 ksi). The ultimate stress of the material model ($\sigma_u$) is 448.16 MPa (65 ksi) occurring at a strain ($\varepsilon_u$) of 0.11. After a strain of 0.205 the material drops to its residual value ($\sigma_R$) of 44.816 MPa (6.5 ksi). Since rupture is not available in ETABS, no rupture strain was included in the STEEL material model.

Due to limitations in both STEEL and Perform3D, it is not possible to get an exact match in the material model between these softwares. The material model used in Perform was a tri-linear model with strength loss. This model consists a linear slope of 199.95 GPa (29000 ksi) with a yield stress of 344.747 MPa (50 ksi) followed by a secondary linear region that ends at a stress of 441.26 MPa (64 ksi) and a strain of 0.05. Following this there is a region of constant stress at 441.26 MPa (64 ksi) until a strain of 0.16. The material model then decreases to a stress of 344.747 MPa (50 ksi) at a strain of 0.207272. The Perform material will then remain at this stress until a strain of 0.218181 after which it drops to 0 stress. It was not possible to obtain the desired residual value of 44.816 MPa (6.5 ksi) in Perform due to a limited number of inputs.

The material model for ETABS has the most flexibility out of all three softwares by allowing the user to input stress and strain coordinates. However, It was found that the more complex material model used in STEEL caused convergence issues following the onset of
nonlinearity due to the difficulty in analysis softwares following more complex material model shapes. Therefore, an identical material model to Perform was chosen with the modification of utilizing the actual residual stress value of 44.816 MPa (6.5 ksi).

![U20 Material Model Comparison](image)

Figure 3-46 - U20 Material Model Comparison

For all three softwares the concrete material model consisted of a Young’s Modulus of 248.55 GPa (3604.997 ksi) with a yield stress of 25.58 MPa (4 ksi), an ultimate stress of 34.47 MPa (5 ksi) occurring at a strain of 0.003, and a residual stress of 3.447 MPa (500 psi).

3.3.5 Software Limitations

3.3.5.1 Perform3D

While conducting several of the linear elastic time history analyses in Perform3D it was noticed that for the specific models, namely the two brace frame and twenty story moment
frame, STEEL and Perform3D yielded similar results for both the elastic stiffness, initial release point, mass, and period; following the removal of the horizontal acceleration the Perform3D model exhibited practically no damping. While the exact method is not perfectly clear it was revealed by CSI customer support that for steel fiber elements 0% of the axial stiffness for Rayleigh damping is used [20]. This was done by the Perform programmers to combat excessive energy dissipation found in their analyses when these axial fibers yield, resulting in an axial expansion. The issue is said to be caused by coupling between the bending and axial affects that is generally not present before yielding or cracking. Through their own analyses it was found that this affect resulted in excessive energy dissipation when using stiffness proportional Rayleigh damping [20]. The decision to remove axial stiffness from steel fiber elements was done to yield a more conservative answer in users’ analyses.

For most users this affect will not cause a large error in their simulation because, in general, fiber elements are only used in small, localized regions. However, in order to achieve a valid comparison between these two softwares the usage of fiber elements throughout the Perform model results in a practically no damping being applied in specific models. As a result, the free vibration analysis for the twenty story model resulted in a solution which could not be properly compared to the results of STEEL. However, this model the static stiffness, initial release point from the analysis can still be compared as well as the results from the pushover analysis.

Additionally, Perform does not include geometric updating. While for most typical analyses this will not result in a considerable difference, when calculating the ultimate capacity of the building as well as determine at what drift the structure experiences geometric instability
the lack of geometric updating can begin to play a large difference. It is therefore expected that at very large strains the results of the STEEL and Perform analyses will diverge and it is expected for some of the more realistically weighted buildings that the STEEL analyses will become unstable at a lower drift. Furthermore, for extremely high drifts it was seen that the results from the STEEL pushover analysis can show a rapid increase in base shear following the collapse of the structure. It is believed this is caused by the manner in which STEEL conducts pushover analyses. Due to the monotonically increasing applied accelerations the amount of force being applied to the structure increases regardless of whether the structure is capable of resisting this force. However, since this force must be resolved in the system the base shear of the structure will continue to increase. Through examining the deformed shape of the structure it is clear that this increase in base shear is not indicative of an increase in structural capacity post-collapse but rather an artifact of conducting a force controlled analysis.

3.3.6 Pushover Analysis Comparison

3.3.6.1 Analysis Information

The pushover analyses for Perform and ETABS were done using a static displacement-controlled nonlinear analysis technique, although the exact method for Perform is not perfectly clear, while the STEEL analysis was done using a very slowly ramping ground acceleration. As force-controlled and displacement-controlled pushover analyses can at times yield significantly different results due to restrictions in the application of the story forces the collapse mechanism for all three structures will be examined to ensure each software is calculating behavior in a similar manner. For all three models no damping was applied during the pushover analysis and the story shear at the ground floor was plotted vs. the roof displacement as opposed to the actual base story shear. For all softwares it was assumed that the gravity and
live loads were applied prior to the initiation of the pushover analysis and pushover was conducted as a subsequent analysis. The analyses were allowed to run until the onset of instability in the pushover curve at which the analysis was terminated and the results plotted.

### 3.3.6.2 Results
The results from the pushover analysis can be seen in Figure 3-47. This image shows very similar behavior from all three softwares up until the point where they no longer converge. STEEL, ETABS, and Perform all have nearly identical linear behavior with nonlinearly occurring at approximately 1% drift. Post yield, The ETABS results follows the STEEL results exactly until the analysis fails to converge after a drift of approximately 1%, as shown in Figure 3-48. The STEEL and Perform analysis show similar building capacities with similar ultimate drifts of approximately 3.5%.

![Figure 3-47 - U20 - Pushover Comparison](image)
3.3.6.3 Discussion

As shown in the previous analysis, all three softwares are capable of producing similar results in these capacity based analyses. However, there exists some limitations in each tool which cause it to only be applicable in certain situations.

From this analysis it is clear that when running models with extensive use of fiber elements ETABS is not capable of providing results post-yield. However, if the user is only interested in the stresses and strains in the structure up until the onset of nonlinearity ETABS provides an excellent tool with simple to use post-processing. Furthermore, if the number of fiber hinges used in the ETABS model is drastically reduced the software would be able to converge at larger displacements. This is because the ETABS analysis engine takes additional steps when a hinge is loaded or unloaded and, as the each element has 8 hinges, this results in a large number of analysis steps taking place. In terms of analysis run time ETABS had the
longest out of all three analysis tools, often taking over 2 hours to complete. This is again due to the large number of analysis steps taken as a result of the fiber hinges throughout the model. Furthermore, ETABS provides the user with a significantly larger amount of post processing information than STEEL or Perform which only provide information that the user requests, thereby increasing the runtime of the analyses.

Unlike the ETABS analysis, Perform was capable of providing analysis results through the instability of the model. The differences between the Perform and STEEL analysis is largely due to the difference in material. While it was possible to achieve very similar curves, the elliptical backbone curve of the STEEL software will produce different results than the tri-linear backbone curve of Perform. The total area under the backbone curve for STEEL is larger than that of ETABS and the ultimate strain of Perform was reduced to 441.26 MPa (64 ksi) to allow for better fitting during the strain hardening region. This results in the STEEL analysis having a larger ultimate capacity than the Perform analysis. For large strains the Perform analysis experiences a drastic drop in story shear while the STEEL analysis has a more gradual decrease. This is again due primarily to the difference in material models. In order to better match the Perform and STEEL backbone curves, it was not possible for the Perform material model to have a non-zero residual stress. Therefore, when the collapse mechanism in the structure is reached the capacity of the failed element reduces to zero and the structure collapses, while in STEEL the capacity of the failed element reduces to the residual value of 44.82 MPa(6.5 ksi) and the structure is still able to find stiffness. Further differences could be explained by the inherent differences between conducting a static displacement controlled analysis and a dynamic force controlled analysis. When specifying a target displacement, as is done in Perform, the story
forces are calculated and increased in a manner which aims to reach that displacement. However, there may be situations post yield where the behavior of the structure may result in story forces on specific floors being applied in the opposite direction in an attempt to keep the target displacement of each floor in the desired pattern. To verify both softwares were determining the collapse mechanism of this structure in the same way the deformed shape was plotted just prior to collapse. An image of this can be seen in Figure 3-49. This image shows for both structures a four story mechanism starting at the ground floor and ending at floor 4 indicating similar behavior for both softwares.
Additionally, the lack of geometric updating in Perform would cause differences as well. However, as the drift at which the structure fails is relatively low, it is expected that this difference will be minimal. Finally, the difference in panel zone implementation, shear deformation only in STEEL versus axial / bending deformation in Perform, will result in STEEL analyzing a more flexible structure, thereby lengthening the pushover curve. The runtime of the Perform model was approximately equal to that of STEEL, roughly 15-30 minutes, however it was a far more difficult model to create. The interface is not as friendly as ETABS and can be difficult to setup and prone to user error. However, it is clear that Perform is more than capable of analyzing structures of this type with fiber elements located throughout.

The STEEL results demonstrate the best ability out of the three softwares to calculate the ultimate capacity of the building as well as continue the analysis through instability. The more complex material model allows for a more realistic solution at large strains and the fiber elements used throughout the model provide a higher resolution of stresses and strains. Like Perform, the runtime of the analysis in STEEL was approximately 15-30 minutes, however through the use of Caltech Virtual Shaker the model was constructed in significantly less time. The ETABS user interface coupled with Caltech Virtual Shaker’s profile system allows for rapid creation of multiple analysis models. Additionally, as the creation of fiber element descriptions is automatic, far less user input is required greatly reducing the chance of user error.
3.3.7 Nepal Time History Analysis

3.3.7.1 Earthquake Information

The ground motions applied to the U20 and J20 structures were taken from the April 2015 Nepal earthquake, also known as the Gorka earthquake. The Nepal earthquake was a magnitude 7.8 killing more than 9,000, injuring an additional 23,000 and was given the maximum Mercalli Intensity value of IX (violent). As shown in Figure 3-50, the epicenter of the earthquake was east of the district of Lamjung with numerous aftershocks recording in the nearby regions. The hypocenter of the earthquake was approximately 15 km (9.3 mi), making the earthquake extremely close to the surface and is therefore classified as “shallow” by USGS standards [23]. In addition to the destruction caused by the earthquake itself, the ground motion also trigged an avalanche on Mount Everest killing an additional 19 [24]. Since the ground motion’s hypocenter was so close to the surface, the area where the strong ground motion could be felt was a relatively small area, as shown in Figure 3-51.

Ground motion displacement and acceleration time histories were obtained from the Center for Engineering Strong Motion Data [25] from site KATNP. Sensor readings from this site as shown in Figure 3-52, yielded maximum accelerations of 0.158g and -0.164g and net displacements 116.9 cm and -139.0 cm for the two orthogonal directions. The frequency content of this ground motion could be classified as long-period and is of the appropriate period and strength to cause significant damage to mid-rise steel structures such as U20 and J20.
Figure 3-50 - Nepal Epicenter Locations [25]

Figure 3-51 - Nepal Intensity Plot [25]
Figure 3-52 - Nepal Ground Motion Sensor Readings [25]
3.3.7.2 **Collapse Vulnerability Estimation**

3.3.7.2.1 **Description**

Prior to running the time history analyses with STEEL, the vulnerability of each structure is estimated using the Peak Filtered Acceleration (PFA) method developed by Song [21]. For long period ground motions Song’s PFA method estimates the susceptibility of a structure to collapse by comparing a ground motion’s peak ground acceleration (PGA) following the application of a low-pass Butterworth filter to the normalized ultimate capacity of a structure. If the ground motion’s PGA exceeds the normalized ultimate capacity, it is predicted the structure will collapse.

The calculations for this technique will now be briefly discussed; however, for a more detailed description see [21]. The cutoff frequency for the low-pass Butterworth filter is calculated as a function of a structure’s ductility ($\mu$). Song defines two displacement values, $d_y$ and $d_{0.5}$. $d_y$ represents the roof displacement value at maximum capacity during a pushover analysis while $d_{0.5}$ is roof displacement at which the structure loses 50% of its ultimate capacity. The ductility of the structure can then be calculated via,

$$\mu = \frac{d_{0.5}}{d_y}$$

From the ductility, the cutoff frequency of the low-pass Butterworth filter can be calculated as,

$$f = \frac{1}{cT_1}$$

Where the cutoff period coefficient, $c$, is defined as,

$$c = 0.1241 \frac{d_{0.5}}{d_y} + 0.6931$$

For long-period type ground motions a second order Butterworth filter is used.
After applying the Butterworth filter, the ground motion’s PGA is found then compared to the ultimate pushover capacity of the structure normalized by its seismic weight. Namely,

\[
\frac{V_{\text{max}}}{W_{\text{seismic}}} g
\]

If this normalized capacity is less than the PGA of the ground motion, it is predicted that the structure will collapse.

3.3.7.2.2 U20

The pushover curves of the U20 structures with both perfect and brittle connections must first be calculated. This was conducted using the same techniques as before and the results can be seen in Figure 3-53. From this plot, it was found that the ultimate capacity of the U20P structure was 8036.8 kN at a roof displacement of 1.23 m and a 50% strength roof displacement of 2.78 m. For the U20B structure, the ultimate capacity was 4703.5 kN at a roof displacement of 0.34 m and a 50% strength roof displacement of 1.71 m. These results are summarized in Table 3-1.

After applying the Butterworth filter to both the U20P and U20B structures, a PGA of 160.34 \( \frac{cm}{s^2} \) and 160.06 \( \frac{cm}{s^2} \) was found for the structures respectively. Finally, normalizing the ultimate capacities of both structures by their seismic weight yielded values of 251.76 \( \frac{cm}{s^2} \) and 147.34 \( \frac{cm}{s^2} \) for the U20P and U20B structures respectively. From Song, it is then predicted that the U20P structure will stand while the U20B structure will collapse following the application of the Nepal ground motion. This information is summarized in Table 3-2.
3.3.7.2.3 J20

The pushover curves of the J20 structure’s with both perfect and brittle connections will now be calculated. This was again conducted using the same techniques as before and the results can be seen in Figure 3-54. From this plot, it was found that the ultimate capacity of the J20P structure was 10742.4 kN at a roof displacement of 1.48 m and a 50% strength roof displacement of 2.98 m. For the J20B structure, the ultimate capacity was 6563.11 kN at a roof displacement of 0.43 m and a 50% strength roof displacement of 1.78 m. These results are summarized in Table 3-1.

After applying the Butterworth filter to both the J20P and J20B structures, a PGA of 160.36 $\frac{cm}{s^2}$ and 160.31 $\frac{cm}{s^2}$ was found for the structures respectively. Finally, normalizing the ultimate capacities of both structures by their seismic weight yielded values of 311.32 $\frac{cm}{s^2}$ and 190.2 $\frac{cm}{s^2}$ for the J20P and J20B structures respectively. It is therefore predicted that both the
J20P and J20B structures will stand following the application of the Nepal ground motion. This information is summarized in Table 3-2.

