Report No. CCEER-07-3

Pre-test Analytical Studies of NEESR-SG 4-Span Bridge Model Using OpenSees

M. Sadrossadat Zadeh
M. Saiid Saiidi

Center of Civil Engineering Earthquake Research
Department of Civil Engineering/258
University of Nevada, Reno
Reno, Nevada 89557

February 2007
Acknowledgements

The study presented in this report is a part of a collaborative Network for Earthquake Engineering Simulation Research-Small Group (NEESR-SG) project funded by the National Science Foundation (NSF) under Grant No. CMS-0420347. The opinions expressed in this article belong solely to the authors and do not necessarily represent the view of the sponsor. The valuable support and comments by Dr. Silvia Mazzonie, the OpenSees-NEESit Simulation User Support Manager and Professor Greg Fenves of the University of California, Berkeley are appreciated. The authors are also indebted to Dr. Nathan Johnson, Dr. Patrick Laplace, and Mr. Mathew Dryden for their input in the course of this study.
Abstract

This study focuses on the development of a 4-span bridge analytical model using OpenSees. Three ¼-scale 4-span bridge models are included for testing on the University of Nevada, Reno (UNR) shake tables as a part of a collaborative Network for Earthquake Engineering Simulation Research-Small Group (NEESR-SG) study entitled “Seismic Performance of Bridge Systems with Conventional and Innovative Design.” The columns of the first bridge model include conventional steel and concrete, while in the other two models, innovative materials will be incorporated in the bridge columns. This pre-test analytical study is intended to predict the performance of the first bridge model during shake table testing and develop motions to be used at the abutments during shake table testing. The bridge piers are of drop-cap type with a continuous post-tensioned deck connected to three, two-column bent caps and roller type connections at the abutments. The bridge model will be tested under bilateral horizontal motions applied by the UNR shake tables at the bridge pier footings and longitudinal motions applied at the abutments through actuators to simulate the abutment interaction with the bridge.

To estimate the bridge-abutment interaction and its effect on the bridge response, three different models are developed. The first model (Model 1) represents a bridge with no abutment interaction. In this model the bridge deck ends are supported on rollers so the bridge deck is free to move in both horizontal directions. The abutment interaction is included in the second bridge model (Model 2). In this model, the bridge deck ends are assumed to be supported on the roller. The abutment consists of the backwall and the backfill soil behind it. The soil stiffness is represented by a nonlinear spring. An initial gap of 0.5 in (12.7 mm) is assumed between the deck and the backwall. The third bridge model (Model 3) is developed to represent the actual bridge test setup with the abutment springs replaced by the actuators at the ends. The displacement histories recorded at the bridge deck end nodes in Model 2 are used as the actuator input at the bridge deck end nodes in Model 3.

This report presents the modeling assumptions and the predicted results. To evaluate the adequacy of the analytical modeling method, an analytical model of a two-span bridge for which test data were available was used. Results were found to be reasonably close. In addition to the response of the two-span bridge, this report summarizes important calculated response parameters for the 4-span bridge.
# Table of Contents

1 Introduction........................................................................................................ 1  
  1.1 Background................................................................................................. 1  
  1.2 Finite Element Analysis Programs............................................................... 2  
  1.3 Bridge Modeling.......................................................................................... 2  
      1.3.1 Bridge-Abutment Interaction............................................................... 3  
  1.3 Objectives and Scope.................................................................................... 4  

2 Bridge Model Descriptions and Output Motions .............................................. 5  
  2.1 Introduction.................................................................................................. 5  
  2.2 Prototype Dimensions ............................................................................... 5  
  2.3 Model Bridge Dimensions ......................................................................... 6  
      2.3.1 Column Dimensions and Details......................................................... 6  
      2.3.2 Bridge Superstructure ......................................................................... 6  
      2.3.3 Bridge Abutment Details .................................................................... 7  
  2.4 Superimposed Dead Loads ......................................................................... 7  
  2.5 Selection of Shake Table Motions ............................................................... 7  
  2.6 Material Properties .................................................................................... 9  

3 OpenSees Computer Program .......................................................................... 9  
  3.1 Introduction.................................................................................................. 9  
  3.2 OpenSees Modeling Capabilities................................................................. 9  
      3.2.1 Elements.............................................................................................. 10  
      3.2.1.1 Elastic Beam-Column Elements...................................................... 10  
      3.2.1.2 Non-Linear Beam Column Elements............................................ 10  
      3.2.1.3 Zero-Length Element..................................................................... 11  
      3.2.1.4 Zero-Length Section Element......................................................... 11  
      3.2.2 Uniaxial Material................................................................................ 11  
      3.2.2.1 Steel01 Material............................................................................ 11  
      3.2.2.2 Steel02 Material............................................................................ 11  
      3.2.2.3 Concrete01 Material...................................................................... 11  
      3.2.2.4 Elasto-Plastic Gap Material............................................................ 12  
      3.2.2.5 Hysteretic Material........................................................................ 12  
      3.2.2.6 Pinching4 Material........................................................................ 12  
      3.2.3 Section Models.................................................................................... 12  
      3.2.4 Load Patterns...................................................................................... 13  
      3.2.4.1 Plain Patterns................................................................................ 13  
      3.2.4.2 Uniform Excitation Pattern............................................................. 13  
      3.2.4.3 Multiple Support Pattern............................................................... 13  
  3.3 OpenSees Analysis Capabilities................................................................. 13  
      3.3.1 Constraints Command......................................................................... 14  
      3.3.2 Numberer Command.......................................................................... 14  
      3.3.3 System Command.............................................................................. 15  
      3.3.4 Solution Algorithm Commands.......................................................... 16  
      3.3.5 Convergence Test Commands............................................................. 16  
      3.3.6 Analysis Command............................................................................ 17
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3.6.1 Static Analysis</td>
<td>18</td>
</tr>
<tr>
<td>3.3.6.2 Dynamic Analysis</td>
<td>18</td>
</tr>
<tr>
<td>3.3.7 Integrator Command</td>
<td>19</td>
</tr>
<tr>
<td>3.3.7.1 Integrator for Static Analysis</td>
<td>19</td>
</tr>
<tr>
<td>3.3.7.2 Integrator for Transient Analysis</td>
<td>20</td>
</tr>
<tr>
<td>3.3.8 Rayleigh Command</td>
<td>21</td>
</tr>
<tr>
<td>3.3.9 Analyze Command</td>
<td>22</td>
</tr>
<tr>
<td>3.3.10 Recorder Commands</td>
<td>22</td>
</tr>
<tr>
<td>4 2-Span Bridge Description and OpenSees Model</td>
<td>24</td>
</tr>
<tr>
<td>4.1 Introduction</td>
<td>24</td>
</tr>
<tr>
<td>4.2 Prototype and Model Bridge Description</td>
<td>24</td>
</tr>
<tr>
<td>4.3 OpenSees Analytical Modeling of the Bridge</td>
<td>25</td>
</tr>
<tr>
<td>4.3.1 Computer Model of Bridge Specimen</td>
<td>25</td>
</tr>
<tr>
<td>4.3.2 Nodal Configuration and Masses</td>
<td>25</td>
</tr>
<tr>
<td>4.3.3 Description of Model</td>
<td>26</td>
</tr>
<tr>
<td>4.3.3.1 Column Element Descriptions</td>
<td>26</td>
</tr>
<tr>
<td>4.3.3.2 Material Properties</td>
<td>26</td>
</tr>
<tr>
<td>4.3.3.3 Zero Length Elements</td>
<td>27</td>
</tr>
<tr>
<td>4.3.3.4 Shake Table Motions</td>
<td>27</td>
</tr>
<tr>
<td>4.3.4 Model Efficiency</td>
<td>27</td>
</tr>
<tr>
<td>4.3.4.1 Fiber Configuration</td>
<td>27</td>
</tr>
<tr>
<td>4.3.5 Damping</td>
<td>27</td>
</tr>
<tr>
<td>4.4 Comparison of Calculated and Measured Response</td>
<td>28</td>
</tr>
<tr>
<td>4.5 Concluding Remarks</td>
<td>28</td>
</tr>
<tr>
<td>5 Bridge Modeling</td>
<td>30</td>
</tr>
<tr>
<td>5.1 Introduction</td>
<td>30</td>
</tr>
<tr>
<td>5.2 Description of Model</td>
<td>30</td>
</tr>
<tr>
<td>5.2.1 Material Models</td>
<td>30</td>
</tr>
<tr>
<td>5.2.2 Stiffness Assumptions for Linear Elastic Beam Members</td>
<td>30</td>
</tr>
<tr>
<td>5.3 Analytical Modeling</td>
<td>31</td>
</tr>
<tr>
<td>5.3.1 Fiber Configuration</td>
<td>31</td>
</tr>
<tr>
<td>5.3.2 Element Configuration</td>
<td>31</td>
</tr>
<tr>
<td>5.3.2.1 Lumped Plasticity</td>
<td>31</td>
</tr>
<tr>
<td>5.3.2.2 Distributed Plasticity</td>
<td>32</td>
</tr>
<tr>
<td>5.3.2.3 Selection of Column Element Type</td>
<td>32</td>
</tr>
<tr>
<td>5.3.3 Analysis Increment and Integration Method</td>
<td>32</td>
</tr>
<tr>
<td>5.4 Moment Curvature Analysis</td>
<td>33</td>
</tr>
<tr>
<td>5.5 Push-Over Analysis</td>
<td>33</td>
</tr>
<tr>
<td>5.6 Dynamic Analysis of the Bridge</td>
<td>34</td>
</tr>
<tr>
<td>5.6.1 Input Motions</td>
<td>34</td>
</tr>
<tr>
<td>5.6.1.1 Loading Pattern</td>
<td>34</td>
</tr>
<tr>
<td>5.6.2 Cap-Beam Deck Connection</td>
<td>35</td>
</tr>
<tr>
<td>5.6.3 Bond-Slip Modeling</td>
<td>35</td>
</tr>
<tr>
<td>5.6.4 Bridge-Abutment Arrangement in Bridge Setup</td>
<td>35</td>
</tr>
<tr>
<td>5.6.4.1 Abutment Back Wall Stiffness</td>
<td>36</td>
</tr>
<tr>
<td>5.6.4.2 Models for Bridge Abutment Interaction</td>
<td>36</td>
</tr>
</tbody>
</table>
List of Tables

Chapter 2
2-1 Bridge Column Axial Loads ................................................................. 54
2-2 Earthquake Level Input (g) ................................................................. 55

Chapter 4
4-1 Comparison of Bent 1-3 Peak Displacements of Measured and OpenSees Predicted ................................................................. 56

Chapter 6
6-1 Estimated Displacement Ductility Capacities of Bents in Bridge Transverse Direction ................................................................. 57
6-2 Bridge specimen modal mass participation factors for modes 1 through 5 ..................................................................................... 57
6-3 Maximum Displacements under Different Level of Motions ............. 58
6-4 Maximum Bent Displacements and Corresponding Displacement Ductility ............................................................................. 59
List of Figures

Chapter 2
2-1 Bridge Model Elevation ................................................................. 60
2-2 Model Bridge Bent Dimensions .................................................. 61
2-3 Bridge Column Section ............................................................... 62
2-4 Post-tensioned Rods for Cap-Beam Deck Connection ................. 62
2-5 Inverted T-Beam .................................................................. 62
2-6 Abutment Seat .................................................................. 63
2-7 Superimposed Mass Arrangement ............................................. 64
2-8 Full Scale Input Acceleration Records with Compressed Time .... 65

Chapter 3
3-1 Steel01 Element .................................................................. 66
3-2 Steel02-Hysteretic Behavior with Isotropic Hardening in Tension .... 66
3-3 Concrete01 Material Parameters .............................................. 67
3-4 Typical Hysteretic Stress-Strain Relation of Concrete01 Model ...... 67
3-5 Elastic-Perfectly Plastic Gap Material ........................................ 68
3-6 Parameters for Hysteretic Material .......................................... 68
3-7 Definition of Pinching4 Uniaxial Material Model ....................... 69
3-8 Fiber Section Segments for a Circular Reinforced Concrete Member .. 69
3-9 Analysis Components in OpenSees ......................................... 70

Chapter 4
4-1 Example of the Prototype Location in a Multi-Span Bridge .......... 71
4-2 2-Span Bridge Model ............................................................... 72
4-3 Bridge Bents .................................................................. 73
4-4 Bridge Nodal Configuration ..................................................... 74
4-5 Superimposed Mass Arrangement ............................................. 75
4-6 OpenSees Column Displacement Prediction for test 12 ............. 76
4-7 OpenSees Column Displacement Prediction for test 13 ............. 77
4-8 OpenSees Column Displacement Prediction for test 14 ............. 78
4-9 OpenSees Column Displacement Prediction for test 15 ............. 79
4-10 OpenSees Column Displacement Prediction for test 16 .......... 80
4-11 OpenSees Column Displacement Prediction for test 17 .......... 81
4-12 OpenSees Column Displacement Prediction for test 18 .......... 82
4-13 OpenSees Column Displacement Prediction for test 19 .......... 83

Chapter 5
5-1(a) Bridge Nodal Configuration ............................................... 84
5-1(b) Bridge Bent Nodal Configuration .......................................... 85
5-2 Column Section Fiber Configuration ........................................ 85
5-3 Moment-Curvature for Column Section with 8 Slices .......... 86
5-4 Moment-Curvature for Column Section with 16 Slices .......... 86
<table>
<thead>
<tr>
<th>Chapter 5</th>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5-5</td>
<td>Moment-Curvature Comparison for Column Sections with 16 and 8 Slices</td>
<td>87</td>
</tr>
<tr>
<td>5-6</td>
<td>Force-Displacement Comparison for Bent 1 with Different Column Element Models</td>
<td>87</td>
</tr>
<tr>
<td>5-7</td>
<td>Moment Curvature for Bridge Column Section</td>
<td>88</td>
</tr>
<tr>
<td>5-8</td>
<td>Pushover Analysis of Bridge Bents</td>
<td>88</td>
</tr>
<tr>
<td>5-9</td>
<td>Post-tensioned Rods for Cap-Beam Deck Connection</td>
<td>89</td>
</tr>
<tr>
<td>5-10</td>
<td>Moment Curvature Curve for Cap Beam-Deck Connection</td>
<td>89</td>
</tr>
<tr>
<td>5-11</td>
<td>Force-Displacement Curve for One Foot Length of Prototype Backwall</td>
<td>90</td>
</tr>
<tr>
<td>5-12</td>
<td>Force-Displacement Backbone for Abutment Backwall in Bridge Model</td>
<td>90</td>
</tr>
<tr>
<td>5-13</td>
<td>Bridge Abutment Deck Interaction Models</td>
<td>91</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter 6</th>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6-1</td>
<td>Bent Pushover</td>
<td>92</td>
</tr>
<tr>
<td>6-2</td>
<td>Pushover for Bent 1</td>
<td>92</td>
</tr>
<tr>
<td>6-3</td>
<td>Pushover for Bent 2</td>
<td>93</td>
</tr>
<tr>
<td>6-4</td>
<td>Pushover for Bent 3</td>
<td>93</td>
</tr>
<tr>
<td>6-5</td>
<td>Elastic mode shapes of shake table bridge specimen</td>
<td>94</td>
</tr>
<tr>
<td>6-6</td>
<td>Bent 1 Displacement in Longitudinal and Transverse Directions</td>
<td>95</td>
</tr>
<tr>
<td>6-7</td>
<td>Bent 2 Displacement in Longitudinal and Transverse Directions</td>
<td>95</td>
</tr>
<tr>
<td>6-8</td>
<td>Bent 3 Displacement in Longitudinal and Transverse Directions</td>
<td>96</td>
</tr>
<tr>
<td>6-9</td>
<td>Bent 1 Displacement</td>
<td>96</td>
</tr>
<tr>
<td>6-10</td>
<td>Bent 2 Displacement</td>
<td>97</td>
</tr>
<tr>
<td>6-11</td>
<td>Bent 3 Displacement</td>
<td>97</td>
</tr>
<tr>
<td>6-12</td>
<td>Bent 1 Resultant Displacement History</td>
<td>98</td>
</tr>
<tr>
<td>6-13</td>
<td>Bent 2 Resultant Displacement History</td>
<td>98</td>
</tr>
<tr>
<td>6-14</td>
<td>Bent 3 Resultant Displacement History</td>
<td>99</td>
</tr>
<tr>
<td>6-15</td>
<td>Abutment Displacement</td>
<td>99</td>
</tr>
<tr>
<td>6-16</td>
<td>Abutment and Deck-end Displacement in Bridge Longitudinal Direction</td>
<td>100</td>
</tr>
<tr>
<td>6-17</td>
<td>Abutment-Deck Gap Size History</td>
<td>100</td>
</tr>
<tr>
<td>6-18</td>
<td>Abutment Gap Element (Actuator) Force</td>
<td>101</td>
</tr>
<tr>
<td>6-19</td>
<td>Bent 1 Displacement (Model 2)</td>
<td>101</td>
</tr>
<tr>
<td>6-20</td>
<td>Bent 2 Displacement (Model 2)</td>
<td>102</td>
</tr>
<tr>
<td>6-21</td>
<td>Bent 3 Displacement (Model 2)</td>
<td>102</td>
</tr>
<tr>
<td>6-22</td>
<td>Bent 1 Displacement in Longitudinal and Transverse Directions</td>
<td>103</td>
</tr>
<tr>
<td>6-23</td>
<td>Bent 2 Displacement in Longitudinal and Transverse Directions</td>
<td>103</td>
</tr>
<tr>
<td>6-24</td>
<td>Bent 3 Displacement in Longitudinal and Transverse Directions</td>
<td>104</td>
</tr>
<tr>
<td>6-25</td>
<td>Bent 1 Resultant Displacement History (Model 2)</td>
<td>104</td>
</tr>
<tr>
<td>6-26</td>
<td>Bent 2 Resultant Displacement History (Model 2)</td>
<td>105</td>
</tr>
<tr>
<td>6-27</td>
<td>Bent 3 Resultant Displacement History (Model 2)</td>
<td>105</td>
</tr>
<tr>
<td>6-28</td>
<td>Abutment Displacement</td>
<td>106</td>
</tr>
<tr>
<td>6-29</td>
<td>Abutment-Deck Gap Size History for Seven Levels of Motions (Model 2)</td>
<td>106</td>
</tr>
<tr>
<td>6-30</td>
<td>Abutment Gap Element Force (Model 2)</td>
<td>107</td>
</tr>
<tr>
<td>6-31</td>
<td>Bent 1 Resultant Displacement History (Model 1)</td>
<td>107</td>
</tr>
<tr>
<td>6-32</td>
<td>Bent 2 Resultant Displacement History (Model 1)</td>
<td>108</td>
</tr>
<tr>
<td>6-33</td>
<td>Bent 3 Resultant Displacement History (Model 1)</td>
<td>108</td>
</tr>
</tbody>
</table>
Comparison between Bent 1 Resultant Displacements in Model 1 & Model 2 during Event 2..............................109
Comparison between Bent 1 Resultant Displacements in Model 1 & Model 2 during Event 7.........................................................109
Comparison between Bent 1 Longitudinal Displacements in Model 1 & Model 2 during Event 2.........................................................110
Comparison between Bent 1 Longitudinal Displacements in Model 1 & Model 2 during Event 7.........................................................110
Bent 1 Resultant Displacement History (Model 3)...........................................111
Bent 2 Resultant Displacement History (Model 3)...........................................111
Bent 3 Resultant Displacement History (Model 3)...........................................112
Comparison between Bent 1 Resultant Displacements in Model 2 & Model 3 during Event 2.........................................................112
Comparison between Bent 1 Resultant Displacements in Model 2 & Model 3 during Event 7.........................................................113
Comparison between Bent 2 Resultant Displacements in Model 2 & Model 3 during Event 2.........................................................113
Comparison between Bent 2 Resultant Displacements in Model 2 & Model 3 during Event 7.........................................................114
Comparison between Bent 3 Resultant Displacements in Model 2 & Model 3 during Event 2.........................................................114
Comparison between Bent 3 Resultant Displacements in Model 2 & Model 3 during Event 7.........................................................115
Abutment Gap Element Force (Model 3)...........................................115
CHAPTER 1
INTRODUCTION

1.1 Background

The extensive damage to bridges during recent earthquakes and the tremendous human and economic loss associated with the disruption of lifeline in urban areas have led to a world wide effort toward improving the seismic performance of the bridges. In addition to structural damage and potential loss of life resulting from an earthquake, a severe economical impact is likely to follow when closure of bridges disrupts the transportation infrastructure. As a result of observed bridge damage in the recent earthquakes, a significant amount of research has been carried out worldwide over the past decade to improve seismic design of bridges. With the inherent weakness in previously adopted design concepts being identified as a lack of emphasis on ductility, recent research has promoted application of the capacity design philosophy for seismic design of bridge structures.

