

**INFLUENCE OF COLUMN-BASE FIXITY ON  
LATERAL DRIFT OF GABLE FRAMES**

Amber Verma

Thesis submitted to the Faculty of the  
Virginia Polytechnic Institute and State University  
in partial fulfillment of the requirements for the degree of

Master of Science

In

Civil Engineering

Finley A. Charney

---

Carin L. Roberts Wollmann

---

Matthew R. Eatherton

---

May 1, 2012

Blacksburg, Virginia

Keywords: Metal Buildings, Column-Base Connection, Rotational Stiffness, Maximum Bearing Pressure

# **INFLUENCE OF COLUMN-BASE FIXITY ON LATERAL DRIFT OF GABLE FRAMES**

Amber Verma

## **ABSTRACT**

In a typical light metal building, the structural members are designed for the forces and moments obtained from the wind drift analysis, which assumes pinned connections at the base. The pinned connections provide no moment at the base and have zero rotational stiffness. However, in reality every connection provides some restraint and has some rotational stiffness. Hence, by considering a modeling assumption of pinned condition, the actual behavior of the connection is not captured and this results in overestimation of lateral drifts and appearance of larger moments at the knee of the gable frames. Since the structural components are designed on the basis of these highly conservative results, the cost of the project increases. This thesis investigates the real behavior of the column base connection and tries to reduce the above stated conservatism by developing a computer program or “wizard” to calculate the initial rotational stiffness of any column base connection.

To observe the actual behavior of a column base connection under different load cases, a number of finite element models were created in SAP2000. Each finite element model of the column base connection contained base plate, column stub, anchor bolts and in some cases grout as its components. The model was mainly subjected to three load cases, namely gravity, wind and gravity plus wind. After performing many analyses, the influence of flexibility of each component on the flexibility of the connection was observed and a list of parameters was created. These parameters are the properties of above mentioned components which characterizes any column base connection. These parameters were then used as inputs to model any configuration of the column base connection in the developed wizard. The wizard uses OpenSees and SAP2000 to analyze the modeled configuration of the connection and provides values of the initial rotational stiffness and maximum bearing pressure for the provided loads. These values can be further used in any structural analysis which is done to calculate the lateral drift of a frame under lateral loads. This will also help in getting results which are less conservative than the results which one gets on assuming pinned condition at the base.

## **Acknowledgements**

I would like to thank Dr. Finley A Charney for being my advisor and committee chair. I really appreciate the fact that he provided a challenging work environment which made me work harder to complete this research successfully. I am really glad that I got an opportunity to work under his tutelage. I have learned lots of things from him as a student and person. I would like to thank Dr. Carin L. Roberts Wollmann for being one of my committee members and amazing teacher. I would also like to thank Dr. Matthew R. Eatherton for serving as one of my committee members and providing valuable information throughout my stay at Virginia Tech.

I would like to thank the Metal Building Manufacturers Association for sponsoring this research project. I would like to express my sincere gratitude to Dr. Lee Shoemaker, who has served as the main contact person from MBMA and provided me valuable information time to time.

I would like to thank Ms. Lindy Cranwell and Ms. Ann W. Crate for providing excellent support to the graduate students.

The research reported in this thesis could not have been possible without the constant support and feedback from Ms. Jeena Rachel Jayamon, graduate student at Virginia Tech. Special thanks to Mr. David Padilla, doctoral student at Virginia Tech for his constant criticisms, which helped me improving the performance of the wizard.

I would specially want to thank my parents, Mr. Dev Kumar Verma and Mrs. Amita Verma for being really supportive of my decisions and actions. Without their patience, sacrifices, love and blessings, I could not have completed this research. I would also like to thank my one and only sister, Ms. Avani Verma for her constant boost and support without which I would not have come this far.

# TABLE OF CONTENTS

Chapter 1 Introduction.....	1
1.1 Introduction .....	1
1.2 Typical Exposed Column Base Connection.....	2
1.3 Column-base connection behavior under flexure.....	3
1.4 Procedure for calculating the rotational stiffness of the connection .....	4
Chapter 2 Literature Review .....	7
2.1 Introduction .....	7
2.2 Design of Axially Loaded Base Plate .....	7
2.2.1 Cantilever Model .....	7
2.2.2 Fling Model .....	8
2.2.3 Murray-Stockwell Model .....	8
2.2.4 Thornton Model.....	9
2.2.5 Eurocode 3 Model .....	9
2.3 Analytical Procedures to calculate the Rotational Stiffness.....	10
2.3.1 Analytical Procedures proposed by Galambos.....	10
2.3.2 Component Method .....	11
2.3.3 Mechanical Model proposed by Jaspart and Vandegans.....	14
2.4 Tests and Experimental Studies on the Column-Base Connection .....	15
2.5 Finite Element Models .....	17
2.6 Stiffness calculations from some research studies .....	19
2.6.1 Geometry and Material Used (same for both the tests) .....	19
2.6.2 Flexure test by Picard and Beaulieu (1985).....	20
2.6.3 Compression-Flexure test by Picard and Beaulieu (1985).....	21
Chapter 3 Testing of Finite Elements .....	23
3.1. Introduction .....	23
3.2 Base Plate Testing .....	23
3.2.1 Physical significance of force-deformation relationships .....	26
3.2.2 Load Application .....	26

3.2.3 Observations .....	33
3.3 Grout Testing.....	33
3.3.1 Observations .....	35
Chapter 4 Finite Element Modeling .....	36
4.1 Introduction .....	36
4.2 Development of Finite Element Models .....	36
Chapter 5 Accessing Different Programs from MATLAB .....	42
5.1 Introduction .....	42
5.2. Running SAP2000 through MATLAB .....	42
5.3. Modeling in OpenSees .....	44
5.3.1 Executing OpenSees Commands.....	45
5.3.2 Running OpenSees through MATLAB .....	48
Chapter 6 Graphical User Interface (GUI) Development.....	50
6.1. Introduction .....	50
6.2 GUI Development Process in MATLAB .....	52
6.2.1 Input Panels .....	55
6.2.1.1 Column Properties .....	55
6.2.1.2 Grout Properties .....	56
6.2.1.3 Base Plate Properties.....	56
6.2.1.4 Material Properties.....	56
6.2.1.5 Bolt Properties .....	56
6.2.1.6 Load Application .....	56
6.2.2 Menu Bar .....	57
6.2.2.1 File .....	57
6.2.2.2 View .....	58
6.2.2.3 Mesh.....	58
6.2.2.4 Analysis.....	59
6.2.3. Action Buttons.....	60
6.2.4. Display Screen and Output Windows.....	61
6.2.4.1 Moment Rotation Plot.....	61
6.2.4.2 Bearing Pressure Distribution Plot.....	62

6.2.4.3 Report.....	63
6.3 Batch Mode Operation .....	64
Chapter 7 Parametric Studies .....	66
7.1 Factors affecting the Stiffness of the Connection .....	66
7.1.1 Axial Load .....	67
7.1.2 Thickness of Base Plate .....	69
7.1.3 Bolt Size .....	70
7.1.4 Number of Bolts .....	72
7.1.5 Effective Length of Anchor Bolts .....	74
7.1.6 Location of Bolts .....	76
7.1.7 Column Size .....	78
7.1.8 Presence of Grout .....	80
7.2 Effects of Tapered Column Section on the Stiffness of the Connection.....	82
7.3 Effect of Pretensioned Bolts on the Stiffness of the Connection .....	85
7.4 Effect of Meshing on the Stiffness of the Connection .....	87
7.5 Analysis Engine: SAP2000 vs OpenSees.....	89
7.6 Comparative Studies .....	90
7.6.1 Comparison with Picard and Beaulieu (1985).....	91
7.6.1.1 Comparison with Flexure test of Picard and Beaulieu (1985).....	91
7.6.1.2 Comparison with Compression-Flexure test of Picard and Beaulieu (1985) .....	93
Chapter 8 Conclusions.....	95
8.1 Scope for Future Research .....	96
REFERENCES .....	99
APPENDIX A: Stiffness Calculations .....	101
APPENDIX B: MBMA SURVEY .....	105
APPENDIX C: USER’S MANUAL .....	110
C.1.1 Introduction .....	110
C.1.2 Computer System Requirements .....	111
C.1.3 Instructions for Installing the Wizard in Windows 7 .....	111
C.3.1 Instructions for Installing the GUI Mode.....	112
C.1.3.2 Instructions for installing the Batch Mode.....	123

C.1.4 Instructions for Uninstalling the Wizard from Windows 7 .....	126
C.2.1 Functioning of the GUI Mode .....	127
C.2.2 Major Components in GUI .....	128
C.2.2.1 Input Panels .....	129
C.2.2.1.1 Column Properties .....	129
C.2.2.1.1.1 W Section Type .....	129
C.2.2.1.1.2 Column Length .....	132
C.2.2.1.1.3 Steel Yield Strength .....	132
C.2.2.1.2 Grout Properties .....	132
C.2.2.1.2.1 Thickness .....	133
C.2.2.1.2.2 Modulus of Elasticity .....	133
C.2.2.1.2.3 Compressive Strength .....	133
C.2.2.1.3 Base Plate Properties .....	133
C.2.2.1.3.1 Length .....	133
C.2.2.1.3.2 Width .....	133
C.2.2.1.3.3 Thickness .....	133
C.2.2.1.3.4 Steel Yield Strength .....	134
C.2.2.1.4 Material Properties .....	134
C.2.2.1.4.1 Steel: Modulus of Elasticity .....	134
C.2.2.1.4.2 Steel: Poisson Ratio .....	134
C.2.2.1.4.3 Concrete: Poisson Ratio .....	134
C.2.2.1.5 Bolt Properties .....	134
C.2.2.1.5.1 Standard Bolt Diameter .....	135
C.2.2.1.5.2 Bolt Area .....	135
C.2.2.1.5.3 Anchor bolt length .....	135
C.2.2.1.5.4 Percentage Effective length .....	135
C.2.2.1.5.5 Pretension Force .....	136
C.2.2.1.5.6 Bolt Position .....	136
C.2.2.1.6 Load Application .....	136
C.2.2.1.6.1 Dead Load (Gravity Load) .....	137
C.2.2.1.6.2 Live load (Wind Load) .....	137

C.2.2.2 Action Buttons .....	138
C.2.2.2.1 Preview Model .....	138
C.2.2.2.2 Analyse.....	139
C.2.2.2.3 Unlock .....	139
C.2.2.2.4 Reset.....	140
C.2.2.2.5 Save Data.....	140
C.2.2.2.6. Load Data .....	140
C.2.2.3 Menu Bar .....	141
C.2.2.3.1 File.....	141
C.2.2.3.1.1 New.....	142
C.2.2.3.1.2 Open.....	142
C.2.2.3.1.3 Save As .....	142
C.2.2.3.1.4 Close .....	142
C.2.2.3.2 View .....	142
C.2.2.3.2.1 Original View .....	143
C.2.2.3.2.2 X-Y view .....	143
C.2.2.3.2.3 Y-Z view.....	143
C.2.2.3.2.4 X-Z view.....	143
C.2.2.3.3 Mesh.....	143
C.2.2.3.3.1 Coarse Mesh .....	144
C.2.2.3.3.2 Fine Mesh .....	144
C.2.2.3.4 Analysis .....	144
C.2.2.3.4.1 SAP2000.....	145
C.2.2.4. Display Screen and Output Windows .....	145
C.2.2.4.1 Moment Rotation Plot .....	146
C.2.2.4.1 Bearing Pressure Distribution .....	147
C.2.2.4.3 Report.....	148
C.3.1 Functioning of the Batch Mode .....	150
C.3.2 Batch Mode Operation.....	150
C.4 Solved Examples .....	156
C.4.1. Example 1 .....	156



C.4.2. Example 2 .....	164
C.4.3. Example 3 .....	171

# LIST OF FIGURES

Figure 1.1. Variation of lateral displacement of a frame with rotational stiffness at the column base .....	1
Figure 1.2. A typical configuration of column base connection.....	2
Figure 1.3. Column base behavior under flexure.....	3
Figure 1.4. Initial and final position of the base plate under flexure .....	4
Figure 1.5. Typical moment rotation curve for flexural load .....	5
Figure 1.6. Column base represented by rotational spring which has a unique moment-rotation response.....	6
Figure 2.1. Equivalent rigid area for Cantilever Model.....	7
Figure 2.2. Yield Lines for Fling Model.....	8
Figure 2.3. Equivalent rigid area shown by shaded region for Murray-Stockwell Model .....	9
Figure 2.4. Eurocode representation of bearing stress on an equivalent rigid plate .....	9
Figure 2.5. Configuration of the base connection considered for applying the component method .....	12
Figure 2.6. An equivalent T-stub Model.....	13
Figure 2.7. Mechanical Model for representing the column base connection (Jaspert and Vandegan, 1998).....	15
Figure 2.8. Finite Element Model (Hamizi and Hannachi 2007) .....	18
Figure 2.9. Anchor bolts positions for tests performed by Picard and Beaulieu (1985).....	20
Figure 2.10. Values of stiffness from different sources on the moment-rotation plot of flexural test.....	21
Figure 2.11. Values of stiffness from different sources on a moment-rotation plot for compression-flexural load.....	22
Figure 3.1. Plan view of the base plate in SAP2000 or OpenSees .....	24
Figure 3.2. Modeled configuration of the base plate in SAP2000 and OpenSees .....	24
Figure 3.3. Linear force-deformation relationship assigned to spring elements.....	25
Figure 3.4. Mild Nonlinear force-deformation relationship assigned to spring elements .....	25
Figure 3.5. Full Nonlinear force-deformation relationship assigned to spring elements.....	25
Figure 3.6. Base Plate subjected to gravity load case .....	27

Figure 3.7. Base Plate subjected to moment load case .....	27
Figure 3.8. Base Plate subjected to gravity loads first in gravity+moment load case .....	28
Figure 3.9. Base Plate subjected to moment loads after gravity loads in gravity+moment load case.....	28
Figure 3.10. Deformation of base-plate under gravity load and linear springs .....	29
Figure 3.11. Deformation of base-plate under gravity load and mild nonlinear springs .....	29
Figure 3.12. Deformation of base-plate under gravity load and full nonlinear springs.....	30
Figure 3.13. Deformation of base-plate under moment load and linear springs.....	30
Figure 3.14. Deformation of base-plate under moment load and mild nonlinear springs .....	31
Figure 3.15. Deformation of base-plate under moment load and full nonlinear springs .....	31
Figure 3.16. Deformation of base-plate under gravity+moment load and linear springs.....	32
Figure 3.17. Deformation of base-plate under gravity+moment load and mild nonlinear springs	32
Figure 3.18. Deformation of base-plate under gravity+moment load and full nonlinear springs.	33
Figure 3.19. Modeled grout for testing under gravity loads in SAP2000 and OpenSees .....	34
Figure 3.20. Deformations along the length of the grout under gravity load .....	34
Figure 4.1. First model consisted of Column Section, Base Plate and Nlink Elements .....	37
Figure 4.2. Second Model consisted of Column Section, Base Plate, Grout and Nlink Elements	37
Figure 4.3. Force-Deformation relationship for compression Nlink elements .....	38
Figure 4.4. Assumed bond stress distribution along anchor rod length.....	39
Figure 4.5. Force-Deformation relationship for tension Nlink elements.....	40
Figure 5.1. SAP2000 API commands defined in MATLAB editor window .....	43
Figure 5.2. OpenSees Interactive Window .....	45
Figure 5.3. Method 1: Defining model at OpenSees Prompt.....	46
Figure 5.4. Method 2: Input file ‘sample.tcl’ is created.....	46
Figure 5.5. Calling input file ‘sample.tcl’ from OpenSees command prompt.....	47
Figure 5.6. MATLAB script executing modeling and analysis commands in OpenSees.....	49
Figure 6.1. A GUI performing task of a simple calculator .....	51
Figure 6.2. Display Window of Column Base Connection Wizard .....	54
Figure 6.3. Major Components of Column Base Connection Wizard.....	54
Figure 6.4. W Section Type list box in Column properties panel.....	55
Figure 6.5. Options provided in File submenu .....	57

Figure 6.6. Options provided in View submenu .....	58
Figure 6.7. Options provided in Mesh submenu .....	59
Figure 6.8. Options provided in Analysis submenu.....	60
Figure 6.9. A typical moment-rotation plot obtained from the wizard.....	62
Figure 6.10. Typical layout of Bearing Pressure Distribution.....	63
Figure 6.11. A typical report which is displayed by the wizard as one of the outputs .....	64
Figure 6.12. A batch mode input file .....	65
Figure 7.1. Test Configuration for observing effect of Axial Load on the Connection Stiffness ..	67
Figure 7.2. Moment Rotation Curves for different Compressive Loads on the Connection .....	68
Figure 7.3. Zoom in view of Figure 7.2.....	69
Figure 7.4. Moment Rotation Curves for different Base Plate Thicknesses.....	70
Figure 7.5. Zoom in view of Figure 7.4.....	70
Figure 7.6. Moment Rotation Curves for different Bolt Sizes.....	71
Figure 7.7. Zoom in view of Figure 7.6.....	72
Figure 7.8. Moment Rotation Curves for different number of Bolts .....	74
Figure 7.9. Zoom in view of Figure 7.8.....	74
Figure 7.10. Moment Rotation Curves for different percentages of effective length.....	75
Figure 7.11. Zoom in view of Figure 7.10.....	76
Figure 7.12. Positions of bolts inside the flanges of the Column .....	77
Figure 7.13. Positions of bolts outside the flanges of the Column .....	77
Figure 7.14. Moment Rotation Curves for two different configurations of Bolts .....	78
Figure 7.15. Zoom in view of Figure 7.14.....	78
Figure 7.16. Moment Rotation Curves for different Column Sections.....	79
Figure 7.17. Zoom in View of Figure 7.16.....	80
Figure 7.18. Moment Rotation Curves for the Connection with Grout and without Grout.....	81
Figure 7.19. Zoom in view of Figure 7.18.....	81
Figure 7.20. Knee of Gable Frame.....	82
Figure 7.21. Point of application of load in the wizard in case of tapered section .....	83
Figure 7.22. Moment Rotation Curves for Tapered Sections with different Web depths at the Knee Joint .....	84
Figure 7.23. Zoom in view of Figure 7.22.....	84

Figure 7.24 Uplift of the base plate of the connection under flexure load .....	85
Figure 7.25. Moment Rotation Curves for connection with and without pretensioned bolts.....	86
Figure 7.26. Zoom in view of Figure 7.25.....	86
Figure 7.27. Moment Rotation Curves for Fine and Coarse Meshing.....	88
Figure 7.28. Zoom in view of Figure 7.27 .....	88
Figure 7.29. Moment Rotation Curves for OpenSees and SAP2000.....	89
Figure 7.30. Zoom in view of Figure 7.29 .....	90
Figure 7.31. Values entered for different parameters as taken by Picard and Beaulieu for flexure test.....	92
Figure 7.32. Values of stiffness obtained from different sources plotted on the moment-rotation plot of flexural test.....	92
Figure 7.33. Values entered for different parameters as taken by Picard and Beaulieu for compression-flexural test .....	93
Figure 7.34. Moment Rotation Curves for compression-flexure test obtained from the Test and the Wizard.....	94
Figure 8.1. Reactions on the base of a gable frame subjected to a lateral load.....	97
Figure C.1. A typical configuration of column base connection .....	110
Figure C.2. 'Connection_Wizard' folder .....	112
Figure C.3. Folder containing installation package for Column Base Connection Wizard.....	113
Figure C.4. 'GUI_pkg' is executed by right clicking and choosing 'Run as administrator'.....	113
Figure C.5. MCR installation instruction described by Step 4 .....	114
Figure C.6. MCR installation instruction described by Step 5 .....	115
Figure C.7. MCR installation instruction described by Step 6 .....	115
Figure C.8. MCR installation instruction described by Step 7 .....	116
Figure C.9. MCR installation instruction described by Step 8 .....	116
Figure C.10. MCR installation instruction described by Step 8 .....	117
Figure C.11. MCR installation instruction described by Step 8 .....	117
Figure C.12. MCR installation instruction described by Step 9 .....	118
Figure C.13. Tcl/Tk installation instruction described by Step 10 .....	119
Figure C.14. Tcl/Tk installation instruction described by Step 11 .....	120