![Figure 3-54 - J20 Pushover results for perfect and brittle connections](image)

**Table 3-1 - PFA Calculation Values**

<table>
<thead>
<tr>
<th>Model</th>
<th>Connection Type</th>
<th>Max Shear</th>
<th>Max Disp (m)</th>
<th>50% Shear</th>
<th>50% Disp (m)</th>
<th>Ductility</th>
<th>c</th>
</tr>
</thead>
<tbody>
<tr>
<td>U20</td>
<td>Perfect Welds</td>
<td>8090.33</td>
<td>1.23</td>
<td>4045.17</td>
<td>2.78</td>
<td>2.25</td>
<td>0.96907254</td>
</tr>
<tr>
<td></td>
<td>Brittle Welds</td>
<td>4703.57</td>
<td>0.34</td>
<td>2351.78</td>
<td>1.71</td>
<td>5.00</td>
<td>1.30511424</td>
</tr>
<tr>
<td>J20</td>
<td>Perfect Welds</td>
<td>10742.40</td>
<td>1.48</td>
<td>5371.20</td>
<td>2.98</td>
<td>2.02</td>
<td>0.93984004</td>
</tr>
<tr>
<td></td>
<td>Brittle Welds</td>
<td>6563.11</td>
<td>0.43</td>
<td>3281.55</td>
<td>1.78</td>
<td>4.12</td>
<td>1.19795847</td>
</tr>
</tbody>
</table>

3.3.7.3 *Time History Simulation*

Following the collapse vulnerability estimation analyses, all four models were subjected to the Nepal ground motion via a time history analysis. All models were created using the Caltech VirtualShaker / SteelConverter tools with the same model parameters discussed
previously. Following each analysis the deformed shape of the structure was plotted along with the location of fiber fractures and element failures.

The results of these models will be broken down into one of three categories: “repairable”, “unrepairable”, and “collapse”. The category a structure is placed in is dependent on its residual interstory drift ratio (RIDR). This value is calculated by dividing the relative horizontal displacement of two adjacent floors by the height of that story. The methods used for categorization are similar to that of Bjornsson [9] in that the drift – repair cost analysis of Iwata et al [26] were used to form a baseline of the RIDR’s necessary to cause a structure to no longer be repairable. Therefore, a structure is determined to no longer be repairable when the maximum RIDR is greater than $\frac{1}{71}$. A structural is considered “collapsed” when there is a complete loss of the lateral force-resisting system. While Bjornsson used a value of $\frac{1}{2000}$ to indicate the transition between “Immediate Occupancy” and “Repairable”, it is believed this value is too conservative as a RIDR of $\frac{1}{400}$ is commonly used as the serviceability limit for wind loading in structures at which no structural damage is expected to occur [27]. Instead, it is assumed that any RIDR less than $\frac{1}{71}$ is Repairable and a reference value and the wind serviceability limit is provided as a reference value.

3.3.7.3.1 U20

An image from the time history analyses of the U20P and U20B structures can be seen in Figure 3-55. The U20P structure experiences no steel fiber fractures or element failures and has a maximum residual interstory drift ratio beyond the prescribed $\frac{1}{71}$ unrepairable limit discussed by Iwata [26] as seen in Figure 3-56. Therefore, although the U20P structure did not collapse
due to the applied ground motions, it is expected that the cost of rehabilitation as described by FEMA-547 [28] would not be cost effective and the structure should be torn down.

The time history analysis of the U20B structure shows extensive fiber fracture and element failure throughout the lower portion of the structure leading to an overall collapse of the system. The collapse mechanism of the structure, as was seen in the pushover analyses done previously, is a six story mechanism spanning from floors G to floor 6 resulting in overall instability of the model. This difference in overall structural behavior emphasizes the importance of the inclusion of fracture in fiber based models.

Figure 3-55 – Snapshot Time History Response of U20 Structure to Nepal Ground Motion
Figure 3.7.3.2 J20

An image from the time history analyses of the J20P and J20B structures can be seen in Figure 3-57 while an image of the residual interstory drift ratio for both structures can be seen in Figure 3-58. These figures show both the perfect and brittle connection models experience no steel fiber fractures or element failures when subjected to the Nepal ground motion and the residual interstory drift ratio is only slightly larger than the maximum allowed wind serviceability interstory drift limit, indicating little to no repair is necessary on the structure. As there are no steel fiber fractures it is expected that the results from both perfect connection and brittle connection models are identical. This increase in structural strength, indicated by the larger ultimate capacity in the pushover curves shown earlier, is due to the extra line of framing present in the model as well as slightly larger sections.
Figure 3-57 - Snapshot of Time History Response of J20 Structure to Nepal Ground Motion (J20P and J20B Results are identical)

Figure 3-58 - J20 - Residual Interstory Drift of Structure Subjected to Nepal Ground Motion (J20P and J20B results are identical)
3.3.7.4 Discussion

Following the completion of the full time history analysis the results of the structure can be compared to the collapse vulnerability estimations using the PFA method. The results of this comparison, summarized in Table 3-2, shows complete agreement between the predictions and the analysis results. The U20P structure stood following the application of the Nepal ground motion, albeit with sufficient residual damage to render repairing the structure inefficient, causing the structure to be placed in the “Unrepairable” category. The U20B structure experienced collapse due to the introduction of fiber fracture modeling and is therefore given a category of “Collapsed”. Finally, both J20 structures were standing following the application of the ground motion and can both be given the category of “Repairable” with residual interstory drifts barely exceeding the wind serviceability limit.

The results of these analyses also demonstrate the importance of the inclusion of fracture in fiber based modeling. Examining the pushover curves of both the U20 and J20 structures reveals how significant the inclusion of brittle connections is on the overall capacity of a model. The U20 and J20 structures experienced an overall reduction in maximum base shear of approximately 42% and 39%, respectively. While this reduction for the J20 structure was not significant enough to cause a difference in the overall behavior of the structure due to its higher overall strength, for U20 this modification was the deference between a model which collapsed and one which was standing but unrepairable. It is therefore imperative that when analyzing a structure at the limits of its capacity that element failure be included.

<table>
<thead>
<tr>
<th>Model</th>
<th>Connection Type</th>
<th>PGA (cm/s²)</th>
<th>V/W*g (cm/s²)</th>
<th>Prediction</th>
<th>Analysis Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>U20</td>
<td>Perfect Welds</td>
<td>160.34</td>
<td>251.76</td>
<td>Standing</td>
<td>Standing</td>
</tr>
<tr>
<td></td>
<td>Brittle Welds</td>
<td>160.34</td>
<td>147.35</td>
<td>Collapse</td>
<td>Collapse</td>
</tr>
<tr>
<td>J20</td>
<td>Perfect Welds</td>
<td>160.357</td>
<td>311.32</td>
<td>Standing</td>
<td>Standing</td>
</tr>
<tr>
<td></td>
<td>Brittle Welds</td>
<td>160.31</td>
<td>190.2</td>
<td>Standing</td>
<td>Standing</td>
</tr>
</tbody>
</table>
3.4 Conclusion

The construction of SteelConverter and VirtualShaker allow STEEL to become available to a wide range of researchers and engineers who had no ability to run fiber-based nonlinear models. VirtualShaker creates an environment where users are able to submit and analyze their models as well as run pre- and post-processing without the expensive overhead cost of maintaining their own servers. These softwares also dramatically increase the productivity of all those who use it by introducing a visual front-end to an otherwise text-based solver.

After analyzing and comparing the results from these 6 models it is clear that STEEL and ETABS agree on the basic properties such as a models elastic stiffness, damping, and onset of nonlinearity. However, when an engineer is interested in determining the ultimate capacity of the building the results from STEEL go well beyond that of ETABS. In all six pushover analyses STEEL was capable of bringing the model out to a drift of at least three times that of ETABS and in nearly all analyses, out until the onset of p-delta instability. This type of information is invaluable to an engineer who is interested in learning what type of ground motions would produce these level of forces in a model. However, in ETABS this type of information with this level of detail would not be possible.

While ETABS would be capable of analyzing structures to this level of drift utilizing their more basic P-M2-M3 hinges, the level of detail gained through the use of fiber modeling is, by their own admission, apparent. According to ETABS the use of fiber hinges allow for a more “natural” behavior than that produced by P-M2-M3 hinges at the expensive of computational intensity [2]. However, when attempting to determine the maximum force a structure is able to
withstand before collapse this increased computation cost is often warranted by the increase of accuracy gained.

Additionally, in terms of time taken to construct the model STEEL far surpasses that of ETABS when the previously discussed SteelConverter and Caltech VirtualShaker are used. The creation of fiber hinges in ETABS is a lengthy and tedious process that is prone to error while in STEEL the fiber assignments are made automatically. Additionally, as ETABS will generate every piece of information possible for every node and element in the structure its runtimes can often be much longer than that of STEEL, especially with respect to time history analyses. However, this reduction of runtime with STEEL comes at the cost of only obtaining requested information. As long as the analyst knows exactly what information about the model is needed a great deal of time savings can be made.

The comparisons between STEEL and Perform demonstrated again STEEL’s ability to accurately calculate a wide range of structures’ behavior when subjected to forces and accelerations which cause nonlinear affects. While the ETABS analyses failed to provide verification beyond yield for all six of the models, the Perform analyses showed strong agreement all the way through p-delta instability. Both softwares are able to provide information to the user on what the ultimate capacity of the structure is, however, with limitations such as no geometric updating and the removal of axial stiffness from stiffness proportional damping in Perform it would be difficult for the user to get accurate results with the same level of precision STEEL provides. To be sure, it is often unnecessary to place fiber elements throughout the structure, thereby reducing the error caused by the removal of axial stiffness, the lack of geometric updating in Perform could potentially be a source for concern.
when conducting analysis where large drifts are to be expected, such as during ultimate capacity tests.

As with ETABS, the time required to create a model in STEEL is far lower than that of Perform. Perform is a very powerful software which is capable of doing many types of analyses, thereby increasing its overall complexity. However, when creating models in the same fashion as STEEL the construction of fiber elements is lengthy and prone to error. The run time between these two softwares is very similar due to, for the most part, Perform only providing output that the user requests.

The comparisons made between STEEL, Perform, and ETABS demonstrate STEEL’s ability to accurate analyze complex structures during nonlinear behavior. When this is combined with SteelConverter and Caltech Virtual Shaker the result is an analysis package that creates the opportunity for professors, students, and engineers to utilize our software with confidence in an environment with a smaller learning curve. It has always been a major goal of Caltech to spread the knowledge created here to other universities. With this analysis package, tools which have been in development for years at Caltech now have the chance to be used effectively throughout the academic community.
3.5 Lessons Learned

While using Perform, ETABS, and STEEL to conduct non-linear analyses, several pitfalls were discovered which, if not handled correctly, could result in significantly different results. It is therefore imperative that any researcher or engineer spend a great deal of time verifying model properties such as the material model, damping, and loading environment, to ensure that they are realistic with the desired level of accuracy to provide reliable results.

The choice of material model has arguably the largest impact on the results of non-linear analysis, particularly those that involve collapse. Ensuring a proper stress-strain relationship for regions such as strain hardening and residual stresses can significantly affect the results of the analysis. In Perform the limitations of a tri-linear stress-strain distribution with five effective points can be very limiting when trying to develop an accurate material model, namely the lack of a constant stress region post yield as seen in STEEL. Additionally, a tri-linear relationship lacks the smoothness in behavior that is provided by an elliptical strain hardening model. This rigidity in Perform’s material model may result in a relationship with lower than actual ultimate stress and a zero residual stress. If Perform allowed for two to three additional stress-strain coordinates it would be possible to create a more accurate material model to better represent materials.

A model’s damping can have a drastic effect on the results of any dynamic analysis. As it is extremely difficult to accurately determine how much damping should exist in a model, it can be very easy to over-damp the model, resulting in excessive energy dissipation and an unconservative analysis. This can particularly be seen in the yielding of elements. When an element yields, the large velocity present can result in an unrealistically large amount of energy
dissipation when using Rayleigh damping. To avoid this affect, STEEL uses an “elastic, perfectly-plastic” dashpot which limits the damper force to a more realistic value, while in Perform the problem was averted by eliminating all forms of axial damping with fiber elements.

When conducting a pushover analysis, the analysis type and the loading distribution used can have a significant effect on not only the ultimate capacity of the structure but on the collapse mechanism itself. Perform and ETABS both utilize a displacement controlled analysis where the user inputs a target displacement and a load pattern indicating the relative amount of displacement desired during the analysis. Issues may arise when, in order to maintain the desired displacement distribution, forces may be applied in the opposite direction resulting in unrealistic stresses and potentially influencing the collapse mechanism in the structure. Although STEEL uses a force controlled analysis, a similar distribution of masses along the height of the structure must be provided in order to determine how much force should be applied at each story. For both analysis types, great care must be given when creating the displacement or mass distribution so as to not incorrectly influence the failure mechanism in the structure.

The results from these analyses show that both Perform and STEEL are valid choices for the study of the collapse of structures. However, in the world of the Design Engineer, time and accuracy are of the utmost importance, and due to ETABS’s graphical user interface, creating models in STEEL is both faster and less error prone when coupled with SteelConverter and Caltech VirtualShaker. Additionally, there are several features present in STEEL that allow for more realistic modeling such as the probabilistic fracture of welds and unmodeled element
stiffnesses. However, for users interested in 3D analyses or a large array of pre-built material models in elements, the wide selection provided by Perform is unmatched.

## Works Cited


[31] A. Massari, Personal Notes of Anthony Massari on STEEL Special Column Damping, California.