Over the past ten years, leading structural engineers have promoted the development and application of performance-based seismic design concepts. Performance-based design (PBD) is gradually finding its way into design codes as more tools to implement it become available. If performance-based design, which is closely linked to damage limit states, is to eventually become a reality, there is a fundamental need to develop procedures that provide a better estimate of structural performance under expected seismic events. In current design codes of practice, bridges are designed to undergo a substantial amount of ductile inelastic deformation without collapse. The bridge superstructure is designed to remain elastic while the inelastic deformations are accommodated in the columns thought the formation of plastic hinges. Based on this design philosophy, bridges maintain their gravity loads carrying capacity and lives are saved. However, localized structural damage is permitted.

In order for the design to be effective and reliable, the structural performance during inelastic response from earthquake excitations should be well understood. The design engineer should be able to calculate the bridge performance using reliable analytical simulation models. However, due to the complexity of structural performance during strong earthquakes and the highly nonlinear response of structural systems, reliability of computer simulation models is highly dependent on appropriate selection of material models and analysis techniques.

This study focuses on the development of a 4-span bridge analytical model using OpenSees. Three ¼-scale 4-span bridge models are planned to be tested at the University of Nevada, Reno (UNR) shake tables as a part of a collaborative Network for Earthquake Engineering Simulation Research-Small Group (NEESR-SG) study entitled “Seismic Performance of Bridge Systems with Conventional and Innovative Design.” This pre-test analytical study is intended to predict the performance of the first bridge model during shake table testing. The bridge piers are of drop-cap type with a continuous post-tensioned deck connected to three, two-column bent caps and roller type connections at the abutments. The bridge model will be tested under bilateral horizontal motions applied by the UNR shake tables at the bridge pier footings and longitudinal motions applied at
the abutments through actuators. The results of the analytical analyses will be used to predict the bridge model capacity and displacements under the planned shake table motions. The analytical model can then be verified and calibrated based on the experimental data obtained from the shake table testing of the physical bridge model.

1.2 Finite Element Analysis Programs

Practicing engineers today usually perform earthquake engineering simulations on a computer using the developments offered by a finite element analysis (FEA) program. Typically, a FEA program bundles all the procedures and program kernels that are developed by an individual organization. Preferred FEA programs are those that are able to accommodate new developments such as element formulation, material relations, analysis algorithms, and solution strategies as technologies and structural theories continue to advance. The introduction of the object-oriented programming approach has brought with it a revolution in the way software is being written. Object-oriented design principles and programming have been utilized in FEA software development to support better data encapsulation and to facilitate code reuse (McKenna 1997).

1.2.1 OpenSees Software

The Pacific Earthquake Engineering Research (PEER) Center has created a software framework for developing applications to simulate the performance of structural and geotechnical systems subjected to earthquake loading. This framework was earlier referred to as "G3" but has been renamed "OpenSees" (Open System for Earthquake Engineering Simulation). Using the internet as a communication channel, the framework provides users the ability to pick and choose the most appropriate methods and software features for solving a problem. The framework also includes data and project management functionalities. A database system is employed to store selected analysis results and to provide flexible data management and data access. Project management functions allow users to access information about previous simulations of related models stored in different sites. Detailed information can be found at http://opensees.berkeley.edu. The OpenSees homepage maintains resources for users and developers, including downloadable source code, extensive documentation, and instructions on how to contribute code. Several links to other research projects that are using the OpenSees tool are provided.

OpenSees can basically be seen as a finite element code for nonlinear static and dynamic structural analyses. A main feature of OpenSees is that the code is developed under the object oriented programming paradigm. In OpenSees, C++ is used, which is an object-oriented programming language. OpenSees also uses Tcl, a pre-Java scripting language created by John Ousterhout (1994) that enables engineers to program their models without having to learn the intricacies of C++. OpenSees can use Tcl scripts to drive the simulations. The scripts provide means for defining structural models, analysis procedures, and selections of the output. OpenSees is thus an interactive environment because of the Tcl interpreter.

Due to the object-oriented design approach, the OpenSees software architecture is very modular and transparent. This allows users and developers in different fields (e.g., engineering, computer science, and numerical analysis) to develop and modify specific modules with relatively little dependence on other modules. Developers do not need to
know everything that is in the framework, thus allowing them to concentrate on making improvements in areas of their own expertise (Fenves et al. 2004).

1.3 Bridge Modeling

Many structural failures of reinforced concrete bridges during earthquakes are due to the poor behavior of the structures in the inelastic range. The safeguard against intense earthquakes depends on reliable inelastic responses of the structural elements which provide mechanisms for dissipation of earthquake energy. This suggests that structures in earthquake zones must be designed for strength and also for ductility. However, the material characteristics of reinforced concrete are still not clearly describable with models, and the design of reinforced concrete piers depends heavily on empirical results (Priestly et al. 1981; Park et al. 1982; Watson and Park 1994; Pincheira et al. 1999).

Reliable evaluations of structural safety against destructive earthquakes are needed, where the dynamic performance of reinforced concrete bridge piers deformed into the ultimate states is accurately described. Recently reported studies include the inelastic analysis of reinforced concrete columns for the monotonic loading case by Yalcin and Saatcoiglu (2000), general nonlinear finite element analysis of concrete bridges subjected to earthquakes by Sritharan et al. (2000), slip effects in reinforced concrete members under earthquakes by Spacons and Limkatanyu (2000), etc. Summaries of earlier work in this area were presented in ASCE International Workshop (1993).

This report presents analytical modeling of a 4-span reinforced concrete bridge model using OpenSees version 1.6.2. The bridge columns are modeled using fiber sections with concrete and steel properties assigned to pre-defined concrete and steel models in OpenSees. The bond-slip effect is added to the column sections at the top and bottom. Semi-rigid connections between the cap-beam and deck are defined. Gap elements are used to represent the 0.5 in (12.7 mm) gap between the abutment and the bridge deck at both ends. To include the deck-abutment interaction, nonlinear force-displacement relationship is assumed for the abutment spring stiffness.

1.3.1 Bridge-Abutment Interaction

Modeling the contribution of bridge abutments to overall bridge seismic response has been the focus of a significant research effort in the past decade. It has been shown by many studies that the abutment response could significantly influence the response of short and medium length bridges. There are basically two groups of methods to determine the abutment stiffness. The first group is based on the Caltrans (1999) procedure for determining longitudinal and transverse stiffness of an abutment (Goal and Chopra 1997). Longitudinal direction abutment stiffness is formulated by combining passive backwall pressure, shear strength of the wall itself, and strength and stiffness of the pile groups (if any) supporting the abutment. These stiffness and strength values have been validated in large scale abutment tests (Maroney et al. 1994). Transverse direction values are computed assuming a seat-type abutment with resistance from the wing walls, shear keys, and pile group.

The second group of methods involves deriving the response of the soil embankment with defined cross section geometry, soil shear moduli, and soil unit weight. Wilson and Tan (1990) determine the stiffness per unit length of the embankment. The total stiffness is then derived from an estimated wing wall length. Wilson and Tan (1990) assume that
abutment longitudinal stiffness is the same as transverse stiffness. Zhang and Makris (2001) generalize the formulation to any embankment geometry by using a similar procedure.

The other fundamental factor governing abutment response is the inertial force it generates during an earthquake. Thus coupled with the stiffness quantities defined above, the abutment becomes a single degree of freedom oscillator attached to each end of the bridge. Determining the mass participating in abutment response is highly uncertain and is usually approximated by a critical length of the embankment. Different researchers have proposed participating lengths that best match recorded data (Werner 1994; Wissawapaisal and Aschheim 2000).

To define the abutment stiffness and strength in the present study, the approach based on the first group is used. In the new design procedure (ATC 1996: SDC 2001) the contribution of the backwall and the foundation supporting the abutment is not included. The procedure the abutment backwalls to be sacrificial and sheared off at the very beginning of the earthquake in order to avoid any damage to the pile foundation. Therefore nonlinear force-displacement capacity of the bridge abutment in a seismic event is developed mainly from the mobilized passive pressure behind the abutment wall. The formulation derived by Shamsabadi et al. (2005) is used in this study to calculate a nonlinear abutment load-displacement relationship. The relationship is a function of the abutment height, embankment soil properties, and the mobilized interface friction angle between the embankment and the bridge. In the present study the backwall is assumed to be sheared off and its mass is considered to be the mass participating in abutment response in the bridge longitudinal direction. A granular type of soil with 30 degree internal friction angle is assumed for the embankment. Details of the abutment modeling are provided in Chapter 5.

1.4 Objectives and Scope

The main objective of this study is to determine the seismic response of a ¼ scale 4-span reinforced concrete bridge system under different level of excitations through an analytical study using OpenSees. This analytical study is a component of a larger multi-university study entitled ‘Seismic Performance of Bridge Systems with Conventional and Innovative Design.’ The analytical model results are used to predict the physical bridge model response on the UNR shake tables. Through this analytical study the capacity of the bridge was estimated and longitudinal abutment displacement histories were found so that they can be applied during the shake table testing of the bridge model.
CHAPTER 2
Bridge Model Description and Input Motions

2.1 Introduction
A prototype bridge is assumed as the basis of the model that is constructed for shake table testing. The main features of the assumed prototype are the same as those of the prototype in a previous study of a two-span bridge (Johnson et al. 2006). The primary reason for using that prototype in the present study is to allow for comparison of the response of the models form the two studies. The prototype bridge is a continuous post-tensioned reinforced concrete box girder structure with drop bent caps. The bridge has 4 spans with three, two-column bents and roller supports at the abutments. There is a 2-in (50-mm) gap between the end of the bridge deck and the abutment backwall. The column heights vary among the piers to make the model asymmetric with respect to the transverse axis passing through the center. Hence, the bridge undergoes in-plane rotation when subjected to seismic load in the transverse direction.

For the shake table testing, it was decided to construct a reduced scaled model of the prototype bridge. Based on the capacity of shake tables at the University of Nevada, Reno, and their spacing, a quarter scale of the prototype is chosen for the bridge model. Components of the model are designed based on the current design codes. In this chapter the selection of the prototype and the model bridge are briefly discussed. The input motions for shake table tests are also presented in this chapter. The descriptions are intended to only provide an overview to facilitate the understanding of this report. Detailed information about the prototype, the model, and the motions will be presented elsewhere.

2.2 Prototype Dimensions
A typical span length for post-tensioned box girder bridges is between 100 ft (30.5m) and 250 ft (76.2m) (Caltrans 2001). Span lengths of 98 ft (29.9m) and 116 ft (35.3m) were selected for the end spans and the middle spans of the bridge, respectively. The prototype bridge deck width and the transverse spacing between columns are determined based on the average section properties for typical highway bridges built in California (Pulido et al. 2002). A width of 41.5 ft (12.6m) is selected for the superstructure section to accommodate two 12 ft (3.7 m) wide lanes and wide shoulders. The spacing between columns in each bent is 25 ft (7.6 m) with 6.25 ft (1.91 m) overhangs. A constant deck depth of 56 in (1.42 m) for the bridge superstructure is assumed which is approximately 4% of the longest bridge span length, as recommended by American Association of State Highways and Transportation Officials (AASHTO) Standard Specification (AASHTO 2002).

A prototype bridge column diameter of 4 ft (1.22 m) is assumed to lead to a representative column axial load index of approximately 7%. Axial load index is defined as ratio of the axial load to the product of column section gross area and the specified concrete compressive strength. The bent cap width is taken as 5 ft (1.52 m) which is 1 ft (304 mm) larger than the bridge column diameter, providing 1/2 ft (152 mm) overhang from the column faces on both sides.

As mentioned earlier, to investigate the effect of in-plane rotation, unequal bent heights were assumed in the bridge, with the first bent having the shortest columns with
20 ft (6.09 m) height. The column heights in the second and third bents are assumed to be 28 ft (8.53 m) and 24 ft (7.31 m), respectively.

### 2.3 Model Bridge Dimensions

The test bridge is a quarter-scale geometric model of the prototype bridge. This is the largest possible scale that can be used to ensure reaching model failure without exceeding the capacity of the shake tables. At the same time this scale allows the utilization of conventional bars as longitudinal steel, and is sufficiently large to accommodate regular concrete instead of micro-concrete. With appropriate scaled prototype motion for the bridge model, which is discussed in Section 2.5, it is anticipated that the response of the model bridge would structurally match the response of the prototype.

To facilitate the identification of different bridge bents and abutments, they are sequentially numbered. The shake tables at University of Nevada, Reno are located in line in the north-south direction. The bridge elevation in Fig. 2-1 is the view from the East, thus showing the south abutment on the left and the north abutment on the right. The south and north abutments are labeled ‘Abutment 1’ and ‘Abutment 2’, respectively. The bent numbering also starts from the south, assigning number 1 to the bent adjacent to the Abutment 1, number 2 to the middle bent, and number 3 to the north bent. Bent 1 is the shortest bent with 5 ft (1.52m) clear column height as compared to Bent 2 with 7 ft (2.13 m) and Bent 3 with 6 ft (1.83 m) clear column height as shown in Fig. 2-2.

#### 2.3.1 Column Dimensions and Details

The bridge columns are circular with 12 in (305 mm) diameter. The columns are reinforced with 16-#3 bars (Fig. 2-3), which provides a longitudinal steel ratio of 1.56 percent. This is a representative steel ratio for typical bridge columns. The concrete cover is 1/2 in (12.7 mm). The lateral reinforcement consisted of W 2.9 (0.192 in (4.9 mm) wire) spiral steel with 1.25 in (31.7 mm) pitch.

#### 2.3.2 Bridge Superstructure

The prototype bridge superstructure is a multi-cell post-tensioned box girder. The ¼ scaled model bridge superstructure with exact details would require a slab thickness of only 2 in (50 mm), which would be difficult to construct. Furthermore, a scaled superstructure duplicating all the details is unnecessary, since the superstructure is not expected to undergo non-linear response during the tests. The superstructure remains essentially elastic and un-cracked due to the longitudinal post-tensioning and its high relative strength and stiffness compared to the substructure. Therefore the bridge model superstructure is constructed with solid rectangular beams, having moments of inertia about the orthogonal axes equal to the prototype moments of inertia multiplied by 1/256 to account for the scaling. Hence the superstructure consists of three solid rectangular shape section beams in each span which are post-tensioned transversely to behave as a unit. Each beam has a cross section of 30 in (760 mm) by 14 in (360 mm) resulting in a 90 in (2.29 m) wide slab with 14 in (360 mm) thickness. The resulting dimensions are shown in Fig. 2-1.

The beams are supported on the projected flanges of the inverted T-beams that form the bent caps, and are connected to the T-beams by two post-tensioned 1¼ inch (31.7 mm) diameter rods as shown in Fig 2-4. The cross section of the inverted T-beam is
shown in Fig. 2-5. The longitudinal superstructure post-tensioning tendons pass through the superstructure beams and the inverted T-beams, making the entire superstructure behave as a unit.

2.3.3 Bridge Abutment Details

In the bridge model the abutment is represented by an L-shaped seat (Fig. 2.6). The bridge deck is supported on rollers on the top of the L-shaped seat to enable deck to move in longitudinal and transverse directions relative to abutment. The L-shaped seat itself is free to move only longitudinally on top of a rigid block by using two sliding rails in the bridge longitudinal direction on top of a rigid block. Two 14 in (356 mm) long W8x67 beam sections as shown in Fig. 2-6 are used to provide the sliding connection. The top flange and the upper portion of the web of the steel beams are embedded into the abutment seat and the bottom flange is inserted into rail groove that is attached to a steel plate supported on the rigid block. The contact surfaces between the rail groove and the bottom flange of the beam section are covered by Teflon sheets to minimize the friction.

Since the bridge model specimen test setup does not include the scaled prototype abutment components, the displacement histories at the top of the abutments due to the ground motion are used as input by actuators which are connected to the abutment seats at both bridge ends. The gap between the end of the model bridge deck and the abutment seat is ½-in (12.7 mm) to match the specified 2-in (50 mm) gap in the prototype.

2.4 Superimposed Dead Loads

The target axial load index for the bridge columns is 7 percent, which represents a typical bridge column axial load level. An axial load index of 7 percent in the model is equivalent to a column axial load of 39.6 kips (176.2 kN). Due to the scaling effect, the weight of the superstructure model provides a smaller axial stress than in the prototype. Some of the required axial load is provided by the post-tensioning and the self weight of the bridge model. The rest is provided by lead bricks and concrete blocks that are attached on the top of the superstructure. To determine the configuration of superimposed weights to produce the target axial load in the columns, an elastic model of the bridge specimen is developed and different trials of locations of masses are checked. It was determined that to produce the target axial load index in the bents the arrangement shown in Fig. 2-7 is required. The resulting axial load indexes for the columns are as listed in Table 2-1.

2.5 Selection of Shake Table Motions

The motions that are used for the shake table tests are based on the 1994 Northridge earthquake as recorded from the ground station at the Century City Country Club North, after they are modified to account for soil-structure interaction effects. The Century City station is owned by the California Strong Motion 38 Instrumentation Program, and is located at 34.063 Latitude, -118.418 Longitude. These motions are the same as those used in the study of a two-span bridge model (Johnson et al. 2006) and are selected for the present study to facilitate the comparison of the performance of the bridge models from the two studies. Both the 90 degree and 360 degree lateral components are used in the present study. Throughout this report, the two components are referenced as motion 1 and motion 2, for the transverse and longitudinal
directions, respectively. The calculation of the motions was a joint effort between researchers from the University of California at Davis and the University of Washington (Shin et. al 2006) as part of a collaborative study on soil foundation structure interaction effects (Wood et. al. 2004). The general method for the motion derivation is discussed in Johnson et al. (2006). The time axis of the prototype motions are compressed by a factor of 2 to account for the quarter scale of the bridge specimen. After the time compression, each record is 20 seconds long. The full scale motions are shown in Fig. 2-8.

The bridge model is analyzed for bi-axial excitations simultaneously in the transverse and longitudinal directions. The input motion protocol is the same as that used in Johnson et al. (2006) and is the same for all three shake tables. The same motions are used in successive runs, but the peak ground acceleration (PGA) is scaled. Motion 1 record is scaled with increasing factors that led to PGAs from 0.075 g to 1.32 g. The same factors are used for motion 2. Since motion 2 has higher PGA, the PGAs in the corresponding motions in the longitudinal direction range from 0.10 g to 1.77 g. These same motions are to be used as the input in shake table testing that will follow this analytical study. The input motions protocol is listed in Table 2-2.

### 2.6 Material Properties

The expected material properties are used in the analysis. The specified 28-day concrete compressive strength is 4500 psi (31 MPa), but expected concrete strength is assumed to be 5000 psi (34.5 MPa). Grade 60 reinforcement steel is specified for mild steel in the bridge model. Based on past experience a yield stress of 68 ksi (469 MPa) at the strain of 0.002 was assumed for steel. The elastic modulus is assumed to be 29,000 ksi (200,000 MPa). The strain hardening for steel is assumed to have a slope of 212 ksi (1464 MPa). No yield plateau is assumed in the constituent stress-strain relationship for steel.
CHAPTER 3
OpenSees Computer Program

3.1 Introduction
The analytical modeling described in this chapter is conducted using the Open System for Earthquake Engineering Simulation (OpenSees) program (OpenSees 2002). This chapter presents the highlights of the important features of the program that are utilized in the present study.