Figure C.15. Tcl/Tk installation instruction described by Step 11 .....	120
Figure C.16. Tcl/Tk installation instruction described by Step 12 .....	121
Figure C.17. Tcl/Tk installation instruction described by Step 12 .....	121
Figure C.18. Execution of 'GUI.exe' to run the wizard.....	122
Figure C.19. Column Base Connection Wizard pops up as user clicks on GUI.exe .....	123
Figure C.20. Folder containing installation package for Batch Mode.....	124
Figure C.21. MCR installation screen.....	124
Figure C.22. 'Batch Mode' folder after the expansion of batch_pkd .....	125
Figure C.23. Orientation of the base plate and column section in the wizard .....	127
Figure C.24. Main GUI window for Column-Base Connection Wizard .....	128
Figure C.25. Components of main GUI.....	129
Figure C.26. GUI window showing selection of W8x35 section from the list of W sections....	130
Figure C.27. GUI showing the option provided in W section type list box for creating a Tapered Section.....	131
Figure C.28. GUI window for creating a Tapered Section.....	131
Figure C.29. Linear variation of bond stress between the concrete and the steel.....	135
Figure C.30. Position of Bolts on the Base Plate .....	136
Figure C.31. The directions of loads applied at point A of the Connection .....	137
Figure C.32. Model display on execution of 'Preview Model' button .....	138
Figure C.33. Meshed Model of designed connection which is analyzed in SAP2000 and OpenSees.....	139
Figure C.34. A dialogue box appears on clicking 'Save Data' button.....	140
Figure C.35. A dialogue box appears on clicking 'Load Data' button .....	141
Figure C.36. Sub-menu options under File menu .....	142
Figure C.37. Sub-menu options under View menu.....	143
Figure C.38. Sub-menu options under Mesh menu .....	144
Figure C.39. Sub-menu options under Analysis menu .....	145
Figure C.40. Moment Rotation Curve obtained from the wizard.....	147
Figure C.41. Typical layout of Bearing Pressure Distribution .....	148
Figure C.42. A typical report which is displayed by the wizard.....	149
Figure C.43. 'Batch Mode' folder.....	150

Figure C.44. <i>Batch.txt</i> , Input file for the batch mode operation of the wizard.....	151
Figure C.45. <i>Instructions.txt</i> file provides annotative explanation for <i>Batch.txt</i> .....	152
Figure C.46. <i>batch.exe</i> is an executable file, which on clicking execute analysis .....	152
Figure C.47. An output file, <i>Results.txt</i> from batch mode operation of the wizard .....	153
Figure C.48. <i>Batch_Tapered.txt</i> , the input file for tapered column batch mode operation .....	154
Figure C.49. <i>Batch_Tapered.txt</i> and <i>batch_tapered.exe</i> .....	155
Figure C.50. Positions of Bolts on the Base-Plate .....	157
Figure C.51. Wizard window with all the properties entered .....	157
Figure C.52. W6X25 selected from W section type drop down list box .....	158
Figure C.53. Coordinates of bolts entered in Bolts Position Panel.....	159
Figure C.54. Display Screen showing the preview of the Modeled Connection .....	160
Figure C.55. Wizard Window after performing the analysis and giving back the results .....	161
Figure C.56. Moment-Rotation plotted by the Wizard .....	161
Figure C.57. Bearing Pressure Distribution on the Base Plate plotted by the Wizard .....	162
Figure C.58. Saving entered data in a file.....	163
Figure C.59. Loading earlier saved data in the Wizard .....	163
Figure C.60. Positions of Bolts on the Base-Plate .....	165
Figure C.61. Tapered Section is chosen from W Section Type list box to model Tapered Section .....	166
Figure C.62. GUI for creating desired tapered section for the Column Stub .....	166
Figure C.63. Display Screen showing the preview of the Modeled Connection .....	168
Figure C.64. Wizard Window after performing the analysis and giving back the results .....	169
Figure C.65. Moment-Rotation for Tapered Column Stub plotted by the Wizard .....	169
Figure C.66. Bearing Pressure Distribution on the Base Plate plotted by the Wizard .....	170
Figure C.67. Input file for running batch mode is labeled as <i>Batch.txt</i> .....	171
Figure C.68. <i>batch.exe</i> is an executable file, which on clicking execute analysis .....	172
Figure C.69. Output file, <i>Results.txt</i> .....	172

# LIST OF TABLES

Table 2-1. Values of Initial rotational stiffness obtained from different sources for the flexure test performed by Picard and Beaulieu (1985).....	21
Table 2-2. Values of Initial rotational stiffness obtained from different sources for the compression-flexure test performed by Picard and Beaulieu (1985).....	22
Table 3-1. Finite Element used in SAP2000 and OpenSees to model different components of the connection .....	23
Table 6-1. Input parameters considered while designing GUI .....	52
Table 7-1. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Compressive Loads on the Connection.....	68
Table 7-2. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Base Plate Thicknesses .....	69
Table 7-3. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Bolt Sizes .....	71
Table 7-4. Different bolt configurations considered for observing the effect of number of bolts on the stiffness of the connection.....	73
Table 7-5. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different number of Bolts.....	73
Table 7-6. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for different percentages of effective length.....	76
Table 7-7. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for two different configurations of Bolts.....	77
Table 7-8. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Column Sections .....	79
Table 7-9. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for with and without Grout .....	80
Table 7-10. Values of Stiffness and Maximum Bearing Pressure for Tapered Sections.....	83
Table 7-11. Values of Stiffness and Maximum Bearing Pressure for the connection with and without Pretensioned Bolts .....	86
Table 7-12. Values of Stiffness and Maximum Bearing Pressure for Coarse and Fine Meshing ..	87



Table 7-13. Values of Stiffness and Maximum Bearing Pressure for OpenSees and SAP2000 ....	90
Table 7-14. Values of Initial rotational stiffness obtained from different sources for the flexure test performed by Picard and Beaulieu (1985) .....	93
Table 7-15. Values of Initial rotational stiffness obtained from different sources for the compression-flexure test performed by Picard and Beaulieu (1985) .....	94
Table B-1 Base Plate Dimensions.....	105
Table B-2. Anchor Bolts Details.....	105
Table B-3. Installation Details .....	106
Table B-4. Column Design Details .....	107
Table B-5. Foundation Design Details.....	109
Table C-1. Computer System Requirements .....	111

# Chapter 1 Introduction

## 1.1 Introduction

In typical analysis of a light metal building, the column-base connections are modeled as pinned connections. However, by considering such a modeling assumption, the actual behavior of the connection is not captured and results in overestimation of lateral drifts and appearance of larger moments at the knee of the gable frames. Bajwa (2010) showed the effect of the rotational stiffness present at the column base on the resulting lateral displacements due to wind loads (Figure 1.1). It can be observed from the plot shown in Figure 1.1 that a pinned support condition results in calculation of lateral displacement ( $\Delta_0$ ), which is on the higher side, while a fixed support condition results in lower displacement ( $\Delta_\infty$ ). In actuality, the stiffness of the connection exists in between the two extremes (pinned and fixed) and is represented by  $K_\theta$  in Figure 1.1. If the actual stiffness ( $K_\theta$ ) is used for the calculation, then one may find that the lateral displacement ( $\Delta_\theta$ ) falls on the curved part of the plot. The presence of a definite stiffness makes every connection partially restrained. The purpose of this research is to study this partially restrained behavior of the column base connection and develop a computer program or “wizard” for estimating the stiffness of any configuration of the column base connection.

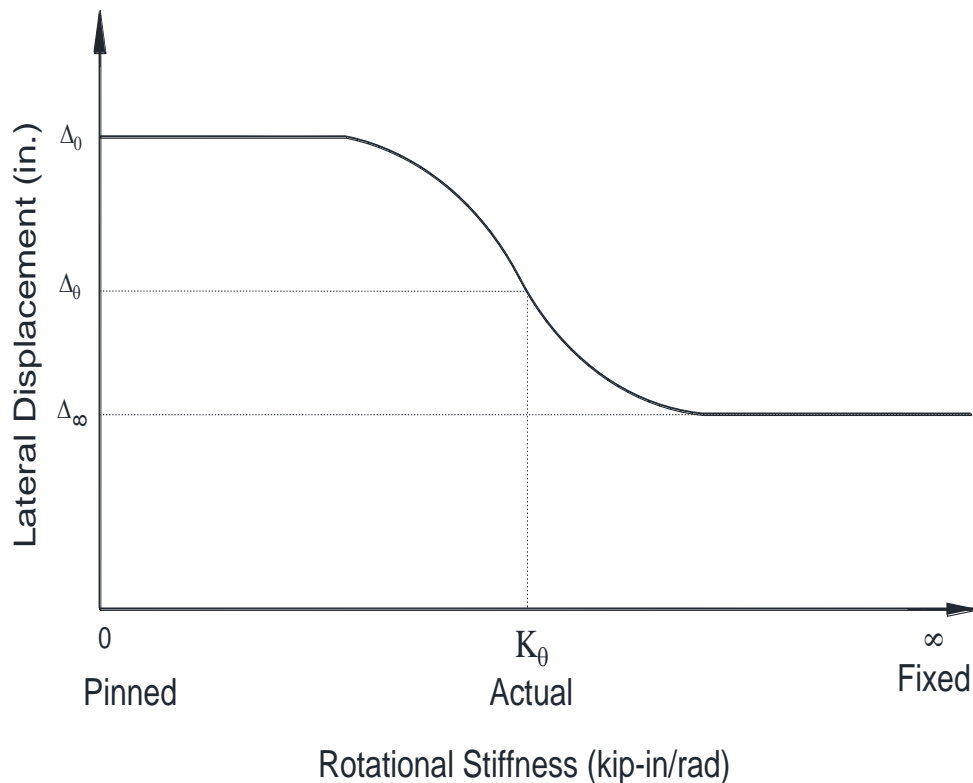


Figure 1.1. Variation of lateral displacement of a frame with rotational stiffness at the column base

## 1.2 Typical Exposed Column Base Connection

A typical exposed column base connection consists of a column and base plate assembly, erected on top of a concrete footing (shown in Figure 1.2). Usually, a concrete grout is also provided between the assembly and footing, however this may not be applicable to the Metal Building Manufacturers Association (MBMA) construction practices. Anchor bolts connect the whole assembly to the footing. Generally, the base plate is welded to the column to prevent any slip due to applied loads. The column can be subjected to any of the three types of loading - axial force, shear force and moment represented by  $P$ ,  $V$  and  $M$ , respectively in Figure 1.2. The resulting deformation under each loading type is contributed by column deformation, grout deformation, base plate bending, anchor bolts deformation and footing deformation. The contribution of each of the above deformations to the resulting deformation of the connection depends upon stiffness of each component constituting the connection. The research reported herein takes account of all of the above stated sources of deformation, except for the footing.

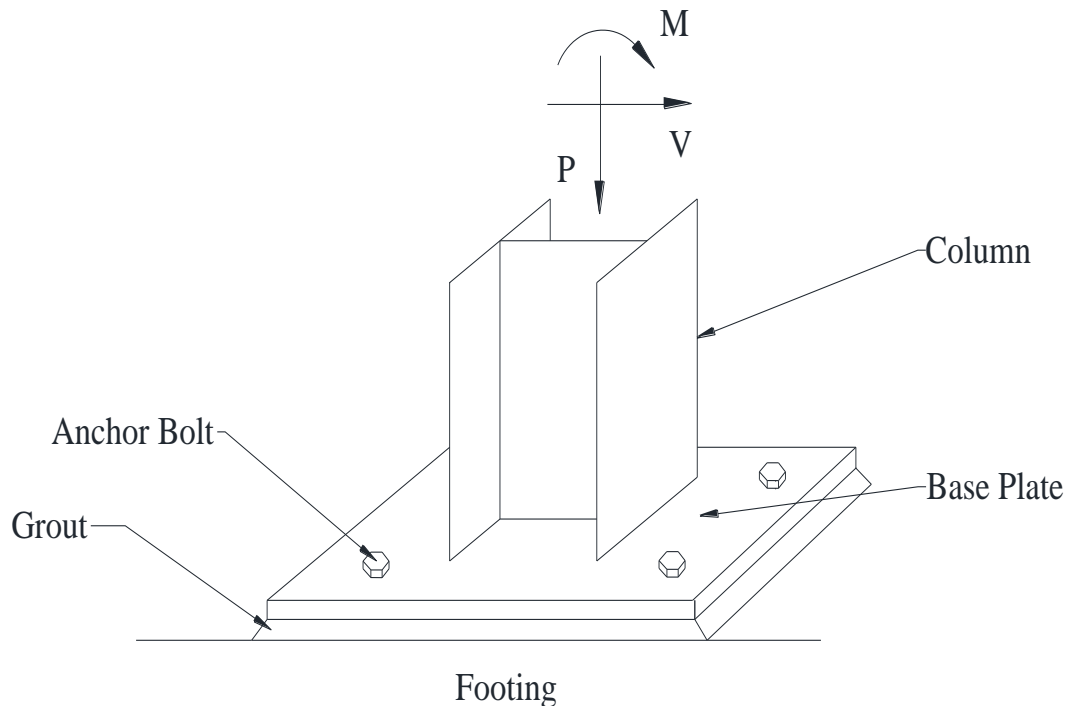


Figure 1.2. A typical configuration of column base connection

These connections are generally subjected to two load cases – gravity and wind. Both gravity and wind load case contain axial loads, shear loads and moments. The gravity load case includes the loads coming from weight of the structural components such as rafter, metal deck, purlins, etc. While performing an analysis, the connection is first subjected to gravity load case and then the wind load case is applied. In other words, these load cases must be applied sequentially (rather than using superposition) because of the base plate uplift which is a nonlinear behavior.

### 1.3 Column-base connection behavior under flexure

Flexure loading has a very profound effect on the column base connection behavior and it's usually accompanied by the application of axial loads, mostly compressive, and occasionally tensile (or uplift). When a moment 'M' is applied on the column base connection, then the column and base plate try to rotate together as they are welded to each other. In doing so, the base plate lifts up and loses contact from one side. Anchor bolts are provided to resist this uplift, and in doing so, the anchor bolts present on the uplift side undergo extension. The base plate experiences tension on the uplifting side and compression on the other due to its restricted deformation. This behavior is shown in Figure 1.3, where the left side of the base plate is under tension and lifts up, while the right side of the base plate is under compression and exerts compressive force on the grout and footing. One may also observe other interactions occurring along with bending of base plate such as concrete in compression, column flange in compression, column web in compression and anchor bolts in tension. The combined effect of these interactions and the stiffness (flexibility) of each component contribute to the stiffness (flexibility) of the whole connection. It should be noted that the research presented herein focuses only on estimating the initial rotational stiffness of the column base connection. However, it should be remembered that in addition to the rotational stiffness, the connection also has axial, shear and torsion stiffness.

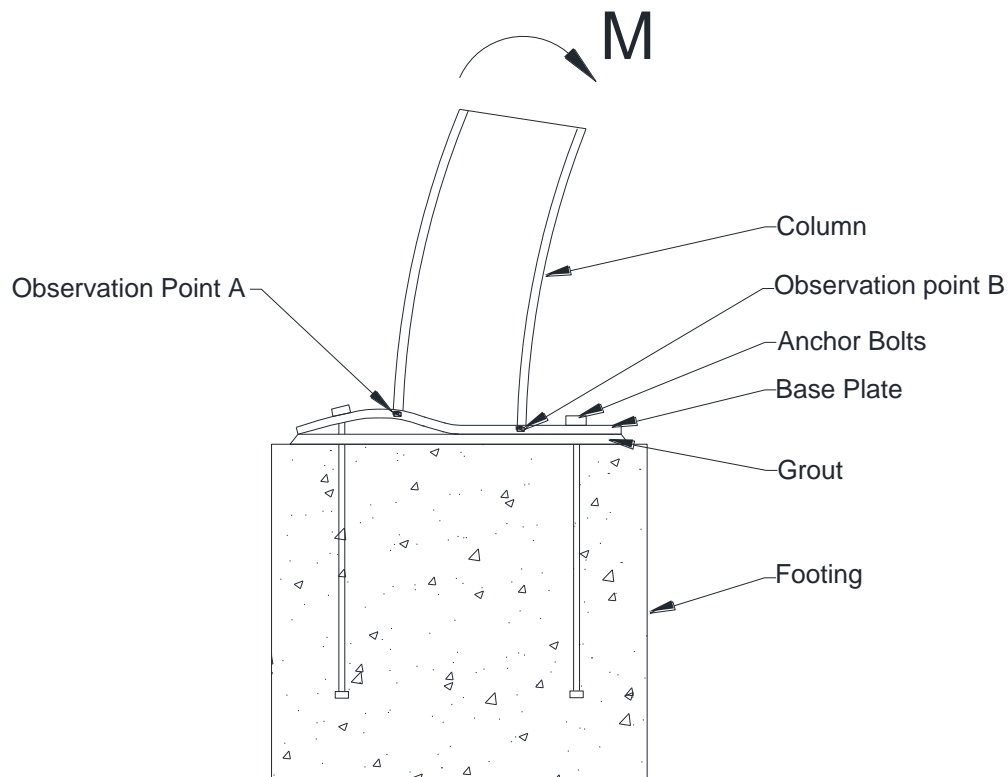


Figure 1.3. Column base behavior under flexure

## 1.4 Procedure for calculating the rotational stiffness of the connection

Under an application of lateral load or moment, the base connections of a frame structure are subjected to flexure. The rotational stiffness at the base connection directly affects the amount of drift which a frame will experience on application of flexural loads. The behavior of the connection under flexure is described in the previous section. Essentially, the whole connection assembly tries to rotate on application of a moment. In this thesis, this rotation is quantified to create a moment rotation plot for each configuration of the base connection. Two observation points are taken on the mid surface of the base plate, right beneath the flanges of the column, represented by A and B in Figure 1.3. Figure 1.4 shows magnified view of mid surface of the base plate and the two points, A and B, on it. Here subscript 'i' corresponds to the initial position of a point (position before the application of the moment) and subscript 'f' represents the final position of a point (position after the application of the moment).

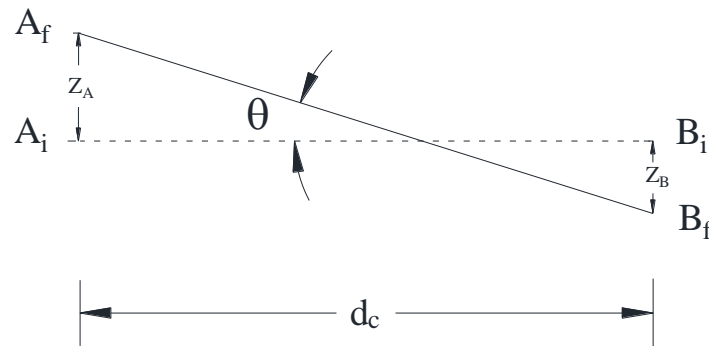


Figure 1.4. Initial and final position of the base plate under flexure

When a clockwise moment is applied on the connection (shown in Figure 1.3), point A is displaced from position  $A_i$  to position  $A_f$  by displacement  $Z_A$ , while point B is displaced from position  $B_i$  to position  $B_f$  by displacement  $Z_B$  (Figure 1.4). The rotation of the connection can be calculated by the following relation.

$$\theta = \frac{(Z_A + Z_B)}{d_c} \quad (1-1)$$

where  $\theta$  is rotation of the connection due to applied moment,  $Z_A$  and  $Z_B$  are 'magnitude' of the displacements of point A and point B respectively and  $d$  is depth of the column. Please note that in this research the rotation is considered positive when the line joining  $A_f$  and  $B_f$  rotates clockwise with respect to the line joining  $A_i$  and  $B_i$ .

By making use of Equation (1-1), one value of rotation can be obtained for each value of the moment applied, which can be designated as  $(M, \theta)$  pair. To capture the complete behavior of the connection, one should obtain as many of these pairs as possible by varying the applied moment.

These pairs can be used to plot a moment-rotation relationship for a particular connection. A typical moment-rotation relationship for flexural loading is shown in Figure 1.5. It should be noted that the plot shown in Figure 1.5 considers the occurrence of various plastic mechanisms in the connection during the loading process. These plastic mechanisms include yielding of the base plate, crushing of the concrete and yielding and rupturing of the anchor bolts. However, in the present research study and in the development process of the wizard for this project, these mechanisms are not taken into account and the focus is more on finding the initial rotational stiffness of the connection, where each component in the connection assembly behaves elastically under service loads.

In this research, the initial slope of any moment rotation curve represents the initial rotational stiffness of the connection. Figure 1.5 shows the initial rotational stiffness represented by  $K_{\theta}$  for a moment-rotation plot.

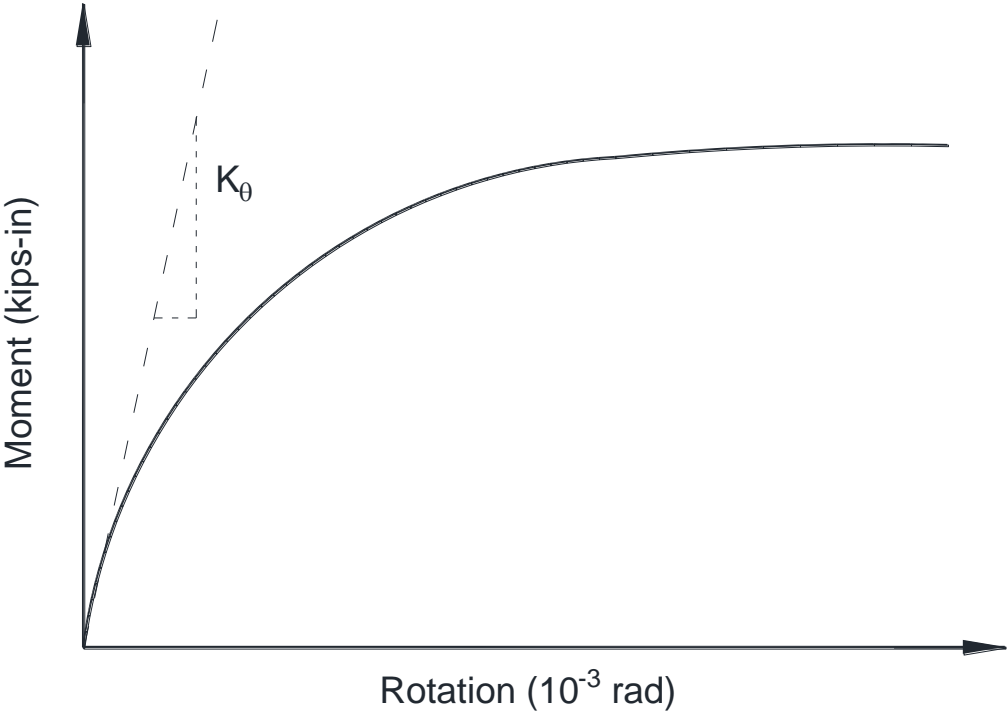


Figure 1.5. Typical moment rotation curve for flexural load

The moment-rotation response can be treated as a unique characteristic of each configuration of the connection. For modeling the restraint condition of an actual column base connection, a rotational spring should be assigned at the column base and the moment-rotation response of the connection should be assigned as the response of this spring. Figure 1.6 shows the above way of modeling the column base connection which represents a partially restrained support condition.