Appendix

Appendix A – Sample 6-Story X-Brace Building .e2k File

$ File $ Desktop\Input Files\Base Models\6_Story_Base_Beams\6_Story_Base_Beams.e2k saved 6/12/2014 12:17:08 PM

$ PROGRAM INFORMATION
PROGRAM "ETABS 2013"  VERSION "13.1.4"

$ CONTROLS
UNITS  "LB"  "IN"  "F"
TITLE2  "6 story concentrically braced frame"
PREFERENCE  MERGETOL 0.1
METHOD "ASCE7-10"  USEDEFAULTMIN "YES"

$ STORIES
STORIES - IN SEQUENCE FROM TOP
STORY "Roof"  HEIGHT 180
STORY "Story6"  HEIGHT 180
STORY "Story5"  HEIGHT 180
STORY "Story4"  HEIGHT 180
STORY "Story3"  HEIGHT 180
STORY "Story2"  HEIGHT 180
STORY "Base"  ELEV 0

$ GRIDS
GRIDSYSTEM "G1"  TYPE "CARTESIAN"  BUBBLESIZE 60
GRID "G1"  LABEL "1"  DIR "X"  COORD 0 VISIBLE "Yes"  BUBBLELOC "End"
GRID "G1"  LABEL "2"  DIR "X"  COORD 360 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "3"  DIR "X"  COORD 720 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "4"  DIR "X"  COORD 1080 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "5"  DIR "X"  COORD 1440 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "A"  DIR "Y"  COORD 0 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "B"  DIR "Y"  COORD 360 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "C"  DIR "Y"  COORD 720 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "D"  DIR "Y"  COORD 1080 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "E"  DIR "Y"  COORD 1440 VISIBLE "Yes"  BUBBLELOC "Start"
GRID "G1"  LABEL "F"  DIR "Y"  COORD 1800 VISIBLE "Yes"  BUBBLELOC "Start"

$ DIAPHRAGM NAMES
DIAPHRAGM "D1"    TYPE RIGID

$ MATERIAL PROPERTIES
MATERIAL  "A992Fy50"    TYPE "Steel"    WEIGHTPERVOLUME 0.2835648
MATERIAL  "A992Fy50"    SYMTYPE "Isotropic"  E 2.9E+07  U 0.3  A 6.49999992674566E-06
MATERIAL  "A992Fy50"  FY 50000  FU 65000 FYE 55000  FUE 71500
MATERIAL  "A992Fy50"  HYSTYPE "Kinematic"  SSTYPE "Simple"  STRAINATHARDENING 0.015 STRAINATULTIMATE 0.11 STRAINATRUPTURE 0.17 FINALSLOPE -0.1
MATERIAL  "4000Psi"    TYPE "Concrete"    WEIGHTPERVOLUME 0.08680555
MATERIAL  "4000Psi"    SYMTYPE "Isotropic"  E 3604997  U 0.2  A 5.50000004295725E-06
MATERIAL  "4000Psi"  FC 4000
MATERIAL  "4000Psi"  HYSTYPE "Takeda"  SSTYPE "Mander"  STRAINATFC 0.00221914 STRAINATULTIMATE 0.005 FINALSLOPE -0.1
MATERIAL  "A615Gr60"    TYPE "Rebar"    WEIGHTPERVOLUME 0.2835648
MATERIAL  "A615Gr60"    SYMTYPE "Uniaxial"  E 2.9E+07  A 6.49999992674566E-06
MATERIAL  "A615Gr60"  FY 60000  FU 90000
MATERIAL  "A615Gr60"  HYSTYPE "Kinematic"  SSTYPE "Simple"  STRAINATULTIMATE 0.01 STRAINATRUPTURE 0.09 FINALSLOPE -0.1

$ REBAR DEFINITIONS
REBARDEFINITION  "#2"  AREA  0.05 DIA  0.25
REBARDEFINITION  "#3"  AREA  0.11 DIA  0.375
REBARDEFINITION  "#4"  AREA  0.2 DIA  0.625
REBARDEFINITION  "#5"  AREA  0.31 DIA  0.625
REBARDEFINITION  "#6"  AREA  0.44 DIA  0.75
REBARDEFINITION  "#7"  AREA  0.6 DIA  0.875
REBARDEFINITION  "#8"  AREA  0.79 DIA  1
REBARDEFINITION  "#9"  AREA  1 DIA  1.25
REBARDEFINITION  "#10"  AREA  1.27 DIA  1.27
REBARDEFINITION  "#11"  AREA  1.56 DIA  1.41
REBARDEFINITION  "#12"  AREA  2.25 DIA  1.693
REBARDEFINITION  "#18"  AREA  4 DIA  2.257

$ FRAME SECTIONS
FRAMESECTION  "W14X500"  MATERIAL "A992Fy50"  SHAPE "W14X500"
FRAMESECTION  "W14X455"  MATERIAL "A992Fy50"  SHAPE "W14X455"
FRAMESECTION  "W14X426"  MATERIAL "A992Fy50"  SHAPE "W14X426"
FRAMESECTION  "W14X398"  MATERIAL "A992Fy50"  SHAPE "W14X398"
FRAMESECTION  "W14X370"  MATERIAL "A992Fy50"  SHAPE "W14X370"
FRAMESECTION  "W14X342"  MATERIAL "A992Fy50"  SHAPE "W14X342"
FRAMESECTION  "W14X311"  MATERIAL "A992Fy50"  SHAPE "W14X311"
FRAMESECTION  "W14X283"  MATERIAL "A992Fy50"  SHAPE "W14X283"
FRAMESECTION  "W14X257"  MATERIAL "A992Fy50"  SHAPE "W14X257"
FRAMESECTION  "W14X233"  MATERIAL "A992Fy50"  SHAPE "W14X233"
FRAMESECTION  "W14X211"  MATERIAL "A992Fy50"  SHAPE "W14X211"
FRAMESECTION  "W14X193"  MATERIAL "A992Fy50"  SHAPE "W14X193"
FRAMESECTION  "W14X176"  MATERIAL "A992Fy50"  SHAPE "W14X176"
FRAMESECTION  "W14X159"  MATERIAL "A992Fy50"  SHAPE "W14X159"
FRAMESECTION  "W14X145"  MATERIAL "A992Fy50"  SHAPE "W14X145"
FRAMESECTION  "W14X132"  MATERIAL "A992Fy50"  SHAPE "W14X132"
FRAMESECTION  "W12X210"  MATERIAL "A992Fy50"  SHAPE "W12X210"
FRAMESECTION  "W12X190"  MATERIAL "A992Fy50"  SHAPE "W12X190"
FRAMESECTION  "W12X170"  MATERIAL "A992Fy50"  SHAPE "W12X170"
FRAMESECTION  "W12X152"  MATERIAL "A992Fy50"  SHAPE "W12X152"
FRAMESECTION  "W12X136"  MATERIAL "A992Fy50"  SHAPE "W12X136"
CONCRETESECTION "ConcCol" LONGBARMATERIAL "A615Gr60" CONFINEBARMATERIAL "A615Gr60" TYPE "Column" PATTERN "R-5-3"
TRANSFREQ "TIES" DESIGNCHECK "DESIGN" COVER 1.5 LONGBARAREA 1 CONFINEBARAREA 0.2 CONFINEBARSspacing 6 HUNCONFINEBARS 3
HUNCONFINEBARS2 3
HUNCONCRETESECTION "ConcBm" LONGBARMATERIAL "A615Gr60" CONFINEBARMATERIAL "A615Gr60" TYPE "Beam" COVERTOP 2.5 COVERBOTTOM 2.5
ATI 0 ARI 0 ATJ 0 ABJ 0

$ SLAB PROPERTIES
SHELLPROP "Slab1" PROPTYPE "Slab" MATERIAL "4000Psi" MODELINGTYPE "ShellThin" SLABTYPE "Slab" SLABTHICKNESS 8
SHELLPROP "Plank1" PROPTYPE "Slab" MATERIAL "4000Psi" MODELINGTYPE "Membrane" ONEDIMLOADDIST "Yes" SLABTYPE "Slab" SLABTHICKNESS 8

$ DECK PROPERTIES
SHELLPROP "Deck1" PROPTYPE "Deck" DECKTYPE "Filled" CONCMATERIAL "4000Psi" DECKMATERIAL "A992Fy50" DECKSLABDEPTH 3.5
DECKSHEARTHICKNESS 0.035 DECKUNITWEIGHT 0.0159722 SHEARSTUDDIAM 0.75 SHEARSTUDHEIGHT 6 SHEARSTUDFE 69000

$ WALL PROPERTIES
SHELLPROP "Wall1" PROPTYPE "Wall" MATERIAL "4000Psi" MODELINGTYPE "ShellThin" WALLTHICKNESS 12

$ LINK PROPERTIES
LINKPROP "Link1" TYPE "LINEAR"
LINKPROP "Link1" DOF "U1"
LINKPROP "Link1" U1STIFF 1

$ PANEL ZONE PROPERTIES
PANELONE "PZone1"

$ PIER/SPANDREL NAMES
PIERNAME "PI" SPANDRELNAME "SI"

$ POINT COORDINATES
POINT "1" 0 0 0
POINT "2" 0 360
POINT "3" 0 720
POINT "4" 0 1080
POINT "5" 0 1440
POINT "6" 0 1800
POINT "7" 360 0
POINT "8" 360 360
POINT "9" 360 720
POINT "10" 360 1080
POINT "11" 360 1440
POINT "12" 360 1800
POINT "13" 720 0
POINT "14" 720 360
POINT "15" 720 720
POINT "16" 720 1080
POINT "17" 720 1440
POINT "18" 720 1800
POINT "19" 1080 0
POINT "20" 1080 360
POINT "21" 1080 720
POINT "22" 1080 1080
POINT "23" 1080 1440
POINT "24" 1080 1800
POINT "25" 1440 0
POINT "26" 1440 360
POINT "27" 1440 720
POINT "28" 1440 1080
POINT "29" 1440 1440
POINT "30" 1440 1800
POINT "31" 0 1440
POINT "32" 0 900
POINT "33" 0 900
POINT "34" 1440 1440
POINT "35" 1440 900
POINT "36" 540 0
POINT "37" 540 0
POINT "38" 540 1800
POINT "39" 900 0
POINT "40" 900 1800

$ LINE CONNECTIVITIES
LINE "C1" COLUMN "1" "1" 1
LINE "C2" COLUMN "2" "2" 1
LINE "C3" COLUMN "3" "3" 1
LINE "C4" COLUMN "4" "4" 1
LINE "C5" COLUMN "5" "5" 1
LINE "C6" COLUMN "6" "6" 1
LINE "C7" COLUMN "7" "7" 1
LINE "C8" COLUMN "8" "8" 1
LINE "C9" COLUMN "9" "9" 1
LINE "C10" COLUMN "10" "10" 1
LINE "C11" COLUMN "11" "11" 1
LINE "C12" COLUMN "12" "12" 1
LINE "C13" COLUMN "13" "13" 1
LINE "C14" COLUMN "14" "14" 1
LINE "C15" COLUMN "15" "15" 1
LINE "C16" COLUMN "16" "16" 1
LINE "C17" COLUMN "17" "17" 1
LINE "C18" COLUMN "18" "18" 1
LINE "C19" COLUMN "19" "19" 1
LINE "C20" COLUMN "20" "20" 1
LINE "C21" COLUMN "21" "21" 1
LINE "C22" COLUMN "22" "22" 1
LINE "C23" COLUMN "23" "23" 1
LINE "C24" COLUMN "24" "24" 1
LINE "C25" COLUMN "25" "25" 1
<table>
<thead>
<tr>
<th>LINE</th>
<th>BRACE</th>
<th>&quot;33&quot;</th>
<th>&quot;4&quot;</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>LINE</td>
<td>D29</td>
<td>BRACE</td>
<td>&quot;34&quot;</td>
<td>&quot;26&quot;</td>
</tr>
<tr>
<td>LINE</td>
<td>D30</td>
<td>BRACE</td>
<td>&quot;34&quot;</td>
<td>&quot;27&quot;</td>
</tr>
<tr>
<td>LINE</td>
<td>D31</td>
<td>BRACE</td>
<td>&quot;35&quot;</td>
<td>&quot;27&quot;</td>
</tr>
<tr>
<td>LINE</td>
<td>D32</td>
<td>BRACE</td>
<td>&quot;35&quot;</td>
<td>&quot;28&quot;</td>
</tr>
</tbody>
</table>

$ GROUPS

GROUP "Column A"
GROUP "Column A" POINT "2" "Roof"
GROUP "Column A" POINT "2" "Story6"
GROUP "Column A" POINT "2" "Story5"
GROUP "Column A" POINT "2" "Story4"
GROUP "Column A" POINT "2" "Story3"
GROUP "Column A" POINT "2" "Base"
GROUP "Column A" POINT "4" "Roof"
GROUP "Column A" POINT "4" "Story6"
GROUP "Column A" POINT "4" "Story5"
GROUP "Column A" POINT "4" "Story4"
GROUP "Column A" POINT "4" "Story2"
GROUP "Column A" POINT "4" "Base"
GROUP "Column A" POINT "7" "Roof"
GROUP "Column A" POINT "7" "Story6"
GROUP "Column A" POINT "7" "Story5"
GROUP "Column A" POINT "7" "Story4"
GROUP "Column A" POINT "7" "Story3"
GROUP "Column A" POINT "7" "Story2"
GROUP "Column A" POINT "7" "Base"
GROUP "Column A" POINT "12" "Roof"
GROUP "Column A" POINT "12" "Story6"
GROUP "Column A" POINT "12" "Story5"
GROUP "Column A" POINT "12" "Story4"
GROUP "Column A" POINT "12" "Story3"
GROUP "Column A" POINT "12" "Story2"
GROUP "Column A" POINT "12" "Base"
GROUP "Column A" POINT "19" "Roof"
GROUP "Column A" POINT "19" "Story6"
GROUP "Column A" POINT "19" "Story5"
GROUP "Column A" POINT "19" "Story4"
GROUP "Column A" POINT "19" "Story3"
GROUP "Column A" POINT "19" "Story2"
GROUP "Column A" POINT "19" "Base"
GROUP "Column A" POINT "24" "Roof"
GROUP "Column A" POINT "24" "Story6"
GROUP "Column A" POINT "24" "Story5"
GROUP "Column A" POINT "24" "Story4"
GROUP "Column A" POINT "24" "Story3"
GROUP "Column A" POINT "24" "Story2"
GROUP "Column A" POINT "24" "Base"
GROUP "Column A" POINT "26" "Roof"
GROUP "Column A" POINT "26" "Story6"
GROUP "Column A" POINT "26" "Story5"
GROUP "Column A" POINT "26" "Story4"
GROUP "Column A" POINT "26" "Story3"
GROUP "Column A" POINT "26" "Story2"
GROUP "Column A" POINT "26" "Base"
GROUP "Column A" POINT "28" "Roof"
GROUP "Column A" POINT "28" "Story6"
GROUP "Column A" POINT "28" "Story5"
GROUP "Column A" POINT "28" "Story4"
GROUP "Column A" POINT "28" "Story3"
GROUP "Column A" POINT "28" "Story2"
GROUP "Column A" POINT "28" "Base"
GROUP "Column A" LINE "C2" "Roof"
GROUP "Column A" LINE "C2" "Story6"
GROUP "Column A" LINE "C2" "Story5"
GROUP "Column A" LINE "C2" "Story4"
GROUP "Column A" LINE "C2" "Story3"
GROUP "Column A" LINE "C2" "Story2"
GROUP "Column A" LINE "C4" "Roof"
GROUP "Column A" LINE "C4" "Story6"
GROUP "Column A" LINE "C4" "Story5"
GROUP "Column A" LINE "C4" "Story4"
GROUP "Column A" LINE "C4" "Story3"
GROUP "Column A" LINE "C4" "Story2"
GROUP "Column A" LINE "C7" "Roof"
GROUP "Column A" LINE "C7" "Story6"
GROUP "Column A" LINE "C7" "Story5"
GROUP "Column A" LINE "C7" "Story4"
GROUP "Column A" LINE "C7" "Story3"
GROUP "Column A" LINE "C7" "Story2"
GROUP "Column A" LINE "C24" "Roof"
GROUP 'Column G' LINE "C16" 'Story2'
GROUP 'Column G' LINE "C17" 'Roof'
GROUP 'Column G' LINE "C17" 'Story6'
GROUP 'Column G' LINE "C17" 'Story5'
GROUP 'Column G' LINE "C17" 'Story4'
GROUP 'Column G' LINE "C17" 'Story3'
GROUP 'Column G' LINE "C20" 'Roof'
GROUP 'Column G' LINE "C20" 'Story6'
GROUP 'Column G' LINE "C20" 'Story5'
GROUP 'Column G' LINE "C20" 'Story4'
GROUP 'Column G' LINE "C20" 'Story3'
GROUP 'Column G' LINE "C20" 'Story2'
GROUP 'Column G' LINE "C21" 'Roof'
GROUP 'Column G' LINE "C21" 'Story6'
GROUP 'Column G' LINE "C21" 'Story5'
GROUP 'Column G' LINE "C21" 'Story4'
GROUP 'Column G' LINE "C21" 'Story3'
GROUP 'Column G' LINE "C21" 'Story2'
GROUP 'Column G' LINE "C22" 'Roof'
GROUP 'Column G' LINE "C22" 'Story6'
GROUP 'Column G' LINE "C22" 'Story5'
GROUP 'Column G' LINE "C22" 'Story4'
GROUP 'Column G' LINE "C22" 'Story3'
GROUP 'Column G' LINE "C22" 'Story2'
GROUP 'Column G' LINE "C23" 'Roof'
GROUP 'Column G' LINE "C23" 'Story6'
GROUP 'Column G' LINE "C23" 'Story5'
GROUP 'Column G' LINE "C23" 'Story4'
GROUP 'Column G' LINE "C23" 'Story3'
GROUP 'Column G' LINE "C23" 'Story2'
GROUP 'Column G' LINE "C24" 'Roof'
GROUP 'Column G' LINE "C24" 'Story6'
GROUP 'Column G' LINE "C24" 'Story5'
GROUP 'Column G' LINE "C24" 'Story4'
GROUP 'Column G' LINE "C24" 'Story3'
GROUP 'Column G' LINE "C24" 'Story2'
GROUP 'Column G' LINE "C25" 'Roof'
GROUP 'Column G' LINE "C25" 'Story6'
GROUP 'Column G' LINE "C25" 'Story5'
GROUP 'Column G' LINE "C25" 'Story4'
GROUP 'Column G' LINE "C25" 'Story3'
GROUP 'Column G' LINE "C25" 'Story2'
GROUP 'Column G' LINE "C26" 'Roof'
GROUP 'Column G' LINE "C26" 'Story6'
GROUP 'Column G' LINE "C26" 'Story5'
GROUP 'Column G' LINE "C26" 'Story4'
GROUP 'Column G' LINE "C26" 'Story3'
GROUP 'Column G' LINE "C26" 'Story2'
GROUP 'Column G' LINE "C27" 'Roof'
GROUP 'Column G' LINE "C27" 'Story6'
GROUP 'Column G' LINE "C27" 'Story5'
GROUP 'Column G' LINE "C27" 'Story4'
GROUP 'Column G' LINE "C27" 'Story3'
GROUP 'Column G' LINE "C27" 'Story2'