OpenSees is an object-oriented software framework for simulation applications in earthquake engineering using finite element methods (McKenna and Fenves 2000). The goal of the OpenSees development is to improve the modeling and computational simulation in earthquake engineering through open-source development. OpenSees is intended to serve as the computational platform for research in performance-based earthquake engineering at the Pacific Earthquake Engineering Research (PEER). The core development team members for the initial development of OpenSees are members of PEER. However, as the program framework grows, it is anticipated that others in PEER and the engineering community will be involved, since the OpenSees source code is freely available. OpenSees has been selected as the simulation component for the George E. Brown Jr. Network for Earthquake Engineering Simulation in the NEESgrid System Integration Project.

The software architecture and open-source approach for OpenSees provide powerful features for advanced simulation of nonlinear structural and geotechnical systems. First, the modeling approach is very flexible. It allows selection and various combinations of a number of different element and material formulations. A second advantage is that there is a wide range of solution procedures and algorithms that the user can adopt to solve difficult nonlinear problems for static and dynamic loads. Another feature is that OpenSees has a fully programmable scripting language for defining models, solution procedures, and post-processing that can provide simple problem solving capability. Finally, OpenSees provides a flexible interface to computer resources, storage and databases, and network communication to take advantage of high-end computing systems. Structural and geotechnical models can be analyzed on computers with single or parallel processors.

OpenSees uses Tcl, a general purpose scripting language that has been extended with commands for OpenSees (Welch et al. 2003). The Tcl language provides useful programming tools, such as variables manipulation, mathematical expression evaluation and control structures. The OpenSees interpreter adds commands to Tcl for finite element analysis. Each of these commands is associated with a C++ procedure that is provided. It is this procedure that is called upon by the interpreter to parse the command.

OpenSees is comprised of a set of modules to perform creation of the finite element model, specification of an analysis procedure, and the output of results. These modules are discussed in this chapter.

3.2 OpenSees Modeling Capabilities
As in any finite element analysis, the first modeling step is to subdivide the system into elements and nodes, to define loads acting on the elements and nodes, and to define nodal constraints. OpenSees includes a general model builder for creating three-
dimensional frame and continuum models using Tcl. The “ModelBuilder” is used to build and add the objects discussed in the following sections. Users can generate models using Tcl procedures. Additional versions of ModelBuilder suited to a particular type of model may also be developed.

3.2.1 Elements

There is variety of elements in OpenSees. Since only Frame elements are used in bridge modeling, only this element type is discussed. This type of element is used to model beams and columns in planar and three-dimensional structures. The Frame element uses a general three-dimensional beam-column formulation which includes the effects of biaxial bending, torsion, axial deformation, and biaxial shear deformations.

A Frame element is modeled as a straight line connecting two points. Each element has its own local coordinate system for defining section properties and loads, and for interpreting output. Element internal forces are produced at the element ends and at a user-specified number of equally spaced output stations along the length of the element. In OpenSees there is an option to specify joint offsets independently at each end of the element. Joint offsets are external to the element and do not have any mass or load. Internally the program creates a fully rigid constraint along the joint offsets. This option can be used in beam-column connections to model the stiffening effect of the connection geometry.

3.2.1.1 Elastic Beam-Column Element

This type of element is used to represent beams and columns with linear elastic behavior. The input properties are: cross sectional area of the element, Young’s modulus, shear modulus, torsional inertia, and the moment of inertia about the local axes of the element. The elastic beam column element is used for the bridge superstructure and the cap beam with uncracked properties, and for the part of the columns outside plastic hinge zones with cracked properties.

3.2.1.2 Non-Linear Beam Column Element

There are basically two types of non-linear beam column elements. The first type is force based element which has force formulation. The second type is displacement based elements which is called ‘dispBeamColumn’ and incorporates distributed plasticity with linear curvature distribution.

In OpenSees there are two forced based element types. The first is ‘nonlinearBeamColumn’ which is based on force formulation and considers the spread of plasticity along the element. The second type is ‘beamWithHinges’ which is based on flexibility formulation and considers concentrated plasticity over specified end hinge lengths and elastic behavior for the rest of the member. The ratios of the element end hinge lengths to the element length are two of the required input for this type of element. The elastic properties are integrated only over the elastic part of the element. Forces and deformations of the inelastic regions are sampled at the hinge midpoints.

The forced based non-linear beam column ‘nonlinearBeamColumn’ is used for the bridge columns in the plastic hinge zones. The rest of the column length is modeled with elastic beam column elements with cracked properties. The cracked column section stiffness (EI) is found from the moment curvature analysis.
3.2.1.3 Zero-Length Element
A zero length element is defined by two nodes at the same location. The nodes are connected by multiple uniaxial material objects to represent a force-deformation relationship for the element. Uniaxial hysteretic properties can be defined for the zero-length element. In the bridge model, zero-length elements were used at the column-footing and column-cap beam connections to model bond-slip moment-rotation properties at column ends.

3.2.1.4 Zero-Length Section Element
This element is defined by two nodes at the same location, but the nodes are connected by a single section force deformation object to represent a force-deformation relationship for the element. The deck abutment connections at the bridge ends which include a gap and abutment elements are modeled with a zero length section element, since they only take axial loads. Uniaxial sections with specified longitudinal force-deformation properties are assigned to the zero length section elements.

3.2.2 Uniaxial Materials
The uniaxial material command is used to construct a uniaxial object which represents uniaxial stress-strain or force-deformation relationship. Different types of uniaxial materials are introduced in OpenSees, but only the types that are used in bridge modeling are discussed in this section.

3.2.2.1 Steel01 Material
This type of material represents steel with bi-linear stress strain properties (Fig. 3-1). The input parameters are the yield strength, initial elastic stiffness, strain hardening ratio, and the optional isotropic hardening.

3.2.2.2 Steel02 Material
This material model is used to construct a uniaxial steel material with isotropic strain hardening. It also provides control over the transition from elastic to plastic portions. The strain hardening effect is optional and can be specified in compression or tension. Figure 3-2 shows stress-strain curve for Steel02 material with strain hardening in tension. This type of material is used for the reinforcement in the bridge columns with fiber section properties, without isotropic strain hardening effect.

3.2.2.3 Concrete01 Material
The Concrete01 material model assumes no tensile strength for concrete (Fig. 3-3). It is a Kent-Scott-Park concrete model (Kent and Park 1971) with degraded linear unloading/reloading stiffness according to the work of Karsan-Jirsa (Karson and Jirsa 1969). The concrete 28 days compressive strength, concrete strain at maximum strength, and concrete crushing strain and strength are the required input for this concrete. The concrete strength remains constant beyond the crushing point. A typical hysteretic stress-strain relationship for Concrete 01 is shown in Fig. 3-4.
3.2.2.4 Elasto-Plastic Gap Material

This material can be used to model a gap in tension or compression as shown in Fig. 3-5. In the bridge model, the gap between the abutment and the deck is modeled as a compression gap material since the force can only be transmitted between the deck and abutment when the gap is closed. The input for this element consists of initial stiffness, yield stress, and the initial gap size. Large values for the initial stiffness and the yield stress were specified to force the gap element to transfer the load to surrounding elements without any energy absorption.

3.2.2.5 Hysteretic Material

This material model simulates a uniaxial tri-linear hysteretic material object with pinching of force and deformation and degraded unloading stiffness as a function of the maximum ductility demand in previous cycles (Fig. 3-6). The degraded unloading stiffness can be modified due to energy damage in the previous cycle as a function of the ratio of final to initial material stiffness for that cycle. The input for this type of material includes the stress and strain for three points of backbone envelope in each of the positive and negative directions. No pinching, damage, and degradation are specified in bridge model for Hysteretic material which is assigned to zero length elements discussed in Section 3.2.1.3.

3.2.2.6 Pinching Material

This is a uniaxial material that represents a pinched load-deformation response and exhibits degradation under cyclic loading (Fig. 3-7). Cyclic degradation of strength and stiffness occurs in three ways; unloading stiffness degradation, reloading stiffness degradation, and strength degradation. In this material model, four positive and four negative force-deformation points are specified. The floating points are given in terms of the ratio of the force and deformation at which unloading and reloading occurs to the maximum force-deformation demand if unloading and reloading degradations are ignored.

3.2.3 Section Models

The ‘Section’ command is used to construct a section force deformation object which represents force-deformation relationship at specified points. In general there are three types of sections in OpenSees as follows:

Elastic: Is used to construct an elastic section object which is defined by material constants and accommodates uncoupled axial and bending response.

Fiber: Is discretized by fibers which collectively define section response (Fig. 3-8). The stress- strain response of fibers is integrated to determine resultant behavior. A fiber section has a general geometric configuration formed by sub regions of simpler, regular shapes (e.g. quadrilateral, circular, and triangular regions), called patches. For modeling reinforced concrete sections, reinforcement bars can be specified by ‘layer’ subcommand. The geometric input for a circular reinforced concrete section is as shown in Fig. 3-8.
Properties of steel and concrete materials, discussed in Sections 3.2.2.1 to 3.2.2.3, can be specified as uniaxial material.

**Uniaxial:** Is used to construct a section which uses a previously-defined uniaxial material object to represent a single section force-deformation response quantity. For example, a uniaxial material can be used to model section moment-curvature behavior. In order to combine multiple uncoupled uniaxial materials to define additional section force-deformation relations, ‘Aggregator’ section command can be used. For example, a fiber section can be combined with an uncoupled shear force-deformation relation by using Aggregator command. This command is used to add torsional and shear stiffness to the column sections defined with fiber sections.

### 3.2.4 Load Patterns

The pattern command is used to construct a load pattern object. Basically there are three types of patterns for load application: plain, uniform excitation, and multiple support excitations.

#### 3.2.4.1 Plain Pattern

All the static constant loads can be defined using plain pattern. The gravity loads including bridge weight and superimposed dead loads are defined as nodal loads using this command.

#### 3.2.4.2 Uniform Excitation Pattern

This command is used when the structure is subjected to a uniform applied load in each direction. The same ground motion record at different support points can be applied to the structure using this command. Uniform excitation pattern is used for ground motion accelerations in the transverse and longitudinal directions for the ‘Model1’ bridge in which abutment-superstructure interaction is ignored. This model is described in Chapter 5.

#### 3.2.4.3 Multiple Support Pattern

This load pattern allows the application of displacement, velocity, and acceleration records to the specified nodes of the model. Coherent and incoherent ground motions can be specified as input to the support nodes using this command. Recorded shake table displacements in previous bridge tests at UNR have shown that the achieved motions could be different from the target motions. This implies that the actual shake table motions at the bridge support points may not be similar, even when target motions are the same. Since the input motions for analytical bridge model are the similar target motions, both ‘Uniform Excitation’ and ‘Multiple Support’ patterns would give the same results. However to make the model capable of taking revised input motions based on actual shake table motions, it is decided to use Multiple Support Pattern for ‘Model2’ bridge analytical model. This pattern is the only choice for ‘Model3’ bridge model since in addition to the different shake table motions, the actuators loads have to be applied at bridge ends in the form of displacement record simultaneously. The bridge Model1, Model2, and Model3 are discussed in Section 5.6.4.1.
3.3 OpenSees Analysis Capabilities

The Analysis objects are the tools to analyze the system. The analysis moves the model from the system state at time ‘t’ to the system state at time ‘t + dt’. The analysis may be a simple linear analysis, a transient, or a variable transient analysis. In OpenSees, each analysis object is composed of several components which define the type of analysis and how the analysis is performed. The component classes are shown in Fig. 3-9 and discussed in this section.

3.3.1 Constraints Command

This command is used to construct the ‘ConstraintHandler’ object. This object determines how the constraint equations are enforced in the analysis. Constraint equations enforce a specified value for a degree-of-freedom (DOF), or a relationship between DOFs. The constraint commands are as follows:
- Plain Constraints which is used to enforce homogeneous single-point constraints, such as the case of homogeneous boundary conditions, where all boundary conditions are fixed.
- Constraint Transformation which transforms the stiffness matrix by condensing out the constrained DOF’s. This method reduces the size of the system for multi-point constraints. This is the recommended method for a transient analysis and is used for the bridge dynamic analysis.

3.3.2 Numberer Command

In a finite element model the node or element numbering pattern can strongly influence the bandwidth. A well planned numbering pattern can minimize the matrix bandwidth. There are two general methods for setting up the order of the equations in the assembly of the structure stiffness matrix (Cook et al. 1989). One uses the node number numerical sequence with the DOFs per node to order the equations. The other method uses the element number numerical sequence with the node numbers of each element of the DOF per node to order the equations. The way to minimize either of these is to plan the numbering pattern so nodes that connect through elements have their equations assembled close together in the structure stiffness matrix. This means that the node numbers used to define an element be close to each other numerically. This concept is followed for the node and element numbering of the bridge model. Additionally, in OpenSees the user can specify an internal numbering algorithm to optimize the solution. The Numberer command is used to construct the DOF_Numberer object. This object determines the mapping between equation numbers and the numbered DOFs. There are two types of Numberers:
- Plain -- The Plain numberer assigns DOFs to the nodes based on how the nodes are stored in the domain. Currently, the user has no control over how nodes are stored. This method is recommended for small problems or when sparse solvers are used, as they do their own internal DOF numbering.
- RCM -- The RCM numberer uses the reverse Cuthill-McKee algorithm (Cuthill and McKee 1969) to number the DOFs. This algorithm optimizes node numbering to reduce bandwidth. It may be used in conjunction with diagonal band storage, although it is most effective if used in conjunction with variable
bandwidth storage. This algorithm optimizes node numbering to reduce bandwidth using a numbering graph. This method will output a warning when the structure is disconnected.

The RCM numberer is used for the bridge analysis.

### 3.3.3 System Command

This command is used to construct the linear system of equations and linear solver objects to store and solve the system of equations in the analysis. The components of a solution strategy are interchangeable, allowing analysts to find sets suited to their particular problem. The linear system of equations may provide a fully populated matrix, if all its elements are non-zero, or sparse matrix, if only small portion of its elements is non-zero. The matrix could also be symmetric or unsymmetric. One of the following available solution methods in OpenSees can be used based on the properties of the linear system of equations. Each solver is tailored to a specific matrix topology.

- **Band SPD**: The ‘system BandSPD’ command is used to construct a symmetric positive definite (SPD) banded system of equations object which is factored and solved during the analysis using the Lapack band spd solver (Anderson et al. 1990).
- **Band General**: The ‘system BandGeneral’ command is used to construct an unsymmetric banded system of equations object which will be factored and solved during the analysis using the Lapack band general solver.
- **Sparse General**: This command is used to construct a general sparse system of equations object which will be factored and solved during the analysis using the SuperLU solver (Demmel et al. 1997).
- **Sparse Symmetric**: This command is used to construct a sparse symmetric positive definite system of equations object which will be factored and solved during the analysis using a sparse solver developed at Stanford University by Kincho Law (Law and Mackay 1993).
- **UmfPack General**: This command is used to construct a general sparse system of equations object which will be factored and solved during the analysis using the Umfpack solver (Davis 2003).
- **Profile SPD**: The ‘system ProfileSPD’ command is used to construct a symmetric positive definite profile system of equations objects which will be factored and solved during the analysis using a profile solver. The profile solver is based on variable bandwidth elimination algorithm.

The advantages of using variable bandwidth storage as opposed to diagonal band storage matrix decomposition are as follows (Jennings and MacKeown, 1992):

1. Greater flexibility is permissible in the choice of ordering scheme for the variables.
2. Optimum variable bandwidth ordering schemes will often provide more efficient decompositions than optimum diagonal band schemes.
3. Dummy variables are never required to obtain efficient decompositions.
Since RCM Numberer is used in the bridge analytical model to reduce the bandwidth, the ‘SparseGeneral’ is used as the system command. It seems that this combination provides an optimized solution.

3.3.4 Solution Algorithm Commands

These commands are used to construct a solution algorithm object, which determines the sequence of steps taken to solve the non-linear equation.

- Linear Algorithm
This command is used to construct a linear algorithm object, which requires only one iteration to solve the system of equations.

- Newton Algorithm
The ‘algorithm Newton’ command is used to construct a solution algorithm object which uses the Newton-Raphson method to advance to the next time step. The most frequently used iteration schemes for the solution of nonlinear finite element equations are some form of Newton-Raphson equations. It is well known that the method usually converges rapidly to a solution, if the initial estimate is sufficiently close to the solution, but otherwise may fail to converge. Despite this limitation, the Newton-Raphson algorithm remains the basis for most modern methods because of its fast ultimate rate of convergence to a solution. In this method the tangent stiffness is updated at each iteration.

- Newton with Line Search Algorithm
The ‘algorithm NewtonLineSearch’ command is used to construct a solution algorithm object which uses the Newton-Raphson method with line search to advance to the next time step. The required input is the limiting ratio between the residuals before and after the incremental update (between 0.5 and 0.8).

- Modified Newton Algorithm
This command is used to construct a solution algorithm object which uses the Modified Newton-Raphson method to advance to the next time step. The difference between this method and the Newton-Raphson method is that the tangent stiffness is updated only at selected steps, thus avoiding expensive calculations needed in multi degree-of-freedom systems. However, more iteration may be needed to reach a prescribed accuracy.

To minimize the required time for the analysis, Modified Newton Algorithm is used as the main solution algorithm in the bridge model. The Newton algorithm is specified as the solution algorithm if the convergence with the first system is not achieved after the specified maximum number of iteration with specified tolerance.

3.3.5 Convergence Test Commands

In any nonlinear analysis, due to the dependence of the stiffness matrix on the displacement, there is an imbalance between the external load and the internal forces of the model. The equilibrium of the nonlinear model is achieved when the unbalanced force or the incremental displacement is zero or sufficiently small. This equilibrium is obtained by an iterative procedure that consists of steps at which the unbalanced force is computed and tested. If the unbalanced force is not sufficiently small, an incremental displacement related to this force is computed, the displacement is adjusted and
imbalance is again evaluated. If the displacement exceeds a certain level, the stiffness matrix is also updated and the process is repeated.

In OpenSees the ‘Test’ command is used to construct a Convergence Test object. Certain solution algorithm objects require a convergence test object to determine if convergence has been achieved at the end of an iteration step. The convergence test is applied to the following equation:

$$K\Delta U = R$$  \hspace{1cm} (3.1)

Where $K$ is the stiffness, $\Delta U$ is the displacement increment, and $R$ is the unbalanced force. In all types of convergence tests in OpenSees, tolerance and maximum number of iterations have to be specified by the user. If the convergence is not achieved within the tolerance limit, more iteration will be performed. Based on the type of the test, the relevant checks can be performed as follows:

- **Norm Unbalance** – The ‘test NormUnbalance’ command is used to construct a test object which tests positive force convergence if the 2-norm of the $R$ vector (the unbalanced force) in the linear system of equation object is less than the specified tolerance.

$$\sqrt{R^*R} < tol$$  \hspace{1cm} (3.1a)

- **Norm Displacement Increment** -- This command is used to construct a test object which tests positive force convergence if the 2-norm of the $\Delta U$ vector (the displacement increment) in the linear system of equation object is less than the specified tolerance.

$$\sqrt{\Delta U^T \Delta U} < tol$$  \hspace{1cm} (3.1b)

- **Energy Increment** – The ‘test EnergyIncr’ command is used to construct a test object which tests positive force convergence if one half of the product of the $\Delta U$ and $R$ vectors (displacement increment and unbalanced force) in the linear system of equation object is less than the specified tolerance.

$$\frac{1}{2}(\Delta U^T R) < tol$$  \hspace{1cm} (3.1c)

The ‘test EnergyIncr’ with a tolerance of 1.0e-7 and 25 maximum number of iterations is used for the bridge analysis.

### 3.3.6 Analysis Command

This command is used to construct the analysis object, which defines what type of analysis is to be performed. The available analysis types in OpenSees are discussed in this section.
3.3.6.1 Static Analysis
The applied static load creates a static response which is proportional to the structure stiffness and applied loads. Even for time varying loads, as long as the frequency of the applied loading is less than about one-third of the lowest natural frequency of the structure, the response is given by the static solution in the proportion to the instantaneous load (Knight 1993). The ‘Static Analysis’ command in OpenSees solves the KU=R problem, without the mass or damping matrices.