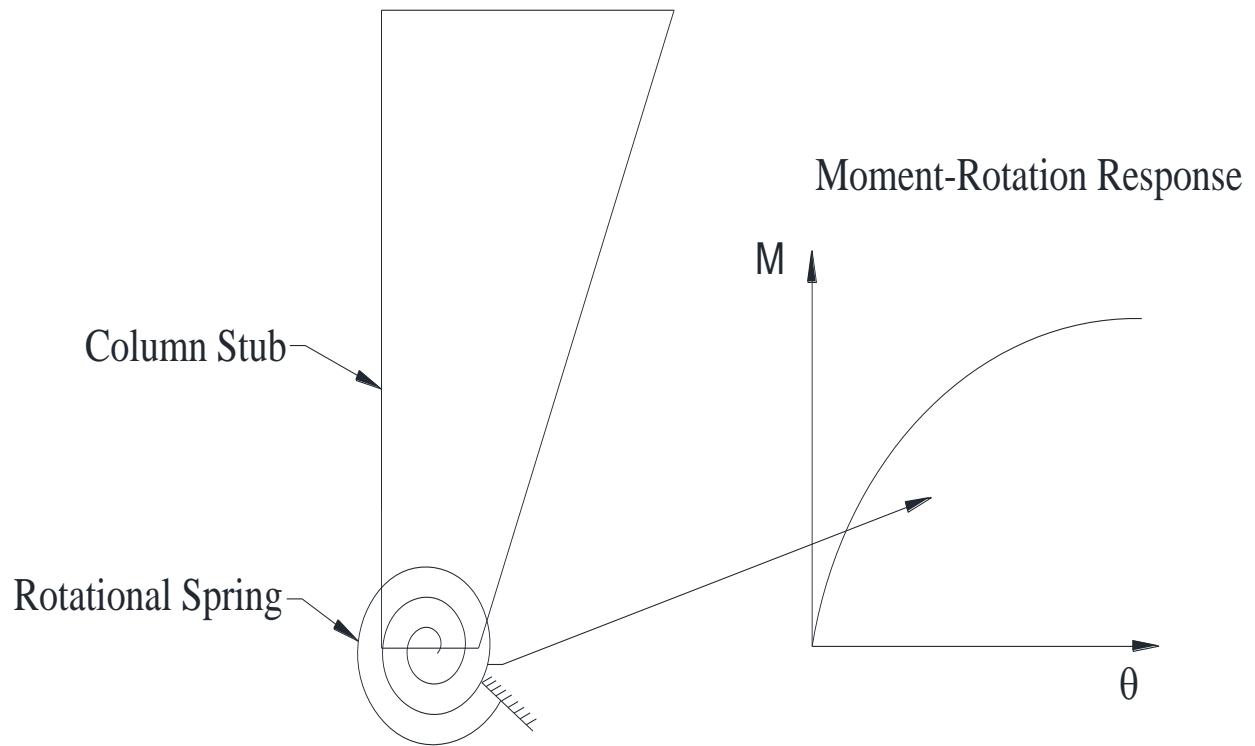


Figure 1.6. Column base represented by rotational spring which has a unique moment-rotation response

This chapter presents some background work of a previous research (Bajwa 2010) which inspired this research. The general behavioral characteristics of the connection are also investigated. The next chapter presents some of the past research works that were performed to understand the behavior of the base connection under various load cases. Some methods are also discussed which are very commonly used for designing various components of the column base connection.

## Chapter 2 Literature Review

### 2.1 Introduction

This chapter reviews some of the past research work that has been performed on column base connections to observe behavior under various load cases. The current design methodologies for designing the base plates are outlined. Some analytical procedures and experimental studies are also presented to understand the influence of various factors in determining the characteristic properties of any base connection. The stiffness values for some particular configurations of the base connection are calculated by using analytical procedures, which are also discussed in detail in this chapter.

### 2.2 Design of Axially Loaded Base Plate

The current methodologies for designing the base plates for axial loads assume pinned support at the column base. However, the actual restraint condition depends on the connection details that are provided at the column base. Chapter 14 of Steel Construction Manual (2010) published by American Institute of Steel Construction (AISC) and AISC Design Guide No.1 *Column Base Plates* (DeWolf and Ricker, 1990) provide some models which can be used to design the steel base plate for axial loads. The main models are discussed briefly in the following sections.

#### 2.2.1 Cantilever Model

This method is suitable for designing the base plates which have much larger dimensions than the column dimensions. The main assumption for this method is uniform bearing pressure variation beneath the plate. This assumption leads to a conservative design of base plates. For a wide flange section, the loaded area is considered as  $0.95 d_c \times 0.80 b_{fc}$  over the base plate (Figure 2.1), where  $d_c$  is column depth and  $b_{fc}$  is flange width of wide flange section. This results into overhanging of the remaining area of the base plate which is assumed as cantilever.

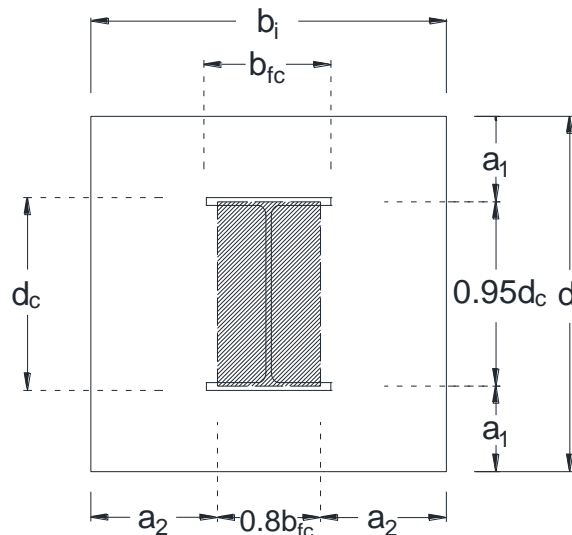


Figure 2.1. Equivalent rigid area for Cantilever Model



### 2.2.2 Fling Model

Fling (1970) recommended this model for wide flange sections only; it is a modification to the cantilever method. A strength check based on yield line theory for a pattern as shown in Figure 2.2, is recommended for a base plate having dimensions almost similar to column. The main assumption is the support condition, which is fixed along the web, simply supported along the flange and free for rest of the plate.

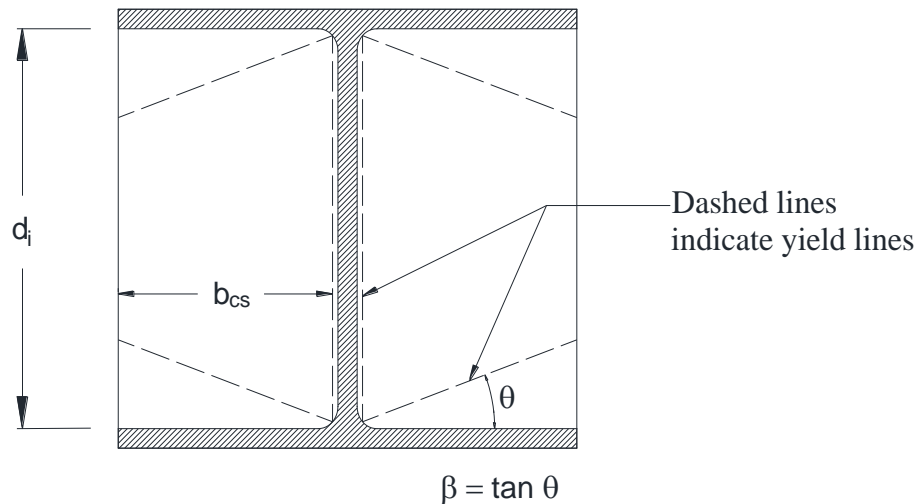


Figure 2.2. Yield lines for Fling Model

### 2.2.3 Murray-Stockwell Model

Stockwell (1975) proposed a model that is applicable to the flexible base plates having approximately similar size as of the column. This type of column base plate configuration is referred to as 'lightly loaded' for a wide flange section. He realized that in case of wide flange sections, the bearing pressure is not uniform on the whole base plate. As a result of this he proposed that only portions of the base plate which are under the column section should be subjected to a uniform bearing pressure equal to bearing capacity of concrete. Murray (1983) performed an experimental and analytical study. He used finite element analysis to model the base plate and gathered more information about the lightly loaded configuration. On the basis of his test results and analytical study, he supported the Stockwell model and defined a bearing area for the base plate, represented by shaded area annotated by  $A_H$  in Figure 2.3. In Figure 2.3,  $b_{fc}$  is the width of flange,  $d_c$  is depth of the wide flange section,  $a_3$  is a dimension which is used to define the bearing area ( $A_H$ ). It is measured from the centerline of the wide flange section.

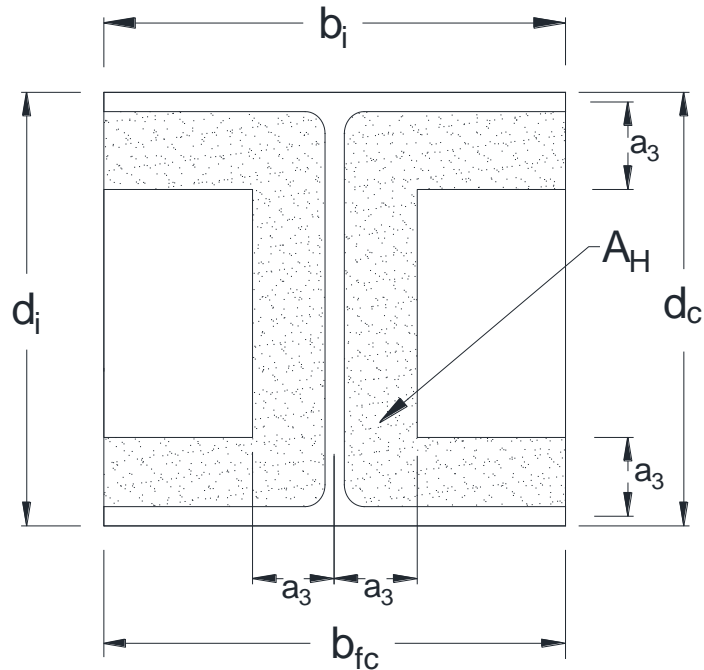


Figure 2.3. Equivalent rigid area shown by shaded region for Murray-Stockwell Model

### 2.2.4 Thornton Model

Thornton (1990) proposed a model which combines the above three models namely the cantilever, Fling and Murray-Stockwell models. Currently, this model is also recommended by AISC Steel Construction Manual. Usually, for large dimension plates, the cantilever model governs, otherwise the Murray-Stockwell model works well. It should be noted that the mathematical relations derived from this method are valid for wide flange sections only.

### 2.2.5 Eurocode 3 Model

This model is described in Annex L of Eurocode 3 (1993). It also uses the same concept of rigid area as used by the Murray-Stockwell model. In this model, the rigid area is extended both inside and outside the section. For a wide flange section, the rigid area is shown by Figure 2.4.

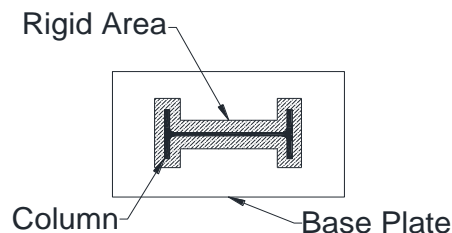


Figure 2.4. Eurocode representation of bearing stress on an equivalent rigid plate

The model is more efficient than Thornton's model in predicting the bearing pressure. However, as it assumes a simplified assumption of stress distribution, this model overestimates the bearing pressure values in comparison to finite element analysis.

The above described models are used for designing the base plates for axial loads and they do not include bending moment acting on the base plate. The methods for designing the base plates with moments are discussed in AISC Design Guide No.1 *Column Base Plates* (DeWolf and Ricker, 1990).

## 2.3 Analytical Procedures to calculate the Rotational Stiffness

### 2.3.1 Analytical Procedures proposed by Galambos

Galambos (1960) proposed two analytical methods for determining the rotational stiffness of the column base connection. The first method only takes account of the behavior of the base plate, anchor bolts and concrete and neglects the rotation of the footing in the soil. The rotational stiffness of the connection ( $K_{con}$ ) is given by the following relation:

$$K_{con} = \frac{M}{\theta} = \frac{b d^2 E_c}{12} \quad (2-1)$$

where  $M$  is moment at the base of the column,  $\theta$  is rotation at the base of the column,  $b$  is width of the base plate,  $d$  is length of the base plate and  $E_c$  is modulus of elasticity of the concrete.

The second method predicts the stiffness of the soil underneath the connection and assumes the footing as a rigid beam and the soil as a series of elastic springs. The rotational stiffness of the soil ( $K_{soil}$ ) is given by following relation:

$$K_{soil} = \frac{M}{\theta} = \frac{q g f^3}{12} \quad (2-2)$$

where  $q$  is modulus of subgrade reaction,  $g$  is width of the concrete footing and  $f$  is length of the concrete footing. It should be noted that Equation (2-2) assumes that the column base connection is completely rigid and does not have internal stiffness. In other words, the connection does not contribute to the stiffness. In order to obtain the rotational stiffness of the whole column base assembly consisting of steel connection plus footing, one may think of adding the two stiffness,  $K_{con}$  and  $K_{soil}$  in series according to Equation (2-3). However, Galambos emphasized to take the minimum of the two values, as both the expressions neglect the influence of the other one. In other words, Equation (2-1) neglects the rotation of the footing and Equation (2-2) assumes the steel connection to be rigid.

$$K_{Total} = \frac{1}{\frac{1}{K_{con}} + \frac{1}{K_{soil}}} \quad (2-3)$$

where,  $K_{Total}$  is rotational stiffness of the whole assembly ,  $K_{con}$  and  $K_{soil}$  are the rotational stiffness of the connection and soil as defined by Equation 2-1 and Equation 2-2, respectively.

### 2.3.2 Component Method

The component Method is an analytical method proposed in Section 6.3 of Eurocode 3 (EN-1993-1-8) for estimating the rotational stiffness of the column base connection. This method is briefly described below.

It contains five parts:-

- (1) Identification: Decomposing connection in different components.
- (2) Characterization: Mechanical properties are determined (stiffness and deformation capacity are some of the common ones).
- (3) Assembly: Each component's mechanical properties are combined to get the mechanical property of the connection (stiffness and deformation capacity of joint).
- (4) Classification: To classify the behavior of the connection under overall analysis as in rigid, pinned or partially rigid.
- (5) Modeling: For taking account of nonlinear properties of the connection under overall analysis.

The behavior of each component is studied by performing a number of tests on it under different load cases. The stiffness coefficients are estimated for the base plate and concrete block in compression, base plate in bending and anchor bolts in tension and shear. Table 6-11 (Eurocode 3, EN-1993-1-8) provides the expressions to calculate the above mentioned stiffness coefficients. As a part of assembly process, the calculated stiffness coefficients are combined to find the rotational stiffness of the connection. Table 6-12 (Eurocode 3, EN-1993-1-8) provides a relation for estimating the rotational stiffness of the column base connection subjected to combined axial force and bending moment. It should be noted that the stiffness coefficients provided by Table 6-11 (Eurocode 3, EN-1993-1-8) have dimensional formula of  $M^0L^1T^0$  and the rotational stiffness expression provided by Table 6-12 (Eurocode 3, EN-1993-1-8) is dependent on the load configuration, hence for different orientations of the loads, the relation will be different. For a compressive axial load and a clockwise moment on the connection, the rotational stiffness ( $S_j$ ) is given by following expression:

$$S_j = \frac{EZ^2}{\mu \left( \frac{1}{k_{T,l}} + \frac{1}{k_{C,r}} \right)} \frac{e}{e+e_k} \quad \text{where, } e_k = \frac{z_{C,r} k_{C,r} - z_{T,l} k_{T,l}}{k_{T,l} + k_{C,r}} \quad (2-4)$$

where  $k_{T,l}$  is the tension stiffness coefficient on the left side (tension side for this load configuration) of the connection and should be taken as equal to the sum of the stiffness coefficients  $k_{15}$  and  $k_{16}$  (Table 6.11, Eurocode 3, EN-1993-1-8) acting on the left hand side of the connection;  $k_{C,r}$  is the compression stiffness coefficient of the right side (compression side for this load configuration) of the connection and should be taken as equal to the stiffness coefficient  $k_{13}$  (Table 6.11, Eurocode 3, EN-1993-1-8) acting on the right hand side of the connection ;  $z$  is the lever arm shown in Figure 2.5;  $E$  is elastic modulus of the steel;  $\mu$  is stiffness ratio, equal to 1.0 for finding the initial rotational stiffness (6.3.1(6) , Eurocode 3, EN-1993-1-8) ;  $e$  is eccentricity of the applied axial load from center of the connection, equal to  $\frac{M_D}{N_D}$  , where  $M_D$  is the design moment and  $N_D$  is the design axial force;  $Z_{C,r}$  is the distance from the center of the connection to the point on the base plate beneath the column flange in compression and  $Z_{T,l}$  is the distance from the center of the connection to the point on the base plate intersecting with bolt position (Figure 2.5).

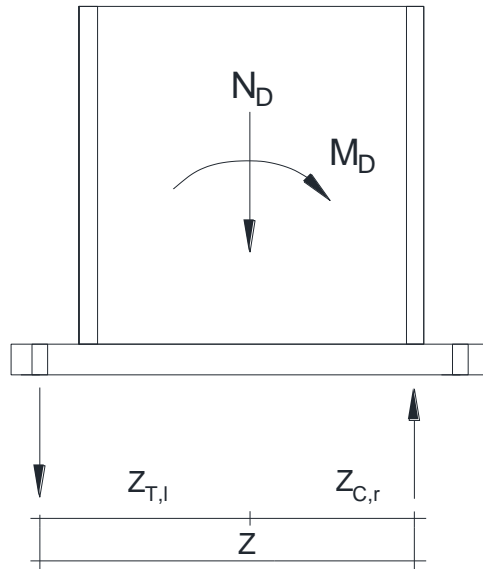


Figure 2.5. Configuration of the base connection considered for applying the component method

The coefficient  $k_{13}$  represents the stiffness coefficient of the concrete in compression (including grout). The expression for  $k_{13}$  (Table 6.11, Eurocode 3, EN-1993-1-8) is presented in Equation (2-5)

$$k_{13} = \frac{E_c \sqrt{b_{eff} l_{eff}}}{1.275 E} \quad (2-5)$$

where  $E_c$  is elastic modulus of the concrete,  $E$  is elastic modulus of the steel,  $l_{eff}$  and  $b_{eff}$  are length and width of the T stub flange shown in Figure 2.6.