$ POINT ASSIGN
POINASSIGN '1' 'Roof' DIAPH "D1"
POINASSIGN '1' 'Story6' DIAPH "D1"
POINASSIGN '1' 'Story5' DIAPH "D1"
POINASSIGN '1' 'Story4' DIAPH "D1"
POINASSIGN '1' 'Story3' DIAPH "D1"
POINASSIGN '1' 'Story2' DIAPH "D1"
POINASSIGN '1' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '2' 'Roof' DIAPH "D1"
POINASSIGN '2' 'Story6' DIAPH "D1"
POINASSIGN '2' 'Story5' DIAPH "D1"
POINASSIGN '2' 'Story4' DIAPH "D1"
POINASSIGN '2' 'Story3' DIAPH "D1"
POINASSIGN '2' 'Story2' DIAPH "D1"
POINASSIGN '2' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '3' 'Roof' DIAPH "D1"
POINASSIGN '3' 'Story6' DIAPH "D1"
POINASSIGN '3' 'Story5' DIAPH "D1"
POINASSIGN '3' 'Story4' DIAPH "D1"
POINASSIGN '3' 'Story3' DIAPH "D1"
POINASSIGN '3' 'Story2' DIAPH "D1"
POINASSIGN '3' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '4' 'Roof' DIAPH "D1"
POINASSIGN '4' 'Story6' DIAPH "D1"
POINASSIGN '4' 'Story5' DIAPH "D1"
POINASSIGN '4' 'Story4' DIAPH "D1"
POINASSIGN '4' 'Story3' DIAPH "D1"
POINASSIGN '4' 'Story2' DIAPH "D1"
POINASSIGN '4' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '5' 'Roof' DIAPH "D1"
POINASSIGN '5' 'Story6' DIAPH "D1"
POINASSIGN '5' 'Story5' DIAPH "D1"
POINASSIGN '5' 'Story4' DIAPH "D1"
POINASSIGN '5' 'Story3' DIAPH "D1"
POINASSIGN '5' 'Story2' DIAPH "D1"
POINASSIGN '5' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '6' 'Roof' DIAPH "D1"
POINASSIGN '6' 'Story6' DIAPH "D1"
POINASSIGN '6' 'Story5' DIAPH "D1"
POINASSIGN '6' 'Story4' DIAPH "D1"
POINASSIGN '6' 'Story3' DIAPH "D1"
POINASSIGN '6' 'Story2' DIAPH "D1"
POINASSIGN '6' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '7' 'Roof' DIAPH "D1"
POINASSIGN '7' 'Story6' DIAPH "D1"
POINASSIGN '7' 'Story5' DIAPH "D1"
POINASSIGN '7' 'Story4' DIAPH "D1"
POINASSIGN '7' 'Story3' DIAPH "D1"
POINASSIGN '7' 'Story2' DIAPH "D1"
POINASSIGN '7' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '8' 'Roof' DIAPH "D1"
POINASSIGN '8' 'Story6' DIAPH "D1"
POINASSIGN '8' 'Story5' DIAPH "D1"
POINASSIGN '8' 'Story4' DIAPH "D1"
POINASSIGN '8' 'Story3' DIAPH "D1"
POINASSIGN '8' 'Story2' DIAPH "D1"
POINASSIGN '8' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '9' 'Roof' DIAPH "D1"
POINASSIGN '9' 'Story6' DIAPH "D1"
POINASSIGN '9' 'Story5' DIAPH "D1"
POINASSIGN '9' 'Story4' DIAPH "D1"
POINASSIGN '9' 'Story3' DIAPH "D1"
POINASSIGN '9' 'Story2' DIAPH "D1"
POINASSIGN '9' 'Base' RESTRAINT 'UX UY UZ' DIAPH "D1"
POINASSIGN '10' 'Roof' DIAPH "D1"
POINTASSIGN "24" "Roof" DIAPRH "D1"
POINTASSIGN "24" "Story6" DIAPRH "D1"
POINTASSIGN "24" "Story5" DIAPRH "D1"
POINTASSIGN "24" "Story4" DIAPRH "D1"
POINTASSIGN "24" "Story3" DIAPRH "D1"
POINTASSIGN "24" "Story2" DIAPRH "D1"
POINTASSIGN "24" "Base" RESTRAINT "UX UY UZ" DIAPRH "D1"
POINTASSIGN "25" "Roof" DIAPRH "D1"
POINTASSIGN "25" "Story6" DIAPRH "D1"
POINTASSIGN "25" "Story5" DIAPRH "D1"
POINTASSIGN "25" "Story4" DIAPRH "D1"
POINTASSIGN "25" "Story3" DIAPRH "D1"
POINTASSIGN "25" "Story2" DIAPRH "D1"
POINTASSIGN "25" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "26" "Roof" DIAPRH "D1"
POINTASSIGN "26" "Story6" DIAPRH "D1"
POINTASSIGN "26" "Story5" DIAPRH "D1"
POINTASSIGN "26" "Story4" DIAPRH "D1"
POINTASSIGN "26" "Story3" DIAPRH "D1"
POINTASSIGN "26" "Story2" DIAPRH "D1"
POINTASSIGN "26" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "27" "Roof" DIAPRH "D1"
POINTASSIGN "27" "Story6" DIAPRH "D1"
POINTASSIGN "27" "Story5" DIAPRH "D1"
POINTASSIGN "27" "Story4" DIAPRH "D1"
POINTASSIGN "27" "Story3" DIAPRH "D1"
POINTASSIGN "27" "Story2" DIAPRH "D1"
POINTASSIGN "27" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "28" "Roof" DIAPRH "D1"
POINTASSIGN "28" "Story6" DIAPRH "D1"
POINTASSIGN "28" "Story5" DIAPRH "D1"
POINTASSIGN "28" "Story4" DIAPRH "D1"
POINTASSIGN "28" "Story3" DIAPRH "D1"
POINTASSIGN "28" "Story2" DIAPRH "D1"
POINTASSIGN "28" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "29" "Roof" DIAPRH "D1"
POINTASSIGN "29" "Story6" DIAPRH "D1"
POINTASSIGN "29" "Story5" DIAPRH "D1"
POINTASSIGN "29" "Story4" DIAPRH "D1"
POINTASSIGN "29" "Story3" DIAPRH "D1"
POINTASSIGN "29" "Story2" DIAPRH "D1"
POINTASSIGN "29" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "30" "Roof" DIAPRH "D1"
POINTASSIGN "30" "Story6" DIAPRH "D1"
POINTASSIGN "30" "Story5" DIAPRH "D1"
POINTASSIGN "30" "Story4" DIAPRH "D1"
POINTASSIGN "30" "Story3" DIAPRH "D1"
POINTASSIGN "30" "Story2" DIAPRH "D1"
POINTASSIGN "30" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "31" "Roof" DIAPRH "D1"
POINTASSIGN "31" "Story6" DIAPRH "D1"
POINTASSIGN "31" "Story5" DIAPRH "D1"
POINTASSIGN "31" "Story4" DIAPRH "D1"
POINTASSIGN "31" "Story3" DIAPRH "D1"
POINTASSIGN "31" "Story2" DIAPRH "D1"
POINTASSIGN "31" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"
POINTASSIGN "32" "Roof" DIAPRH "D1"
POINTASSIGN "32" "Story6" DIAPRH "D1"
POINTASSIGN "32" "Story5" DIAPRH "D1"
POINTASSIGN "32" "Story4" DIAPRH "D1"
POINTASSIGN "32" "Story3" DIAPRH "D1"
POINTASSIGN "32" "Story2" DIAPRH "D1"
POINTASSIGN "32" "Base" RESTRAINT "OX UY UZ" DIAPRH "D1"

$ LINE ASSIGNS
LINEASSIGN "C1" "Roof" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C1" "Story6" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C1" "Story5" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C1" "Story4" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C1" "Story3" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C1" "Story2" SECTION "W14X48" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Roof" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Story6" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Story5" SECTION "W14X193" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Story4" SECTION "W14X193" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Story3" SECTION "W14X193" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C2" "Story2" SECTION "W14X193" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Roof" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Story6" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Story5" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Story4" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Story3" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C3" "Story2" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C4" "Roof" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C4" "Story6" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C4" "Story5" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LINEASSIGN "C4" "Story4" SECTION "W14X74" MINNUMSTA 3 MESH "POINTSANDLINES"
LOAD PATTERNS

LOADPATTERN "Dead" TYPE "Dead" SELFWEIGHT 1
LOADPATTERN "Live" TYPE "Live" SELFWEIGHT 0
LOADPATTERN "Eqx" TYPE "Seismic" SELWEIGHT 0
LOADPATTERN "Windy" TYPE "Wind" SELFWEIGHT 0
LOADPATTERN "Roof" TYPE "Roof" SELWEIGHT 0

SECTION "W12X19" RELEASE "T1 M2I M2J M3I M3J" CARDINALPT 8 MAXSTASPC 24 MESH "POINTS AND LINES"
<table>
<thead>
<tr>
<th>POINT OBJECT LOADS</th>
<th>USERLOAD SET 1</th>
<th>Story6</th>
<th>D1</th>
<th>FX 392000</th>
<th>ECC 0.05</th>
</tr>
</thead>
<tbody>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story5</td>
<td>D1</td>
<td>FX 306700</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story4</td>
<td>D1</td>
<td>FX 223500</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story3</td>
<td>D1</td>
<td>FX 143100</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story2</td>
<td>D1</td>
<td>FX 66700</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Roof</td>
<td>D1</td>
<td>FX 391700</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story6</td>
<td>D1</td>
<td>FX 392000</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story5</td>
<td>D1</td>
<td>FX 306700</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story4</td>
<td>D1</td>
<td>FX 223500</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story3</td>
<td>D1</td>
<td>FX 143100</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Story2</td>
<td>D1</td>
<td>FX 66700</td>
<td>ECC 0.05</td>
</tr>
<tr>
<td>POINT OBJECT LOADS</td>
<td>USERLOAD SET 1</td>
<td>Roof</td>
<td>D1</td>
<td>FX 391700</td>
<td>ECC 0.05</td>
</tr>
</tbody>
</table>

**WIND** "User Loads" REVERSIBLE "Yes"

<table>
<thead>
<tr>
<th>POINT EXPOSURE WIND</th>
<th>&quot;SET&quot;</th>
<th>Angle 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>WIND &quot;User Loads&quot;</td>
<td>&quot;SET&quot;</td>
<td>Angle 0</td>
</tr>
</tbody>
</table>

$ POINT OBJECT LOADS

| POINT OBJECT LOADS | USERLOAD SET 1 | Story6 | D1 | FX 25800 | XLOC 720 | YLOC 900 |
|--------------------|-----------------|--------|----|-----------|----------|
| POINT OBJECT LOADS | USERLOAD SET 1 | Story5 | D1 | FX 45700 | XLOC 720 | YLOC 900 |
| POINT OBJECT LOADS | USERLOAD SET 1 | Story4 | D1 | FX 38400 | XLOC 720 | YLOC 900 |

| POINT OBJECT LOADS | USERLOAD SET 1 | Story6 | D1 | FX 25800 | XLOC 720 | YLOC 900 |
|--------------------|-----------------|--------|----|-----------|----------|
| POINT OBJECT LOADS | USERLOAD SET 1 | Story5 | D1 | FX 45700 | XLOC 720 | YLOC 900 |
| POINT OBJECT LOADS | USERLOAD SET 1 | Story4 | D1 | FX 38400 | XLOC 720 | YLOC 900 |

| POINT OBJECT LOADS | USERLOAD SET 1 | Story6 | D1 | FX 25800 | XLOC 720 | YLOC 900 |
|--------------------|-----------------|--------|----|-----------|----------|
| POINT OBJECT LOADS | USERLOAD SET 1 | Story5 | D1 | FX 45700 | XLOC 720 | YLOC 900 |
| POINT OBJECT LOADS | USERLOAD SET 1 | Story4 | D1 | FX 38400 | XLOC 720 | YLOC 900 |

| POINT OBJECT LOADS | USERLOAD SET 1 | Story6 | D1 | FX 25800 | XLOC 720 | YLOC 900 |
|--------------------|-----------------|--------|----|-----------|----------|
| POINT OBJECT LOADS | USERLOAD SET 1 | Story5 | D1 | FX 45700 | XLOC 720 | YLOC 900 |
| POINT OBJECT LOADS | USERLOAD SET 1 | Story4 | D1 | FX 38400 | XLOC 720 | YLOC 900 |