3.3.6.2 Dynamic Analysis
When the frequency of the applied load exceeds approximately one-third of the lowest frequency of the structure, a different solution technique has to be used to include the inertial effects due to material mass and damping effects (Knight 1993). There are several different procedures involved in doing dynamic analysis that depend on the type of solution the user is seeking.

In the most general case, the problem to solve is in the form of Equation 3.2. The solution is the time dependent response of all structural nodes by inclusion of the equivalent inertial forces and damping forces in the equation. The inertial forces are the product of mass and acceleration, and the damping forces are the product of damping coefficient and velocity. The general equation is:

\[
([M] \ddot{D} + [C] \dot{D} + [K] D) = F
\]  

(3.2)

Where, \([M]\) represents the structure mass matrix, \([\ddot{D}]\) is the nodal acceleration vector, \([C]\) is the structure damping matrix, \([\dot{D}]\) is the nodal velocity vector, \([K]\) is the structure stiffness matrix, \([D]\) is the nodal displacement vector, and \([F]\) is the applied time varying nodal load vector.

Different types of dynamic analysis are discussed in the following sections.

- Eigenvalue Analysis
The eigenvalue problem derives from Equation 3.2 without the damping and applied forces. The structure vibration starts by an initial condition of displacement, velocity or acceleration.

The number of independent eigenvalues is the same as the number of degrees of freedom in the finite element model. Each of the eigenvalues has an independent eigenvector or mode shape. However, only a few of the lowest eigenvalues are needed in the analysis, because the contribution of higher eigenvalues to the response is negligible.

In OpenSees bridge analysis the ‘Eigen’ command is used to perform a generalized eigenvalue problem to determine the eigenvalues and the corresponding eigenvectors with a specified damping of 2%.

- Transient Response Analysis
If the input loading function is an arbitrary time dependent function, then a transient response analysis has to be performed. There are two general approaches to solve the transient response problem. One of these is direct integration of the system equations
after approximation by a finite difference or finite element method in the time dimension for the velocity and acceleration components. The direct integration approach will involve the total set of system equations and requires many time steps with a complete solution in each step. This can become a large computing task for significant size problems. The Direct Integration Method is discussed in Section 3.3.7.2.

The second approach is called Modal Superposition. The basis of this approach is an assumption that superposition of the mode shapes corresponding to the lower natural frequencies adequately represents the dynamic response of the structure. The complete response is found by the summation of a fraction of the low frequency mode shapes. This requires a transformation of the equations from nodal displacement coordinates to a set of modal coordinates. The transformation changes the set of system equations that consists one equation for each DOF to a set of modal equations involving the selected number of mode shapes. This results in much fewer equations.

In OpenSees the transient response analysis is based on direct integration analysis which is performed by using ‘Transient Analysis’ command. This analysis object is constructed with the component objects previously created by the analyst. If none has been created, default objects for solution algorithm, transient integrator, constraint handler, DOF numberer, and linear system of equations are constructed and used. The time steps in the analysis and in the output are constant.

Another option to perform transient analysis is by using ‘Variable Transient Analysis’ command. This command performs the same analysis type as the transient analysis object. The time step, however, is variable. The time step in the output is also variable. This method is used when there are convergence problems with the transient analysis object at a peak or when the time step is too small. This type of transient analysis is used in the bridge dynamic analysis.

3.3.7 Integrator Command

This command is used to construct the Integrator object. The Integrator object is used for the following:

- determine the predictive step for time $t+\Delta t$
- specify the tangent stiffness matrix and the residual force vector at any iteration
- determine the corrective step based on the displacement increment $dU$

The type of integrator used in the analysis depends on whether it is a static analysis or transient analysis.

3.3.7.1 Integrator for Static Analysis

In OpenSees there are four types of static integrators to perform static analysis. The integrators are used based on the type of control on each analysis step. The load and displacement control integrator commands are discussed here because they were used in the present study.

- Load Control

The command ‘Integrator LoadControl’ is used to construct a static integrator object of type Load Control. The first load increment has to be given with this command. The optional additional input data are: factor relating load increment at subsequent time steps
Displacement Control

The command ‘Integrator DisplacementControl’ is used to construct a static integrator object of the type Displacement Control. The required input data are: node number and the DOF the response of which controls the solution and the first displacement increment. The optional input data are: factor relating displacement increment at subsequent time steps (default is 1.0) and the minimum and maximum displacement increments (default is the first displacement increment for both).

### 3.3.7.2 Integrator for Transient Analysis

As discussed earlier, the transient analysis in OpenSees utilizes direct integration. In direct integration all the individual equations (Equation 3.2) are integrated using a numerical step-by-step procedure. The term ‘direct’ means that prior to the numerical integration, no transformation of the equations into a different form is carried out. In essence, direct numerical integration is based on two ideas. First, instead if trying to satisfy Eq. 3.2 at any time \( t \), it is aimed to satisfy it only at discrete time interval \( \Delta t \). This means that, basically, equilibrium, which includes the effect of inertia and damping forces, is sought at discrete time points within the interval of solution. Therefore, it appears that all solution techniques employed in static analysis can also be used in direct integration. The second idea is that the variation form of displacement, velocities, and accelerations within each time interval \( \Delta t \) is pre-assumed.

A large number of direct integration methods are available (Bathe 1982). Two direct integration schemes for transient analysis are available in OpenSees: the Newmark Method (Newmark 1959)) and Hilbert-Hughes-Taylor Method (Hilbert et al. 1977). The choice is largely determined by the accuracy characteristics of the method, i.e. the accuracy that can be obtained in the integration for a given time step \( \Delta t \).

#### Newmark Method

This is the time integration method preferred for large scale linear analysis and is based on the classical Newmark method. The Newmark integration scheme uses the following equations:

\[
\begin{align*}
\dot{D}_{t+\Delta t} &= \dot{D}_t + [(1 - \gamma)\ddot{D}_t + \gamma_{t+\Delta t}\dddot{D}]\Delta t \\
D_{t+\Delta t} &= D_t + \dot{D}_t\Delta t + \left[\frac{1}{2} - \beta\right]\ddot{D}_t + \beta\dddot{D}_{t+\Delta t}\Delta t^2
\end{align*}
\]  

(3.3a)  

(3.3b)

Where \( D, \dot{D}, \) and \( \ddot{D} \) are the displacement, velocity, and acceleration vectors of the finite element assemblage. \( \beta \) and \( \gamma \) are parameters that control accuracy and stability of the results. The integration scheme is unconditionally stable provided that \( \gamma \geq 0.5 \) and \( \beta \geq 0.25(\gamma + 0.5)^2 \). Newmark originally proposed as an unconditionally stable scheme using the constant-average-acceleration method, in which case \( \gamma = 0.5 \) and \( \beta = 0.25 \), which relates to a linear variation of acceleration over the time interval \( t \) to \( \Delta t \). It has been shown that the Newmark method with these proposed parameter values, has the
most desirable accuracy characteristics (Bathe 1982). In OpenSees, user can specify these parameters. The Newmark proposed values for $\gamma$ and $\beta$ have been used as input for ‘integrator Newmark’ command in OpenSees bridge model analysis.

- Hilbert-Hughes-Taylor Method
  This command is used to construct a TransientIntegrator object of type HHT. The HHT method is a generalized of the Newmark method and reduces to Newmark method for $\alpha=0$ which represents constant acceleration. The HHT method is useful in structural dynamic simulation incorporating many degrees of freedom and in which it is desirable to numerically dampen-out the response at high frequencies (Hilbert et al. 1977).

### 3.3.8 Rayleigh command

In Eq. 3.2, it is assumed that damping is of viscous type with damping forces that are proportional to nodal velocities. It is known that damping is also a function of displacement amplitude, accelerations, coulomb friction, and other energy loss mechanisms. These effects require damping terms in the equation that involve the displacement as well as the acceleration vectors. However, these terms are relatively small and a velocity dependent damping force is an acceptable approximation.

OpenSees utilizes proportional damping according to Eq. 3.4.

$$\quad [C] = \alpha[M] + \beta[K] \quad \quad (3.4)$$

Where $\alpha$ and $\beta$ are constants to be determined based on damping ratios for two modes of vibration. The ‘Rayleigh’ command in OpenSees is used to assign damping to all elements and nodes. The damping matrix $[C]$ is specified as combination of stiffness and mass-proportional damping matrices:

$$\quad C = \alpha M \cdot M + \beta K \cdot K_{\text{current}} + \beta K_{\text{init}} \cdot K_{\text{init}} + \beta K_{\text{comm}} \cdot K_{\text{lastcommit}} \quad \quad (3.5)$$

Where:

- $\alpha M$ Mass proportion Raleigh damping parameter (taken 0 in the bridge analysis)
- $M$ Mass matrix used to calculate Raleigh damping
- $\beta K$ Stiffness proportional damping parameter (taken 0 in the bridge analysis)
- $K_{\text{current}}$ Stiffness matrix at current state determination used to calculate Raleigh damping
- $\beta K_{\text{init}}$ Initial stiffness proportional damping parameter (taken 0 in the bridge analysis)
- $K_{\text{init}}$ Stiffness matrix at initial state determination used to calculate Rayleigh damping
- $\beta K_{\text{comm}}$ Committed stiffness proportional damping (taken as $\frac{2 \times 0.02}{\sqrt{[\text{eigen}1]}}$)
- $K_{\text{lastcommit}}$ Stiffness matrix at last committed state determination used to calculate Rayleigh damping
In the above equation there are three different stiffness values that can be used: the current value, the initial value, or the stiffness at the last committed state. The stiffness at the last committed state was used for the bridge analysis in this study.

3.3.9 Analyze Command

The analysis is executed using the ‘analyze’ command. This command moves the analysis forward by the specified number of steps. The required input is the number of load steps. The time step increment is required if transient analysis or variable transient analysis is performed. The minimum time step, maximum time step, and the number of iteration performed at each step are required only if variable transient analysis is performed.

3.3.10 Recorder Commands

The recorder commands are used to construct a Recorder object, which is used to monitor items of interest. Some of the most common recorder types in OpenSees are discussed in this section.

- Node Recorder
  The ‘recorder Node’ command records the displacement, velocity, acceleration, and incremental displacement at the nodes. The required inputs are the DOF being recorded and the response type. The available response types are: displacement, velocity, acceleration, incremental displacement, and eigenvectors. The optional input parameters are: the output file name (default is screen output), pseudo time, and node numbers where response is being recorded (default is all).

- Envelope Node Recorder
  The ‘recorder EnvelopeNode’ records the envelope of displacement, velocity, acceleration, and incremental displacement at the nodes. The envelope consists of the minimum, maximum, and the maximum absolute value of the specified response type. The tag for a start and an end node is an optional input if the response is requested for a range of nodes between the two specified nodes.

- Drift Recorder
  The ‘recorder Drift’ records the displacement drift between two nodes. The drift is taken as the ratio between the prescribed relative displacement component and the specified distance between the nodes. The input parameters include the node numbers, their nodal DOF to monitor the drift, and the perpendicular global direction from which length is determined.

- Element Recorder
  The ‘recorder Element’ command is used to monitor forces at two ends of the element in global or local coordinates. The element recorder can also be used to monitor the element section response at the integration points along the element. The section response includes force, deformation, stiffness, and the stress-strain relationship. For fiber sections, the stress strain of a fiber at the given location can be monitored if the section number is followed by the ‘fiber,’ its coordinate, and ‘stressStrain’ in the recorder command.
- Envelope Element Recorder

The format of ‘recorder EnvelopeElement’ command is similar to the element recorder, but it gives only the minimum, maximum, and the maximum absolute value of specified response type.
24

CHAPTER 4
2-Span Bridge Description and OpenSees Model

4.1 Introduction
This chapter discusses the analytical model for a 2-span bridge which was tested on UNR shake tables. The 2-span bridge test was part of a collaborative NEES demonstration study to investigate soil-foundation-structure-interaction (SFSI) of bridge systems. The major focus of the research was on the multiple shake table testing of a reinforced concrete bridge system including the analytical modeling of bridges and investigation of bridge system response (Johnson et al. 2006).

There are similarities in the dimensions of the 2-span and the 4-span bridge models. The column section and the deck sizes are the same in both models. The span lengths of the 2-span bridge are the same as the length of the interior spans in the 4-span bridge. However, the 2-span bridge has monolithic cap-beam and deck connections, while 4-span bridge is a drop cap-beam type. There are deck abutment interactions at the 4-span bridge deck ends, but in the 2-span bridge model there are no abutments. Despite differences between the two models, the column elements which are the key components in the nonlinear behavior in both models have the same section properties and close overall heights. The same element types and material models were used to develop the OpenSees analytical models for both bridges. Due to the availability of the measured data from 2-span bridge shake table test, the analytical results can readily be compared with the measured data. The accuracy of the 2-span bridge analytical model in predicting the measured response would give an indication on how accurate the analytical model would be for the 4-span bridge test, since similar node configuration, element types, and material models have been used in both analytical models.

The achieved 2-span bridge shake table motions from high amplitude tests (tests12 through 19) were input to the model in sequence as an attempt to compare the bridge calculated displacements with the measured displacements obtained from shake table tests. This chapter provides a brief description of the test model and presents the analytical and experimental results.

4.2 Prototype and Model Bridge Description
The prototype for the experimental studies (Fig. 4-1) was a two-span frame of a cast-in-place post-tensioned reinforced concrete box girder bridge. The span lengths were 120ft (37m), and the substructure was composed of 1.2 m (4 ft) diameter 2-column piers on extended pile foundations. Columns have a 1.56 percent longitudinal steel ratio and a spiral reinforcement ratio of 0.9 percent (Johnson et al. 2006).

The bridge specimen was a quarter scale model of the bridge. The total height of the specimen from the bottom of the footing to the top of the superstructure was 10.75 ft (3.28 m) and the total length was 67.3 ft (20.5 m). Span lengths were 30 ft (9.14 m) and the columns of the three bents had clear heights of 6 ft, 8 ft, and 5 ft (1.83, 2.44, and 1.52 m) with the tallest bent in the middle (Figs. 4.2 and 4.3). The superstructure was composed of a solid slab that was post-tensioned in both the longitudinal and transverse directions of the bridge. It was designed to remain un-cracked, and its stiffness matched the stiffness of the prototype superstructure about both orthogonal bending axes.
4.3 OpenSees Analytical Modeling of the Bridge

The OpenSees model was created with the intent to use standard analytical methods to model displacement and force response for different tests, particularly for the 4-span bridge test. The correlation between the analytical model and the measured displacements would give an indication of the reliability of the OpenSees predicted results for the 4-span bridge model to be tested in the future.

During the analytical studies it was noted that variations of the bond slip deformation properties at the column end sections could improve the correlation between the calculated and measured response. However, this was not done because identifying the best fit input parameters was not the intent of the study. Furthermore, the column failure was not modeled. This is in part because the material models in the fiber sections were defined to have gradual strength loss at failure to aid in convergence and to prevent computational instability. Another reason for exclusion of column failure points was that fiber models describing the column sections do not allow for direct modeling of spiral rupture in the plastic hinge zone.

4.3.1 Computer Model of Bridge Specimen

The OpenSees model was a 3-dimensional assemblage of linear and nonlinear elements connected at the nodes. Nonlinear column elements that were defined between nodes are the same as those discussed in Sections 3.2.1.2 and 5.3.3. All nonlinear deformations in the computer models were assumed to take place in the columns. This assumption was based on the fact the superstructure and cap beams were strong and stiff in comparison to the columns. The modulus of elasticity in the linear parts of the columns was based on the measured unconfined concrete compressive strength. Gross section properties were used for the superstructure. The cap beams were also designed to remain elastic because they were rigidly connected to the post-tensioned solid superstructure. The gross section properties were used for the cap beams.

Fiber elements were used to model the nonlinear behavior of the columns. The defined ‘aggregator’ option in OpenSees was used to add cracked section shear and torsional properties to the column fiber element sections.

4.3.2 Nodal Configuration and Masses

The OpenSees model was composed of linear beam column elements combined with nonlinear column fiber section elements that connected a three-dimensional assemblage of nodes. The nodal configuration is shown in Fig. 4-4.

The orientation of the model is such that the global x, y, and z directions are in the longitudinal, vertical, and transverse, directions of the structure, respectively. Nodes and elements were located at centerlines of the bridge components. The columns were assumed to be fully fixed at the base.

The superstructure mass was lumped at the nodes defined at the tenth points of the span between columns and at the nodes defined along the cantilevers based on their tributary area. Superimposed masses were lumped at the nodes defined at the center of each concrete block or lead pallet. The center of mass node for each imposed load was connected with a rigid beam column element vertically to the centerline of the superstructure (Fig. 4.5). Rotational inertial masses were not included.
4.3.3 Description of Model

As shown in Fig. 4-4, the bridge was modeled using a three dimensional assemblage of nodes connected by two-dimensional beam-column elements. Tributary masses were lumped at the nodes. All elements, except columns, were assumed to behave linearly. The damping was specified using mass and stiffness proportional coefficients that were calculated for two percent damping. P-delta effects were included in the analysis.

4.3.3.1 Column Element Descriptions

The columns were modeled with nonlinear elements defined using fiber elements. The constitutive relationships of the fibers were specified for confined concrete, unconfined concrete, and steel using measured material properties as discussed in the next section. Constant cracked section properties for both shear and torsion were specified. The cracked shear stiffness was specified based on the truss analogy (Park and Paulay 1975). At both ends of the columns zero length elements were defined. The bond slip properties in terms of moment-rotation curves were incorporated in zero length elements. The bond-slip model proposed by Wehbe et al. (1997) was used in the analysis. To achieve the moment-rotation curve for bond slip modeling, section moment-curvature analysis was performed. The resulting curve was then modified based on the steel strain at the extreme section fiber at three points: cracking, yield, and the ultimate. The modified tri-linear moment-rotation curve was specified as a property of a hysteretic material in the positive and negative directions. Details of the calculations are similar to those of the 4-span bridge column bond slip properties as presented in Appendix A.

4.3.3.2 Material Properties

Three stress-strain curves were used to define the fibers in the inelastic fiber sections for the longitudinal column reinforcement, unconfined concrete, and confined concrete. These constitutive relationships were specified as multi-linear curves to match the measured curves. There are three options in OpenSees to specify concrete material properties. The first model (Concrete01) was used in the analysis. This is a uniaxial Kent-Scott-Park concrete material object with degraded linear unloading/reloading stiffness according to the work of Karsan-Jirsa and has no tensile strength as discussed in Section 3.2.2.3. Both the confined and unconfined concrete was defined using this model. As mentioned in Section 3.2.2.3, Concret01 model assumes a constant strength after concrete crushing. Linear strength reduction for the unconfined concrete was defined after the peak strain of 0.002 up to 0.006 at zero stress. The strain of 0.005 was defined at the peak stress of the confined concrete. After the crushing strain, strength was assumed to be linearly reduced to a stress of 70 percent of the peak stress at a strain of 0.017, after which the stress remained constant. Mander’s model (Mander et al.1988) was used to define the properties of the confined concrete.

The bi-linear steel material (Steel01) was used to model the reinforcements. The initial modulus of elasticity of 30,000 ksi (2x10^5 MPa) was assumed up to the yielding stress of 66.5 ksi (458 MPa). The strain hardening slope was defined as 0.2% of the initial modulus of elasticity.
4.3.3.3 Zero Length Elements
Apart from the column nonlinear elements, the rest of the bridge model elements were assumed to be linear elastic. The zero length elements at the column ends were used for the bond slip modeling in terms of moment-rotation relationship. Uncracked properties in shear and torsion were assumed for the zero length elements.

4.3.3.4 Shake Table Motions
Although the target motions for the shake tables were identical, the achieved shake table motions in the test were incoherent, because of the interaction between the bridge and shake tables. In OpenSees the multiple support excitations can be used to input the motions in terms of acceleration, velocity, or displacement. In this study the achieved shake table displacement records were used as input motions to the bridge footings in the OpenSees model by utilizing the multiple support ground motion pattern.