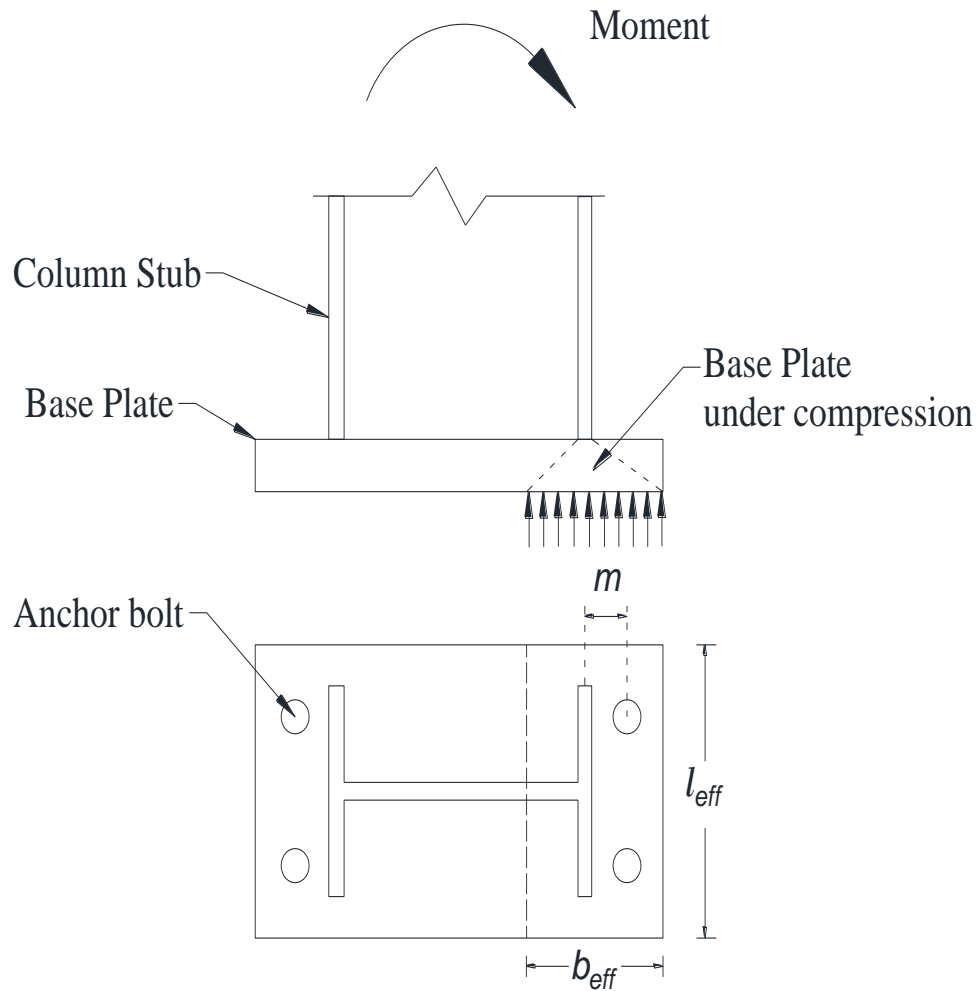


Figure 2.6. An equivalent T-stub Model

$k_{15}$  and  $k_{16}$  represent the stiffness coefficients for the base plate in bending and anchor bolts in tension, respectively. Table 6.11 (Eurocode 3, EN-1993-1-8) provides two expressions for both  $k_{15}$  and  $k_{16}$  based on the presence and absence of prying force. In presence of the prying force (contact between the T-stub model and concrete surface is present), the connection parameters should satisfy the following inequality:

$$\frac{A_b}{L_b} \geq \frac{l_{eff} t_p^3}{8.82 m^3} \quad (2-6)$$

where  $t_p$  is the base plate thickness,  $A_b$  is the area of the anchor rods,  $L_b$  is the length of the anchor rods and  $m$  is the dimension of the T-stub as shown in Figure 2.6.

The stiffness coefficient ( $k_{15}$ ) for the base plate in bending in presence of the prying force is given by the following relation:

$$k_{15} = \frac{0.85 l_{eff} * t_p^3}{m^3} \quad (2-7)$$

The stiffness coefficient ( $k_{16}$ ) for the anchor bolts in tension in presence of the prying force is given by the following relation:

$$k_{16} = \frac{1.6 A_b}{L_b} \quad (2-8)$$

In case of no prying force (no contact between the T- stub model and concrete surface is present), the connection parameters should satisfy the following inequality:

$$\frac{A_b}{L_b} \leq \frac{l_{eff} t_p^3}{8.82 m^3} \quad (2-9)$$

The stiffness coefficient ( $k_{15}$ ) for the base plate in bending in absence of the prying force is given by the following relation:

$$k_{15} = \frac{0.425 l_{eff} * t_p^3}{m^3} \quad (2-10)$$

The stiffness coefficient ( $k_{16}$ ) for the anchor bolts in tension in presence of the prying force is given by the following relation:

$$k_{16} = \frac{2A_b}{L_b} \quad (2-11)$$

Hence, the sum of  $k_{15}$  and  $k_{16}$  gives  $k_{T,l}$  and  $k_{13}$  is equal to  $k_{C,r}$ . Both  $k_{T,l}$  and  $k_{C,r}$  are substituted back in Equation (2-4) to obtain the rotational stiffness of the connection for combined axial load and bending moment.

Typically, under high bending moment and low compressive load, the base plate in bending and the anchor bolts in tension contribute more to the stiffness of the connection than any other components. While under high compressive load and low bending moment, the concrete in compression contribute more to the stiffness of the connection.

### 2.3.3 Mechanical Model proposed by Jaspart and Vandegans

Jaspart and Vandegans (1998) realized the complexity of the column-base components and instead of proposing an analytical model, they presented a mechanical model to simulate the actual behavior of the column-base (Figure 2.7). The model contains different types of linear and nonlinear spring elements for simulating the behavior of different components like base plate, anchor bolts and concrete block. There are exactly four types of spring elements used in this model. The springs represented by number 1 are extensional springs which mimic the deformation of column section. Number 2 represents the extensional springs which act as anchor bolts and resist the uplift of the base plate. Number 3 represents the extensional springs for

transferring the compression on the base plate to the concrete below. Lastly, number 4 represents the spring elements, which are subjected to bending due to occurrence of plastic deformation on the compression side of the base plate. Please note in Figure 2.7, 'N' and 'M' are axial force and moment applied on the column stub, respectively.

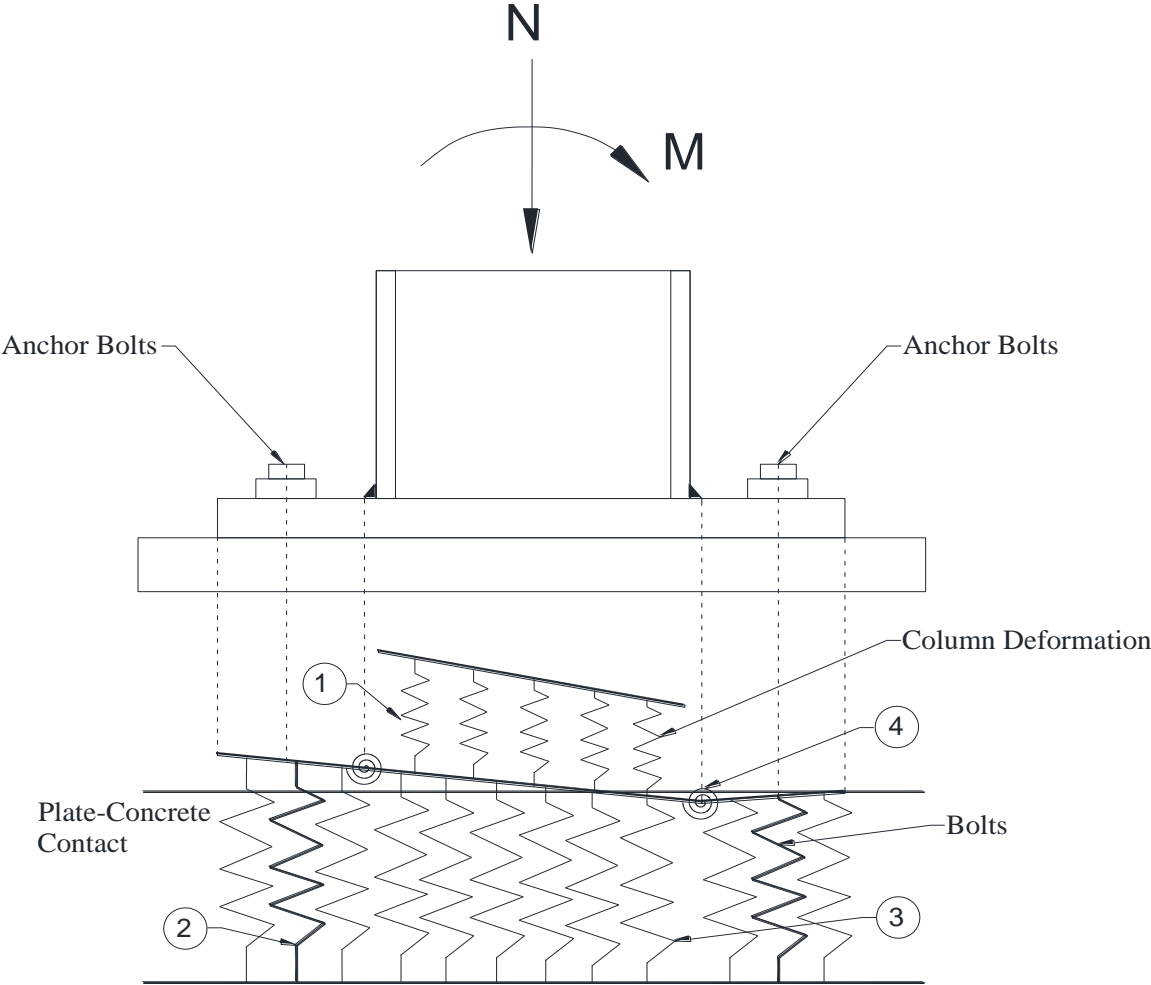


Figure 2.7. Mechanical Model for representing the column base connection (Jaspart and Vandegan, 1998)

They also performed a number of tests on the column base connection to compare the prediction of the model with that of the tests. The model predicted the initial stiffness and yield strength of connection in a good agreement with their test results. However, for higher values of the rotation (near collapse), the correlation became poor as each component displayed complex behavior close to collapse.

### 2.4 Tests and Experimental Studies on the Column-Base Connection

Numerous experimental studies were done by the past researchers on the column base to capture the actual behavior of the connection. The typical test assembly consisted of a column section with a base plate welded to its base, installed over a concrete block and tightened by two or four



symmetrically attached anchor bolts. A grout layer was placed in between the base plate and concrete block in most of the tests, except few in which the behavior was to be observed without the grout. This arrangement was subjected to various combinations of moments, axial and shear forces to observe the behavior of the column base under these load cases. Most of the tests were performed under varying magnitude of the moment and axial force and were aimed to take account of the stiffness that is provided by the connection.

Hon and Melchers (1988) studied the column bases under bending moments by performing tests. According to them the column base behavior could be best described by P-M- $\phi$  relationship, where P is axial force, M is moment applied and  $\phi$  is relative rotation between the column and base respectively. The behavior was supposed to be governed by (i) base plate thickness (ii) bolt size (iii) anchor bolt length. This governing influence by the above mentioned factors was observed through a number of experimental tests for the combination of different column sizes, bolt sizes and base plate thicknesses. It was concluded that the plate thickness had a profound effect on the stiffness of the connection for thin base plates. While in case of thick base plates, the bolts controlled the failure mode. In another words, the stiffness increased as the bolt size was increased. Column size was also found out to be a governing factor and an increment in it in the form of dimensions or weight, resulted in higher resistance and stiffness. These observations demonstrated the existence of substantial amount of stiffness in the column-base connection.

Ermopoulos and Stamatopoulos (1996) proposed a similar kind of mathematical model. It took account of material property of the concrete, anchor bolts and plate, geometry of the concrete base and magnitude of axial load on the column in addition to the parameters mentioned in the above model. By varying values of different parameters, a series of M- $\phi$  curves (where M is moment applied and  $\phi$  is observed rotation) were obtained through experiments. These curves could be replotted to get C- $\phi$  curves (where C is rotational stiffness of the connection). In order to utilize the curves more efficiently, they put forward following nonlinear relationship between M and  $\phi$ .

$$M(\phi) = a * \frac{M_o \phi}{\phi_o + \phi} \quad (2-11)$$

where  $a$  is a coefficient obtained through a curve fitting method and  $(M_o, \phi_o)$  are the coordinates of the characteristic point for each curve.

The coordinates of characteristic point were obtained by intersection point of the tangents from linear and nonlinear parts of an M- $\phi$  curve. The above equation, if used in structural analysis, takes account of the degree of fixity in each support instead of assuming pinned or fixed support condition. In another words, these curves can be used to perform more accurate structural analysis of the frames.

Gresnigt et al. (2008) performed several tests in Stevin Laboratory of Delft University to understand the column base behavior and to take account of the stiffness under shear and axial loads. It was found out that in most of the cases the shear force can be resisted by the friction or shear and bending of the anchor bolts. It was observed that when the normal load was highly compressive, the friction was sufficient enough to resist the shear. However for a lightly loaded column and high horizontal deformation, the anchor bolts played a critical role and transferred the shear to the concrete footing. It was found out that the grout layer was weak in resisting the bearing pressure, so considerable amount of bending of the anchor bolts took place to develop the shear resistance and stiffness. An analytical model was also proposed by the authors to predict the horizontal deformation and shear force for a given connection.

Gresnigt et al. (2008) also studied the effect of ductility of the bolts on the column behavior. It was found out that if the failure was governed by the bolts, then higher ductility led to more horizontal deformation, which resulted in higher shear resistance and consequently higher ultimate strength. Also the shear resistance was found out to be dependent on characteristics of the anchor bolts rather than on thickness of the grout layer. However, the horizontal deflection was dependent on the thickness of grout layer. So a better grout gave a lower deformation.

Picard and Beaulieu (1985) performed tests to observe the behavior of column base under axial loads. They signified the connection fixity by defining a fixity factor. Theoretical and experimental results were compared to draw out intriguing conclusions. According to them, an increase in stiffness was observed when the column base connection was subjected to a compressive axial load. Moreover in absence of an axial load, the flexural rigidity of the connection was enough to be incorporated in the design process. And this could result in significant decrease in lateral displacement and P- $\Delta$  effects in the analysis.

## **2.5 Finite Element Models**

Although some of the models mentioned in the previous section correlated well with the experiment results, there was still scope for improving the obtained results, since most of them were using rigid plate or equivalent rigid plate assumption. The advancement in finite element analysis and computation techniques was used to simulate the actual behavior of the column base connection. Various linear and nonlinear elements were used to design different components of the connection and analyses were performed. Krishnamurthy and Thambiratnam (1990) performed a two dimensional finite element study on the base plates with plate thickness and load eccentricity as parameters. This analysis used several finite elements such as bars, springs, constant strain triangle, hybrid rectangle, sub-parametric quadrilateral, and sub-parametric solid blocks. After the analysis, they found out that the actual behavior of the base plate was quite different than its anticipated behavior as rigid plate. According to them, due to thickness and flexible nature of the base plate, the bearing pressure varies nonlinearly and its maximum occurs

at a point beneath the compression flange in thin plates and close to the edge in thick plates. Moreover, the maximum bending stress for the finite element model was about half the theoretical value predicted by the rigid plate assumption and classical design methods. In another words, designing by rigid plate assumption was giving overestimates of bearing pressure and bending stresses.

Hamizi and Hannachi (2007) developed a finite element model simulating the test setups used by Picard and Beaulieu (1985). The model was designed to take account of the nonlinear interaction between the grout and base plate. For the loading direction shown in the Figure 2.8, finite element analyses were performed and a series of moment-rotation plots were obtained. These plots were then compared with the plots obtained by experimental tests performed by Picard and Beaulieu (1985). The correlation between the two was found in good agreement.

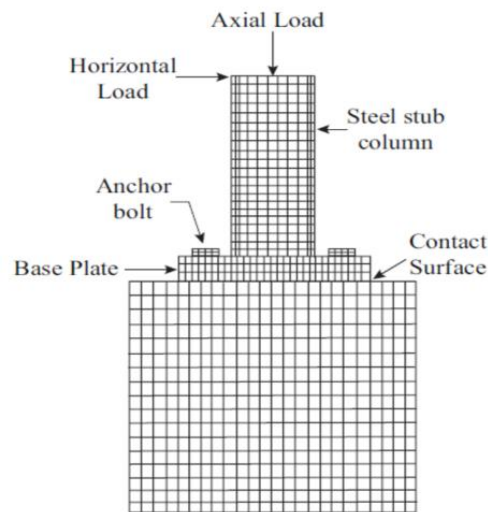


Figure 2.8. Finite Element Model (Hamizi and Hannachi 2007)

Dumas, Beaulieu and Picard (2006) came up with analytical characterization equations for three most common column-base connections to incorporate the behavior of the connection into analysis software. The procedure is based on curve standardization theory. In this theory, a number of moment-rotation curves are used to develop an equation which fits them. As the available standard experiment plots of moment-rotation curves from the past researches were not enough, the authors created a number of finite element models to represent the behavior. The finite element models were cross checked with the existing experiment results and then used independently to produce additional moment-rotation curves. These additional moment-rotation curves were needed to observe the effect of various changing parameters like axial load, base plate thickness, column section depth, anchor bolt diameter and anchor bolt eccentricity.

In conclusion, finite element analysis showed good correlation with the test results and proved to be a viable method for simulating the actual behavior of various components like base plate, anchor bolts and concrete block interacting with each other under various loading conditions.

## **2.6 Stiffness calculations from some research studies**

From the above presented experiments, analytical procedures and models, some of the studies are selected to compare their stiffness predictions with the stiffness values obtained from the wizard for the configurations of the connection used in the these study. This comparison provides a fair idea that how closely the wizard is able to correlate with the experimental studies and analytical models. In this section, only the selected studies are presented and the comparisons with the values obtained from the wizard are shown in Section 7.5 of this thesis. The three studies which are selected for the comparative studies are listed below.

- Analytical procedure by Galambos (1960) (Section 2.3.1)
- Eurocode 3 procedure (Section 2.3.2)
- Experiments performed by Picard and Beaulieu (1985) (Section 2.5)

As mentioned in Section 2.5, Picard and Beaulieu (1985) performed a number of tests to understand the behavior of the column base connection under different load cases and tried to quantify the fixity of it. Since detailed information of their experimental configurations of the connection is available, two of their test configurations are chosen as reference configurations for the comparative studies. The first test is a flexure test in which only a bending moment was applied to the connection. The second test is a compression-flexure test in which a compressive axial load was applied in addition to a bending moment. The geometrical arrangement of different components of the connection is the same in both the tests. The stiffness of the connection is calculated for each of the above test by using the analytical procedure of Galambos and Eurocode 3. The calculations are shown in Appendix A. These calculated values of the stiffness are plotted on the respective moment rotation curve obtained by Picard and Beaulieu for both the tests and compared with their stiffness prediction. The details of the connection used by them for both the tests are provided below.

### **2.6.1 Geometry and Material Used (same for both the tests)**

A W6X25 column section having a length of 4 ft was attached to a base plate which had a length of 11.81 in., a width of 7.48 in. and a thickness of 1.14 in. A standard diameter of 3/4 in. was taken for the bolts and the anchor rod length was taken as 17.72 in. The grout provided underneath the base plate had a thickness of 0.787 in., a compressive strength of 3.625 ksi (kips per square inch) and a modulus of elasticity of concrete as 3230 ksi. There was no pretension force provided in the anchor bolts as it was assumed that that they were snug tight. The positions of the anchor bolts on the base plate are shown in Figure 2.9. The material was steel for the column, base plate and bolts, which had a modulus of elasticity as 29000 ksi and poisson ratio of 0.3. Yield strengths for the steel used for the column and base plate were 43.5 ksi and 36.26 ksi respectively. The percentage of length of the anchor rod which is effective in providing the stiffness was taken as 100 %.

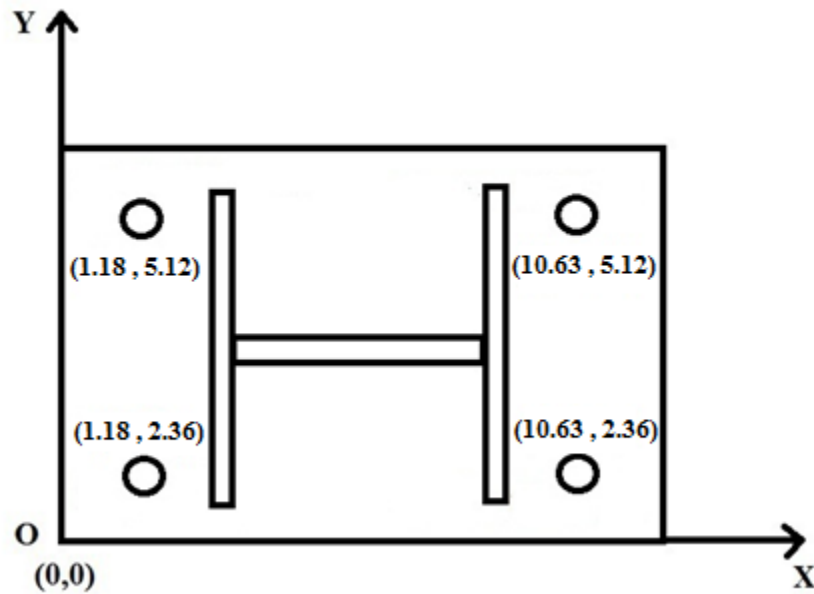


Figure 2.9. Anchor bolts positions for tests performed by Picard and Beaulieu (1985)

### 2.6.2 Flexure test by Picard and Beaulieu (1985)

In this flexure test, a bending moment was applied at top of the column till the failure point. The connection failed at a moment of 540 kips-in. Figure 2.10 shows the moment rotation curve corresponding to the flexure test performed by Picard and Beaulieu. The initial rotational stiffness predicted by them is shown by the red line. The stiffness of the same configuration is also calculated by using Equation 2-1 and it is shown by the green line. The slope of these colored line or in other words the values of the predicted stiffness by the two methods are summarized in Table 2-1. It can be observed from the Figure 2.10 and Table 2-1 that the stiffness for the test predicted by Galambos is much higher than the actual stiffness predicted by Picard and Beaulieu. The stiffness estimated by the wizard for this test is shown in Section 7.5.