<p>| POINT OBJECT LOADS | USERLOAD SET 1 | Story6 | D1 | FX 25800 | XLOC 720 | YLOC 900 |
|--------------------|-----------------|--------|----|-----------|----------|
| POINT OBJECT LOADS | USERLOAD SET 1 | Story5 | D1 | FX 45700 | XLOC 720 | YLOC 900 |
| POINT OBJECT LOADS | USERLOAD SET 1 | Story4 | D1 | FX 38400 | XLOC 720 | YLOC 900 |</p>
<table>
<thead>
<tr>
<th>POINTLOAD</th>
<th>Name</th>
<th>Type</th>
<th>LC</th>
<th>FZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>Story1</td>
<td>0</td>
<td>dead</td>
<td>96</td>
</tr>
<tr>
<td>22</td>
<td>Story2</td>
<td>0</td>
<td>live</td>
<td>60</td>
</tr>
<tr>
<td>22</td>
<td>Story3</td>
<td>0</td>
<td>dead</td>
<td>40</td>
</tr>
<tr>
<td>22</td>
<td>Story4</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>22</td>
<td>Story5</td>
<td>0</td>
<td>dead</td>
<td>10</td>
</tr>
<tr>
<td>22</td>
<td>Story6</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>23</td>
<td>Roof</td>
<td>0</td>
<td>dead</td>
<td>96</td>
</tr>
<tr>
<td>23</td>
<td>Story2</td>
<td>0</td>
<td>live</td>
<td>60</td>
</tr>
<tr>
<td>23</td>
<td>Story3</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>23</td>
<td>Story4</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>23</td>
<td>Story5</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>23</td>
<td>Story6</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>24</td>
<td>Roof</td>
<td>0</td>
<td>dead</td>
<td>96</td>
</tr>
<tr>
<td>24</td>
<td>Story2</td>
<td>0</td>
<td>live</td>
<td>60</td>
</tr>
<tr>
<td>24</td>
<td>Story3</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>24</td>
<td>Story4</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>24</td>
<td>Story5</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>24</td>
<td>Story6</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>25</td>
<td>Roof</td>
<td>0</td>
<td>live</td>
<td>96</td>
</tr>
<tr>
<td>25</td>
<td>Story2</td>
<td>0</td>
<td>live</td>
<td>60</td>
</tr>
<tr>
<td>25</td>
<td>Story3</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>25</td>
<td>Story4</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
<tr>
<td>25</td>
<td>Story5</td>
<td>0</td>
<td>live</td>
<td>10</td>
</tr>
<tr>
<td>25</td>
<td>Story6</td>
<td>0</td>
<td>live</td>
<td>50</td>
</tr>
</tbody>
</table>
$ WALL DESIGN PREFERENCES
$ WALKING ACCELERATION LIMIT 0.005 MI
$ COMPOSITE DESIGN PREFERENCES
$ 0.75 PHIShearSeismic 0.6 PHIShearJoint 0.85 TLMCDeflectionLimitABS 1 240
$ STEEL DESIGN PREFERENCES
$ LOAD COMBINATIONS
$ LOAD CASES
$ GENERALIZED DISPLACEMENTS
$ LOAD COMBINATIONS
$ CONCRETE DESIGN PREFERENCES
$ STEEL DESIGN PREFERENCES
$ COMPOSITE DESIGN PREFERENCES
$ FUNCTIONS
$ $ ANALYSIS OPTIONS
$ $ FRAME OBJECT LOADS
$ $ CONCRETE DESIGN PREFERENCES
$ $ COMPUTATIONAL OPTIONS
$ $ CONCRETE DESIGN PREFERENCES
$ $ COMPUTATIONAL OPTIONS
$ $ CONCRETE DESIGN PREFERENCES
$ $ STEEL DESIGN PREFERENCES
$ $ STEEL DESIGN PREFERENCES
6 story concentrically braced frame (for Steel) with beams.
Appendix B – Sample SteelConverter Configuration File

%Config File for SteelConverter

%All characters following % until carriage return are ignored
%All White space ignored
%Config option name surrounded by `[]' and must be written in quotes


%%%%%%%%%%%%%%%% Program Output Information %%%%%%%%%%%%%%%%%
[DEBUGLEVEL] 0 % 0 = Only Errors, 1 = Element Creation, 2 = Parsing Info, 3 = All Debug Info
[SECTIONCONVERSION] yes % Toggle to enable or disable output of section conversion table (yes or no)
[MASSCONVERSION] yes % Toggle to enable or disable output of material conversion table (yes or no)

%%%%%%%%%%%%%%%% Model Information %%%%%%%%%%%%%%%%%
[TITLE] 6_Story_Base_Beams % Title of Model (Name output data will be saved to)
[SAVELOC] /Users/Chris/Desk saved to (dont include trailing / in directory)
[ETABSTITLE] 6_Story_Base_Beams.e2k % Title of Etabs File (Name of e2k file to be read from)
[ETABSLOC] /Users/Chris/Desk/Input Files/Base Models/6_Story_Base_Beams % Location of Etabs Input File (dont include trailing / in directory)
[PRIMARYETABSDIR] X % Direction in the Etabs model to use for primary frames
[STEELELEMENT] A1341.xml % Section Database (Use Capitol X,Y,Z)

%%%%%%%%%%%%%%%% Analysis Options %%%%%%%%%%%%%%%%%
[MTP] 40 % Maximum number of turning points in Hysteric Models (use 20)
[NDIM] 200000 % Maximum number of turning point locations (Use 100000)
[NSS] 10 % Number of static load steps
[BETA] 0.25 % Newmark Integration Parameter (0 = Central Difference, 0.25 = Const Average, 0.166 = Linear Average)
[GAMMA] 0.5 % Newmark Integration Parameter (0.5)
[A0] 0 % Damping Parameter (C = A0*M + A1*K) (Assumed to be 0 when using special columns to model damping)
[FIRSTMODELPERIOD] 0.005 % Period of the first mode of the structure. If left blank program assumes T = 0.1*N
[BASEESHEAR] 3000 % Pushover Base Shear
[BASESHARDRIFT] 0.1 % Determine ratio (Hall says 2.5)
[BASESHARDRIFT] 0.36 % Ratio between F_push and F_eq_des (F_push/F_eq_des) Must run pushover analyses to
[BASEDRIFT] 0.005 % Drift of the base floor, should use Etabs model to determine quantity (Default is 1/400*Story_Height)
[UT] 0.005 % Time Step for dynamic analysis (Only used if no EQ data is provided)
[INT] 1 % Output interval for response time histories on unit 8 (1 means every step, 2 means every other)
[IPRT] 5 % Togle to also output response time histories to unit 4 (1 = yes, 0 = no)
[FIRSTMODELPERIOD] 161 % IPORT time step at which current dynamic analysis ends (If empty then uses NDS)

%%%%%%%%%%%%%%%% Convergence Options %%%%%%%%%%%%%%%%%
[M1G] 20 % Maximum number of global iterations (Use 20)
[TOL1] 0.2 % Force Tolerance for global iterations
[TOL3] 0.2 % Moment Tolerance for global iterations
[TOL5] 2.0 % Force Tolerance for local iterations
[TOL7] 1 % Moment tolerance for local iterations
[ALPHAVC] 100000000 % Connection Element Stiffness
[A3] 1 % Multiplier of yield strength of Floor-to-floor spring to give yield strength of floor-
to-floor shear dampers

%%%%%%%%%%%%%%%% Vertical Constraint Options %%%%%%%%%%%%%%%%%
%In this area the value of alphavc for the vertical connection element stiffness is inputed using the following form
% [ALPHAVC] (x, y, z) alphavc
% Where (x, y, z) are the coordinates of the column to be given a vertical stiffness of alphavc
% additionally the entry of
% % [ALPHAVCDEF] alphavc
% Must exist where alphavc is the vertical stiffness to be given to all column elements which do not have a specific
% stiffness given

[ALPHAVCDEF] 100000000

%%%%%%%%%%%%%%%% Fiber Options %%%%%%%%%%%%%%%%%
[E1] 0.17 % Axial Load Eccentricity factor for braces
[NSEFRBC] 8 % Number of fiber segments for beam or column (Use 8)
[NSEFRBR] 7 % Number of Fiber Segments for Braces (Use 7)
[HILF] 20 % Maximum number of element iterations (Use 20)

%%%%%%%%%%%%%%%% Load Options %%%%%%%%%%%%%%%%%
[LOADCOMBO] Load % Name of ETABS load combination to use for loads on steel model (Do not use combinations of combinations)
[MASCOMBO] Mass of combinations % Name of ETABS load combination to use for mass on steel model (Do not use combinations of combinations)

%%%%%%%%%%%%%%%% Extra Response Time Histories %%%%%%%%%%%%%%%%%
%In this area extra response time histories are place. SteelConverter automatically outputs displacement time history
% of every node in the building if enabled.

PlotAll 0 % Toggle for plotting displacements of every node (1 = yes, 0 = no)
PlotSecondary 0 % Searches through secondary direction for nodes matching coordinates of primary (1 = yes, 0 = no)

% Extra time histories in the form
% [ExTH] (x1, y1, z1) (x2, y2, z2) OutputType OutputValue
% Where (x1, y1, z1) are the Etabs Coordinates of the first node for the time history (Required)
% (x2, y2, z2) are the Etabs Coordinates of the second node for the time history (Required for element base output)
% OutputType 1 = Nodal Response History
% OutputValue 1 = Steel X Direction
% 2 = Steel Y Direction
% 3 = Beam Rotation
% 4 = Column Rotation
% 2 = Panel Zone History
% OutputValue 1 = Panel Zone Moment
% 2 = Panel Zone Plastic Rotation
% 3 = Beam/Column/Brace Element History
% OutputValue 1 = Moment At Node 1 (According to Config Input)
% 2 = Moment at Node 2 (According to Config Input)
% 3 = Plastic Rotation at Node 1 (According to Config Input)
% 4 = Plastic Rotation at Node 2 (According to Config Input)
% 5 = Axial Force in Element
% 6 = Plastic Axial Displacement in Element

| [ExTH] (0, 0, 0) | 1 | 1 |
| [ExTH] (0, 0, 180) | 1 |
| [ExTH] (0, 0, 360) | 1 |
| [ExTH] (0, 0, 540) | 1 |
| [ExTH] (0, 0, 720) | 1 |
| [ExTH] (0, 0, 900) | 1 |
| [ExTH] (0, 0, 1080) | 1 |
| [ExTH] (360, 0, 0) | 1 |
| [ExTH] (360, 0, 180) | 1 |
| [ExTH] (360, 0, 360) | 1 |
| [ExTH] (360, 0, 540) | 1 |
| [ExTH] (360, 0, 720) | 1 |
| [ExTH] (360, 0, 900) | 1 |
| [ExTH] (360, 0, 1080) | 1 |
| [ExTH] (540, 0, 180) | 1 |
| [ExTH] (540, 0, 360) | 1 |
| [ExTH] (540, 0, 540) | 1 |
| [ExTH] (540, 0, 720) | 1 |
| [ExTH] (540, 0, 900) | 1 |
| [ExTH] (540, 0, 1080) | 1 |
| [ExTH] (720, 0, 0) | 1 |
| [ExTH] (720, 0, 180) | 1 |
| [ExTH] (720, 0, 360) | 1 |
| [ExTH] (720, 0, 540) | 1 |
| [ExTH] (720, 0, 720) | 1 |
| [ExTH] (720, 0, 900) | 1 |
| [ExTH] (720, 0, 1080) | 1 |
| [ExTH] (900, 0, 180) | 1 |
| [ExTH] (900, 0, 360) | 1 |
| [ExTH] (900, 0, 540) | 1 |
| [ExTH] (900, 0, 720) | 1 |
| [ExTH] (900, 0, 900) | 1 |
| [ExTH] (900, 0, 1080) | 1 |
| [ExTH] (1080, 0, 0) | 1 |
| [ExTH] (1080, 0, 180) | 1 |
| [ExTH] (1080, 0, 360) | 1 |
| [ExTH] (1080, 0, 540) | 1 |
| [ExTH] (1080, 0, 720) | 1 |
| [ExTH] (1080, 0, 900) | 1 |
| [ExTH] (1080, 0, 1080) | 1 |
| [ExTH] (1440, 0, 0) | 1 |
| [ExTH] (1440, 0, 180) | 1 |
| [ExTH] (1440, 0, 360) | 1 |
| [ExTH] (1440, 0, 540) | 1 |
| [ExTH] (1440, 0, 720) | 1 |
| [ExTH] (1440, 0, 900) | 1 |
| [ExTH] (1440, 0, 1080) | 1 |
| [ExTH] (0, 0, 0) | 2 |
| [ExTH] (0, 0, 180) | 2 |
| [ExTH] (0, 0, 360) | 2 |
| [ExTH] (0, 0, 540) | 2 |
| [ExTH] (0, 0, 720) | 2 |
| [ExTH] (0, 0, 900) | 2 |
| [ExTH] (0, 0, 1080) | 2 |
| [ExTH] (360, 0, 0) | 2 |
| [ExTH] (360, 0, 180) | 2 |
| [ExTH] (360, 0, 360) | 2 |
| [ExTH] (360, 0, 540) | 2 |
| [ExTH] (360, 0, 720) | 2 |
| [ExTH] (360, 0, 900) | 2 |
| [ExTH] (360, 0, 1080) | 2 |
| [ExTH] (540, 0, 180) | 2 |
| [ExTH] (540, 0, 360) | 2 |
| [ExTH] (540, 0, 540) | 2 |
| [ExTH] (540, 0, 720) | 2 |
| [ExTH] (720, 0, 0) | 2 |
| [ExTH] (720, 0, 180) | 2 |
| [ExTH] (720, 0, 360) | 2 |
| [ExTH] (720, 0, 540) | 2 |
| [ExTH] (720, 0, 720) | 2 |
| [ExTH] (720, 0, 900) | 2 |
| [ExTH] (720, 0, 1080) | 2 |
| [ExTH] (900, 0, 180) | 2 |
| [ExTH] (900, 0, 360) | 2 |
| [ExTH] (900, 0, 540) | 2 |
| [ExTH] (900, 0, 720) | 2 |
| [ExTH] (900, 0, 900) | 2 |
| [ExTH] (900, 0, 1080) | 2 |
| [ExTH] (1080, 0, 180) | 2 |
| [ExTH] (1080, 0, 360) | 2 |
| [ExTH] (1080, 0, 540) | 2 |
| [ExTH] (1080, 0, 720) | 2 |
| [ExTH] (1080, 0, 900) | 2 |
| [ExTH] (1080, 0, 1080) | 2 |
[ExTH] (1080, 0, 720) 1 2
[ExTH] (1080, 0, 900) 1 2
[ExTH] (1080, 0, 1080) 1 2
[ExTH] (1440, 0, 0) 1 2
[ExTH] (1440, 0, 360) 1 2
[ExTH] (1440, 0, 540) 1 2
[ExTH] (1440, 0, 720) 1 2
[ExTH] (1440, 0, 900) 1 2
[ExTH] (1440, 0, 1080) 1 2

<<<<<<<<<<< Material Models >>>>>>>>>>
% Add Explanation
[SteelMat] 11600000 24000 % Default Shear Modulus of Steel and Shear Yield Stress of Steel
[DefWallShearMod] 998899 % Default Shear Modulus to use for Basement Wall Elements
[NumMaterial] 2 % Number of Material Models, Default is 2
% Material Input of the form [MAT] E ES SIGY SIGU EPSU EPSU PRAT RES
% Where
% E Youngs Modulus for Material I for Beam/Col/Brace Elements
% ES Initial Strain Hardening Modulus ...
% SIGY Yield Stress ...
% SIGU Ultimate Stress
% EPSU Strain at onset of Strain hardening ...
% PRAT Poisson's Ratio
% RES Residual Stress
[MAT] 29000000 580000 50000 6500 0.012 0.16 0.3 0.1
[MAT] 29000000 580000 50000 65000 0.012 0.16 0.3 0.1
% Add Explanation
[ConcreteMat] 3000000 4000 0.1 % Modulus Crushing Stress Percentage of Crushing for Tension
[MATERIALCONV] A992Fy50 1 % Conversion between ETABS material and Steel Material
[MATERIALCONV] 4000Psi 0