4.3.4 Model Efficiency
To perform inelastic modeling, efficiency of the model was an important consideration. This included the configuration of nodes, type of elements, and parameters to perform the analysis. Variable transient type of analysis with a specified time step of 0.001 second was used. The 0.0005 second and 0.01 second were specified as the minimum and maximum time steps respectively with a relative convergence tolerance for iteration of $10^{-8}$. The integration method used for the analysis was the Newmark’s $\beta$ method using integration parameters $\gamma=0.5$ and $\beta=0.25$, which correspond to the average acceleration method.

4.3.4.1 Fiber Configuration
Fiber configurations for the column sections were chosen based on the parametric study discussed in Section 5.3.1. The goal was to specify the minimal number of slices and layers to obtain reasonably accurate results.

The parametric study showed that there was no considerable change in push over analysis results if more than 8 slices and 7 layers were used for the column fiber sections. Therefore this optimum configuration was used in the analysis as shown in Fig. 5.8.

4.3.5 Damping
The recommended damping ratio for cracked reinforced concrete structures is 3 to 5%. This range is recommended for working stress level where the stress is no more than approximately one-half of the steel yield stress. For the stress level at or just below the yield point the recommended damping ratio is in the range of 7 to 10% (Bathe 1982). These recommended damping ratios can be used for the linearly elastic analysis with classical damping. The damping matrix must be defined completely if classical modal analysis is not applicable, such as analysis of nonlinear systems.

Damping in OpenSees model was specified through mass and stiffness proportional damping. The Rayleigh damping method was used to calculate the damping coefficients of the classical damping matrix based on the calculated periods of the first and third transverse modes. Since the bridge model is analyzed for input base displacement histories, the mass proportion damping can be greatly overestimated (Wilson 2002). Unlike the acceleration input motions in which the nodal displacements are relative to the
The results obtained from OpenSees analysis were compared to the response of the shake table specimen for the achieved high amplitude tests 12 through 19 motions to evaluate the accuracy of the OpenSees model. The direct comparisons of the computer model and the specimen response were made on the displacement histories of the three bents in the transverse direction. Figures 4-6 through 4-13 are plots of the calculated displacement histories compared with the measured results. The peak negative and positive displacements for tests 12 through 19 are listed in Table 4-1. The table also lists the percent difference between the analytical and experimental peak displacements for each test and the average percent difference for each bent.

As shown in the response plots, the OpenSees estimated displacement histories that were reasonably accurate for tests 12 through the part of test 19 until bent 3 failed. The model accurately calculated yielding of the three bents. The response of the bridge bents to the lowest amplitude motion (test 12) was overestimated by OpenSees model. It also overestimated the response of bents 2 and 3 for tests 12-14, while for higher amplitude motions the displacements were mostly underestimated (Table 4.1). The response of the bridge to the higher amplitude motions (tests 15-18) is of more importance since it corresponds to more than 80% of the maximum bent displacement at bridge failure (test 19). As shown in Table 4.1, the measured displacements for tests 15 to 18 are underestimated by OpenSees model by 20%, 28%, and 13% for bents 1, 2, and 3 respectively with an average of 20%. The significant deviation between the measured and calculated displacements in test 19 (Fig. 4-13) is because of the bridge failure during this test.

4.5 Concluding Remarks
The following conclusions can be drawn from the comparison of the calculated and measured displacement histories of the three bents:

- The analytical model could predict the measured response with reasonable accuracy. The displacements of the bents were overestimated by analytical model for the low amplitude tests by approximately 30% on average. Whereas for high amplitude tests the response of the bents was underestimated by about 20% on average.

- Similar bond slip modal can be used in the 4-span bridge model. The material properties, however, have to be revised to derive moment-rotation curve based on the measured material properties used in the 4-span bridge model.

- The multiple support ground motion pattern in OpenSees is a suitable option to be used for shake table displacement record input to the 4-span bridge. This pattern can also be used for the actuator displacement motion input to perform computer analysis of the bridge after the shake table tests.
Although the calculated displacements showed reasonably consistent correlation with the measured data for the entire range of test amplitudes, in general the OpenSees model underestimated the bridge response for high amplitude motions by 20%. The same outcome can be expected from the analytical model for the 4-span bridge.
CHAPTER 5
Bridge Modeling

5.1 Introduction
This chapter discusses the analytical models that are developed for column sectional analysis, pushover, and dynamic analysis of the 4-span bridge. The analytical modeling of the bridge is aimed at predicting bridge performance during shake table testing and to provide a platform to help select the shake table testing protocol. The configuration of the bridge, type of beam and column elements, and material properties used in the model are presented in this chapter.

5.2 Description of Model
The bridge is modeled using a three dimensional assemblage of nodes connected by two-dimensional beam-column elements (Fig. 5-1). The orientation of the model is such that the global x, y, and z directions are in the longitudinal, vertical, and transverse directions of the structure, respectively. Nodes and elements are located at the centerlines of the bridge components. The superstructure node arrangement is defined such that all imposed weights are located at the defined nodes. Column bases are assumed to be fixed because they will be rigidly attached to the shake tables. Tributary masses are lumped at the nodes. Rotational inertial masses are not included. All elements, except columns, are assumed to behave linearly. Damping is specified using mass and stiffness proportional coefficients that are calculated for two percent damping at the first and third modes. The P-delta effects are included in the analysis.

5.2.1 Material Models
The 28 day concrete compressive strength of 5 ksi (34.5 MPa) is specified for the bridge columns and superstructure. The specified concrete strength is used as the unconfined concrete strength in the OpenSees bridge model. Opensees ‘Concrete01’ uniaxial material model is used for confined and unconfined concrete. This uniaxial material model is based on Kent-Scott-Park (Kent and Park 1971) model with degrading linear unloading/reloading stiffness according to the work of Karsan-Jirsa (Karsan and Jirsa 1969). This model assumes no tensile strength for concrete. For unconfined concrete, 0.002 strain at peak stress and 0.006 strain at ultimate are used. The confined concrete properties are calculated based on Mander’s model (Mander et al. 1988), with 6.56 ksi (45.2 MPa) peak stress at 0.005 strain and 5.1 ksi (35.1 MPa) stress at the ultimate strain of 0.0169.

The reinforcing bars are modeled as “steel02” with bi-linear stress strain relationship to account for strain hardening effect. The bi-linear curve has a slope of 29000 ksi (199810 MPa) at initial stage up to yield stress of 68 ksi (469 MPa), and the slope of 212 ksi (1461 MPa) at strain hardening stage. The specified stress-strain relationship for reinforcing bars is symmetric for tension and compression.

5.2.2 Stiffness Assumptions for Linear Elastic Beam Members
The concrete stiffness for linear elements is specified based on the unconfined concrete strength with the assumed 28 day cylinder strength of 5 ksi (34.5 MPa). The post-tensioned superstructure, which is designed to remain elastic and uncracked, is
modeled with gross section properties. The cap beams are modeled as elastic elements using gross section properties.

5.3 Analytical Modeling
Since inelastic modeling can be computationally demanding, model efficiency is an important consideration. The analysis time is affected by the number of fibers and fiber segments in the column plastic hinges defined at the top and bottom regions of all columns. A parametric study is conducted to determine the type of bridge column elements, the minimal number of the fibers, and the largest time step that can be used to obtain stable and convergent results.

5.3.1 Fiber Configuration
The diameter of all the columns is 12 inch (304.8 mm). The longitudinal reinforcement consists of 16-#3 reinforcing bars with concrete cover of 0.5 inch (12.7 mm) on the spirals. Fiber configuration for column section is as shown in Fig. 5-2. To determine the optimum number of fibers moment curvature analysis of the column section is performed using OpenSees. Moment curvature analysis results are discussed in Section 5.4. The goal is to specify a minimum number of slices and layers to obtain convergent results. A minimum of three layers for the column section is required to represent concrete core, reinforcement, and concrete cover. Since considering only one layer for the column core would not give accurate results, three layers are considered for the column core in the parametric study as a minimum. A combination of three to nine with equal thickness core layers with eight and sixteen slices is studied. Two layers are considered for the concrete cover. Comparison is made using moment curvature relationships for all combinations. Figure 5-3 shows the moment curvature plots for column section with 8 slices and different number of core layers. It can be seen that the stiffness and strength of the section increases by increasing the number of layers. The higher the number of layers, the more accurate results are expected. The improvement, however, is not significant when the number of layers exceeds seven. Therefore it is decided to use seven layers for the column core. Figure 5-4 shows the moment curvature plots for the 16- slice column with the same core layer configurations as those for those of the 8-slice column section (Figure 5-3). No change is observed when the number of slices is increased from 8 to 16 (Fig. 5-5). Therefore, the column section with 8 slices, 7 layers of core, and 2 layers cover is chosen for the analysis (Fig.5-2).

5.3.2 Element Configuration
Once the optimum configuration of fibers is determined, another parametric study is conducted using OpenSees pushover analysis to determine the optimum element type and configuration. There are two main options for modeling columns with inelastic behavior: lumped plasticity and distributed plasticity. The effect of using different models for the plastic hinge is discussed in the following sections.

5.3.2.1 Lumped Plasticity
In the lumped plasticity model, plastic deformations are assumed to be concentrated at the specified locations in the plastic hinge zones. In OpenSees ‘Beam with Hinges’ element can be used for this purpose. This element is based on flexibility formulation,
and assumes plasticity to be spread over specified hinge lengths at the element ends. The input for this type of element consists of element end node numbers, the hinge lengths at both ends, and the properties of the hinges and the element. Fiber section can be used in the column plastic hinge area to define the hinge properties.

5.3.2.2 Distributed Plasticity

In the distributed plasticity model, the plasticity is assumed to spread along the entire element length. There are basically two types of nonlinear beam column elements in OpenSees. The first type is a force-based element called ‘Nonlinear Beam Column,’ which is based on force formulation and considers the spread of plasticity along the element. The integration along the element is based on Gauss-Lobatto quadrature rule, with two integration points at the element ends (Davis and Rabinowitz 1984). The second type is called ‘dispBeamColumn’ which is a displacement based element with distributed plasticity with linear curvature distribution. The integration in this element types is based on the Gauss-Legendre quadrature rule.

5.3.2.3 Selection of Column Element Type

Bent 1 is chosen as a test case to investigate the effect of column nonlinear element type on the force-displacement relationship. Pushover analysis is performed on Bent 1 with distributed plasticity and lumped plasticity column models for comparison. Nonlinear beam column element with distributed plasticity is used in the first model for the entire column. In the second model lumped plasticity is used over the column plastic hinge length at column ends and the rest of the column is modeled with elastic beam column elements with cracked section properties. Figure 5-6 shows nearly the same initial strength for both models as expected and a slightly higher strength for the model with lumped plasticity beyond yielding. The computational time required for Bent 1 pushover analysis with lumped plasticity model is approximately 50% of the time required for the distributed plasticity model. Since the lumped plasticity model requires less computational time and leads to nearly the same result, the column lumped plasticity model was first selected for both pushover analysis of the bents and dynamic analysis of the entire bridge. However, the dynamic analysis of the bridge with columns modeled with lumped plasticity was stopped during the Event 4 input motion with ‘no convergence’ message. Therefore distributed plasticity model is used for the bridge columns to perform dynamic analysis.

5.3.3 Analysis Increment and Integration Method

In pushover analysis of the bridge bents, the vertical load is applied in 10 steps with a static integrator object called ‘Integrator Load Control.’ The specified lateral load is applied in 0.001 inch (0.025 mm) increments using “Displacement Control” integrator object. The computational time for Bent1 pushover analysis using a 2.8 GHz processor is approximately 5 minutes.

For dynamic analysis of the bridge model a variable time step is specified. The initial time step of 0.001 second is chosen based on the required minimum time step for full scale motion to converge. The minimum and maximum specified time steps are 0.0001 and 0.002 respectively. If the analysis fails to converge with the initial time step, the stiffness would change to its initial value with smaller time step. The average run time for
a 20 second earthquake analysis, which is the duration of the full scale motion, using a 2.8 GHz processor is nearly 2 hours.

The integration method used for the analysis is Newmark’s β method (Newmark 1959) using γ=0.5 and β=0.25 as discussed in chapter 3. The system of equation storage and solver is specified as “system ProfileSPD”. This command is used to construct a symmetric positive definite profile system of equation objects. This profile solver is based on variable bandwidth elimination algorithm which provides an efficient solution for large structures as discussed in Chapter 3.

5.4 Moment-Curvature Analysis

The moment-curvature analysis of reinforced concrete cross sections is based on equilibrium of forces and compatibility of strains. Thus the moment-curvature relationship can be obtained by finding a strain profile for the cross section that would result in equilibrium between the applied axial load and the internal forces in the concrete and steel. The strain profile resulting in equilibrium of forces is used to calculate the curvature. The corresponding moment is then found by summing the moments of the forces developed in the cross section about the cross section plastic centroidal axis.

In order to compare the moment curvature results obtained from a beam section and a fiber section analysis, two computer programs are used: RCMC version2 (Wehbe and Saiidi 2003), and Opensees (OpenSees 2002). The bridge column section is analyzed. The column diameter is 12 in (304.8 mm) with the details described in Section 2.3.1. The same material properties are used as input for both models. The fiber section shown in Fig. 5-2 is used for Opensees analysis. The results are compared in Fig. 5-7. It can be seen that the column yield strength in RCMC is higher than in Opensees model, but after spalling of the concrete cover the strength in RCMC model drops and reaches the same strength as that in Opensees at about 0.0037 in-1 (0.00014 mm-1) curvature. The difference between the two models is due to the difference in type of sections, and concrete models in the two programs. A lower strength for fiber section as compared to a beam section is expected since, as mentioned in Section 5-3-1, by increasing the number of fiber elements, the column strength is increased. The Opensees model shows higher strength after concrete cover spalling. This is due to the fact that RCMC removes the unconfined concrete once the cover concrete reaches the ultimate strain, while Concrete01 model in Opensees maintains some level of stress in concrete even after concrete ultimate strain is reached.

5.5 Push-Over Analysis

The in-plane lateral load response of each bridge bent is determined using push-over analysis. Since the bridge bents have different heights and axial forces, a separate push-over analysis is performed for each bent to determine the bent force-displacement relationship and to identify the most critical bridge bent. The force-displacement curves are shown in Fig. 5-8. The calculated vertical loads of 81.1 kips (361 kN) for bents 1 and 3 and 78.8 kips (351 kN) for bent 2 are used in the analysis based on the bridge superstructure weight, the superimposed dead load, and reactions resulting from the superstructure prestress force. The stiffness of bents 1, 2, and 3 in the elastic portion of the force-displacement curves are 111 kips/in (19.4 kN/mm), 42 kips/in (7.4 kN/mm), and 70 kips/in (12.3 kN/mm) respectively (Fig. 5-7). The stiffness of the bents is
approximately inversely proportional to the cube of their heights, as expected for elastic columns with both ends fixed against rotation.

In the non-linear portion of the force-displacement curves, bent 3 with the same axial load as bent 1 has approximately 18% less strength than Bent 1. Considering that Bent 1 is 20% shorter than Bent 3, the difference between their peak lateral strength is justifiable because when the plastic hinges are formed at the bottom and top of the columns, the frame lateral capacities are inversely proportional to their height. Although Bent 2 has slightly lower axial load, its capacity is the lowest among the other bents because it is the tallest. The same justification applies to the difference between the capacity of the Bent 2 and the capacity of the other bents with respect to their heights. Bent 2 is 40% taller than Bent 1 resulting in approximately 70% decrease in the lateral strength of Bent 2 as compared to Bent 1 (Fig. 5-8).

The strength of the bents is reduced after their peak strength point. This is due to the specified concrete properties and the P-delta effects. However, the slope of the strength reduction is small because of the specified bi-linear stress strain curve for steel with continuous strain hardening effect beyond the elastic portion in OpenSees 'steel02' model. In addition concrete strength capacity in ‘Concrete01’ model beyond the ultimate stress point remains constant with no failure point. These material model properties used in the OpenSees push-over analysis are consistent with the trend of force-displacement curves in Fig. 5-8 with no failure points.

5.6 Dynamic Analysis of the Bridge

5.6.1 Input Motions

The input motions used for the dynamic analysis are taken from a previous study (Johnson et al. 2006) at University of Nevada, Reno (UNR) for shake table testing of a two-span bridge, as mentioned in Section 2.5. The full scale transverse and longitudinal acceleration inputs are as shown in Fig. 2-8 in which the time axis for both motions is compressed by 0.5 to account for the ¼ scaled bridge model. The transverse record is scaled by its peak acceleration and applied in sequential runs of increasing amplitude with peak ground acceleration (PGA) from 0.075 g to 1.32g. The same scale factors are used for the longitudinal motion. The corresponding PGAs for longitudinal sequential motions are from 0.10g to 1.77g as shown in Table 2-2.

5.6.1.1 Loading Pattern

Two input loading pattern options are available in OpenSees: the “uniform excitation,” and the “multiple support excitation”. The uniform excitation pattern formulates the inertial loads for the transient analysis in the direction that input accelerations are specified. This load pattern is not used in the analysis because of two reasons: First is the inevitable deviations between the actual shake table motions and the target motions at different supports, which are mainly due to the interaction between the bridge and the shake tables, and second is that in addition to the longitudinal shake table motions at the pier bases, the abutment actuators apply abutment displacements to the bridge deck simultaneously. The uniform excitation pattern is not capable of handling simultaneous acceleration and displacement inputs. The multiple support excitation option, however, can handle different input records and types of simultaneous motions at different locations. This pattern is used in the analysis.
5.6.2 Cap-Beam Deck Connection

The connection between the cap beam and the deck in the test model is provided by two post-tensioned 1¼ in (31.7 mm) diameter bars at each bent, as shown in Fig 5-9. The bars have ultimate tensile strength of 187.5 kips (834 kN). The required post-tensioning force is calculated based on preventing the deck from sliding on the cap beam. Pushover analysis for different bents showed that Bent 1 has the highest lateral capacity of 49 kips (218 kN) among the bents. Hence the maximum shear transfer between the bents and the superstructure takes place at this bent. Therefore the lateral capacity of Bent 1 is used to calculate the post-tensioning force required in the connecting bars. Based on the weight of the deck and superimposed loads on Bent 1 and assuming a friction coefficient of 0.5 between the cap beam and deck with a safety factor of 2.5, the required post-tensioning force in each bar is found to be 159 kips (708 kN).

Moment curvature analysis of the connection bent cap/superstructure interface section is performed. The moment-curvature curve as shown in Fig. 5-10 has three kinks which divide the curve in parts with different slopes. The initial stiffness corresponds to the connection under full compression due to post-tensioning force in connecting bars. The stiffness then reduces when the applied moment balances the compression stress at one side of the connection and the superstructure starts to separate from the bent cap at tension side. The yielding of steel is the cause of further reduction in stiffness beyond the second kink point. The slope of moment-curvature curve remains positive due to steel strain hardening up to the peak point where the steel bars fail. Beyond the peak point the connection undergoes rapid strength reduction until concrete failure. Linear elastic behavior for the connection is assumed based on the idealized bilinear relationship as shown in Fig. 5-10.

5.6.3 Bond-Slip Modeling

The strains associated with stresses along the tensile reinforcing bar development length in reinforced concrete members create a concentrated elongation of the bar at the connection interface. This effect is referred to as bond slip. In this study Wehbe’s method (Wehbe et al.1997) is used to calculate bond slip at the connections at the top and bottom of the columns. The tri-linear bond-slip characteristic of the column ends are derived from moment curvature relationships (Fig.5-7), using cracking, yielding, and ultimate points of the column section. Details of bond-slip calculation are presented in Appendix A.

5.6.4 Bridge Abutment Arrangement in Test Setup

The prototype bridge has a seat type abutment. The abutment consists of a backwall connected to the top of the abutment rigid wall. There is a gap of 2 in (50.8 mm) between the backwall and the end of the bridge deck which is supported on rollers on the top of the abutment rigid wall in the prototype. In normal design practice, the abutment backwall is designed to break under large longitudinal earthquake motions. This helps in limiting the damage to the abutment rigid wall and preventing any damage to the lower part of abutment and the supporting piles. The same assumptions are used to design the abutment in the ¼ scale bridge test setup. The backwall in the scaled bridge model is represented by an L-shaped segment which is referred to as ‘abutment seat’ in this report.
The connection between the abutment seat and the abutment rigid wall is made in such a way to restrain all the degrees of freedom of the abutment seat except for sliding in the longitudinal direction of the bridge. The bridge deck at both ends is supported on frictionless Teflon sheets attached to the top of the horizontal surfaces of the abutment seats to represent roller type connections.