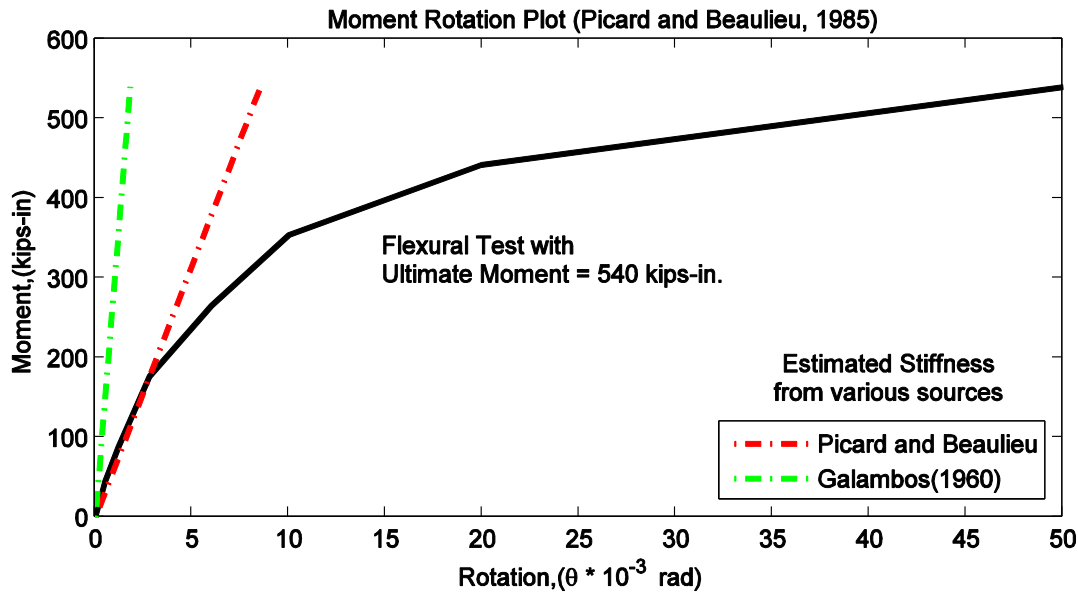


Figure 2.10. Values of stiffness from different sources on the moment-rotation plot of flexural test

Table 2-1. Values of Initial rotational stiffness obtained from different sources for the flexure test performed by Picard and Beaulieu (1985)

Source	Initial Rotational Stiffness (Kips-in/rad.)
Picard and Beaulieu experiment (1985)	58065
Galambos analytical procedure(1960)	298205

### 2.6.3 Compression-Flexure test by Picard and Beaulieu (1985)

The second test which is chosen for doing the comparative study is a compression-flexure test performed by Picard and Beaulieu. A compressive axial load and a moment were applied at the column top. The axial load was kept constant at 142.2 kips and the moment was increased till the failure point. The ultimate moment achieved was 1707.31 kips-in. The connection details at the base were same as given by Section 2.6.1. Figure 2.11 shows the moment-rotation curve obtained by Picard and Beaulieu; the stiffness predicted by them is presented by the red line. The green line on the figure represents the stiffness predicted by Eurocode. Table 2-2 summarizes the values of the initial rotational stiffness predicted by the two methods. The stiffness predicted by the wizard for this test is shown in Section 7.5. It can also be observed from the Figure 2.11 and Table 2-2 that the stiffness predicted by Eurocode is higher than the actual stiffness.

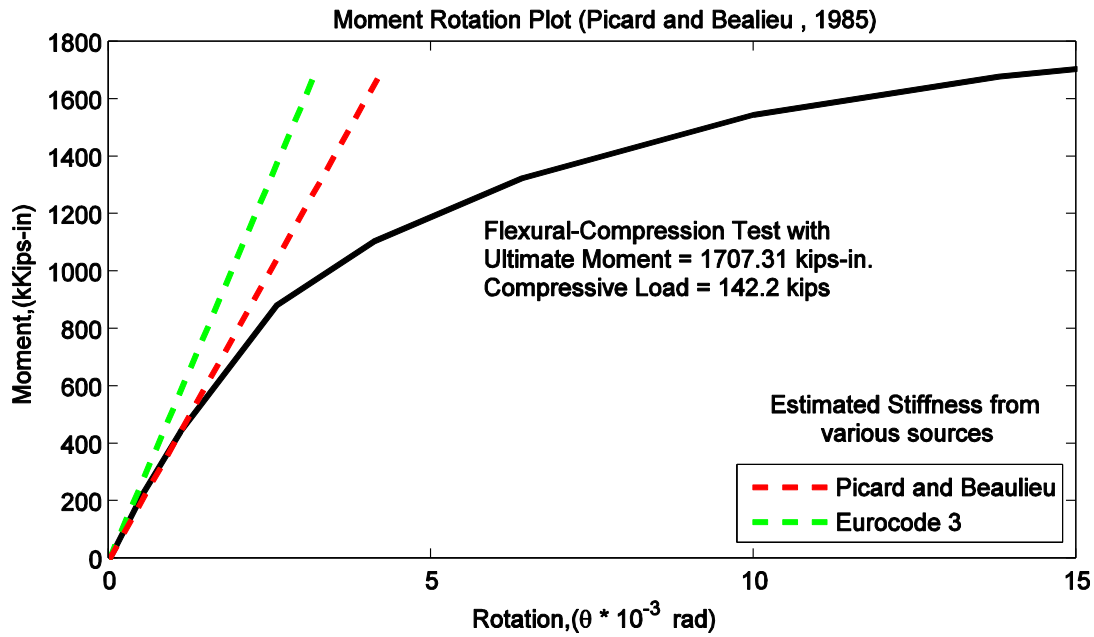


Figure 2.11. Values of stiffness from different sources on a moment-rotation plot for compression-flexural load

Table 2-2. Values of Initial rotational stiffness obtained from different sources for the compression-flexure test performed by Picard and Beaulieu (1985)

Method	Initial Rotational Stiffness (kips-in/rad)
Picard and Beaulieu (1985)	402309
Eurocode 3	529708

Note that the values of the stiffness obtained from the wizard for the above two tests is presented in Chapter 7. The reader is advised to refer Section 7.5 which compares the stiffness values obtained from the wizard with values obtained by these above methods. This literature review provides a quick glance of the research works that have been performed in recent years to understand the actual behavior of the column base connection. The next chapter investigates the responses of different finite elements under various load cases. In this research, these finite elements are used to model different components of the connection.

## Chapter 3 Testing of Finite Elements

### 3.1. Introduction

The column-base connection wizard uses either Open System for Earthquake Engineering Simulation (OpenSees) (University of California, Berkley 1999, 2000) or SAP2000 (Computer and Structures, Inc. 1976-2011) as its analysis engine. OpenSees is selected as one of the analysis engines due to its free availability and versatility in performing complex structural analysis. SAP2000 is chosen because it is commonly used is most of the structural firms for performing analysis of various types of structures. To make sure that the user gets the same estimate for the stiffness from the two programs, the finite elements used for modeling the connection in both the software packages were tested under different load cases. The sole objective behind this kind of basic testing was to understand and compare the response of the finite element used for modeling each component in both the programs. Table 3-1 shows the finite elements used in both SAP2000 and OpenSees to model different components of the connection.

Table 3-1. Finite Element used in SAP2000 and OpenSees to model different components of the connection

Component	Finite Element Used	
	SAP2000	OpenSees
Base Plate	Thick Shell	ShellMITC4
Grout	Solid	8 node Brick
Column Section	Thick Shell	ShellMITC4
Anchor Bolts	Nlink Elements	2 node Truss Element

### 3.2 Base Plate Testing

Behavior of the base plate under various load cases is the most important factor affecting the behavior of overall connection. This demands for the testing of finite element used for modeling of the base plate in both the programs under different load cases. The base plate was modeled using thick shell in SAP2000 and shellMITC4 element in OpenSees. The modeled base plate was assigned a plan area of 12 in. by 8 in. and a thickness of 0.5 in. Figure 3.1 shows the plan view of the modeled base plate. To test the flexibility of the base plate, it was connected to the rigid base with spring elements. These spring elements are used for simulating the interaction between the base plate and the grout in preliminary models of the whole connection, which is described in detail in the next chapter. These spring elements were attached to the base plate at one end and restrained at other end. Figure 3.2 shows the modeled configuration of the base plate.



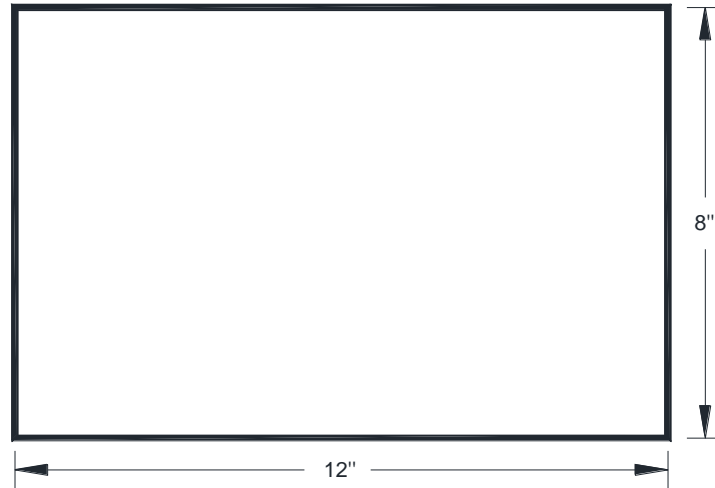


Figure 3.1. Plan view of the base plate in SAP2000 or OpenSees

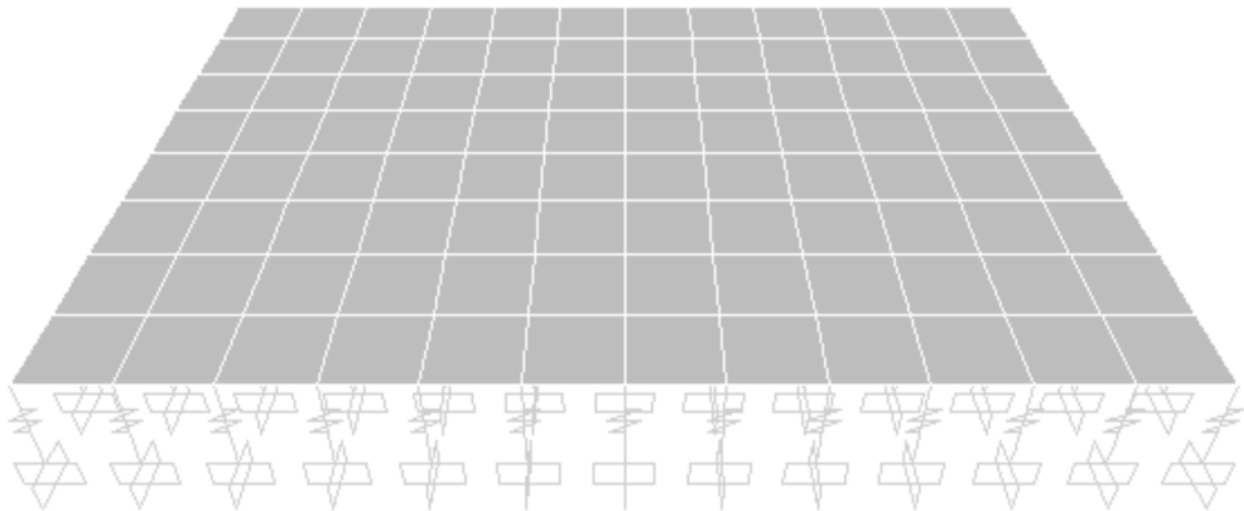


Figure 3.2. Modeled configuration of the base plate in SAP2000 and OpenSees

In SAP2000, link elements and in OpenSees, truss elements were used as spring elements. These spring elements are usually assigned with a force deformation relationship which defines their response to any applied axial load. For the testing purpose, three different force-deformation relationships (see Figure 3.3, 3.4 and 3.5) are considered. Figure 3.3 shows a linearly varying force deformation relationship. Figure 3.4 shows a nonlinear force deformation relationship which has mild nonlinearity. Figure 3.5 also shows a nonlinear force deformation relationship which has high nonlinearity. Note that the slope of force-deformation relationship represents stiffness. In all relationships shown below, ' $K_t$ ' and ' $K_c$ ' represent the stiffness exhibited by a spring element in tension and compression, respectively.

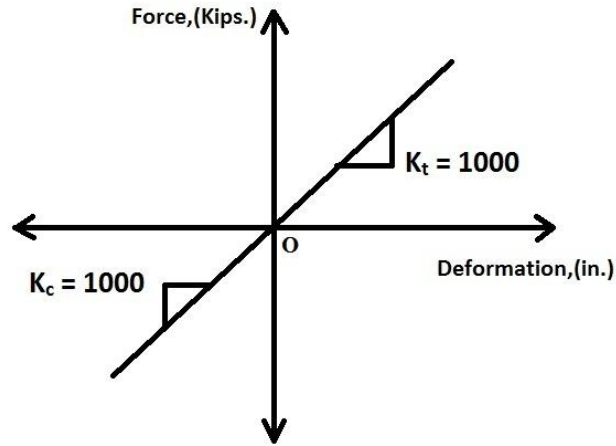


Figure 3.3. Linear force-deformation relationship assigned to spring elements

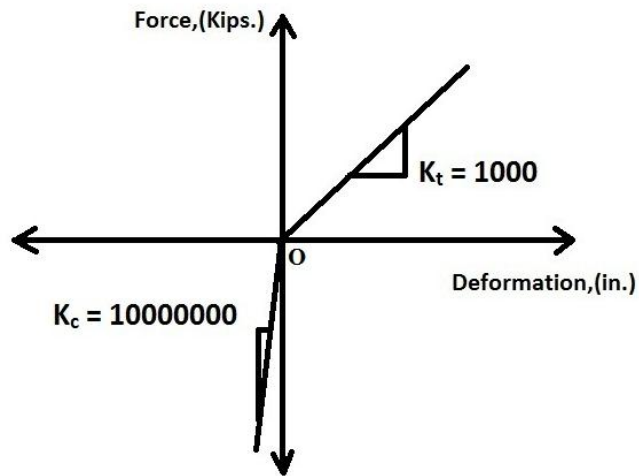


Figure 3.4. Mild Nonlinear force-deformation relationship assigned to spring elements

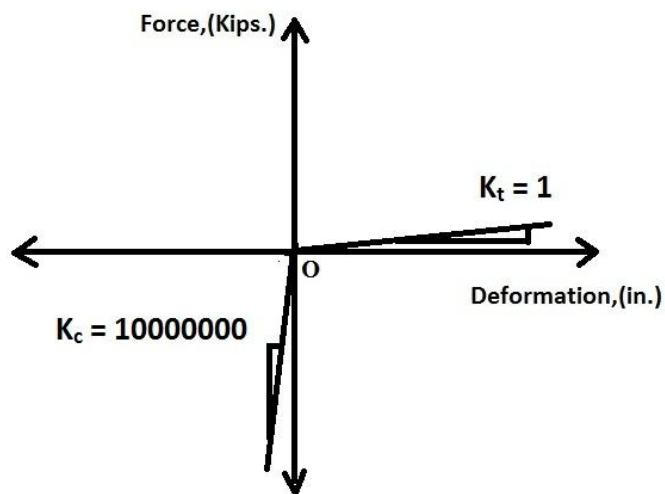


Figure 3.5. Full Nonlinear force-deformation relationship assigned to spring elements

### 3.2.1 Physical significance of force-deformation relationships

Figures 3.3, 3.4 and 3.5 show three different force-deformation relationships. Each of these has a physical significance. The right side of the origin 'O' in these plots represents the response in tension and the left side represents the response in compression. In other words, the response of a spring element in tension is represented by the line having slope  $K_t$ , drawn on right of the origin and the response in compression is represented by the line having slope  $K_c$ , drawn on left of the origin in each plot. If the spring element is assigned with the force-deformation relationship shown by Figure 3.3 then the magnitude of the deformation under an equal magnitude of tensile or compressive force will be same as the slope of the lines representing the responses at the two sides is same. If the spring element is assigned with the force-deformation relationship shown by Figure 3.4 then the magnitude of the deformation under a tensile force will be more than the magnitude of the deformation under a compressive force which has the same magnitude as the tensile force. Lastly, if the spring element is assigned with the force-deformation relationship shown by Figure 3.5 then the magnitude of the deformation under a tensile force will be much more than the magnitude of the deformation under a compressive force which has the same magnitude as the tensile force. Note that the slope of the line on right side of the origin in Figure 3.4 is larger than the slope of the line on right side of the origin in Figure 3.5. This means that the response in tension in Figure 3.4 is stiffer than the response in tension in Figure 3.5.

### 3.2.2 Load Application

The sub assembly shown in Figure 3.2 was tested under three load cases in both SAP2000 and OpenSees. The three load cases are listed below.

- Gravity: 1 kip/in<sup>2</sup> uniformly distributed load was applied in the direction of gravity as shown in Figure 3.6.
- Moment: Applied load varied from 0.6 kip/in<sup>2</sup> (at left edge of the base plate) to 0.1 kip/in<sup>2</sup> (at the center) in the upward direction and varied from 0.1 kip/in<sup>2</sup> (at the center) to 0.6 kip/in<sup>2</sup> (at right edge of the base plate) in the downward direction as shown in Figure 3.7.
- Gravity + Moment: First a uniformly distributed load of 1 kip/in<sup>2</sup> was applied in the direction of gravity and then a moment load was applied in the clockwise direction. The moment load was created by the application of varying distributed load along the length of the base plate. The uniformly distributed load was varied from 6 kips/in<sup>2</sup>(at left edge of the base plate) to 1 kip/in<sup>2</sup> (at the center) acts in upward direction and 1 kip/in<sup>2</sup> (at the center) to 6 kips/in<sup>2</sup> (at right edge of the base plate)acts in downward direction on the base plate as shown in Figures 3.8 and 3.9.