<<<<<<<<<<< Foundation Nodes >>>>>>>>>>
% Need to give properties for foundation nodes, Input must be given for a default and for any specific springs
% [DefFndNode] ALP STRH STRVU STRVD
% [FndNode] Name ALP STRH STRVU STRVD
% Where
% Name Name of Spring Element in Etabs
% ALP Post-Yield Stiffness Ratio for Foundation Springs
% STRH Yield Stiffness of Horizontal Spring
% STRVU Yield Strength of Vertical Spring in Upward Direction
% STRVD Yield Strength of Vertical Spring in Downward Direction
[DefFndNode] 0.15 827072 413423.6 827072
[FndNode] F1 0.15 827072 413423.6 827072

<<<<<<<<<<< IPC, FRAC segment lengths Beam/Col Elements >>>>>>>>>>
% Represent Segment lengths for Beams and Column Elements input of the form
% [Frac-BC] val1 len1
% [Frac-BC] val2 len2
% ...%
% Final row must be 0 0, default is
% [Frac-BC] 0 0
% [Frac-BC] 1 0.03
% [Frac-BC] 1 0.06
% [Frac-BC] 1 0.16
% [Frac-BC] 2 0.25
% [Frac-BC] 1 0.16
% [Frac-BC] 1 0.06
% [Frac-BC] 1 0.03
% [Frac-BC] 0 0
[Frac-BC] 1 0.03
[Frac-BC] 1 0.06
[Frac-BC] 1 0.16
[Frac-BC] 2 0.25
[Frac-BC] 1 0.16
[Frac-BC] 1 0.06
[Frac-BC] 1 0.03
[Frac-BC] 0 0

<<<<<<<<<<< IPC, FRAC segment lengths Brace Elements >>>>>>>>>>
% Represent Segment lengths for Brace Elements input of the form
% [Frac-BR] val1 len1
% [Frac-BR] val2 len2
% ...%
% Final row must be 0 0, default is
% [Frac-BR] 0 0
% [Frac-BR] 1 0.25
% [Frac-BR] 1 0.16
% [Frac-BR] 1 0.07
% [Frac-BR] 1 0.04
% [Frac-BR] 1 0.07
% [Frac-BR] 1 0.16
% [Frac-BR] 1 0.25
[Frac-BR] 0 0
[Frac-BR] 1 0.25
[Frac-BR] 1 0.16
[Frac-BR] 1 0.07
[Frac-BR] 1 0.04
[Frac-BR] 1 0.07
[Frac-BR] 1 0.16
[FRAC-BR] 1 0.25
[FRAC-BR] 0 0

% Scale factor for ground accelerations. Uncomment to override ETABS value
[GAMULT] 386.4
286	  
	  

Appendix  C  -­‐  Sample  STEEL  for001  Input  File  
6_Story_Base_Beams
420 594 0 0 30 5 180 10 ASNI4 82 20 180 40 200000
0.005 0.25 0.5 0 0.001114 13.464 1 386.4 0.2 0.2 2 1 1e+08
0.17 8 7 20
5 1 ASNI3
293 10000 900 1 1 1 1 0 -37420 29950 29950
361 10000 1080 1 1 1 1 0 -32670 26375 26375
233 10000 720 1 1 1 1 0 -37070 29775 29775
165 10000 540 1 1 1 1 0 -36820 29650 29650
105 10000 360 1 1 1 1 0 -36670 29575 29575
37 10000 180 1 1 1 1 0 -36520 29500 29500
1 10000 0 0 0 1 1 0 0 0 0
300 12000 900 1 1 1 1 0 -61100 48950 48950
366 12000 1080 1 1 1 1 0 -53160 42700 42700
238 12000 720 1 1 1 1 0 -60550 48675 48675
172 12000 540 1 1 1 1 0 -60250 48525 48525
110 12000 360 1 1 1 1 0 -60050 48425 48425
44 12000 180 1 1 1 1 0 -60000 48400 48400
6 12000 0 0 0 1 1 0 0 0 0
305 14000 900 1 1 1 1 0 -61100 48950 48950
371 14000 1080 1 1 1 1 0 -53160 42700 42700
243 14000 720 1 1 1 1 0 -60550 48675 48675
177 14000 540 1 1 1 1 0 -60250 48525 48525
115 14000 360 1 1 1 1 0 -60050 48425 48425
49 14000 180 1 1 1 1 0 -60000 48400 48400
11 14000 0 0 0 1 1 0 0 0 0
310 16000 900 1 1 1 1 0 -61100 48950 48950
376 16000 1080 1 1 1 1 0 -53160 42700 42700
248 16000 720 1 1 1 1 0 -60550 48675 48675
182 16000 540 1 1 1 1 0 -60250 48525 48525
120 16000 360 1 1 1 1 0 -60050 48425 48425
54 16000 180 1 1 1 1 0 -60000 48400 48400
16 16000 0 0 0 1 1 0 0 0 0
315 18000 900 1 1 1 1 0 -61100 48950 48950
381 18000 1080 1 1 1 1 0 -53160 42700 42700
253 18000 720 1 1 1 1 0 -60550 48675 48675
187 18000 540 1 1 1 1 0 -60250 48525 48525
125 18000 360 1 1 1 1 0 -60050 48425 48425
59 18000 180 1 1 1 1 0 -60000 48400 48400
21 18000 0 0 0 1 1 0 0 0 0
320 20000 900 1 1 1 1 0 -37420 29950 29950
386 20000 1080 1 1 1 1 0 -32670 26375 26375
258 20000 720 1 1 1 1 0 -37070 29775 29775
192 20000 540 1 1 1 1 0 -36820 29650 29650
130 20000 360 1 1 1 1 0 -36670 29575 29575
64 20000 180 1 1 1 1 0 -36520 29500 29500
26 20000 0 0 0 1 1 0 0 0 0
294 10360 900 1 1 1 1 0 -61100 48950 48950
362 10360 1080 1 1 1 1 0 -53160 42700 42700
234 10360 720 1 1 1 1 0 -60550 48675 48675
166 10360 540 1 1 1 1 0 -60250 48525 48525
106 10360 360 1 1 1 1 0 -60050 48425 48425
38 10360 180 1 1 1 1 0 -60000 48400 48400
2 10360 0 0 0 1 1 0 0 0 0
301 12360 900 1 1 1 1 0 -97920 78400 78400
367 12360 1080 1 1 1 1 0 -84360 67300 67300
239 12360 720 1 1 1 1 0 -97320 78100 78100
173 12360 540 1 1 1 1 0 -97320 78100 78100
111 12360 360 1 1 1 1 0 -97320 78100 78100
45 12360 180 1 1 1 1 0 -97320 78100 78100
7 12360 0 0 0 1 1 0 0 0 0
306 14360 900 1 1 1 1 0 -97920 78400 78400
372 14360 1080 1 1 1 1 0 -84360 67300 67300
244 14360 720 1 1 1 1 0 -97320 78100 78100
178 14360 540 1 1 1 1 0 -97320 78100 78100
116 14360 360 1 1 1 1 0 -97320 78100 78100
50 14360 180 1 1 1 1 0 -97320 78100 78100
12 14360 0 0 0 1 1 0 0 0 0
311 16360 900 1 1 1 1 0 -97920 78400 78400
377 16360 1080 1 1 1 1 0 -84360 67300 67300
249 16360 720 1 1 1 1 0 -97320 78100 78100
183 16360 540 1 1 1 1 0 -97320 78100 78100
121 16360 360 1 1 1 1 0 -97320 78100 78100
55 16360 180 1 1 1 1 0 -97320 78100 78100
17 16360 0 0 0 1 1 0 0 0 0
316 18360 900 1 1 1 1 0 -97920 78400 78400
382 18360 1080 1 1 1 1 0 -84360 67300 67300
254 18360 720 1 1 1 1 0 -97320 78100 78100
188 18360 540 1 1 1 1 0 -97320 78100 78100
126 18360 360 1 1 1 1 0 -97320 78100 78100
60 18360 180 1 1 1 1 0 -97320 78100 78100
22 18360 0 0 0 1 1 0 0 0 0
321 20360 900 1 1 1 1 0 -61100 48950 48950
387 20360 1080 1 1 1 1 0 -53160 42700 42700
259 20360 720 1 1 1 1 0 -60550 48675 48675
193 20360 540 1 1 1 1 0 -60250 48525 48525
131 20360 360 1 1 1 1 0 -60050 48425 48425
65 20360 180 1 1 1 1 0 -60000 48400 48400
27 20360 0 0 0 1 1 0 0 0 0
296 10720 900 1 1 1 1 0 -61100 48950 48950
363 10720 1080 1 1 1 1 0 -53160 42700 42700


301

4 8 3 4 5 6 0 0 0 0 0 0 0.3
4 1 1 2 7 8 0 0 0 0 0 0 0.0
4 1 3 4 5 6 0 0 0 0 0 0 0.3
5 8 5 6 0 0 0 0 0 0 0 0 0.3
5 1 5 6 0 0 0 0 0 0 0 0 0.3
6 8 1 2 3 4 7 8 9 10 0 0 0.0
6 8 5 6 0 0 0 0 0 0 0 0.3
6 1 5 6 0 0 0 0 0 0 0 0 0.3
7 8 5 6 0 0 0 0 0 0 0 0 0.3
7 1 1 2 3 4 7 8 9 10 0 0 0.0
7 1 5 6 0 0 0 0 0 0 0 0 0.3
8 8 1 2 3 4 7 8 9 10 0 0 0.0
8 8 5 6 0 0 0 0 0 0 0 0 0.3
8 1 1 2 3 4 7 8 9 10 0 0 0.0
8 1 5 6 0 0 0 0 0 0 0 0 0.3
9 8 3 4 5 6 0 0 0 0 0 0 0.3
9 7 3 4 5 6 0 0 0 0 0 0 0.3
9 1 3 4 5 6 0 0 0 0 0 0 0.3
9 2 3 4 5 6 0 0 0 0 0 0 0.3
10 8 1 2 7 8 3 6 0 0 0 0 0.0
10 7 1 2 7 8 3 6 0 0 0 0 0.0
10 8 4 5 0 0 0 0 0 0 0 0 0.6
10 7 4 5 0 0 0 0 0 0 0 0 0.6
11 1 1 2 7 8 3 6 0 0 0 0 0.0
11 2 1 2 7 8 3 6 0 0 0 0 0.0
11 1 4 5 0 0 0 0 0 0 0 0 0.6
11 2 4 5 0 0 0 0 0 0 0 0 0.6
12 8 1 2 7 8 3 6 0 0 0 0 0.0
12 7 1 2 7 8 3 6 0 0 0 0 0.0
12 8 4 5 0 0 0 0 0 0 0 0 0.6
12 7 4 5 0 0 0 0 0 0 0 0 0.6
12 1 1 2 7 8 3 6 0 0 0 0 0.0
12 2 1 2 7 8 3 6 0 0 0 0 0.0
12 1 4 5 0 0 0 0 0 0 0 0 0.6
12 2 4 5 0 0 0 0 0 0 0 0 0.6
0 0 0 0 0 0 0 0 0 0 0 0 0