The vertical walls of the abutment seats are connected to the actuators at each bridge end. The actuators simulate the backwall movement during the earthquake motion and exert force on the bridge deck when the ½ in (12.7 mm) gap between the superstructure and the backwall closes. Since the abutment backwall is assumed to be sheared off from the abutment rigid wall at the early stages of the motion, the movement of the actuators in the bridge model represents the movement of the backwall after failure of its connection to the base. During shaking the separated back wall can slide back and forth due to the soil pressure on one side and the pushing of the bridge deck on the other side when the gap closes.

A similar assumption is made for the lateral shear keys at the abutment. The lateral shear keys are assumed to have failed thus not functioning under high amplitude motions. Therefore in bridge test setup the superstructure is free to move in the transverse direction at the abutment without any restraints. The longitudinal movement is also unrestrained until the gap between the superstructure and back wall is closed.

5.6.4.1 Abutment Back Wall Stiffness

The separated backwall is pushed against the backfill soil when the bridge deck moves towards the abutment and the gap is closed. A nonlinear force deformation relationship for backfill soil is defined based on the soil type, soil coefficient of friction, and the contact area with soil. The force deformation curve is first developed for the prototype bridge and then modified based on the ¼ scale bridge abutment. Sand with 30 degrees internal coefficient of friction is assumed as the backfill soil.

The height and the thickness of the prototype backwall are 5.5 ft (1676 mm) and 1 ft (304.8 mm), respectively. The force deformation curve is derived based on the formulation proposed by Shamsabadi et al. (2005) for unit length of the backwall as shown in Fig. 5-11. The curve is idealized by a tri-linear relationship first and then modified to account for the ¼ scale of the model bridge by multiplying the forces by 7.5'/4 to obtain the result in kips (2286 mm/4 for result in kN). Note that the abutment width in the bridge model is 7.5’ (2286 mm). The backwall mass in the scaled model is taken as 1/16 of the backwall mass in the prototype to keep the stresses the same as those in the prototype.

In addition to the compressive force displacement relationship due to the passive soil pressure, which defines the interaction between the backwall and the soil when backwall is pushed against the soil, a tensile stiffness for the backwall is defined. The tensile stiffness represents the friction force at the backwall base when it moves away from the soil and towards the deck. Tensile stiffness is found to be approximately 10% of the backfill soil compressive strength. The resulting backbone curve is shown in Fig. 5-12.

5.6.4.2 Models for Bridge-Abutment Interaction

In order to estimate the bridge-abutment interaction and its effect on the bridge response, three different models are developed (Fig. 5-13). The first model (Model 1)
represented a bridge with no abutment interaction. In this model the bridge deck ends are supported on rollers so the bridge deck is free to move in both horizontal directions.

In second bridge model (Model 2), the abutment soil is represented by a nonlinear spring with the backwall stiffness that is discussed in Section 5.6.4.1. The spring stiffness is assigned to a zero length element that is used to connect the backwall to a fixed point. A uniaxial elastic gap element is placed between the deck and the backwall spring to model the gap between them. The uniaxial element has an initial gap of 0.5 in (12.7 mm) and can take only compression loads. A large value for modulus of elasticity is specified for the element to let the abutment deformations take place only in the abutment nonlinear soil spring.

The third bridge model (Model 3) is developed to represent the actual bridge test setup with the abutment springs replaced by the actuators at the ends. The elastic gap elements at both ends are included in the model. The displacement histories recorded at the backwall nodes in Model 2 were used as the actuator input at the end of the elastic gap elements. Although the response of Model 3 differs from the response of Model 2 because of energy dissipation in the abutment spring in Model 2 (which is not included in Model 3), the difference is expected to be minimal because of the small proportion of the energy dissipation in the abutment spring to the energy dissipation in the entire bridge.
CHAPTER 6
Analytical Results

6.1 Introduction

The results obtained from the OpenSees analytical studies are presented in this chapter. The input motions to the bridge model specimen will include shake table bidirectional motions and actuator longitudinal motions at bridge ends to represent the bridge-abutment interaction. In the analytical model the bridge abutments are modeled as nodes connected to the springs which represent the abutment stiffness. The results of the abutment node displacements in the analytical model will be used as actuator input displacement record for the shake table testing.

The moment curvature, push over, and dynamic analysis results are presented in this chapter. The moment curvature analysis results are used as a basis to establish the bond-slip relationship at the column end connection points. The push over analysis of the bridge three bents is performed to identify bent stiffness in transverse direction. The cracking, yield, and ultimate force and displacements are specified from the force-deformation curves. Finally the dynamic results are presented which include the abutment motions and the bridge response. Comparison between the bent displacement obtained from push over analysis and the bent displacement achieved in dynamic analysis is made to identify the level of motion that would lead to cracking, yielding, and failure of the columns. This information is useful to plan the shake table testing.

The 2-span bridge model (discussed in Chapter 4) was only subjected to transverse motion and did not include the abutment interaction. However, the reasonable correlation between the measured and OpenSees predicted displacements provides some confidence that this model is reasonably accurate in estimating the 4-span bridge response.

6.2 Moment Curvature Analysis

The column sectional analysis is performed to calculate the level of stress and strain in steel and concrete under increasing moment and define the moment curvature curve for the column section (as discussed in Sec 5.4). The curve is then used to define the moment rotation curve for the column plastic hinge zone. Finally the moment rotation property of the column section is modified to account for bond slip at the column ends. The modification is done at the following three level of the applied moment: concrete cracking, yielding of steel, and section ultimate strength. The resulting tri-linear curve is then used as the backbone curve for the hysteretic material property of the zero length element defined at the column ends to incorporate bond-slip properties.

6.3 Pushover Analysis

Load-deflection curves from pushover analyses of the three bents are shown in Fig. 6-1. After yielding in each curve, a slight kink is observed due to the crushing of the unconfined concrete. Beyond this point, the bents continue to resist higher loads mainly due to reinforcement strain hardening. The points at which confined concrete crushes are marked in the figure. The confined concrete crushing is at displacements of approximately 3.8, 6.34, and 5.0 in (96, 161, and 127 mm) for bent 1, bent2, and bent 3,
respectively. Since OpenSees Concrete01 model does not allow for removal of concrete beyond the crushing point, no sudden decrease in force can be observed in Fig. 6-1.

Elasto-plastic idealizations of pushovers are plotted in Figs. 6-2 to 6-4. The yield point is determined by passing the initial slope through the first yield point of longitudinal reinforcement and balancing the areas between the idealized plot and the pushover plot. To define the ultimate displacement for the bents, two methods are reviewed. The first is to assume ultimate displacement point where the lateral force decreases to 85 percent of the maximum force. Unrealistically high displacement values are calculated by this method, because the concrete model in OpenSees has no abrupt loss in concrete strength after crushing. The other method to define ultimate displacement is to take it at the point of confined concrete crushing as shown on the pushover curves in Fig. 6-1. The displacement ductility capacities for the bents are shown in Table 6.1. The ductility capacity of Bent 2 is 8.9 which is the highest among the bents.

6.4 Modal Shapes of Superstructure
To have a better understanding of the response of the bridge during the shake table testing, an eigenvalue analysis is performed. This includes calculation of the periods and the elastic modal shapes. Since the bridge is excited in both the longitudinal and transverse directions, modal shapes and periods in both directions are considered. The calculated superstructure modal shapes for the first five modes are shown in Fig. 6-5. To determine the contribution of each mode to the response mass participation factors are calculated. The periods and mass participation factors for the mass in each direction for the first five modes are presented in Table 6-2. Only the first five modes are listed because as shown in Table 6-2 the first five modes include more than 99 percent of the modal response in both directions.

The first mode has a period of 0.42 second and a participation factor of 0.182. This mode is primarily in-plane rotation of the superstructure with slight translation and peak amplitude at the right abutment. Bent 3 (the bent with medium height) has the maximum transverse displacement among the bents in this mode. Mode 2 is in the longitudinal direction with a period of 0.36 second. The longitudinal response of the bridge is dominated by this mode, which has a mass participation of 0.99. Mode 3 shows the primary transverse mode, which has a period of 0.33 second and a mass participation factor of 0.815. This mode is primarily superstructure translation with slight in-plane rotation and peak amplitude at bent 1, which is the shortest bent. The fourth and fifth modes have periods of 0.18 and 0.10 second and mass participation factors of 0.002 and 0.001, respectively. These modes are superstructure bending in the transverse direction. Because the superstructure is very stiff in comparison to the substructure, the third transverse mode plays very little role in the transverse displacement response of the bridge model.

6.5 Dynamic Analysis of Model 2
6.5.1 Longitudinal and Transverse Displacement of Bents
The full scale acceleration records, discussed in Section 2.5, are input in the transverse and longitudinal directions of the bridge model. The bent top displacements in the longitudinal and transverse directions are shown in Figs. 6.6 to 6.8. The maximum
positive and negative displacements for bent 1 in transverse are 1.65” (42 mm) and 1.4” (36 mm), respectively. In the longitudinal direction the maximum displacements are 2.1” (53 mm) and 2.7” (68 mm) in positive and negative directions, respectively. Comparison between the longitudinal displacements (Fig. 6-6) and abutment-deck gap size (Fig. 6-17) indicates that the gap is closed at the time of maximum displacement. This implies that the bridge peak longitudinal displacements are influenced by the stiffness of the abutments.

Due to the large axial stiffness of the bridge deck the displacements of all three bents in the longitudinal direction of the bridge are the same. In the transverse direction, the trends in the displacement histories are similar but the peaks are different. This is due to the bridge deck in-plane rotation which is caused by the unequal stiffness of the bents.

Although the target motions for the shake table tests are planned to be the same for all three shake tables, past experience with 2-span bridge model testing at UNR shows that the actual motions may be slightly incoherent in multiple shake table tests. Because torsional modes might be more sensitive to incoherent motions, a more significant in-plane rotational effects might be observed in the actual tests than what is predicted by the analytical models. The current analytical model will be revised based on the achieved shake table motions to check the accuracy of the model in duplicating the bridge measured response.

6.5.1.1 Bent Displacements

The displacements of top of the bents in the horizontal plane are shown in Figs. 6-9 to 6-11. The displacement trends show that the high simultaneous displacements in both directions occur when the bridge is moving in the negative longitudinal direction (southward). To calculate the ductility demand, it can be assumed that the columns would be under the same flexural demands for the same displacements of the bridge deck in either longitudinal or transverse directions. This assumption is valid only if no cap beam rotation is expected during the shaking about the x and z horizontal axes. Due to the high bridge deck stiffness, the cap beam rotation about the bridge longitudinal axis is expected to be small. However, slight cap beam rotation may occur about the bridge transverse axis because of elongation of the post-tensioned rods connecting cap-beam to the deck. This rotation is ignored in the ductility demand calculations in the pretest analyses. Therefore the resultant horizontal bent displacement is used to calculate its ductility demand and compare with ductility capacity.

In order to identify the maximum bent displacement, resultants of longitudinal and transverse displacement histories are calculated on time scale. The resultant displacements for the bents are shown in Figs. 6-12 to 6-14. The peak resultant displacement values are close to the peak longitudinal direction values (Figs. 6-6 to 6-8), meaning that the maximum displacements in longitudinal and transverse directions do not occur at the same time. The ductility demand is estimated by dividing the resultant displacements (from Figs. 6-12 to 6-14) to the bent yield displacements obtained from the idealized pushover curves (Figs. 6-2 to 6-4) for in-plane load analysis of each bent. The maximum ductility demand for the full scale biaxial motions for the Bent 1, Bent 2, and Bent 3 are 5.8, 3.9, and 4.7, respectively. This is a crude estimate of the displacement ductility because it is based on the pushover curves of the bents in the transverse
direction of the bridge. Because the column top fixity is different in the longitudinal and transverse direction of the bridge, the yield displacements would be different.

6.5.2 Abutment Displacement Histories

The abutment back wall is assumed to be the only part of the abutment that has interaction with the bridge deck during shaking. As mentioned in Sec. 5.6.4.2, according to Caltrans (2001) guidelines the back wall is designed to be sheared off from the bottom portion of the abutment under moderate level of shaking. The reason for this design concept is to save the lower portion of the abutment and the piling system from excessive rotation and/or damage which is expensive to repair.

The bridge abutment at each bridge end is represented by one node. The mass of the bridge back wall is lumped at the back wall nodes. The back wall nodes are connected to the bridge deck ends with gap elements with a 0.5” (12.7 mm) gap size. The abutment stiffness springs are then defined to be connected to the back wall nodes at one end and to a fixed point at the other end. The assumed stiffness of the abutment and type of elements used in the model are discussed in Section 5.6.4.

The displacements of the abutments in the longitudinal direction of the bridge are shown in Fig. 6-15. The left abutment is at the south end of the bridge model. The abutment displacements in the north direction are positive and in the south direction are negative. Since both abutments are under the same input motions, their displacements are essentially the same before the deck-abutment gap closure. The directions of abutments displacement are different after the gap closure (Fig. 6-15). The left (south) abutment moves towards the south and the right (north) abutment moves towards the north after the gap closure. The outward displacements of the abutments are resisted by the back fill soil. In the model the abutment spring compressive strength is defined based on the passive soil pressure resistance when the back wall is pushed against the soil. As discussed in Section 3.2, the tension spring stiffness is calculated based on the friction force between the separated back wall and the top of the abutment, which is about 1/10 of its compressive stiffness.

The displacements of the bridge left end deck node and the left abutment are shown in Fig. 6-16. The positive and negative displacements show the movements in north and south directions respectively. Because of different mass and stiffness properties, even before the first contact between the deck and the abutment, their displacements do not match. The abutment is forced to move with the deck when the gap is closed, as expected.

6.5.3 Abutment-Deck Gap Size History

The gap size history which is the relative displacement between the bridge deck and the abutment is shown in Fig. 6-17. The initial gap size at both ends is 0.5” (12.7 mm). In the first two seconds of the motion, since there is no contact between the deck and the abutment the gap size is fluctuating about initial gap size. Between two to seven seconds numerous gap closures occur at both bridge ends. A maximum separation of 3.4” (86 mm) between the right abutment and the bridge deck is recorded at about five seconds. At the left abutment the maximum gap size is about 2.6” (66 mm). No gap closure is seen after about seven seconds and the gap size remains close to 0.5” (12.7 mm) with no significant permanent displacements at the end of the shaking.
6.5.4 Deck-Abutment Interaction Force

The force transmitted between the deck and the abutment at gap closures is taken as the force in the gap element. The gap element force history is shown in Fig. 6-18. Since the high forces are only generated at the time of gap closure, the vertical lines in Fig. 6-18 shows the instant force transmission between the deck and the abutment at the times that they are in contact. Maximum forces of about 231 kips (1228 kN) and 248 kips (1104 kN) are recorded at the left (south) and right (north) bridge ends, respectively. These forces are unlikely to be realistic because they occur at a very short period of time (less than 0.005 sec) which is not representative of the condition of the actual test. In addition, the energy loss during the bridge and the abutment impact is not considered in the OpenSees analysis.

6.5.5 Bridge Response under Multiple Motions of Increasing Amplitudes

The main goal of this analytical study is to predict the bridge model response during shake table testing. In order to capture the bridge response from initial yielding of reinforcement through column failure, the motions will be applied to the bridge initially at low amplitudes and then with scaled up amplitudes for the successive tests until failure according to the loading protocol shown in Table 6-3. The full scale motion record in the transverse direction is scaled by its peak acceleration and applied in sequential runs of increasing amplitude with PGA from 0.075 g to 1.32 g. The same scale factors are applied to the full scale motion in the longitudinal direction leading to motions with PGA of 0.10g to 1.77g (as discussed in Chapter 2), and is input to the model with transverse motion simultaneously. In addition the abutment actuators apply prescribed displacements that are estimated in the study presented in this report. The input abutment motions for each are obtained from the corresponding analytical results.

6.5.5.1 Bent Displacements

The maximum simultaneous bi-directional displacements occur in the positive and negative (transverse and longitudinal) directions (Figs. 6-19 to 6-21), which are the North-West and South-East directions of the bridge shake table test setup, respectively. The bridge bent displacements in both horizontal directions are shown in Figs. 6-22 to 6-24 and the resultant displacements are calculated and shown in Figs. 6.25 to 6.27. The maximum displacements of the bents for all 7 events are shown in Table 6-3. The only pre-yield motion is the first motion with maximum GPA of 0.075g and 0.10g in the transverse and longitudinal directions, respectively. In the second run all three bents are fully yielded with maximum displacements of 1.0” (25 mm). Based on the yield displacements (Table 6-1), the ductility demand for all three bents are calculated and tabulated in Table 6-4. Bent 1 is close to failure at Event 5. The failure of Bents 1 & 3 is expected to occur in Events 6 & 7, respectively.

6.5.5.2 Abutment Displacement and Gap Size

Both abutments experience residual displacements starting in Event 2 (Fig. 6-28). Larger bent displacements occur in outward directions where the abutments are pushed against the back fill soil (i.e. left abutment in the negative direction and right abutment in the positive direction). The residual displacement increases in the north direction for the
right (north) abutment with a maximum of 0.3” (7.6 mm) at the end of Event 6. The residual displacements are reduced during Event 7. The gap size history is shown in Fig.6-29. No gap closures occur in the first run. Fewer gap closures occur in the second run as compared to the subsequent runs. The gap sizes at both ends are close to each other for the first three runs. However, the gap size at the right abutment is larger than the left abutment gap for the last four runs with a maximum of 4.9 in (124 mm) and 6.7 in (170 mm) at the left and right abutments, respectively.

6.5.5.3 Deck-Abutment Interaction Force
The calculated force in the end gap elements are shown in Fig. 6-30. No force is transmitted between the bridge and the abutment in Event 1 because the gap between them remains open during this event. The maximum force of 290 kips (1290 kN) obtained at the left abutment at Event 5 is increased to 300 kips (1335 kN) in Event 6. However, for the right abutment the force decreases from 260 kips (1157 kN) to 230 kips (1023 kN) from Event 5 to Event 6. The highest gap element forces of 630 kips (2804 kN) and 320 kips (1424 kN) are recorded during Event 7 at the left and right abutments respectively. As mentioned in Sec. 6.5.4, these forces are unlikely to be realistic.

6.6 Dynamic Analysis of other Bridge Analytical Models
6.6.1 Model 1
As mentioned in Chapter 5, to study the effect of the abutment interaction, Model 1 analytical model with no abutment interaction is developed. In this model the bridge deck ends are supported on the rollers with no back wall interaction. The displacement histories for the three bents are shown in Figs. 6-31 to 6-33. Comparison between the bent displacements between Model 1 and 2 shows about 8% lower maximum displacements in Event 7 in Model 1 (Figs. 6-25 to 6-27). No significant change in displacement patterns is seen between the two models. The maximum displacement comparison for Events 2 and 7 are shown in Figs. 6-34 and 6-35, respectively. Since the abutment spring is defined only in the longitudinal direction of the bridge in Model 2, the longitudinal motions are compared in Figs. 6-36 and 6-37. It can be seen that despite some deviations at the peak points, the general longitudinal displacement trends in both models are similar. This implies that the bridge motions are not greatly influenced by the abutment interaction.