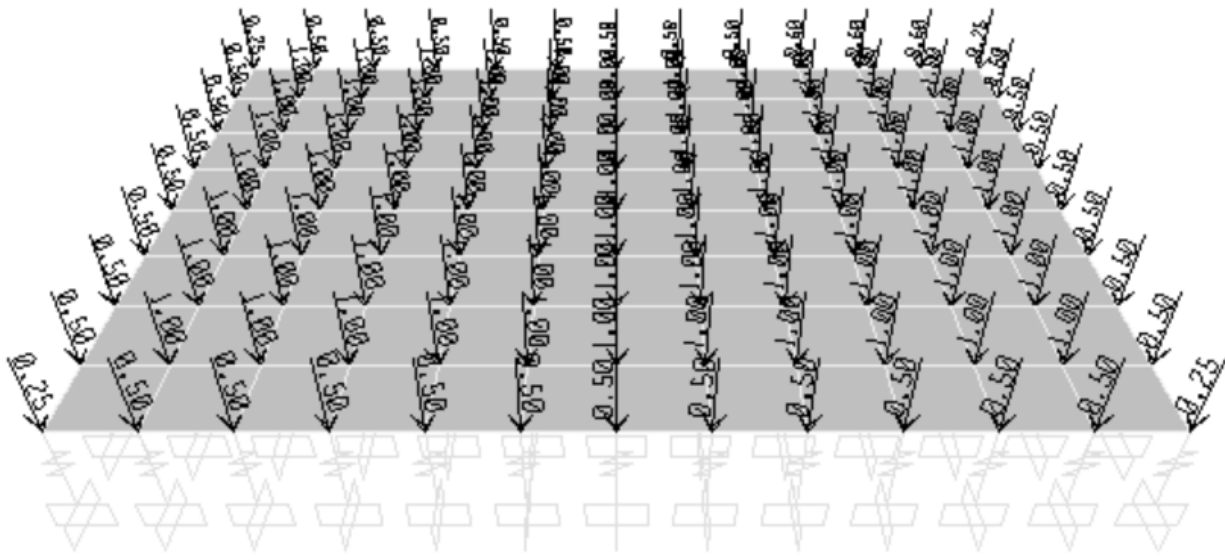


Figure 3.6. Base Plate subjected to gravity load case

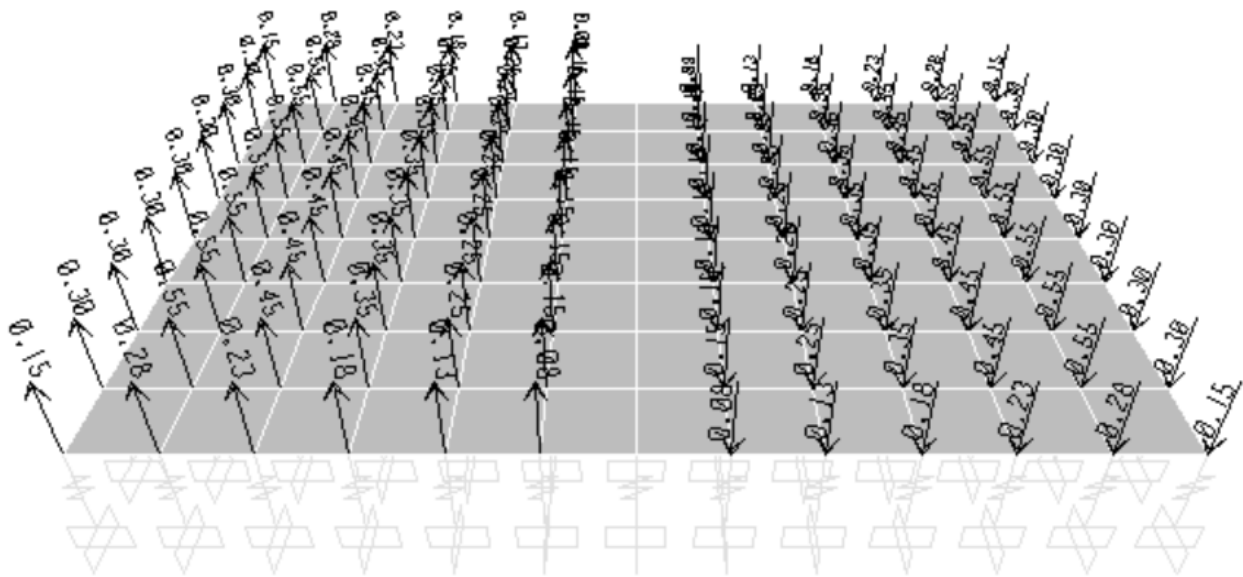


Figure 3.7. Base Plate subjected to moment load case

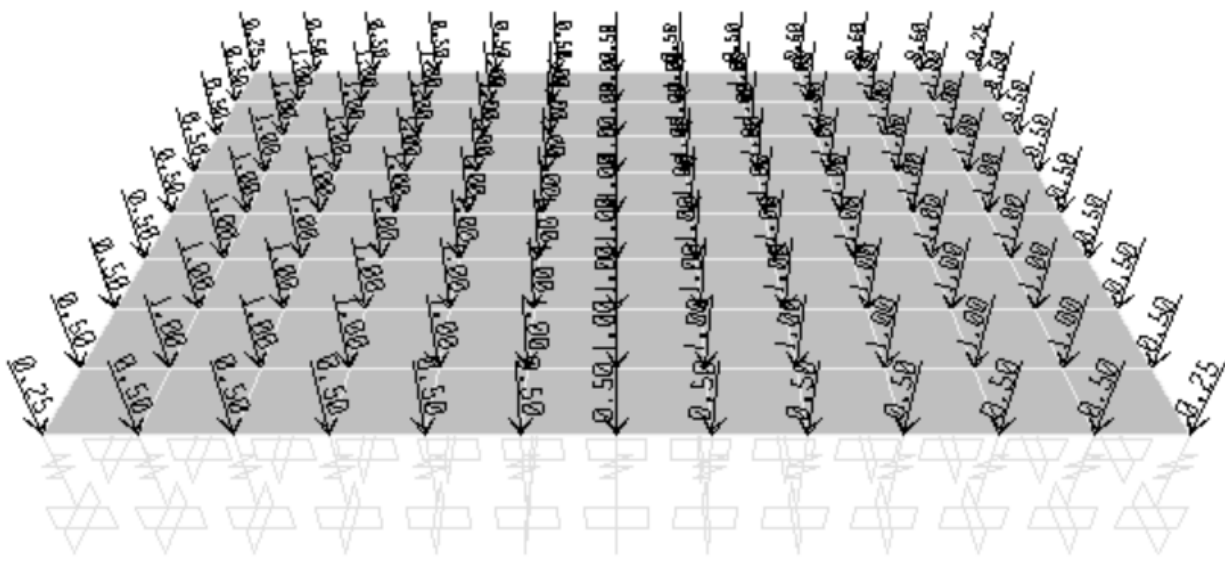


Figure 3.8. Base Plate subjected to gravity loads first in gravity+moment load case

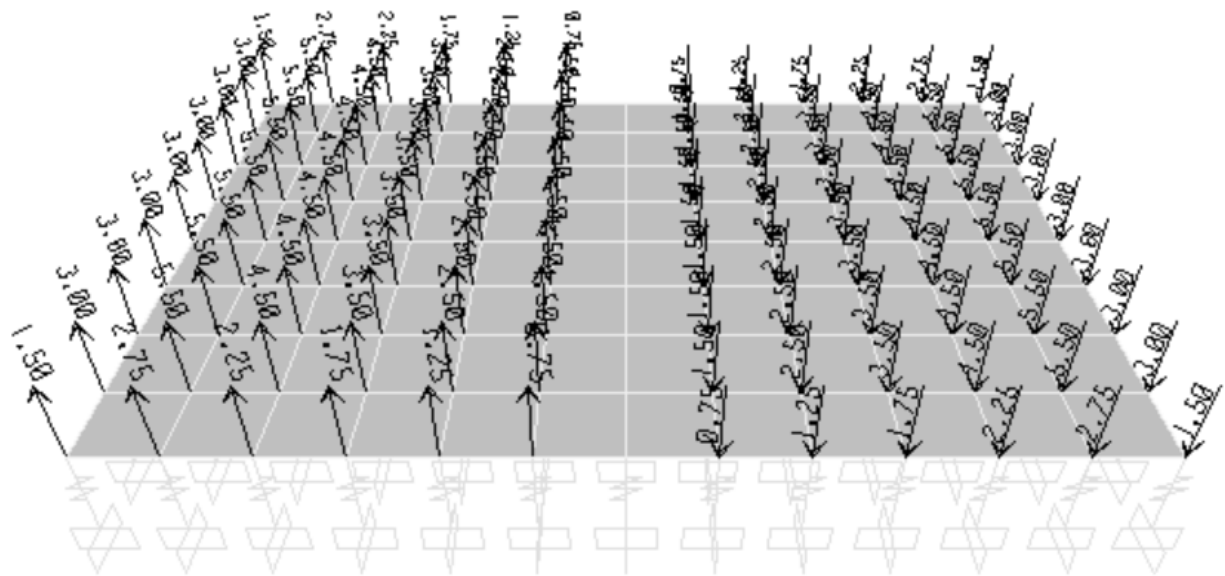


Figure 3.9. Base Plate subjected to moment loads after gravity loads in gravity+moment load case

There were total nine models analyzed in each program, as for each of the three load cases, three force-deformation relationships were assigned to the spring elements. The following plots show the comparison of the responses produced by SAP2000 and OpenSees for the above load cases. Each plot presents the deformation along length of the base plate for a specific load case and a specific force-deformation relationship.

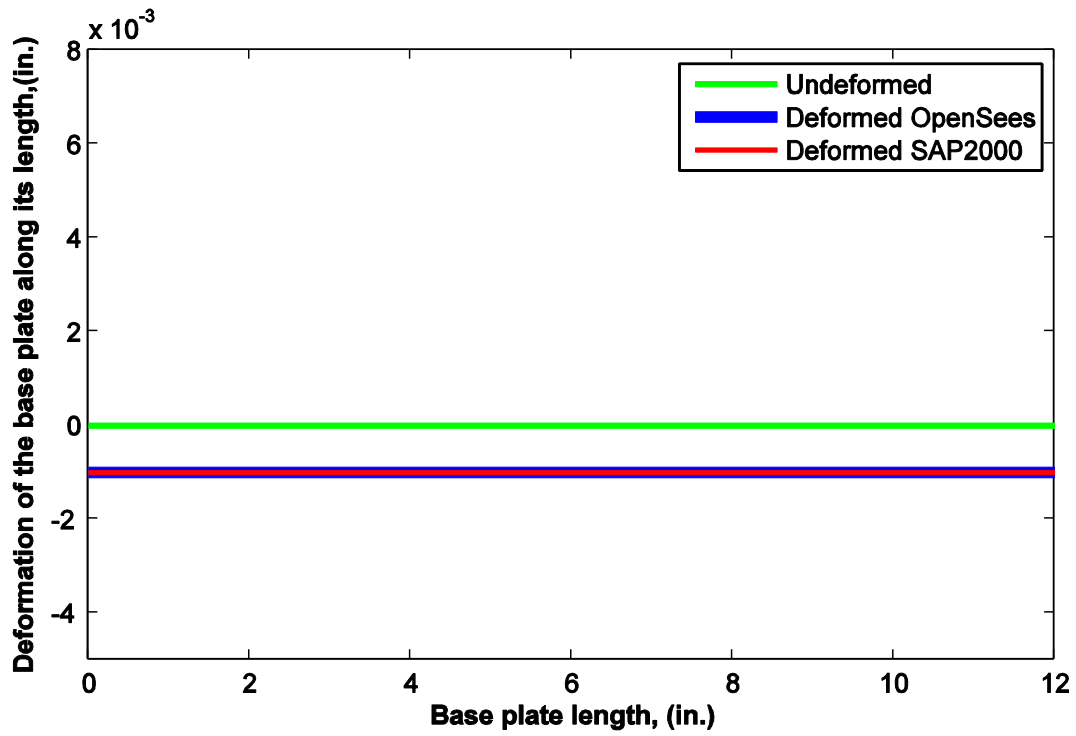


Figure 3.10. Deformation of base-plate under gravity load and linear springs

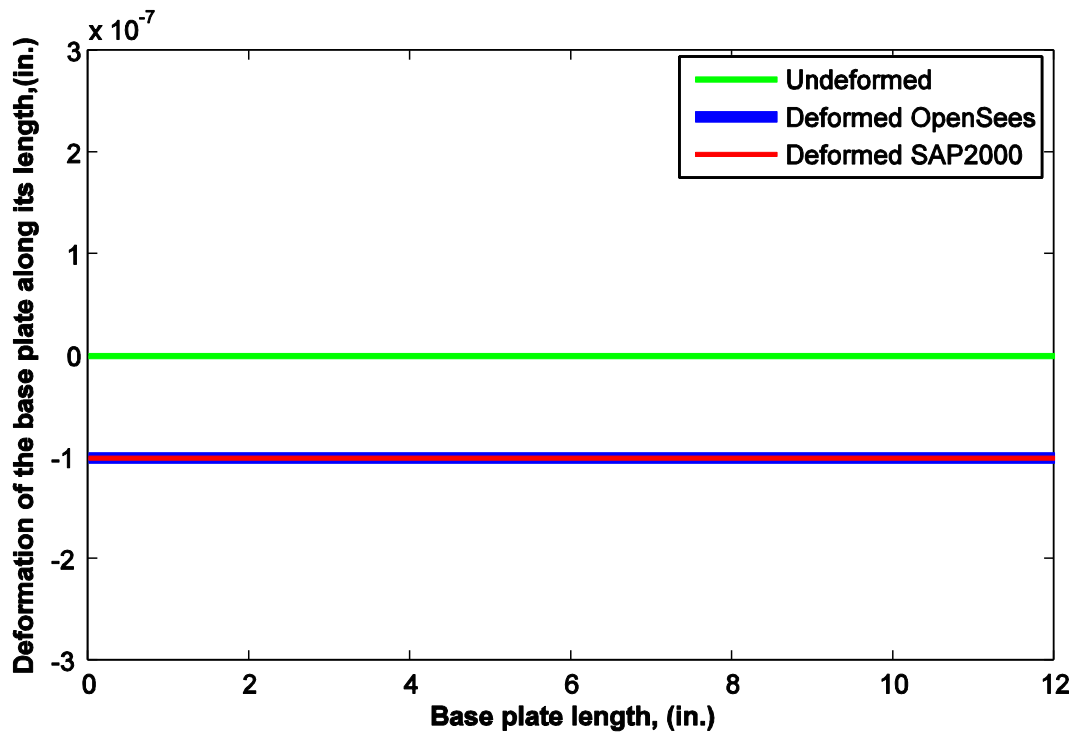


Figure 3.11. Deformation of base-plate under gravity load and mild nonlinear springs

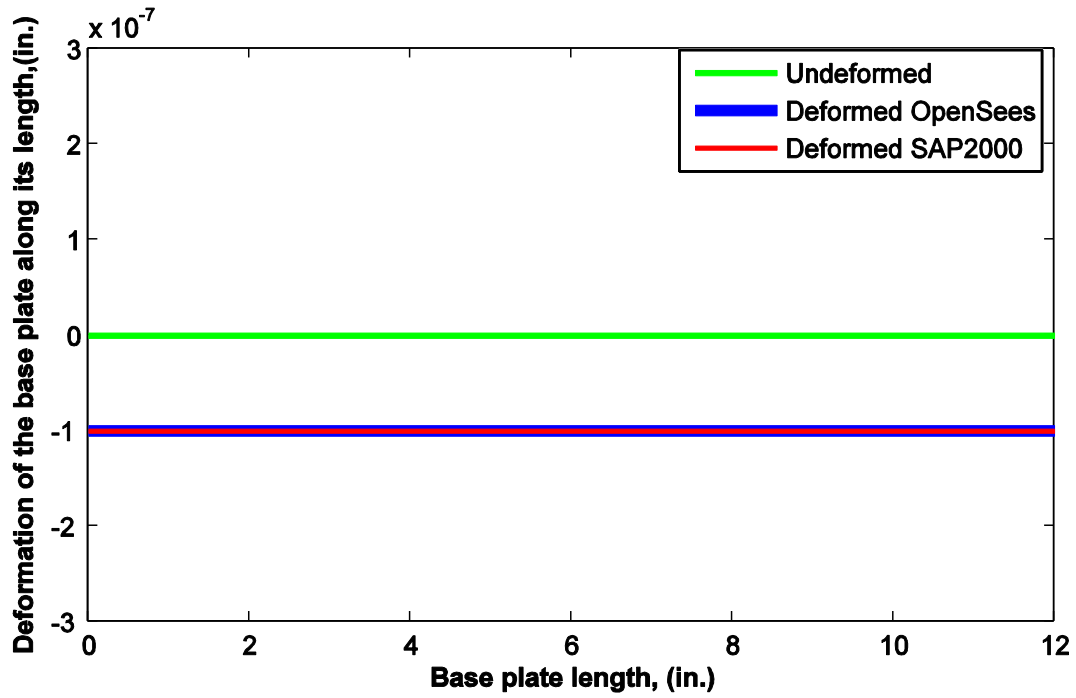


Figure 3.12. Deformation of base-plate under gravity load and full nonlinear springs

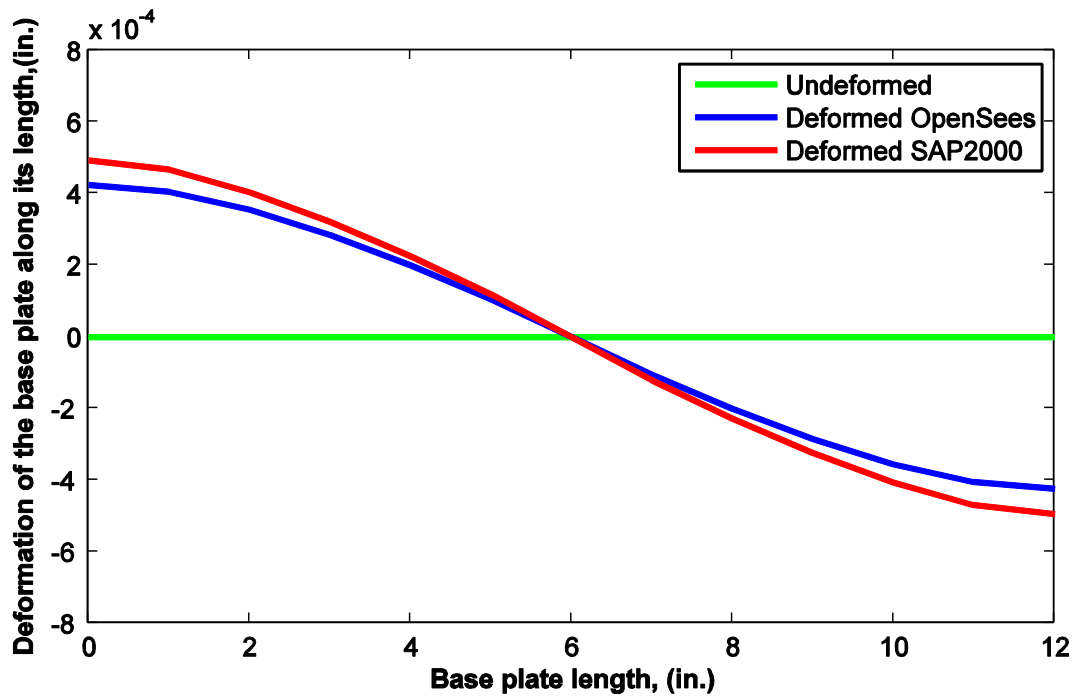


Figure 3.13. Deformation of base-plate under moment load and linear springs

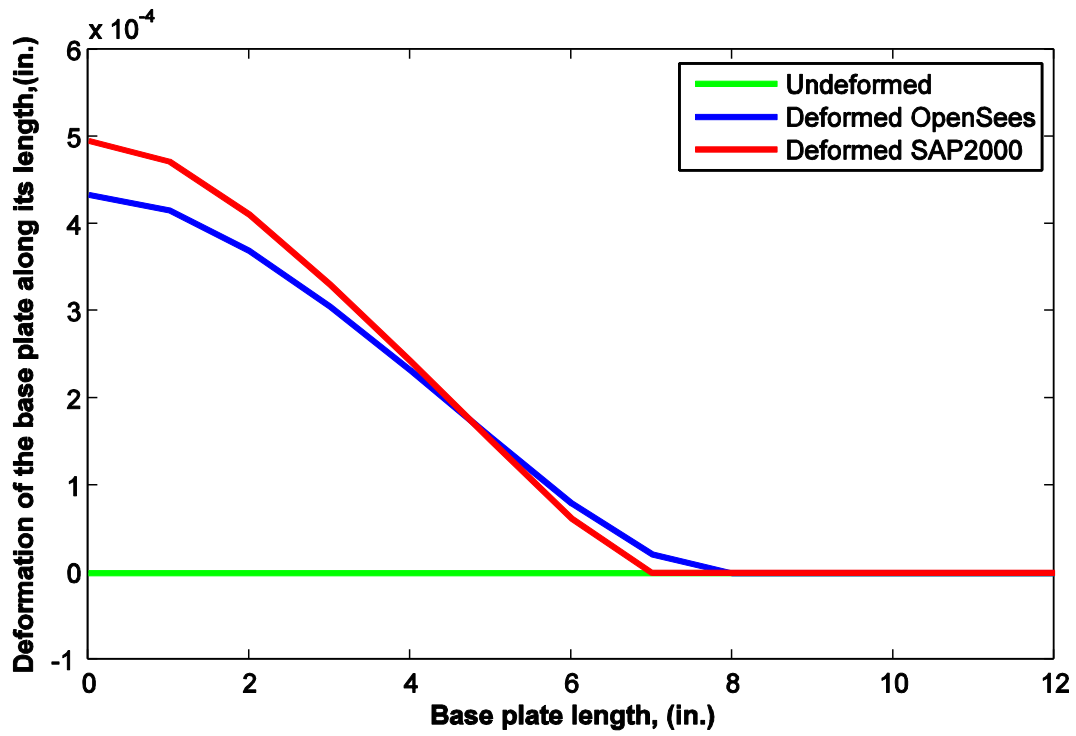


Figure 3.14. Deformation of base-plate under moment load and mild nonlinear springs

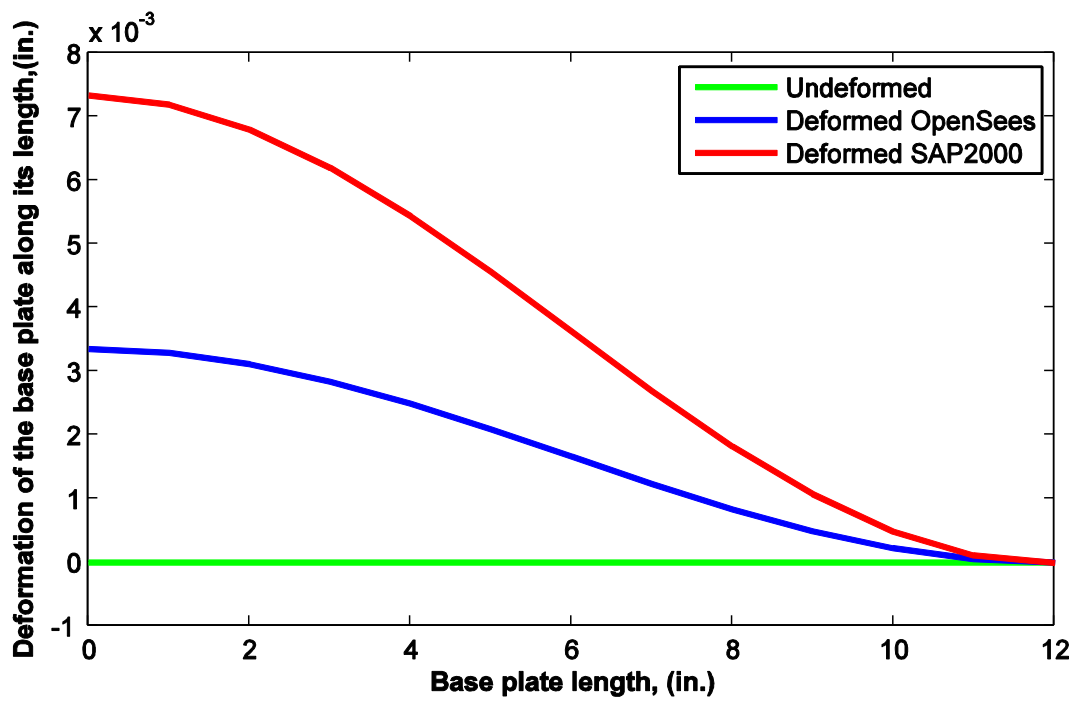


Figure 3.15. Deformation of base-plate under moment load and full nonlinear springs