1 0.03
1 0.06
1 0.16
2 0.25
1 0.16
1 0.06
1 0.03
0 0

1 0.25
1 0.16
1 0.07
1 0.04
1 0.07
1 0.16
1 0.25
0 0

386.4
### Appendix D – Sample STEEL Section Conversion File

<table>
<thead>
<tr>
<th>Section No.</th>
<th>Width</th>
<th>Thickness</th>
<th>Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>10000</td>
<td>20</td>
<td>0.581</td>
<td>12</td>
</tr>
<tr>
<td>10001</td>
<td>20</td>
<td>0.465</td>
<td>12</td>
</tr>
<tr>
<td>10002</td>
<td>20</td>
<td>0.349</td>
<td>12</td>
</tr>
<tr>
<td>10003</td>
<td>20</td>
<td>0.291</td>
<td>12</td>
</tr>
<tr>
<td>10004</td>
<td>20</td>
<td>0.581</td>
<td>8</td>
</tr>
<tr>
<td>10005</td>
<td>20</td>
<td>0.465</td>
<td>8</td>
</tr>
<tr>
<td>10006</td>
<td>20</td>
<td>0.349</td>
<td>8</td>
</tr>
<tr>
<td>10007</td>
<td>20</td>
<td>0.291</td>
<td>8</td>
</tr>
<tr>
<td>10008</td>
<td>20</td>
<td>0.465</td>
<td>4</td>
</tr>
<tr>
<td>10009</td>
<td>20</td>
<td>0.349</td>
<td>4</td>
</tr>
<tr>
<td>10010</td>
<td>20</td>
<td>0.291</td>
<td>4</td>
</tr>
<tr>
<td>10011</td>
<td>20</td>
<td>0.233</td>
<td>4</td>
</tr>
<tr>
<td>10012</td>
<td>18</td>
<td>0.581</td>
<td>6</td>
</tr>
<tr>
<td>10013</td>
<td>18</td>
<td>0.465</td>
<td>6</td>
</tr>
<tr>
<td>10014</td>
<td>18</td>
<td>0.349</td>
<td>6</td>
</tr>
<tr>
<td>10015</td>
<td>18</td>
<td>0.291</td>
<td>6</td>
</tr>
<tr>
<td>10016</td>
<td>18</td>
<td>0.233</td>
<td>6</td>
</tr>
<tr>
<td>10017</td>
<td>16</td>
<td>0.581</td>
<td>16</td>
</tr>
<tr>
<td>10018</td>
<td>16</td>
<td>0.465</td>
<td>16</td>
</tr>
<tr>
<td>10019</td>
<td>16</td>
<td>0.349</td>
<td>16</td>
</tr>
<tr>
<td>10020</td>
<td>16</td>
<td>0.291</td>
<td>16</td>
</tr>
<tr>
<td>10021</td>
<td>16</td>
<td>0.233</td>
<td>16</td>
</tr>
<tr>
<td>10022</td>
<td>16</td>
<td>0.465</td>
<td>12</td>
</tr>
<tr>
<td>10023</td>
<td>16</td>
<td>0.349</td>
<td>12</td>
</tr>
<tr>
<td>10024</td>
<td>16</td>
<td>0.291</td>
<td>12</td>
</tr>
<tr>
<td>10025</td>
<td>16</td>
<td>0.233</td>
<td>12</td>
</tr>
<tr>
<td>10026</td>
<td>14</td>
<td>0.581</td>
<td>14</td>
</tr>
<tr>
<td>10027</td>
<td>14</td>
<td>0.465</td>
<td>14</td>
</tr>
<tr>
<td>10028</td>
<td>14</td>
<td>0.349</td>
<td>14</td>
</tr>
<tr>
<td>10029</td>
<td>14</td>
<td>0.291</td>
<td>14</td>
</tr>
<tr>
<td>10030</td>
<td>14</td>
<td>0.233</td>
<td>14</td>
</tr>
<tr>
<td>10031</td>
<td>14</td>
<td>0.233</td>
<td>12</td>
</tr>
<tr>
<td>10032</td>
<td>12</td>
<td>0.581</td>
<td>12</td>
</tr>
<tr>
<td>10033</td>
<td>12</td>
<td>0.465</td>
<td>12</td>
</tr>
<tr>
<td>10034</td>
<td>12</td>
<td>0.349</td>
<td>12</td>
</tr>
<tr>
<td>10035</td>
<td>12</td>
<td>0.291</td>
<td>12</td>
</tr>
<tr>
<td>10036</td>
<td>12</td>
<td>0.233</td>
<td>12</td>
</tr>
<tr>
<td>10037</td>
<td>12</td>
<td>0.465</td>
<td>8</td>
</tr>
<tr>
<td>10038</td>
<td>12</td>
<td>0.349</td>
<td>8</td>
</tr>
<tr>
<td>10039</td>
<td>12</td>
<td>0.291</td>
<td>8</td>
</tr>
<tr>
<td>10040</td>
<td>12</td>
<td>0.233</td>
<td>8</td>
</tr>
<tr>
<td>10041</td>
<td>12</td>
<td>0.581</td>
<td>6</td>
</tr>
<tr>
<td>10042</td>
<td>12</td>
<td>0.465</td>
<td>6</td>
</tr>
<tr>
<td>10043</td>
<td>12</td>
<td>0.349</td>
<td>6</td>
</tr>
<tr>
<td>10044</td>
<td>12</td>
<td>0.291</td>
<td>6</td>
</tr>
<tr>
<td>10045</td>
<td>12</td>
<td>0.233</td>
<td>6</td>
</tr>
<tr>
<td>10046</td>
<td>12</td>
<td>0.233</td>
<td>4</td>
</tr>
<tr>
<td>10047</td>
<td>12</td>
<td>0.581</td>
<td>4</td>
</tr>
<tr>
<td>10048</td>
<td>12</td>
<td>0.465</td>
<td>4</td>
</tr>
<tr>
<td>10049</td>
<td>12</td>
<td>0.349</td>
<td>4</td>
</tr>
<tr>
<td>10050</td>
<td>12</td>
<td>0.291</td>
<td>4</td>
</tr>
<tr>
<td>10051</td>
<td>12</td>
<td>0.233</td>
<td>4</td>
</tr>
<tr>
<td>10052</td>
<td>12</td>
<td>0.233</td>
<td>3.5</td>
</tr>
<tr>
<td>10053</td>
<td>12</td>
<td>0.233</td>
<td>3</td>
</tr>
<tr>
<td>10054</td>
<td>12</td>
<td>0.233</td>
<td>2</td>
</tr>
<tr>
<td>10055</td>
<td>12</td>
<td>0.233</td>
<td>2</td>
</tr>
<tr>
<td>10056</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10057</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10058</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10059</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10060</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10061</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10062</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10063</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10064</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10065</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10066</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10067</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10068</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10069</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10070</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10071</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10072</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10073</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10074</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10075</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10076</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10077</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10078</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10079</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10080</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10081</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10082</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10083</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10084</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10085</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10086</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10087</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10088</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10089</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>10090</td>
<td>12</td>
<td>0.233</td>
<td>1.75</td>
</tr>
<tr>
<td>Index</td>
<td>Value 1</td>
<td>Value 2</td>
<td>Value 3</td>
</tr>
<tr>
<td>--------</td>
<td>-----------</td>
<td>-----------</td>
<td>-----------</td>
</tr>
<tr>
<td>10091</td>
<td>2.233</td>
<td>2.233</td>
<td>2.233</td>
</tr>
<tr>
<td>10092</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10093</td>
<td>0.581</td>
<td>0.581</td>
<td></td>
</tr>
<tr>
<td>10094</td>
<td>0.465</td>
<td>0.465</td>
<td></td>
</tr>
<tr>
<td>10095</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10096</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10097</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10098</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10099</td>
<td>0.581</td>
<td>0.581</td>
<td></td>
</tr>
<tr>
<td>10100</td>
<td>0.465</td>
<td>0.465</td>
<td></td>
</tr>
<tr>
<td>10101</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10102</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10103</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10104</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10105</td>
<td>0.581</td>
<td>0.581</td>
<td></td>
</tr>
<tr>
<td>10106</td>
<td>0.465</td>
<td>0.465</td>
<td></td>
</tr>
<tr>
<td>10107</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10108</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10109</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10110</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10111</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10112</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10113</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10114</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10115</td>
<td>0.581</td>
<td>0.581</td>
<td></td>
</tr>
<tr>
<td>10116</td>
<td>0.465</td>
<td>0.465</td>
<td></td>
</tr>
<tr>
<td>10117</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10118</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10119</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10120</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10121</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10122</td>
<td>0.465</td>
<td>0.465</td>
<td></td>
</tr>
<tr>
<td>10123</td>
<td>0.349</td>
<td>0.349</td>
<td></td>
</tr>
<tr>
<td>10124</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10125</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10126</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10127</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10128</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10129</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10130</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10131</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10132</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10133</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10134</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10135</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10136</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10137</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10138</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10139</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10140</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10141</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10142</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10143</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10144</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10145</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10146</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10147</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10148</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10149</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10150</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10151</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10152</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10153</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10154</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10155</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10156</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10157</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10158</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10159</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10160</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10161</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10162</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10163</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10164</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10165</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10166</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10167</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10168</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10169</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10170</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10171</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10172</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10173</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10174</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10175</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10176</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10177</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10178</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10179</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10180</td>
<td>0.291</td>
<td>0.291</td>
<td></td>
</tr>
<tr>
<td>10181</td>
<td>0.233</td>
<td>0.233</td>
<td></td>
</tr>
<tr>
<td>10182</td>
<td>0.174</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>10183</td>
<td>0.116</td>
<td>0.116</td>
<td></td>
</tr>
<tr>
<td>10285</td>
<td>4.5 0.233 4.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10286</td>
<td>4.5 0.174 4.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10287</td>
<td>4.5 0.116 4.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10288</td>
<td>4.5 0.465 4.5 0.465</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10289</td>
<td>4.5 0.349 4.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10290</td>
<td>4.5 0.291 4.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10291</td>
<td>4.5 0.233 4.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10292</td>
<td>4.5 0.174 4.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10293</td>
<td>4.5 0.116 4.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10294</td>
<td>4.5 0.349 4.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10295</td>
<td>4.5 0.291 4.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10296</td>
<td>4.5 0.233 4.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10297</td>
<td>4.5 0.174 4.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10298</td>
<td>4.5 0.116 4.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10299</td>
<td>4.5 0.349 4.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10300</td>
<td>4.5 0.291 4.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10301</td>
<td>4.5 0.233 4.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10302</td>
<td>4.5 0.174 4.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10303</td>
<td>4.5 0.116 4.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10304</td>
<td>4.5 0.349 4.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10305</td>
<td>4.5 0.291 4.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10306</td>
<td>4.5 0.233 4.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10307</td>
<td>4.5 0.174 4.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10308</td>
<td>4.5 0.116 4.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10309</td>
<td>3.5 0.349 3.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10310</td>
<td>3.5 0.291 3.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10314</td>
<td>3.5 0.349 3.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10315</td>
<td>3.5 0.291 3.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10316</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10317</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10318</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10322</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10323</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10324</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10325</td>
<td>3.5 0.349 3.5 0.349</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10326</td>
<td>3.5 0.291 3.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10327</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10328</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10329</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10330</td>
<td>3.5 0.291 3.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10331</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10332</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10333</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10334</td>
<td>3.5 0.291 3.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10335</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10336</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10337</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10338</td>
<td>3.5 0.233 3.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10339</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10340</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10341</td>
<td>3.5 0.174 3.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10342</td>
<td>3.5 0.116 3.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10343</td>
<td>2.5 0.291 2.5 0.291</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10344</td>
<td>2.5 0.233 2.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10345</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10346</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10347</td>
<td>2.5 0.233 2.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10348</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10349</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10350</td>
<td>2.5 0.233 2.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10351</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10352</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10353</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10354</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10355</td>
<td>2.5 0.233 2.5 0.233</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10356</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10357</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10358</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10359</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10360</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10361</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10362</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10363</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10364</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10365</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10366</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10367</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10368</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10369</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10370</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10371</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10372</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10373</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10374</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10375</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10376</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10377</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10378</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10379</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10380</td>
<td>2.5 0.174 2.5 0.174</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10381</td>
<td>2.5 0.116 2.5 0.116</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Row</td>
<td>Value 1</td>
<td>Value 2</td>
<td>Value 3</td>
</tr>
<tr>
<td>-----</td>
<td>---------</td>
<td>---------</td>
<td>---------</td>
</tr>
<tr>
<td>1</td>
<td>23.7</td>
<td>0.415</td>
<td>8.97</td>
</tr>
<tr>
<td>2</td>
<td>23.7</td>
<td>0.43</td>
<td>7.04</td>
</tr>
<tr>
<td>3</td>
<td>23.6</td>
<td>0.395</td>
<td>7.01</td>
</tr>
<tr>
<td>4</td>
<td>23.0</td>
<td>0.91</td>
<td>12.6</td>
</tr>
<tr>
<td>5</td>
<td>22.7</td>
<td>0.83</td>
<td>12.5</td>
</tr>
<tr>
<td>6</td>
<td>22.5</td>
<td>0.75</td>
<td>12.4</td>
</tr>
<tr>
<td>7</td>
<td>22.1</td>
<td>0.72</td>
<td>12.5</td>
</tr>
<tr>
<td>8</td>
<td>21.8</td>
<td>0.65</td>
<td>12.4</td>
</tr>
<tr>
<td>9</td>
<td>21.7</td>
<td>0.6</td>
<td>12.4</td>
</tr>
<tr>
<td>10</td>
<td>21.5</td>
<td>0.55</td>
<td>12.3</td>
</tr>
<tr>
<td>11</td>
<td>21.4</td>
<td>0.5</td>
<td>12.3</td>
</tr>
<tr>
<td>12</td>
<td>21.4</td>
<td>0.58</td>
<td>8.42</td>
</tr>
<tr>
<td>13</td>
<td>21.4</td>
<td>0.515</td>
<td>8.36</td>
</tr>
<tr>
<td>14</td>
<td>21.2</td>
<td>0.455</td>
<td>8.2</td>
</tr>
<tr>
<td>15</td>
<td>21.1</td>
<td>0.43</td>
<td>8.27</td>
</tr>
<tr>
<td>16</td>
<td>21.0</td>
<td>0.4</td>
<td>8.24</td>
</tr>
<tr>
<td>17</td>
<td>20.8</td>
<td>0.375</td>
<td>8.22</td>
</tr>
<tr>
<td>18</td>
<td>20.6</td>
<td>0.35</td>
<td>8.14</td>
</tr>
<tr>
<td>19</td>
<td>20.4</td>
<td>0.375</td>
<td>8.1</td>
</tr>
<tr>
<td>20</td>
<td>20.3</td>
<td>0.36</td>
<td>8.06</td>
</tr>
<tr>
<td>21</td>
<td>20.2</td>
<td>0.355</td>
<td>8.03</td>
</tr>
<tr>
<td>22</td>
<td>20.1</td>
<td>0.35</td>
<td>8.02</td>
</tr>
<tr>
<td>23</td>
<td>20.0</td>
<td>0.345</td>
<td>8.0</td>
</tr>
<tr>
<td>24</td>
<td>19.9</td>
<td>0.34</td>
<td>7.97</td>
</tr>
<tr>
<td>25</td>
<td>19.8</td>
<td>0.335</td>
<td>7.94</td>
</tr>
<tr>
<td>26</td>
<td>19.7</td>
<td>0.33</td>
<td>7.91</td>
</tr>
<tr>
<td>27</td>
<td>19.6</td>
<td>0.325</td>
<td>7.88</td>
</tr>
<tr>
<td>28</td>
<td>19.5</td>
<td>0.32</td>
<td>7.85</td>
</tr>
<tr>
<td>29</td>
<td>19.4</td>
<td>0.315</td>
<td>7.82</td>
</tr>
<tr>
<td>30</td>
<td>19.3</td>
<td>0.31</td>
<td>7.79</td>
</tr>
<tr>
<td>31</td>
<td>19.2</td>
<td>0.3</td>
<td>7.76</td>
</tr>
<tr>
<td>32</td>
<td>19.1</td>
<td>0.295</td>
<td>7.73</td>
</tr>
<tr>
<td>33</td>
<td>19.0</td>
<td>0.29</td>
<td>7.7</td>
</tr>
<tr>
<td>34</td>
<td>18.9</td>
<td>0.285</td>
<td>7.67</td>
</tr>
<tr>
<td>35</td>
<td>18.8</td>
<td>0.28</td>
<td>7.64</td>
</tr>
<tr>
<td>36</td>
<td>18.7</td>
<td>0.275</td>
<td>7.61</td>
</tr>
<tr>
<td>37</td>
<td>18.6</td>
<td>0.27</td>
<td>7.58</td>
</tr>
<tr>
<td>38</td>
<td>18.5</td>
<td>0.265</td>
<td>7.55</td>
</tr>
<tr>
<td>39</td>
<td>18.4</td>
<td>0.26</td>
<td>7.52</td>
</tr>
<tr>
<td>40</td>
<td>18.3</td>
<td>0.255</td>
<td>7.49</td>
</tr>
<tr>
<td>41</td>
<td>18.2</td>
<td>0.25</td>
<td>7.46</td>
</tr>
<tr>
<td>42</td>
<td>18.1</td>
<td>0.245</td>
<td>7.43</td>
</tr>
<tr>
<td>43</td>
<td>18.0</td>
<td>0.24</td>
<td>7.4</td>
</tr>
<tr>
<td>44</td>
<td>17.9</td>
<td>0.235</td>
<td>7.37</td>
</tr>
<tr>
<td>45</td>
<td>17.8</td>
<td>0.23</td>
<td>7.34</td>
</tr>
<tr>
<td>46</td>
<td>17.7</td>
<td>0.225</td>
<td>7.31</td>
</tr>
<tr>
<td>47</td>
<td>17.6</td>
<td>0.22</td>
<td>7.28</td>
</tr>
<tr>
<td>48</td>
<td>17.5</td>
<td>0.215</td>
<td>7.25</td>
</tr>
<tr>
<td>49</td>
<td>17.4</td>
<td>0.21</td>
<td>7.22</td>
</tr>
<tr>
<td>50</td>
<td>17.3</td>
<td>0.205</td>
<td>7.19</td>
</tr>
<tr>
<td>51</td>
<td>17.2</td>
<td>0.2</td>
<td>7.16</td>
</tr>
<tr>
<td>52</td>
<td>17.1</td>
<td>0.195</td>
<td>7.13</td>
</tr>
<tr>
<td>53</td>
<td>17.0</td>
<td>0.19</td>
<td>7.1</td>
</tr>
<tr>
<td>54</td>
<td>16.9</td>
<td>0.185</td>
<td>7.07</td>
</tr>
<tr>
<td>55</td>
<td>16.8</td>
<td>0.18</td>
<td>7.04</td>
</tr>
<tr>
<td>56</td>
<td>16.7</td>
<td>0.175</td>
<td>7.01</td>
</tr>
<tr>
<td>57</td>
<td>16.6</td>
<td>0.17</td>
<td>6.98</td>
</tr>
<tr>
<td>58</td>
<td>16.5</td>
<td>0.165</td>
<td>6.95</td>
</tr>
<tr>
<td>59</td>
<td>16.4</td>
<td>0.16</td>
<td>6.92</td>
</tr>
<tr>
<td>60</td>
<td>16.3</td>
<td>0.155</td>
<td>6.89</td>
</tr>
<tr>
<td>61</td>
<td>16.2</td>
<td>0.15</td>
<td>6.86</td>
</tr>
<tr>
<td>62</td>
<td>16.1</td>
<td>0.145</td>
<td>6.83</td>
</tr>
<tr>
<td>63</td>
<td>16.0</td>
<td>0.14</td>
<td>6.8</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>10673</td>
<td>12</td>
<td>0.428</td>
<td>5.08</td>
</tr>
<tr>
<td>10674</td>
<td>12</td>
<td>0.35</td>
<td>5</td>
</tr>
<tr>
<td>10675</td>
<td>10</td>
<td>0.594</td>
<td>4.94</td>
</tr>
<tr>
<td>10676</td>
<td>10</td>
<td>0.311</td>
<td>4.66</td>
</tr>
<tr>
<td>10677</td>
<td>8</td>
<td>0.441</td>
<td>4.17</td>
</tr>
<tr>
<td>10678</td>
<td>8</td>
<td>0.271</td>
<td>4</td>
</tr>
<tr>
<td>10679</td>
<td>6</td>
<td>0.465</td>
<td>3.57</td>
</tr>
<tr>
<td>10680</td>
<td>6</td>
<td>0.232</td>
<td>3.33</td>
</tr>
<tr>
<td>10681</td>
<td>5</td>
<td>0.214</td>
<td>3</td>
</tr>
<tr>
<td>10682</td>
<td>4</td>
<td>0.326</td>
<td>2.8</td>
</tr>
<tr>
<td>10683</td>
<td>4</td>
<td>0.193</td>
<td>2.66</td>
</tr>
<tr>
<td>10684</td>
<td>3</td>
<td>0.349</td>
<td>2.51</td>
</tr>
<tr>
<td>10685</td>
<td>3</td>
<td>0.17</td>
<td>2.33</td>
</tr>
<tr>
<td>10686</td>
<td>18.3</td>
<td>1.13</td>
<td>18.1</td>
</tr>
<tr>
<td>10687</td>
<td>18</td>
<td>1.18</td>
<td>1</td>
</tr>
<tr>
<td>10688</td>
<td>17.7</td>
<td>0.87</td>
<td>17.9</td>
</tr>
<tr>
<td>10689</td>
<td>17.5</td>
<td>0.75</td>
<td>17.8</td>
</tr>
<tr>
<td>10690</td>
<td>16.5</td>
<td>1.13</td>
<td>16.3</td>
</tr>
<tr>
<td>10691</td>
<td>16.3</td>
<td>1</td>
<td>16.1</td>
</tr>
<tr>
<td>10692</td>
<td>16</td>
<td>0.875</td>
<td>16</td>
</tr>
<tr>
<td>10693</td>
<td>15.8</td>
<td>0.75</td>
<td>15.9</td>
</tr>
<tr>
<td>10694</td>
<td>15.5</td>
<td>0.625</td>
<td>15.8</td>
</tr>
<tr>
<td>10695</td>
<td>15.3</td>
<td>0.54</td>
<td>15.7</td>
</tr>
<tr>
<td>10696</td>
<td>14.2</td>
<td>0.805</td>
<td>14.9</td>
</tr>
<tr>
<td>10697</td>
<td>14</td>
<td>0.705</td>
<td>14.8</td>
</tr>
<tr>
<td>10698</td>
<td>13.8</td>
<td>0.615</td>
<td>14.7</td>
</tr>
<tr>
<td>10699</td>
<td>13.6</td>
<td>0.505</td>
<td>14.6</td>
</tr>
<tr>
<td>10700</td>
<td>12.3</td>
<td>0.685</td>
<td>12.3</td>
</tr>
<tr>
<td>10701</td>
<td>12.1</td>
<td>0.605</td>
<td>12.2</td>
</tr>
<tr>
<td>10702</td>
<td>11.9</td>
<td>0.515</td>
<td>12.1</td>
</tr>
<tr>
<td>10703</td>
<td>11.8</td>
<td>0.435</td>
<td>12</td>
</tr>
<tr>
<td>10704</td>
<td>9.99</td>
<td>0.565</td>
<td>10.2</td>
</tr>
<tr>
<td>10705</td>
<td>9.7</td>
<td>0.415</td>
<td>10.1</td>
</tr>
<tr>
<td>10706</td>
<td>8.02</td>
<td>0.445</td>
<td>8.16</td>
</tr>
<tr>
<td>10707</td>
<td>18.000000</td>
<td>0.500000</td>
<td>18.000000</td>
</tr>
<tr>
<td>10708</td>
<td>18.000000</td>
<td>0.250000</td>
<td>6.000000</td>
</tr>
</tbody>
</table>
Appendix E – Sample STEEL Slab Conversion File