6.6.2 Model 3
The abutment interaction in this model is represented by the actuators at both bridge ends. The bridge abutment absolute displacement histories recorded in Model 2 are used as actuator inputs to the bridge deck at both ends. This model represents the bridge model test setup on the UNR shake tables. The maximum displacement histories for the three bents are shown in Figs. 6-38 to 6-40. In general, the displacements are reduced in Model 3 as compared to Model 2 (Figs. 6-25 to 6-27) during each event. The maximum displacement of Bent 1 in Model 3 during Event 7 is approximately 27% less than the Bent 1 displacement in Model 2. Comparison between the two models is shown in Figs. 6-41 and 6-42 for Event 2 and Event 7 respectively. In general, the bent displacements in Model 3 are smaller than the bent displacements in Model 2 (Figs.6-43 to 6-46). Before the gap closure, the displacements in both models are similar. After first few gap
closures, the slight energy dissipation in the abutment spring in Model 2 (which is not represented in Model 3) alters the response of the bridge and its relative displacements to the abutment. This results in stronger impacts between the bridge and the abutment in Model 3 and consequently reduction in peak displacements of the bridge as compared to Model 2. As shown in Figure 6-47, the actuator forces in Model 3 are significantly higher than in Model 2 (Fig. 6-30).
CHAPTER 7
Summary and Conclusions

7.1 Summary
This report presents a pre-test analytical study of a 4-span reinforced concrete bridge model to be tested on the three shake table system at University of Nevada, Reno. The objectives of the study are to two folds: first to provide a reliable analytical model for use for future seismic study of the bridges, and second, to use the analytical model to predict the response of the bridge model. A ¼-scale of the prototype bridge has been designed for the shake table testing. The bridge model has three two-column bents with variable heights, thus undergoes in-plane rotation under lateral seismic loads. The main goal of shake table testing of the bridge is to study the bridge system response under earthquakes.

The motions that are used in the shake table tests are based on the 1994 Northridge earthquake recorded at the Century City Country Club North ground station. These records were modified to account for soil-structure-foundation interaction effects. These motions are the same as those used in the study of a two-span bridge model (Johnson et al. 2006) and are selected for the present study to facilitate the comparison of the performance of the bridge models from the two studies. In the two-span bridge test only the motion in transverse was included. The input motion was the 90-degree component of the modified record (Motion 1). However, in the present study both the 90-degree (Motion 1) and 360-degree (Motion 2) lateral motion components are used. The general method for the motion derivation is discussed in Johnson et al. (2006).

The bridge model is analyzed for bi-axial excitations simultaneously in the transverse and longitudinal directions. The transverse input motion protocol is the same as that used in Johnson et al. (2006) and is the same for all three shake tables. The same motions are used in successive runs, but the peak ground acceleration (PGA) is scaled up in subsequent runs. Motion 1 record was scaled with increasing factors that led to PGAs from 0.075 g to 1.32 g. The same factors were used for Motion 2. Since Motion 2 has higher PGA, the PGAs in the corresponding motions in the longitudinal direction range from 0.10 g to 1.77 g. These same motions will be used as the shake table input to the bridge model that will be tested on the UNR shake tables.

An analytical model similar to the model developed for the 4-span bridge is developed for the 2-span bridge. Comparison is made between the measured and the analytical model output to verify the accuracy of the analytical model. It is believed that this comparison will give an indication of the accuracy of the analytical model for the 4-span bridge, due to the similarity between their analytical and shake table test models. The same element types and material models are used to develop the OpenSees analytical models for both bridges. The achieved shake table motions from high amplitude tests (tests12 through 19) were used in the study of the two-span bridge model. The evaluate the performance of the analytical model, the superstructure calculated displacement histories were compared with the measured displacement histories obtained from shake table tests.
Three analytical models are developed for the 4-span bridge. The first model (Model 1) does not include the abutment interaction. In this model the bridge deck is supported on rollers at both ends. The bridge abutment stiffness is incorporated in the second analytical model (Model 2) to study the bridge response with deck-abutment interaction. The abutment is represented by a node with an allocated mass equivalent to the mass of the abutment backwall. The abutment node is connected to the bridge deck through a gap element incorporating 0.5 in (12.7 mm) gap between the deck and the abutment. To include the interaction between the backwall and the back fill soil, the abutment node is connected to a fixed point through a longitudinal spring. The spring stiffness in compression is specified as the soil stiffness when the gap closes and the backwall is pushed against the soil. However the tensile stiffness of the spring is defined based on the friction between the backwall and the lower part of the abutment, assuming that the backwall is sheared and separated from the abutment during seismic motions.

The bond-slip effect at the top and bottom of the columns are included in the model. The bond-slip property is modeled with a ‘Hysteretic’ material in OpenSees. The same type of material is used for the bridge superstructure and cap beam connection. The connection in the bridge test model is provided by pre-tensioned rods. Based on the section properties at the connection, a tri-linear moment-rotation curve is defined and used as the Hysteretic material backbone curve in OpenSees model. To model the 0.5 in (12.7 mm) gap between the bridge deck ends and the abutments, ‘Elasto-Plastic Gap Material’ is used. This material which is associated to a zero length element is connected to another zero length element with Hysteretic tri-linear material properties representing the abutment stiffness based on the assumed backfill soil properties.

The output displacement histories of the abutment node in Model 2 will be used in the bridge shake table test as the actuator input representing the abutment interaction. To simulate the bridge test setup, a third mode, Model 3, is developed. The abutment springs in Model 2 are replaced by the “actuators” in Model 3. The actuators at both bridge ends apply the abutment node displacement history recorded from Model 2, simultaneous with the transverse and longitudinal accelerations applied to the bridge pier footings. Model 3 is developed to compare the response of the bridge with abutment actuators and the bridge with abutment springs and no actuators (Model 2), because the energy dissipation in the abutment springs in Model 2 could alter the response of the bridge. However, the difference between the two bridge responses is found to be negligible because of the low energy dissipation in the abutment springs compared to the hysteretic energy dissipation in the entire bridge.

7.2 Conclusions

The analytical modeling of the bridge shows that OpenSees is capable to predict the seismic response of the highly nonlinear structural systems with reasonable accuracy. Many important conclusions related to OpenSees analytical modeling of bridges subjected to seismic loads are arrived at in this study. The conclusions are placed in two categories, one on OpenSees modeling aspects of the study and the other on analytical results.
7.2.1 OpenSees Modeling

Section and Element Types

Fiber section in OpenSees is a powerful type of section to be used for the non-linear elements such as bridge columns. Comparison between the analysis time required for the bridge with elastic property along the element with lumped plastic hinges at both ends, and the bridge with non-linear columns composed of fiber element shows negligible difference. Thus, because of higher accuracy obtained from non-linear columns with fiber section elements, this type of modeling is used to model the columns.

To increase the efficiency of the model, the number of slices and layers for the fiber sections has to be minimized, while maintaining a reasonable level of accuracy. The optimized number of slices for a circular fiber section is eight. The accuracy of the analysis is not improved by defining more than eight slices. However, the analysis accuracy is improved by increasing the number of layers. Seven layers for the column core and two for the concrete cover is found to be an optimum number with reasonable accuracy.

Load Patterns

Multiple support excitation is a preferable load pattern choice for structures with multiple sources of excitations as compared to uniform excitation. The multiple support excitation is not only capable of taking motions at different bridge piers and abutment locations but also it can take simultaneous motions in terms of acceleration, velocity, or displacement at different locations. This pattern is the only choice for Model 3 since simultaneous displacement input and acceleration input are required at abutments and footings, respectively. This pattern is also the only choice for incoherent motions at the bridge footings, such as shake table testing of multi-span bridges. The past shake table tests of the two-span bridge showed that the measured shake table displacements were not the same despite the identical input motions fed into the control system of all the shake tables.

System Commands

There are more than six different commands in OpenSees to construct the linear system of equations and linear solver objects to store and solve the system of equations. The ‘system ProfileSPD’ command is used to construct a symmetric positive definite profile system of equations objects. This profile solver is based on variable bandwidth elimination algorithm. The main advantage of using variable bandwidth storage as opposed to diagonal band storage matrix decomposition is its optimum variable bandwidth ordering schemes which will often provide more efficient decompositions.
Solution Algorithm

The combination of Newton-Raphson method and Modified Newton-Raphson method is the most efficient model to construct a solution algorithm. The Newton-Raphson method converges rapidly to a solution, if the initial estimate is sufficiently close to the solution, but otherwise fails to converge. In this method the tangent stiffness is updated at each iteration. In the modified Newton-Raphson method the tangent stiffness is updated only at selected steps, thus avoiding lengthy calculations needed in multi degree-of-freedom systems. However, more iteration may be needed to reach a prescribed accuracy. To minimize the required time for the analysis, Modified Newton Algorithm is used as the main solution algorithm in the bridge model. The Newton algorithm is specified as the solution algorithm if the convergence with the first system is not achieved after the specified maximum number of iteration with specified tolerance.

Analysis Command

Since the input loading function is an arbitrary time dependent function, then a transient response analysis has to be performed. The ‘Transient Analysis’ in OpenSees performs the analysis with constant, user specified time step. This method is inappropriate for the bridge analysis. The analysis stops at early stages with ‘no convergence’ message when the analysis time step is 0.001 sec. Decreasing the time step to 0.0005 sec significantly increases the analysis time, and still the analysis stops with ‘no convergence’ message but at a later stage. The best way to overcome this problem is to use ‘Variable Transient Analysis’ option. This command performs the same analysis type as the transient analysis object. The time step, however, is variable. This method is used when there are convergence problems with the transient analysis object at a peak or when the time step is relatively small. The analysis is successfully performed by using the variable transient analysis to the end of all the input motions with the specified time step of 0.001 sec and minimum and maximum time steps of 0.0001 sec and 0.005 sec, respectively.

7.2.2 Analytical Results

Two-span Bridge Model

The two-span bridge analytical model displacement histories are compared with the shake table test data. The following conclusions can be drawn from the comparison of the predicted and measured displacement histories:

- The analytical model predicted the measured response with reasonable accuracy in terms of waveforms.
- Although the calculated displacements showed reasonably consistent correlation with the measured data for the entire range of test amplitudes, in general the analytical model underestimated the measured peak displacements by 12% on average.
4-Span Bridge Model

Although the results obtained from the bridge analytical model need to be verified with the measured data from the shake table testing, the following conclusions can be drawn from the pre-test analytical studies.

- Comparison between the bridge bent ductility capacity (obtained from push-over analysis) and the ductility demand (obtained from the dynamic analysis) shows that the bridge model may only survive up to Event 6 with a PGA of 0.95g and 1.27g in the transverse and longitudinal directions, respectively. Bent 1 with the shortest columns is most vulnerable to failure followed sequentially by Bent 3 and Bent 2.

- The dominant overall maximum displacement of the bridge is towards the North (longitudinal direction of the bridge) up to Event 4 and will then shift toward North-West close to failure at higher level motions.

- No major change in response of the bridge with no abutment spring (Model 1) and the bridge with the abutment spring (Model 2) is observed. Thus elimination of the abutment interaction causes negligible change in bridge response in terms of the displacement waveforms. The maximum longitudinal displacement was increased by approximately 20% when the abutments were added. The effect of abutment interaction could have been more visible by assuming higher stiffness for the backfill soil, smaller deck-abutment gap, and a non-zero flexural stiffness for the backwall (as opposed to a sheared off backwall with zero stiffness).

- In general, the bent displacements in Model 3 are smaller than the bent displacements in Model 2. The difference in the response is mainly due to energy dissipation in the abutment spring in Model 2 which is not represented in Model 3.

- Based on the 6.7 in (170 mm) maximum longitudinal relative displacement between the abutment and bridge deck (gap size) in Event 7 and by considering a reasonable factor of safety, the design width for the abutment seat can be determined for the shake table bridge model testing.

- The maximum recorded actuator (abutment spring) force of 630 kips (2804 kN) seems to be unrealistic because the impact energy loss due to energy absorbed by the actuator and the abutment seat is not considered in the analytical model. The estimated force is not recommended to be used as the required actuator capacity for the bridge testing.
REFERENCES


Caltrans, 1999. Seismic Design Criteria 1.1, California Department of Transportation, California.


Table 2-1 Bridge Column Axial Loads

<table>
<thead>
<tr>
<th>Column Location</th>
<th>Post Tensioning Kips (kN)</th>
<th>Self Weight Kips (kN)</th>
<th>External Weights kips (kN)</th>
<th>Total Kips (kN)</th>
<th>Capacity Kips (kN)</th>
<th>Index %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bent 1</td>
<td>-8.9 (-39.6)</td>
<td>20.5 (91.2)</td>
<td>29.6 (131.7)</td>
<td>41.1 (182.9)</td>
<td>565.5 (2516.5)</td>
<td>7.27</td>
</tr>
<tr>
<td>Bent 2</td>
<td>8.7 (38.7)</td>
<td>22.3 (99.2)</td>
<td>8.5 (37.8)</td>
<td>39.4 (175.3)</td>
<td>565.5 (2516.5)</td>
<td>6.97</td>
</tr>
<tr>
<td>Bent 3</td>
<td>-8.9 (-39.6)</td>
<td>20.5 (91.2)</td>
<td>29.6 (131.7)</td>
<td>41.1 (182.9)</td>
<td>565.5 (2516.5)</td>
<td>7.27</td>
</tr>
</tbody>
</table>
## Table 2-2 Earthquake Level Input (g)

<table>
<thead>
<tr>
<th>Event No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Unscaled Motion PGA (g)</strong></td>
<td>Coeff</td>
<td>Scaled PGA (g)</td>
<td>Coeff</td>
<td>Scaled PGA (g)</td>
<td>Coeff</td>
<td>Scaled PGA (g)</td>
<td>Coeff</td>
</tr>
<tr>
<td>Transverse Direction</td>
<td>0.47</td>
<td>0.158</td>
<td>0.075</td>
<td>0.316</td>
<td>0.15</td>
<td>0.527</td>
<td>0.25</td>
</tr>
<tr>
<td>Longitudinal Direction</td>
<td>0.63</td>
<td>0.158</td>
<td>0.10</td>
<td>0.316</td>
<td>0.20</td>
<td>0.527</td>
<td>0.33</td>
</tr>
</tbody>
</table>
Table 4.1 Comparison of Bent 1-3 Peak Displacements of Measured and OpenSees

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>Max 0.20 5.1</td>
<td>-0.20 -5.1</td>
<td>Max 0.17 4.3</td>
<td>-0.16 -4.0</td>
<td>Max 0.19 4.8</td>
<td>-0.18 -4.6</td>
<td>+83 +40</td>
<td>+11 +12</td>
<td>+83 +91</td>
</tr>
<tr>
<td></td>
<td>Min -0.20 -5.1</td>
<td>-0.28 -7.1</td>
<td>Min -0.16 -4.0</td>
<td>-0.18 -4.6</td>
<td>Min -0.12 -3.1</td>
<td>-0.23 -5.8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>Max 0.58 14.7</td>
<td>-0.61 -15.5</td>
<td>Max 0.37 9.3</td>
<td>-0.43 -10.9</td>
<td>Max 0.39 10.9</td>
<td>-0.42 -10.7</td>
<td>+91 +5</td>
<td>+5 +2</td>
<td>+11 +22</td>
</tr>
<tr>
<td></td>
<td>Min -0.61 -15.5</td>
<td>-0.55 -13.9</td>
<td>Min -0.43 -10.9</td>
<td>-0.42 -10.7</td>
<td>Min -0.31 -7.9</td>
<td>-0.38 -20.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>Max 0.75 19.1</td>
<td>-0.72 -18.3</td>
<td>Max 0.56 14.2</td>
<td>-0.56 -14.2</td>
<td>Max 0.49 12.6</td>
<td>-0.50 -12.7</td>
<td>+11 +19</td>
<td>+9 +24</td>
<td>+33 +24</td>
</tr>
<tr>
<td></td>
<td>Min -0.72 -18.3</td>
<td>-0.52 -13.2</td>
<td>Min -0.56 -14.2</td>
<td>-0.50 -12.7</td>
<td>Min -0.65 16.5</td>
<td>-0.62 -15.8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>Max 1.13 28.8</td>
<td>-1.54 -39.1</td>
<td>Max 1.11 28.3</td>
<td>-1.25 -31.7</td>
<td>Max 1.01 26.2</td>
<td>-1.03 -26.2</td>
<td>+10 +28</td>
<td>+7 +19</td>
<td>+6 +17</td>
</tr>
<tr>
<td></td>
<td>Min -1.24 -31.7</td>
<td>-1.25 -31.7</td>
<td>Min -1.03 -26.2</td>
<td>-1.03 -26.2</td>
<td>Min -1.21 -30.7</td>
<td>-1.21 -30.7</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Max 2.63 66.9</td>
<td>-2.22 -56.3</td>
<td>Max 2.30 58.3</td>
<td>-1.77 -45.0</td>
<td>Max 1.47 37.3</td>
<td>-1.52 -38.6</td>
<td>+15 +31</td>
<td>+9 +36</td>
<td>+15 +17</td>
</tr>
<tr>
<td></td>
<td>Min -2.22 -56.3</td>
<td>-1.97 -50.0</td>
<td>Min -1.77 -45.0</td>
<td>-1.52 -38.6</td>
<td>Min -1.60 -40.6</td>
<td>-1.64 -41.6</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>Max 1.98 50.3</td>
<td>-1.30 -33.0</td>
<td>Max 1.95 49.4</td>
<td>-1.42 -36.1</td>
<td>Max 0.85 21.6</td>
<td>-1.10 -27.9</td>
<td>+28 +50</td>
<td>+7 +56</td>
<td>+22 +2</td>
</tr>
<tr>
<td></td>
<td>Min -1.30 -33.0</td>
<td>-1.37 -34.8</td>
<td>Min -1.42 -36.1</td>
<td>-1.10 -27.9</td>
<td>Min -1.39 -35.3</td>
<td>-1.39 -35.3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>Max 2.28 57.8</td>
<td>-2.78 -70.6</td>
<td>Max 3.37 85.6</td>
<td>-2.75 -69.8</td>
<td>Max 1.97 50.0</td>
<td>-1.91 -48.5</td>
<td>+30 +5</td>
<td>+24 +41</td>
<td>+23 +17</td>
</tr>
<tr>
<td></td>
<td>Min -2.78 -70.6</td>
<td>-2.34 -59.4</td>
<td>Min -2.75 -69.8</td>
<td>-1.91 -48.5</td>
<td>Min -2.91 -73.9</td>
<td>-2.91 -73.9</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>Max 3.52 89.5</td>
<td>-2.79 -70.9</td>
<td>Max 2.99 75.9</td>
<td>-4.27 -108.4</td>
<td>Max 1.73 43.9</td>
<td>-2.26 -57.4</td>
<td>+25 +83</td>
<td>+7 +73</td>
<td>+25 +30</td>
</tr>
<tr>
<td></td>
<td>Min -2.79 -70.9</td>
<td>-2.44 -62.0</td>
<td>Min -4.27 -108.4</td>
<td>-2.26 -57.4</td>
<td>Min -4.71 -119.6</td>
<td>-3.26 -82.8</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 6-1 Estimated Displacement Ductility Capacities of Bents in Bridge Transverse Direction

<table>
<thead>
<tr>
<th></th>
<th>Yield Displacement in</th>
<th>Yield Displacement mm</th>
<th>Ultimate Displacement in</th>
<th>Ultimate Displacement mm</th>
<th>Displacement Ductility</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bent 1</td>
<td>0.48</td>
<td>12.2</td>
<td>3.83</td>
<td>97.3</td>
<td>8.0</td>
</tr>
<tr>
<td>Bent 2</td>
<td>0.71</td>
<td>18.0</td>
<td>6.34</td>
<td>161.0</td>
<td>8.9</td>
</tr>
<tr>
<td>Bent 3</td>
<td>0.6</td>
<td>15.2</td>
<td>5.02</td>
<td>127.5</td>
<td>8.4</td>
</tr>
</tbody>
</table>

6-2 Bridge specimen modal mass participation factors for modes 1 through 5

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.42</td>
<td>0.000</td>
<td>0.182</td>
<td>0.000</td>
<td>0.179</td>
<td>0.000</td>
<td>0.817</td>
</tr>
<tr>
<td>2</td>
<td>0.36</td>
<td>0.990</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.179</td>
<td>0.000</td>
</tr>
<tr>
<td>3</td>
<td>0.33</td>
<td>0.000</td>
<td>0.815</td>
<td>0.000</td>
<td>0.794</td>
<td>0.000</td>
<td>0.176</td>
</tr>
<tr>
<td>4</td>
<td>0.18</td>
<td>0.000</td>
<td>0.002</td>
<td>0.000</td>
<td>0.001</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>5</td>
<td>0.10</td>
<td>0.000</td>
<td>0.001</td>
<td>0.000</td>
<td>0.001</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>
Table 6-3 Maximum Displacements under Different Level of Motions