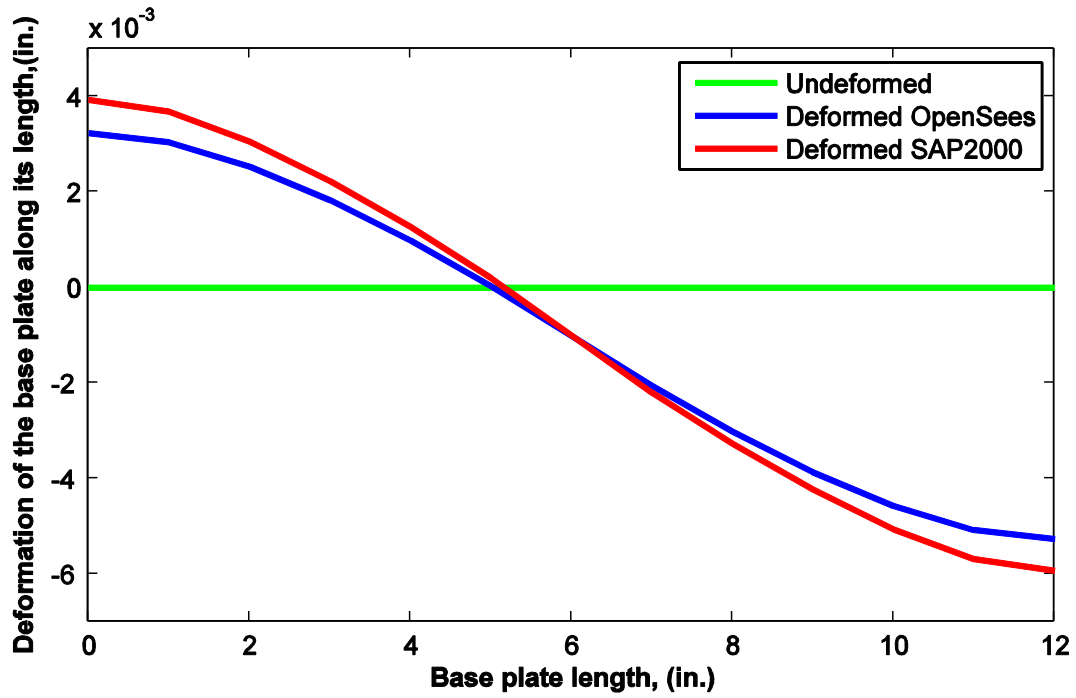


Figure 3.16. Deformation of base-plate under gravity+moment load and linear springs

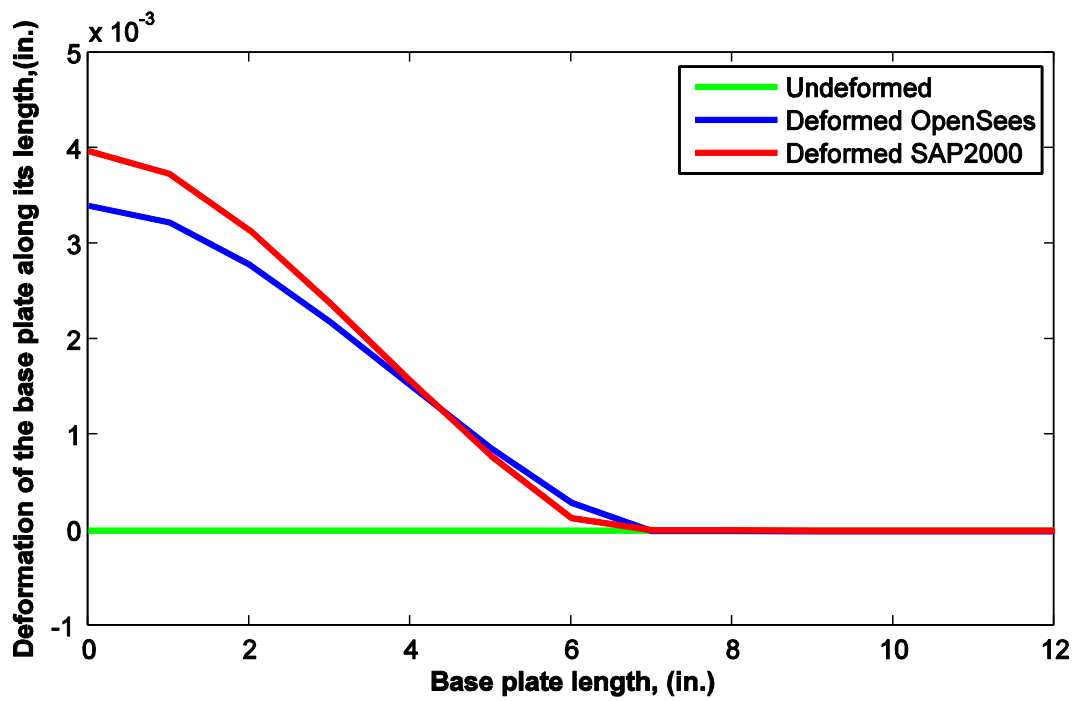


Figure 3.17. Deformation of base-plate under gravity+moment load and mild nonlinear springs

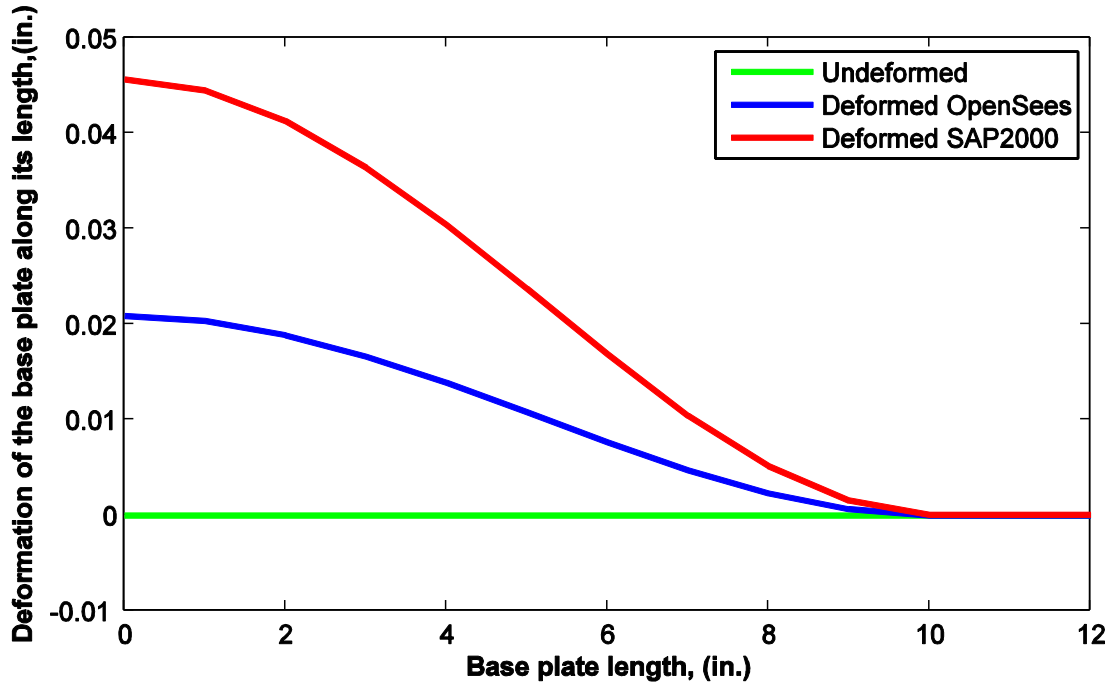


Figure 3.18. Deformation of base-plate under gravity+moment load and full nonlinear springs

### 3.2.3 Observations

Figure 3.10 shows that under gravity load case and with linear spring elements, the base plate in both SAP2000 and OpenSees gave almost the same deformations, if not identical, along the length of the base plate. This trend was further reinforced by Figures 3.11 and 3.12, which showed that under the gravity load case and with nonlinear force-deformation relationships assigned to the spring elements, the base plate deformations along the length of the plate were identical from both the programs. Figures 3.13 and 3.14 show that the deformations along length of the base plate were almost same under the moment load case with linear and mild nonlinear spring elements. However, by looking at Figure 3.15, it can be seen that there was some difference in values from the two programs when full nonlinear spring elements were used. Figures 3.16, 3.17 and 3.18 show similar trends as discussed above for the moment plus gravity load case. The deformations were identical for linear and mild spring elements along the length but little apart for full nonlinear spring elements.

### 3.3 Grout Testing

The wizard developed for this project provides the user with an option to model grout beneath the base plate. In that case, the grout also becomes a component of the column-base connection. This demands for the testing of finite element used for modeling the grout in both the programs (SAP2000 and OpenSees). However, the testing will not be as rigorous as for the base plate because the grout in the connection only resists compression in the base plate and remains

ineffective for tension in the base plate. This meant that the testing should only be done for the gravity loads as they are major contributor in subjecting the base plate under compression. In this wizard, SAP2000 uses solid element and OpenSees uses brick element for modeling the grout. The modeled grout for testing had 12 in. by 8 in. plan area and 1 in. thickness as shown in Figure 3.19; it was also meshed in four parts along the thickness to capture the variation in deformation more effectively. The plot in Figure 3.20 shows the deformations along the length of the grout under gravity load of 1 kip/in<sup>2</sup> uniformly distributed over the grout's upper surface (shown in Figure 3.19).

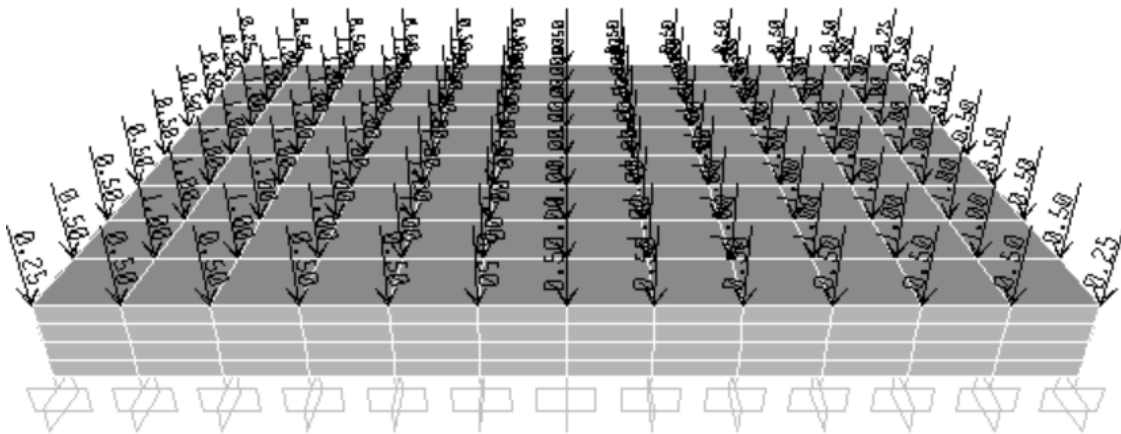


Figure 3.19. Modeled grout for testing under gravity loads in SAP2000 and OpenSees

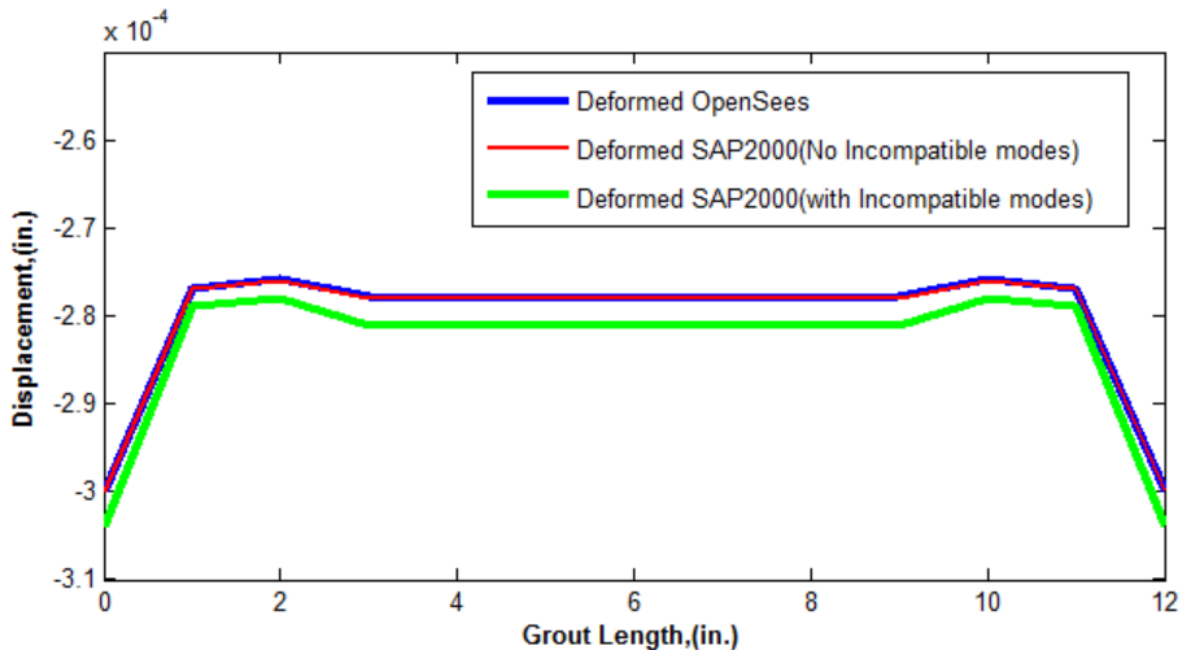


Figure 3.20. Deformations along the length of the grout under gravity load

### **3.3.1 Observations**

Figure 3.20 shows that the grout deformations along the length under the gravity load are almost same from the two programs. However, there is an intriguing observation shown by this plot. When an option given as 'incompatibility mode' for the solid elements in SAP2000 was switched off and the analysis was done again, then the values coming from OpenSees matched exactly with the values from SAP2000. The 'incompatibility mode' option in SAP2000 is provided to improve the bending behavior of a brick or solid element; when it is included, the bending modes are incorporated in the stiffness formulation of the element. Hence, the possible reason for the slight difference in the values obtained from OpenSees and SAP2000 (with 'incompatibility mode' included) can be attributed to the lack of incompatibility mode in OpenSees.

The next chapter discusses the development process of the preliminary models that were developed to understand the connection behavior under different load cases.

## **Chapter 4 Finite Element Modeling**

### **4.1 Introduction**

The objectives of this research are to explore the behavior of column-base connection under various load cases and develop a wizard which estimates a rotational stiffness of the column-base connection for specified input parameters. Two different programs, SAP2000 and OpenSees, are used as processors for estimating this stiffness. The first step to fulfill these objectives was to manually create three dimensional finite element models of a typical column-base connection in SAP2000 and OpenSees with various components such as column section, base-plate, anchor bolt and grout. It should be noted that some of the models also included the foundation as a component of the connection. The motive behind the inclusion of foundation was to observe its influence on the stiffness of the connection. In the modeling process, the above mentioned components were modeled using common dimensions, which were provided by the MBMA survey (see Appendix B). The interactions between various components were simulated to the level best to emulate the actual behavior of the connection under various load cases. The models were analyzed under two load cases, namely dead or gravity load case and live or wind load case. The results obtained from both SAP2000 and OpenSees were compared; they gave an insight into the behavior of the column-base connection under different load cases. In second phase of this research, a wizard was developed for entering the user defined values for various parameters of the column-base connection. The development process of the wizard and the considered parameters are presented in detail in Chapter 6. The following sections focus on the models that were developed manually in SAP2000 and OpenSees to understand the behavior of the connection.

### **4.2 Development of Finite Element Models**

To understand the behavior of the column-base connection, a three dimensional finite element model of a typical column-base connection which can closely simulate the actual behavior was created. There were two models created in SAP2000. The first model shown in Figure 4.1, included a column section, a base-plate and nonlinear spring elements (also referred as Nlink elements in SAP2000) as its major structural components. The second model shown in Figure 4.2, included a column section, a base plate, a grout layer and nonlinear spring elements. Please note that in both the models, the foundation was not included. However in some of the preliminary models the foundation was included to observe its influence on the flexibility of the connection. It was observed that due to its bulky size, the foundation contributes very less to the flexibility of the connection. Hence it was decided to replace the foundation with the fixed support. This also helped in reducing the analysis time. The dimensions for modeling different components of the above mentioned models were taken from the MBMA survey (see Appendix B). The dimensions, which were most common between various companies, were taken for the preliminary modeling process because the dimensions provided by each company were different for different components. The selection of finite elements for modeling different components was done keeping in mind that it should capture the actual behavior. The shell elements were used to model the column and base-plate to observe their flexible behavior. The solid or brick

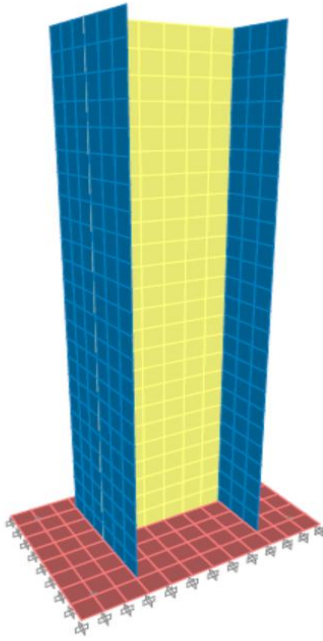


Figure 4.1. First model consisted of Column Section, Base Plate and NLink Elements

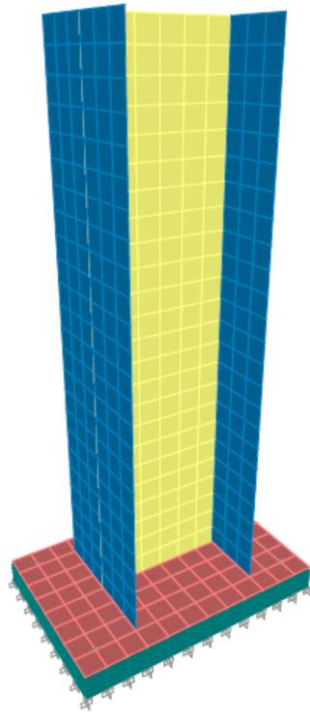


Figure 4.2. Second Model consisted of Column Section, Base Plate, Grout and Nlink Elements

elements were used to model the grout to observe the variation of the stresses with thickness of the component. The nonlinear interaction between the base-plate and the grout was simulated by using nonlinear spring (or Nlink) elements.

Thambiratnam and Paramasivam (1986) observed that the base plate bends under an eccentric loading or moment. The compression side of the base plate comes in contact with the grout and transfers the load. On the other hand, the tension side of base plate lifts up and loses contact with the grout. In short, compression on the base plate is resisted by the grout and tension is resisted by the anchor bolts present at the tension side of the base plate. As the area of the base plate under compression is variable and depends on the load configuration and base plate properties, the nonlinear spring elements (say compression Nlink elements) having force-deformation relationship shown in Figure 4.3 were attached to each node of the base plate at one end and the other end was fully restrained in first model and connected to the grout in the second model. These compression elements have very high stiffness in compression and near zero stiffness in tension. In essence, this force-deformation relationship represents that only compression in the base plate can be transferred to the grout and the tension will only result in uplifting of the base plate and losing contact with the grout. Note that the compression elements were attached to every node of the base plate except the nodes at the bolts' positions.

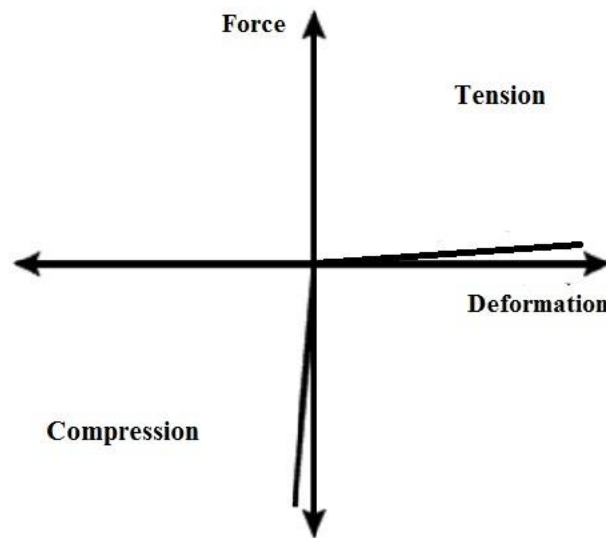


Figure 4.3. Force-Deformation relationship for compression Nlink elements

As mentioned above that the anchor bolts are responsible for resisting the uplift of the base plate, this seeks special attention in modeling of the anchor bolts. Since the objective of this project is to find the initial rotational stiffness of the connection, it is assumed that most of the components behave elastically under service loads. Hence, the same assumption is applied in modeling the anchor bolts. The tensile stiffness of an anchor bolt behaving elastically can be estimated by the relation given in Equation 4-1. This equation assumes the anchor bolt as a one dimensional long bar having an effective length anchored into the concrete foundation passing through the grout.

The effective length depends upon the variation of bond stress between the foundation and anchor rod.

$$K = \frac{A_{rod} * E}{L_{eff}} \quad (4-1)$$

where  $K$  is stiffness of anchor bolt,  $A_{rod}$  is area of cross section of anchor bolt,  $E$  is modulus of elasticity of steel and  $L_{eff}$  is effective length of anchor rod which depends on bond stress variation.