111 0 0 0 0
Appendix F – Sample STEEL Seed File (for029)
Appendix G – Sample STEEL Grid Conversion File

<table>
<thead>
<tr>
<th>Grid Name</th>
<th>Constant Direction: X. Etabs Coordinate:</th>
<th>Steel Coordinate:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>360.</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>720.</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>1080.</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>1440.</td>
</tr>
<tr>
<td>A</td>
<td></td>
<td>0.</td>
</tr>
<tr>
<td>B</td>
<td></td>
<td>360.</td>
</tr>
<tr>
<td>C</td>
<td></td>
<td>720.</td>
</tr>
<tr>
<td>D</td>
<td></td>
<td>1080.</td>
</tr>
<tr>
<td>E</td>
<td></td>
<td>1440.</td>
</tr>
<tr>
<td>F</td>
<td></td>
<td>1800.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Grid Name</th>
<th>Constant Direction: Y. Etabs Coordinate:</th>
<th>Steel Coordinate:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>10000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>12000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>14000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>16000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>18000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20000</td>
</tr>
</tbody>
</table>
Appendix H – Sample STEEL Material Conversion File

<table>
<thead>
<tr>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConcBm</td>
</tr>
<tr>
<td>ConcCol</td>
</tr>
<tr>
<td>SteelBm</td>
</tr>
<tr>
<td>SteelCol</td>
</tr>
<tr>
<td>W10X12</td>
</tr>
<tr>
<td>W10X15</td>
</tr>
<tr>
<td>W10X17</td>
</tr>
<tr>
<td>W10X19</td>
</tr>
<tr>
<td>W12X106</td>
</tr>
<tr>
<td>W12X120</td>
</tr>
<tr>
<td>W12X136</td>
</tr>
<tr>
<td>W12X14</td>
</tr>
<tr>
<td>W12X152</td>
</tr>
<tr>
<td>W12X16</td>
</tr>
<tr>
<td>W12X170</td>
</tr>
<tr>
<td>W12X19</td>
</tr>
<tr>
<td>W12X190</td>
</tr>
<tr>
<td>W12X210</td>
</tr>
<tr>
<td>W12X22</td>
</tr>
<tr>
<td>W12X26</td>
</tr>
<tr>
<td>W12X96</td>
</tr>
<tr>
<td>W14X132</td>
</tr>
<tr>
<td>W14X145</td>
</tr>
<tr>
<td>W14X159</td>
</tr>
<tr>
<td>W14X176</td>
</tr>
<tr>
<td>W14X193</td>
</tr>
<tr>
<td>W14X211</td>
</tr>
<tr>
<td>W14X22</td>
</tr>
<tr>
<td>W14X233</td>
</tr>
<tr>
<td>W14X257</td>
</tr>
<tr>
<td>W14X26</td>
</tr>
<tr>
<td>W14X283</td>
</tr>
<tr>
<td>W14X30</td>
</tr>
<tr>
<td>W14X311</td>
</tr>
<tr>
<td>W14X342</td>
</tr>
<tr>
<td>W14X370</td>
</tr>
<tr>
<td>W14X398</td>
</tr>
<tr>
<td>W14X426</td>
</tr>
<tr>
<td>W14X455</td>
</tr>
<tr>
<td>W14X48</td>
</tr>
<tr>
<td>W14X500</td>
</tr>
<tr>
<td>W14X53</td>
</tr>
<tr>
<td>W14X61</td>
</tr>
<tr>
<td>W14X68</td>
</tr>
<tr>
<td>W14X74</td>
</tr>
<tr>
<td>W14X82</td>
</tr>
<tr>
<td>W14X90</td>
</tr>
<tr>
<td>W14X99</td>
</tr>
<tr>
<td>W16X12</td>
</tr>
<tr>
<td>W16X31</td>
</tr>
<tr>
<td>W18X35</td>
</tr>
<tr>
<td>W18X40</td>
</tr>
<tr>
<td>W18X50</td>
</tr>
<tr>
<td>W18X55</td>
</tr>
<tr>
<td>W18X60</td>
</tr>
<tr>
<td>W21X44</td>
</tr>
<tr>
<td>W21X55</td>
</tr>
<tr>
<td>W21X57</td>
</tr>
<tr>
<td>W21X62</td>
</tr>
<tr>
<td>W21X68</td>
</tr>
<tr>
<td>W21X93</td>
</tr>
<tr>
<td>W24X55</td>
</tr>
<tr>
<td>W24X62</td>
</tr>
<tr>
<td>W24X68</td>
</tr>
<tr>
<td>W24X76</td>
</tr>
<tr>
<td>W24X84</td>
</tr>
<tr>
<td>W24X94</td>
</tr>
<tr>
<td>W27X102</td>
</tr>
<tr>
<td>W27X114</td>
</tr>
<tr>
<td>W27X129</td>
</tr>
<tr>
<td>W27X84</td>
</tr>
<tr>
<td>W27X94</td>
</tr>
<tr>
<td>W30X108</td>
</tr>
<tr>
<td>W30X116</td>
</tr>
<tr>
<td>W30X124</td>
</tr>
<tr>
<td>W30X132</td>
</tr>
<tr>
<td>W33X130</td>
</tr>
<tr>
<td>W33X141</td>
</tr>
<tr>
<td>W33X152</td>
</tr>
<tr>
<td>W36X150</td>
</tr>
<tr>
<td>W36X160</td>
</tr>
<tr>
<td>W36X170</td>
</tr>
<tr>
<td>W36X182</td>
</tr>
</tbody>
</table>
## Appendix I – Sample STEEL Section Conversion File

<table>
<thead>
<tr>
<th>HSS10X10</th>
<th>HSS10X10X1/2</th>
<th>HSS10X10X1/4</th>
<th>HSS10X10X3/16</th>
<th>HSS10X10X5/16</th>
<th>HSS10X10X7/8</th>
<th>HSS10X10X3/4</th>
<th>HSS10X10X5/8</th>
</tr>
</thead>
<tbody>
<tr>
<td>10094</td>
<td>10097</td>
<td>10098</td>
<td>10095</td>
<td>10096</td>
<td>10099</td>
<td>10100</td>
<td>10101</td>
</tr>
<tr>
<td>10122</td>
<td>10125</td>
<td>10127</td>
<td>10126</td>
<td>10123</td>
<td>10124</td>
<td>10125</td>
<td>10126</td>
</tr>
<tr>
<td>10127</td>
<td>10129</td>
<td>10131</td>
<td>10116</td>
<td>10119</td>
<td>10121</td>
<td>10122</td>
<td>10123</td>
</tr>
<tr>
<td>10132</td>
<td>10134</td>
<td>10136</td>
<td>10118</td>
<td>10120</td>
<td>10124</td>
<td>10130</td>
<td>10132</td>
</tr>
<tr>
<td>10133</td>
<td>10135</td>
<td>10137</td>
<td>10115</td>
<td>10117</td>
<td>10118</td>
<td>10128</td>
<td>10131</td>
</tr>
<tr>
<td>10139</td>
<td>10141</td>
<td>10143</td>
<td>10113</td>
<td>10119</td>
<td>10121</td>
<td>10129</td>
<td>10131</td>
</tr>
<tr>
<td>10145</td>
<td>10147</td>
<td>10149</td>
<td>10122</td>
<td>10124</td>
<td>10126</td>
<td>10134</td>
<td>10136</td>
</tr>
<tr>
<td>10147</td>
<td>10149</td>
<td>10151</td>
<td>10132</td>
<td>10134</td>
<td>10136</td>
<td>10144</td>
<td>10147</td>
</tr>
<tr>
<td>10149</td>
<td>10151</td>
<td>10153</td>
<td>10136</td>
<td>10139</td>
<td>10141</td>
<td>10151</td>
<td>10153</td>
</tr>
<tr>
<td>10151</td>
<td>10153</td>
<td>10155</td>
<td>10139</td>
<td>10142</td>
<td>10144</td>
<td>10153</td>
<td>10155</td>
</tr>
<tr>
<td>10153</td>
<td>10155</td>
<td>10157</td>
<td>10142</td>
<td>10145</td>
<td>10147</td>
<td>10153</td>
<td>10155</td>
</tr>
<tr>
<td>10155</td>
<td>10157</td>
<td>10159</td>
<td>10145</td>
<td>10148</td>
<td>10150</td>
<td>10157</td>
<td>10159</td>
</tr>
<tr>
<td>10157</td>
<td>10159</td>
<td>10161</td>
<td>10148</td>
<td>10151</td>
<td>10153</td>
<td>10159</td>
<td>10161</td>
</tr>
<tr>
<td>10159</td>
<td>10161</td>
<td>10163</td>
<td>10151</td>
<td>10154</td>
<td>10156</td>
<td>10159</td>
<td>10161</td>
</tr>
<tr>
<td>10161</td>
<td>10163</td>
<td>10165</td>
<td>10154</td>
<td>10157</td>
<td>10159</td>
<td>10161</td>
<td>10163</td>
</tr>
<tr>
<td>Code</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X</td>
<td>1/2</td>
<td>1/4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>---</td>
<td>------</td>
<td>----</td>
<td>------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10315</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X2-1/2X5/16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X2X1/4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X2X1/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X2X3/16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X1/4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X1/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X3/16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X32</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X64</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS3</td>
<td>-</td>
<td>1/2X3-1/2X128</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Code</th>
<th>HSS4</th>
<th>-</th>
<th>1/2X</th>
<th>1/2</th>
<th>1/4</th>
</tr>
</thead>
<tbody>
<tr>
<td>10315</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X2-1/2X5/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X2X1/4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X2X1/8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X2X3/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X1/4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X1/8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X3/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X64</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS4</td>
<td>-</td>
<td>1/2X3-1/2X128</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Code</th>
<th>HSS5</th>
<th>-</th>
<th>1/2X</th>
<th>1/2</th>
<th>1/4</th>
</tr>
</thead>
<tbody>
<tr>
<td>10315</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X2-1/2X5/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X2X1/4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X2X1/8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X2X3/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X1/4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X1/8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X3/16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10313</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10312</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10319</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X16</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10321</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10320</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X64</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10311</td>
<td>HSS5</td>
<td>-</td>
<td>1/2X3-1/2X128</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>