<table>
<thead>
<tr>
<th>Event No.</th>
<th>Earthquake Level (g)</th>
<th>Bent 1 Disp. in (mm)</th>
<th>Bent 2 Disp. in (mm)</th>
<th>Bent 3 Disp. in (mm)</th>
<th>Max. Gap Size in (mm)</th>
<th>Comm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.075</td>
<td>0.10</td>
<td>0.40 (10.2)</td>
<td>0.41 (10.4)</td>
<td>0.45 (11.4)</td>
<td>0.28 (7.1)</td>
</tr>
<tr>
<td>2</td>
<td>0.15</td>
<td>0.20</td>
<td>0.67 (17.0)</td>
<td>0.95 (24.1)</td>
<td>1.0 (25.4)</td>
<td>0.60 (15.2)</td>
</tr>
<tr>
<td>3</td>
<td>0.25</td>
<td>0.33</td>
<td>0.93 (23.6)</td>
<td>1.60 (40.6)</td>
<td>1.60 (40.6)</td>
<td>0.80 (20.3)</td>
</tr>
<tr>
<td>4</td>
<td>0.50</td>
<td>0.66</td>
<td>1.60 (40.6)</td>
<td>2.70 (68.6)</td>
<td>2.60 (66.0)</td>
<td>1.65 (41.9)</td>
</tr>
<tr>
<td>5</td>
<td>0.75</td>
<td>1.00</td>
<td>2.60 (66.0)</td>
<td>3.40 (86.4)</td>
<td>3.70 (94.0)</td>
<td>2.65 (67.3)</td>
</tr>
<tr>
<td>6</td>
<td>0.95</td>
<td>1.27</td>
<td>3.60 (91.4)</td>
<td>4.00 (101.6)</td>
<td>4.35 (110.5)</td>
<td>3.70 (94.0)</td>
</tr>
<tr>
<td>7</td>
<td>1.32</td>
<td>1.77</td>
<td>4.50 (114)</td>
<td>5.40 (137)</td>
<td>5.82 (148)</td>
<td>4.40 (112)</td>
</tr>
</tbody>
</table>
Table 6-4 Maximum Bent Displacements and Corresponding Displacement Ductilities

<table>
<thead>
<tr>
<th>Event No.</th>
<th>Bent 1</th>
<th>Bent 2</th>
<th>Bent 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Displacement in mm</td>
<td>μΔ</td>
<td>Displacement in mm</td>
</tr>
<tr>
<td>1</td>
<td>0.45</td>
<td>11</td>
<td>0.9</td>
</tr>
<tr>
<td>2</td>
<td>1.00</td>
<td>25</td>
<td>2.1</td>
</tr>
<tr>
<td>3</td>
<td>1.60</td>
<td>41</td>
<td>3.3</td>
</tr>
<tr>
<td>4</td>
<td>2.60</td>
<td>66</td>
<td>5.4</td>
</tr>
<tr>
<td>5</td>
<td>3.70</td>
<td>94</td>
<td>7.7</td>
</tr>
<tr>
<td>6</td>
<td>4.35</td>
<td>111</td>
<td>9.1</td>
</tr>
<tr>
<td>7</td>
<td>5.82</td>
<td>148</td>
<td>12.1</td>
</tr>
</tbody>
</table>
Figure 2-1 Bridge Model Elevation
Figure 2-2 Model Bridge Bent Dimensions
Figure 2-3 Bridge Column Section

Figure 2-4 Post-tensioned Rods for Cap-Beam Deck Connection

Figure 2-5 Inverted T-Beam
Figure 2-6  Abutment Seat
Figure 2-7 Superimposed Mass Arrangement
Fig. 2-8 Full Scale Input Acceleration Records with Compressed Time
Figure 3-1 Steel01 Element

Figure 3-2 Steel02-Hysteretic Behavior with Isotropic Hardening in Tension
Figure 3-3 Concrete01 Material Parameters

Figure 3-4 Typical Hysteretic Stress-Strain Relation of Concrete01 Model
Figure 3-5 Elastic-Perfectly Plastic Gap Material

Figure 3-6 Parameters for Hysteretic Material
Figure 3-7 Definition of Pinching 4 Uniaxial Material Model

Figure 3-8 Fiber Section Segments for a Circular Reinforced Concrete Member
Figure 3-9 Analysis Components in OpenSees
Fig. 4.1 Example of the Prototype Location in a Multi-Span Bridge
Fig. 4.2 2-Span Bridge Model
Fig. 4.3 Bridge Bents
Fig. 4.4 Bridge Nodal Configuration
Fig. 4.5  Superimposed Mass Arrangement
Fig. 4-6 OpenSees Column Displacement Prediction for test 12
Fig. 4-7: OpenSees Column Displacement Prediction for test 13
Fig. 4-8. OpenSees Column Displacement Prediction for test 14.
Fig. 4-9  OpenSees Column Displacement Prediction for test 15
Fig. 4-10 OpenSees Column Displacement Prediction for test 16
Fig. 4.11 OpenSees Column Displacement Prediction for test 17
Fig. 4.12 OpenSees Column Displacement Prediction for test 18

OpenSees Predicted
Measured

Time (sec)

Displacement (in)

Displacement (mm)
Fig. 4-13 OpenSees Column Displacement Prediction for test 19
Fig. 5-1(a) Bridge Nodal Configuration
Fig. 5-1(b) Bridge Bent Nodal Configuration

Fig. 5-2 Column Section Fiber Configuration
Fig. 5-3 Moment-Curvature for Column Section with 8 Slices

Fig. 5-4 Moment-Curvature for Column Section with 16 Slices
Fig. 5-5 Moment-Curvature Comparison for Column Sections with 16 and 8 Slices

Fig. 5-6 Force-Displacement Comparison for Bent 1 with Different Column Element Models
Fig. 5-7 Moment Curvature for Bridge Column Section

Fig. 5-8 Pushover Analysis of Bridge Bents
Fig. 5-9 Post-tensioned Rods for Cap-Beam Deck Connection

![Diagram of post-tensioned rods for cap-beam deck connection]

Fig. 5-10 Moment Curvature Curve for Cap Beam-Deck Connection

![Graph showing moment-curvature relationship]

Fig 5-10  Moment Curvature Curve for Cap Beam-Deck Connection
Fig. 5-11 Force-Displacement Curve for One Foot Length of Prototype Backwall

Fig. 5-12 Force-Displacement Backbone Curve for Backwall in Bridge Model
Fig. 5-13 Bridge Abutment Deck Interaction Models
**Figure 6-1** Bent Pushover Relationship

**Figure 6-2** Pushover for Bent 1
Fig. 6-3  Pushover for Bent 2

Displacement (mm)

Force (kips)

OpenSees Pushover
Idealized

Displacement (in)

Force (kips)

Fig. 6-4  Pushover for Bent 3

Displacement (mm)

Force (kN)

OpenSees Pushover
Idealized

Displacement (in)

Force (kN)
Fig. 6-5 Elastic mode shapes of shake table bridge specimen
Figure 6-6 Bent 1 Displacement in Longitudinal and Transverse Directions

Figure 6-7 Bent 2 Displacement in Longitudinal and Transverse Directions
Figure 6-8 Bent 3 Displacement in Longitudinal and Transverse Directions

Figure 6-9 Bent 1 Displacement
Figure 6-10 Bent 2 Displacement

Figure 6-11 Bent 3 Displacement
Figure 6-12 Bent 1 Resultant Displacement History

Figure 6-13 Bent 2 Resultant Displacement History
Figure 6-14 Bent 3 Resultant Displacement History

Figure 6-15 Abutment Displacement
Figure 6-16 Abutment and Deck-end Displacement in Bridge Longitudinal Direction

Figure 6-17 Abutment-Deck Gap Size History
Figure 6-18 Abutment Gap Element (Actuator) Force

Figure 6-19 Bent 1 Displacement (Model 2)
Figure 6-20 Bent 2 Displacement (Model 2)

Figure 6-21 Bent 3 Displacement (Model 2)
Figure 6-22  Bent 1 Displacement in Longitudinal and Transverse Directions

Figure 6-23  Bent 2 Displacement in Longitudinal and Transverse Directions
Figure 6-24 Bent 3 Displacement in Longitudinal and Transverse Directions

Figure 6-25 Bent 1 Resultant Displacement History (Model 2)
Figure 6-26 Bent 2 Resultant Displacement History (Model 2)

Figure 6-27 Bent 3 Resultant Displacement History (Model 2)
Figure 6-28 Abutment Displacement

Figure 6-29 Abutment-Deck Gap Size History for Seven Levels of Motions (Model 2)
Figure 6-30 Abutment Gap Element Force (Model 2)

Figure 6-31 Bent 1 Resultant Displacement History (Model 1)
Figure 6-32 Bent 2 Resultant Displacement History (Model 1)

Figure 6-33 Bent 3 Resultant Displacement History (Model 1)
Figure 6-34 Comparison between Bent 1 Resultant Displacements in Model 1 & Model 2 during Event 2

Figure 6-35 Comparison between Bent 1 Resultant Displacements in Model 1 & Model 2 during Event 7
Figure 6-36 Comparison between Bent 1 Longitudinal Displacements in Model 1 & Model 2 during Event 2

Figure 6-37 Comparison between Bent 1 Longitudinal Displacements in Model 1 & Model 2 during Event 7
Figure 6-38 Bent 1 Resultant Displacement History (Model 3)

Figure 6-39 Bent 2 Resultant Displacement History (Model 3)
Figure 6-40 Bent 3 Resultant Displacement History (Model 3)

Figure 6-41 Comparison between Bent 1 Resultant Displacements in Model 2 & Model 3 during Event 2
Figure 6-42 Comparison between Bent 1 Resultant Displacements in Model 2 & Model 3 during Event 7

Figure 6-43 Comparison between Bent 2 Resultant Displacements in Model 2 & Model 3 during Event 2
Figure 6-44 Comparison between Bent 2 Resultant Displacements in Model 2 & Model 3 during Event 7

Figure 6-45 Comparison between Bent 3 Resultant Displacements in Model 2 & Model 3 during Event 2
Figure 6-46 Comparison between Bent 3 Resultant Displacements in Model 2 & Model 3 during Event 7

Figure 6-47 Abutment Gap Element Force (Model 3)
APPENDIX A
Bond-Slip Calculation for Bridge Column Ends

Bond-Slip Model
At cracking:

\[
fr = \frac{My}{I} \frac{P}{A}
\]

\[
fr = 7.5\sqrt{5000} = 530.3 \text{ psi}
\]

\[
I = \frac{1\pi d^4}{64} = \frac{\pi (12)^4}{64} = 1018 \text{ in}^4
\]

\[
A = \frac{\pi (12)^2}{4} = 113 \text{ in}^2
\]

\[
P = \frac{53 \times 10^3}{113} = 471 \text{ psi}
\]

\[
M_{cr} = \frac{(530.3 + 471)(1018)}{6} = 169892 \text{ lb-in}
\]

From section analysis (RCMC): \((\varepsilon_s)_{cr} = 0.000153\)

\[
f_s = E_s \varepsilon_s = 29,000(0.000153) = 4.44 \text{ ksi}
\]

For bar #3: \(u = \frac{9.5\sqrt{f_c'}}{d_b} \leq 800 \text{ psi}\)

\[
u = \frac{9.5\sqrt{5000}}{0.375} = 1791 > 800 \Rightarrow u = 800 \text{ psi}
\]

From RCMC: Neutral axis depth \(c=8 \text{ in}\)

\(d=12-(0.5+0.192+0.375/2)=11.1 \text{ in}\)

\(d-c=11.1-8=3.1 \text{ in}\)

\[
\delta = \frac{d_b f_s^2}{8E_s u} = \frac{0.375(4.44)^2}{8(29,000)(0.8)} = 0.00004 \text{ in}
\]

\[
\theta_{cr} = \frac{\delta}{d-c} = \frac{0.00004}{3.1} = 0.0000128 \text{ rad}
\]

At yielding:

\[M_y=563 \text{ k-in} \quad \varepsilon_y = \frac{68,000}{29 \times 10^6} = 0.00234\]

From RCMC: \(c=4.53 \text{ in}\)

\(d-c=11.1-4.53=6.59 \text{ in}\)

\[
\delta = \frac{0.375(68)^2}{8(29,000)(0.8)} = 0.00934 \text{ in}
\]

\[
\theta_y = \frac{0.000964}{6.59} = 0.00142 \text{ rad}
\]

At concrete cover spalling:
From RCMC: 
\[ M = 698k - in \]
\[ \varepsilon_s = 0.00562 \]
\[ f_s = 68.7ksi \]
\[ l_1 = \frac{(f_s - f_y)d_b}{4u} = \frac{(68.7 - 68)0.375}{4(0.8)} = 0.0815 \text{ in} \]
\[ l_2 = \frac{f_s d_b}{4u} = \frac{68(0.375)}{4(0.8)} = 7.97 \text{ in} \]
\[ \delta = \frac{\varepsilon_y l_2}{2} + \frac{(\varepsilon_s + \varepsilon_y)l_2}{2} = \frac{0.00234(7.97)}{2} + \frac{(0.00562 + 0.00234)(0.0815)}{2} = 0.00964 \text{ in} \]

From RCMC: 
\[ c = 3.85 \text{ in} \]
\[ d - c = 11.1 - 3.85 = 7.27 \text{ in} \]
\[ \theta = \frac{0.00964}{7.27} = 0.001326 \text{ rad} \]

At Ultimate:
\[ M = 704.7k - in \]
From RCMC: 
\[ \varepsilon_s = 0.047 \]
\[ f_s = 77.5ksi \]
\[ l_1 = \frac{(77.5 - 68)(0.375)}{4 \times (0.8)} = 1.11 \text{ in} \]
\[ l_2 = 7.97 \text{ in} \]
\[ \delta = 0.00932 + \frac{(0.047 - 0.00234)(1.11)}{2} = 0.036 \text{ in} \]

From RCMC: 
\[ c = 3.58 \text{ in} \]
\[ d - c = 11.1 - 3.58 = 7.53 \text{ in} \]
\[ \theta_{ult} = \frac{0.036}{7.53} = 0.00478 \text{ in} \]

A tri-linear curve based on the average between cracking and yielding, yielding, and ultimate points was used to represent bond-slip behavior of the column ends.
# LIST OF CCEER PUBLICATIONS

<table>
<thead>
<tr>
<th>Report No.</th>
<th>Publication</th>
</tr>
</thead>
</table>


CCEER-99-14 Ahmad M. Itani, Jose A. Zepeda, and Elizabeth A. Ware "Cyclic Behavior of Steel Moment Frame Connections for the Moscone Center Expansion," December 1999.
<table>
<thead>
<tr>
<th>Report No.</th>
<th>Authors</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>CCEER 00-3</td>
<td>McElhaney, B., M. Saiidi, and D. Sanders</td>
<td>“Shake Table Testing of Flared Bridge Columns With Steel Jacket Retrofit,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER-00-3, January 2000.</td>
</tr>
<tr>
<td>CCEER 00-5</td>
<td>Itani, A., and M. Saiidi</td>
<td>“Seismic Evaluation of Steel Joints for UCLA Center for Health Science Westwood Replacement Hospital,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER-00-5, February 2000.</td>
</tr>
<tr>
<td>CCEER 00-6</td>
<td>Will, J. and D. Sanders</td>
<td>“High Performance Concrete Using Nevada Aggregates,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER-00-6, May 2000.</td>
</tr>
<tr>
<td>CCEER 00-7</td>
<td>French, C., and M. Saiidi</td>
<td>“A Comparison of Static and Dynamic Performance of Models of Flared Bridge Columns,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER-00-7, October 2000.</td>
</tr>
<tr>
<td>CCEER 00-8</td>
<td>Itani, A., H. Sedarat</td>
<td>“Seismic Analysis of the AISI LRFD Design Example of Steel Highway Bridges,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER-00-8, November 2000.</td>
</tr>
<tr>
<td>CCEER 00-9</td>
<td>Moore, J., D. Sanders, and M. Saiidi</td>
<td>“Shake Table Testing of 1960’s Two Column Bent with Hinges Bases,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER 00-09, December 2000.</td>
</tr>
<tr>
<td>CCEER 00-10</td>
<td>Asthana, M., D. Sanders, and M. Saiidi</td>
<td>“One-Way Reinforced Concrete Bridge Column Hinges in the Weak Direction,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER 00-10, April 2001.</td>
</tr>
<tr>
<td>CCEER 01-1</td>
<td>Ah Sha, H., D. Sanders, M. Saiidi</td>
<td>“Early Age Shrinkage and Cracking of Nevada Concrete Bridge Decks,” Civil Engineering Department, University of Nevada, Reno, Report No. CCEER 01-01, May 2001.</td>
</tr>
<tr>
<td>Report No.</td>
<td>Title</td>
<td>Authors</td>
</tr>
<tr>
<td>-----------</td>
<td>----------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>CCEER 01-4</td>
<td>Analysis and Retrofit of Fixed Flared Columns with Glass Fiber-Reinforced Plastic Jacketing</td>
<td>Saiidi, M., J. Mortensen, and F. Martinovic</td>
</tr>
<tr>
<td>CCEER 01-5</td>
<td>Performance of A Full-Scale Two-Story Wood Frame Structure Supported on Ever-Level Isolators</td>
<td>Saiidi, M., A. Itani, I. Buckle, and Z. Cheng</td>
</tr>
<tr>
<td>CCEER 01-6</td>
<td>Experimental Study and Analysis of Retrofitted Flexure and Shear Dominated Circular Reinforced Concrete Bridge Columns Subjected to Shake Table Excitation</td>
<td>Laplace, P., D. Sanders, and M. Saiidi</td>
</tr>
<tr>
<td>CCEER 01-7</td>
<td>Removal and Replacement of Cast-in-Place, Post-tensioned, Box Girder Bridge</td>
<td>Reppi, F., and D. Sanders</td>
</tr>
<tr>
<td>CCEER 02-1</td>
<td>Seismic Performance and Retrofitting of Reinforced Concrete Bridge Bents</td>
<td>Pulido, C., M. Saiidi, D. Sanders, and A. Itani</td>
</tr>
<tr>
<td>CCEER 02-2</td>
<td>Influence of Ground Motion Incoherency on Earthquake Response of Multi-Support Structures</td>
<td>Yang, Q., M. Saiidi, H. Wang, and A. Itani</td>
</tr>
<tr>
<td>CCEER 02-3</td>
<td>A Preliminary Study of Shake Table Response of A Two-Column Bridge Bent on Flexible Footings</td>
<td>M. Saiidi, B. Gopalakrishnan, E. Reinhardt, and R. Siddharthan</td>
</tr>
<tr>
<td>CCEER 02-4</td>
<td>Not Published</td>
<td></td>
</tr>
<tr>
<td>CCEER 02-5</td>
<td>Evaluation of Concrete Mixes for Filling the Steel Arches in the Galena Creek Bridge</td>
<td>Banghart, A., Sanders, D., Saiidi, M.</td>
</tr>
<tr>
<td>CCEER 02-6</td>
<td>Cyclic Behavior of Shear Links and Tower Shaft Assembly of San Francisco – Oakland Bay Bridge Tower</td>
<td>Dusicka, P., Itani, A., Buckle, I. G.</td>
</tr>
<tr>
<td>CCEER 02-7</td>
<td>A Performance-Based Design Method for Confinement in Circular Columns</td>
<td>Mortensen, J., and M. Saiidi</td>
</tr>
<tr>
<td>CCEER 03-1</td>
<td>User’s manual for SPMC v. 1.0 : A Computer Program for Moment-Curvature Analysis of Reinforced Concrete Sections with Interlocking Spirals</td>
<td>Wehbe, N., and M. Saiidi</td>
</tr>
<tr>
<td>CCEER 03-2</td>
<td>User’s manual for RCMC v. 2.0 : A Computer Program for Moment-Curvature Analysis of Confined and Unconfined Reinforced Concrete Sections</td>
<td>Wehbe, N., and M. Saiidi</td>
</tr>
</tbody>
</table>


Research, Department of Civil Engineering, University of Nevada, Reno, Nevada, Report No. CCEER-04-6, August 2004.