For the two models above, a linearly varying bond stress was assumed along the anchor rod length (shown in Figure 4.4) and the value of stiffness for this stress distribution is given by following relation (Equation 4-2).

$$K = \frac{A_{rod} * E}{L/2} \quad (4-2)$$

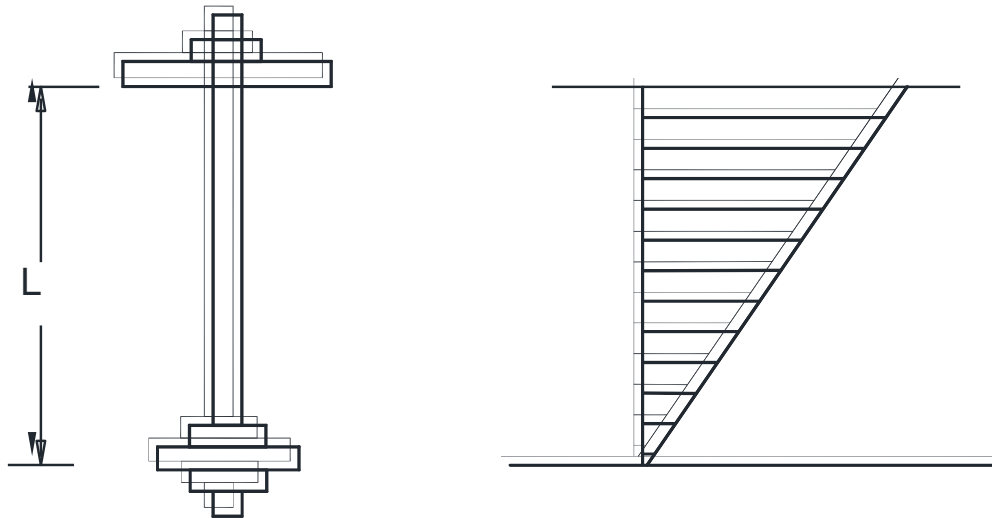


Figure 4.4. Assumed bond stress distribution along anchor rod length

The anchor bolts were modeled by using the nonlinear spring elements (say tension Nlink elements). These nonlinear spring elements were attached to the nodes representing the bolts' positions on the base plate at one end and restrained at the other. It was assumed the anchor bolts only resist tension in the base plate. These tension Nlink elements were assigned with a force-deformation relationship that represents the stiffness of an anchor bolt (given by Equation 4-1) on tension side and near infinite stiffness on compression side as shown in Figure 4.5 (refer Section 3.2.1 for detailed explanation on interpreting a force-deformation relationship). In



essence, the anchor bolts resist tension with some stiffness and transfer any compression on them to the grout without any deformation. An important difference between compression and tension Nlink elements in addition to the force-deformation relationship was connectivity at their ends. The compression elements were assigned with the base plate as one end and the grout or fixed support as other (depending upon the presence or absence of the grout). On the other hand, the tension elements were assigned with the base plate as one end and the other end was always restrained. This restraint was provided to simulate the anchored end condition of the bolts inside the concrete foundation. It was also assumed that there was no slip between the column stub and base plate and the contact interaction between the two was simulated by meshing them together.

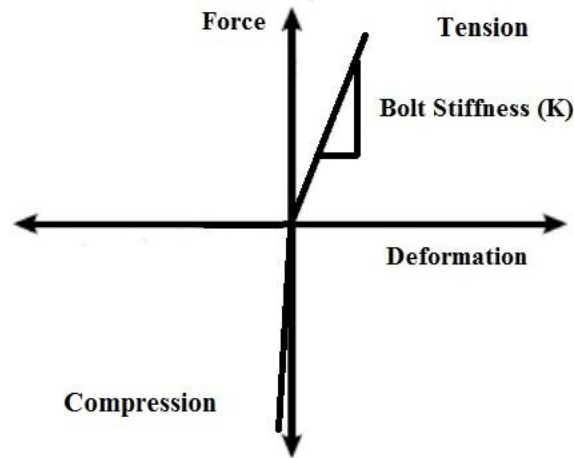


Figure 4.5. Force-Deformation relationship for tension Nlink elements

To observe the base-plate and grout deformation more closely, these components are meshed into small elements. These elements were created with aspect ratio (the ratio of any two dimensions of a finite element) less than four is to one (4:1) with the aim to obtain good results from the analysis. The lines of intersection of the column section planes and base plate were taken as boundary lines for creating these small meshed elements. The compression Nlink elements were drawn from the nodes of the finite elements on the base plate to the nodes of the finite elements on the grout top surface (when the grout is present), while the tension Nlink elements were drawn from the nodes of the finite elements on the base plate to the fixed support. It is to be noted that when the grout is not present in the model, the compression Nlinks elements are drawn from the nodes on base plate to the fixed support.

The two models were subjected to two loading cases – gravity load case and wind load case. The gravity load includes the loads which do not change with time and are fixed in magnitude. The weight of the structural components fall in this category. The gravity loads may not be act concentrically on the connection in reality, which results in unbalanced moments acting on the

connection. Therefore usually an axial force and a moment form a gravity load case. The wind load case includes the loads resulting from the application of wind on the structure. It usually includes a lateral force, an uplifting force or a downward force and an overturning moment. The application point for both the load cases was centroid of the cross section at the top of the column stub.

A nonlinear analysis was performed for each of the two load cases. The motive behind performing a nonlinear analysis was to observe the behavior of the connection at the base plate-grout interface. Since a nonlinear force-deformation relationship was assigned to the spring elements connecting the base plate and grout, a nonlinear analysis was the best possible way to capture the nonlinearity of the connection. The deformed shape of the base plate was observed under different load configurations. Some new models were created by changing the properties of different components of the connection. After performing lots of analyses on different models, it was concluded that the rationale behind the modeling is reasonable and can be carried further to find the rotational stiffness of the connection.

To achieve the final goal of this research, which is to develop a wizard for finding the stiffness of the connection, a graphical user interface (GUI) has to be developed. To develop the GUI, a graphical user interface building software is required. Since MATLAB (MathWorks, 1984-2011) is a very robust program and has the capability by which one can design the GUI's, it is selected as GUI building tool for this research. The designed GUI has to interact with selected analysis program which is either SAP2000 or OpenSees. This simply means that prior to designing of the GUI, one should know how to access the above mentioned analysis programs from MATLAB. The next chapter presents different ways and procedures by which one can access these programs from MATLAB.

## Chapter 5 Accessing Different Programs from MATLAB

### 5.1 Introduction

Since SAP2000 and OpenSees are used as analysis engines and MATLAB is used for developing the Graphical User Interface (GUI) for this research, this means that MATLAB should somehow be able to access the above two programs for modeling and performing analysis. This chapter describes the ways in which the connectivity between the two programs and MATLAB can be developed.

### 5.2. Running SAP2000 through MATLAB

SAP2000 provides us with a programming interface known as Application Programming Interface (API) through which a developer can access SAP2000 from external programs such as Microsoft excel, Visual Basic, Visual C#, Visual C++, Intel Visual Fortran, MATLAB etc. This API consists of set of commands which access the drawing objects, editing objects, analyzing objects and output objects present in SAP2000. A developer can choose a program of his choice from the list of above given programs and write routines for modeling and analyzing a structure by making use of syntaxes (or commands) and data types of that particular chosen program and API commands of SAP2000. API is advantageous for the developers who wish to use SAP2000 as an analysis tool for their own program, since they can obtain any analysis information of the structure without directly interfacing with the SAP2000 window.

For this research project, MATLAB has been chosen as program for interacting with the API due its popularity and numerical versatility. Moreover, it can be used to develop the GUI which can be used to input the values of various parameters of a structure to be modeled and display the processed results to the user.

As a first step in using the SAP2000 API, two files having `.m` extension (MATLAB script files), were written in MATLAB which automatically create, analyze the two models (mentioned in Section 4.2) in SAP2000 and fetch the results of analyses into the MATLAB command window. The content of each `.m` file starts with a set of systematic commands which develop connectivity between MATLAB and SAP2000. There are two important commands which should be defined to access the API through MATLAB. First is to pass the data to SAP2000 in one dimensional arrays. This can be declared through MATLAB by using following command:

```
feature('COM_SafeArraySingleDim', 1);           (#1)
```

Secondly, all the arrays should be passed *by reference* instead of passing *by value*. This simply means that SAP2000 accesses the original arrays defined in MATLAB rather than their copies. Moreover, whatever changes are done in SAP2000, they are reflected in the original array defined in MATLAB. This can be declared in MATLAB by using following command.

```
feature('COM_PassSafeArrayByRef', 1);           (#2)
```

After declaring the above two commands, a SAP2000 object named as *SapObject* is created. This object is used to access SAP2000 by making use of the API commands. The following commands can be used to create a SAP2000 object and start a SAP2000 application.

```
SapObject = actxserver('SAP2000.SapObject');    (#3)
```

```
SapObject.ApplicationStart;                    (#4)
```

```
SapModel = SapObject.SapModel;                (#5)
```

Figure 5.1 shows a snapshot of the MATLAB editor defining the above commands and setting up an interface between MATLAB and SAP2000.

```
%% clean workspace & command window
%clear;
%clc;

%% pass data to Sap2000 as one-dimensional arrays
feature('COM_SafeArraySingleDim', 1);

%% pass non-scalar arrays to Sap2000 API by reference
feature('COM_PassSafeArrayByRef', 1);

%% create Sap2000 object
SapObject = actxserver('sap2000.SapObject');

%% start Sap2000 application
SapObject.ApplicationStart;

%% create SapModel object
SapModel = SapObject.SapModel;
```

Figure 5.1. SAP2000 API commands defined in MATLAB editor window

By making use of API commands, the model is created automatically from MATLAB by providing the dimensions and meshing information in the form of arrays. The material properties and sections for various components of the column base connection are also defined in arrays and passed to SAP2000 in the form of commands. Most of the commands can be written in generalized form as shown by Command #6. Here *ret* is a variable defined in MATLAB which gives 0 on correct execution of a command, otherwise returns 1.

```
ret = SapModel.(Command line 1).(Command line 2).(Input argument 1,
Input argument 2,.....);                                (#6)
```

After execution of a number of modeling commands which generally have the form of Command #6, the modeling process is completed. For performing analyses and getting results back to the MATLAB command window, some specific commands are executed (Commands #7, #8 and #9). The first command simply runs the analysis. It should be noted that various load cases can be analyzed in a single analysis. The second command deselects all the load cases which are analyzed and the third command selects the specific load case which is desired for obtaining the results.

```
ret = SapModel.Analyze.RunAnalysis();                    (#7)
```

```
ret = SapModel.Results.Setup.DeselectAllCasesAndCombosForOutput; (#8)
```

```
ret = SapModel.Results.Setup.SetCaseSelectedForOutput('Load Case');
                                                                    (#9)
```

Finally, the connection server, which is established between SAP and MATLAB, is closed and a null value is given to all the earlier defined objects such as SapObject and SapModel. This process is very essential, as without doing this one cannot establish the server next time on execution of the *.m* file.

```
ret = SapObject.ApplicationExit(false());                (#10)
```

```
SapModel = 0;                                           (#11)
```

```
SapObject = 0;                                          (#12)
```

A GUI window is created for this research to enter all the parameters which are required to model the connection in SAP2000 (or OpenSees). The GUI window takes the values of all the parameters from the user and place them in the commands defined in above described *.m* files. When the analysis is triggered from the GUI, these commands model and analyze the connection according to the input values. The output of the analysis from SAP2000 is brought back to MATLAB and processed further to produce some plots such as moment rotation plot, bearing pressure plot and provide quantities such as stiffness of the connection and maximum bearing pressure on the base plate. Further discussion on development and working of the GUI developed for this research can be found in Chapter 6.

### 5.3. Modeling in OpenSees

The Open System for Earthquake Engineering Simulation better known as OpenSees is an open source object oriented program that allows users to perform finite element analysis of structural systems. This is mainly used for structural systems which are subjected to earthquakes. It is

primarily written in C++, C and FORTRAN. This program is chosen for our analysis due to its free availability and versatility in performing complex structural analysis. It is a powerful tool to perform dynamic analysis in addition to static analysis. Figure 5.2 shows an interacting window of OpenSees.

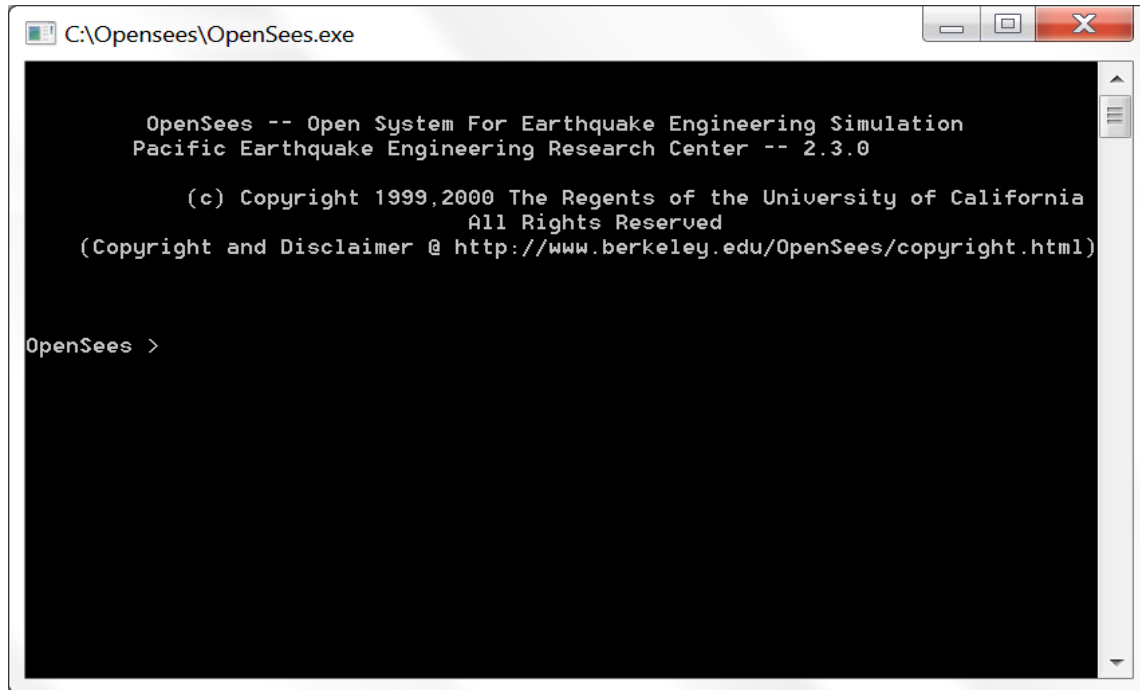


Figure 5.2. OpenSees Interactive Window

OpenSees is extended by using the Tcl language, which is a string based general purpose command language. Therefore, one has to download a *dynamic link library* file (*.dll* file) for the Tcl interpreter along with the OpenSees program. OpenSees provides the user with a command library which consists of following Tcl command types:

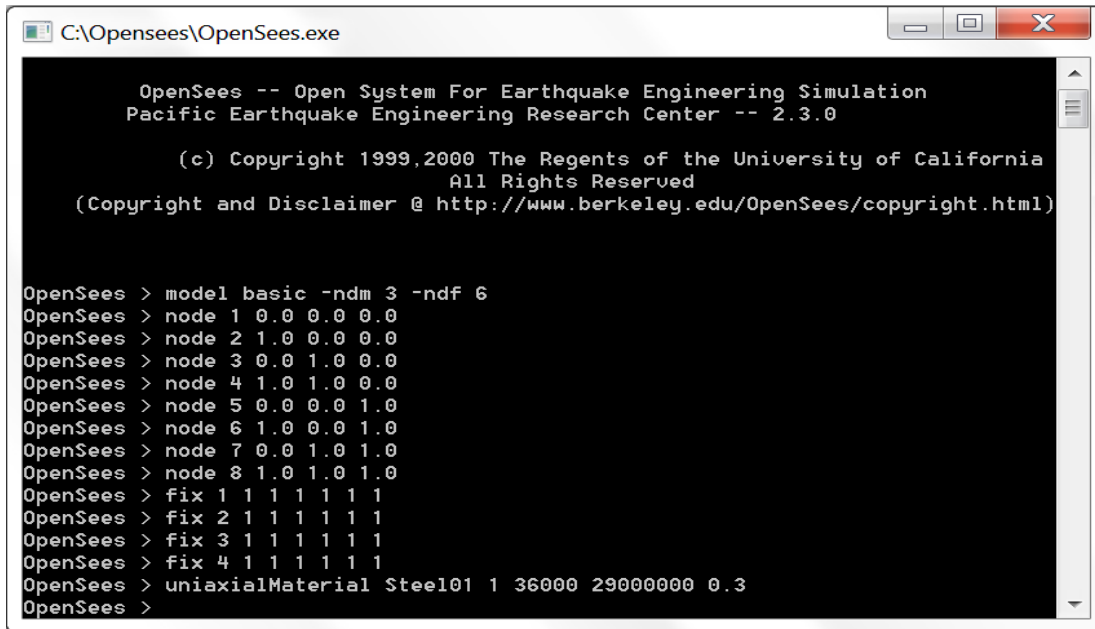
- Modeling Commands
- Analysis Commands
- Output Commands
- Misc Commands
- Database Commands

The above given Tcl command types enable a user to model and analyze any desired structure, and process results as per his requirements.

### 5.3.1 Executing OpenSees Commands

There are three ways to execute OpenSees or Tcl commands. The first method is to input or write appropriate commands on the interactive window displayed by OpenSees shown in Figure 5.2. A sample set of commands for defining a structural configuration looks something like Figure 5.3. The interactive window looks just like command prompt, hence it is referred as the OpenSees

command prompt in rest of the thesis. The reader is advised not to confuse the OpenSees command prompt with the MS-DOS or Windows command prompt.



```
C:\Opensees\OpenSees.exe

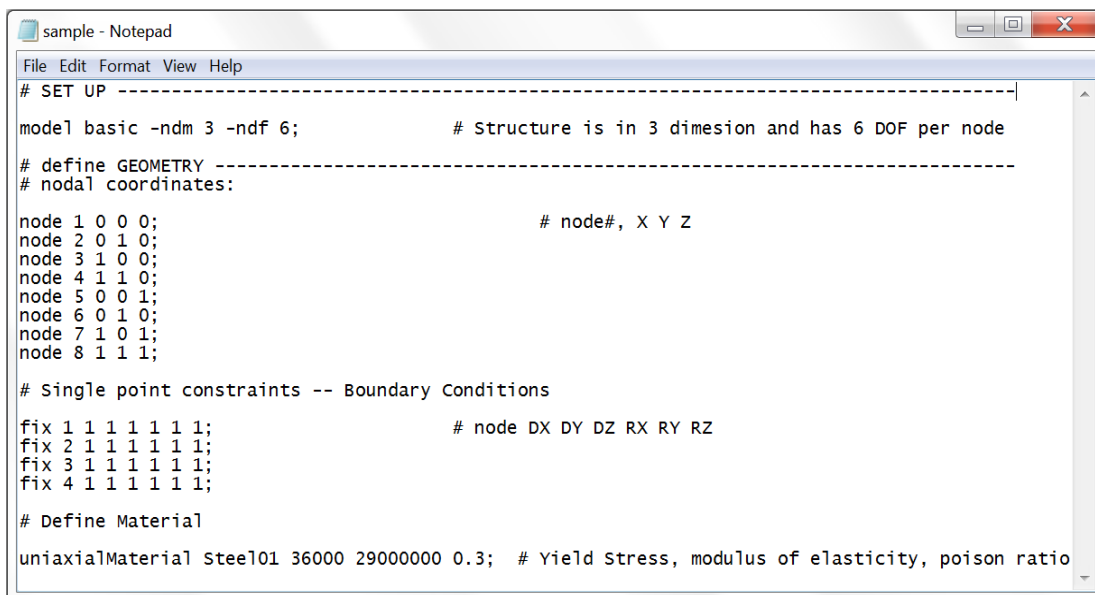
OpenSees -- Open System For Earthquake Engineering Simulation
Pacific Earthquake Engineering Research Center -- 2.3.0

(c) Copyright 1999,2000 The Regents of the University of California
All Rights Reserved
(Copyright and Disclaimer @ http://www.berkeley.edu/OpenSees/copyright.html)

OpenSees > model basic -ndm 3 -ndf 6
OpenSees > node 1 0.0 0.0 0.0
OpenSees > node 2 1.0 0.0 0.0
OpenSees > node 3 0.0 1.0 0.0
OpenSees > node 4 1.0 1.0 0.0
OpenSees > node 5 0.0 0.0 1.0
OpenSees > node 6 1.0 0.0 1.0
OpenSees > node 7 0.0 1.0 1.0
OpenSees > node 8 1.0 1.0 1.0
OpenSees > fix 1 1 1 1 1 1
OpenSees > fix 2 1 1 1 1 1
OpenSees > fix 3 1 1 1 1 1
OpenSees > fix 4 1 1 1 1 1
OpenSees > uniaxialMaterial Steel01 1 36000 29000000 0.3
OpenSees >
```

Figure 5.3. Method 1: Defining model at OpenSees Prompt

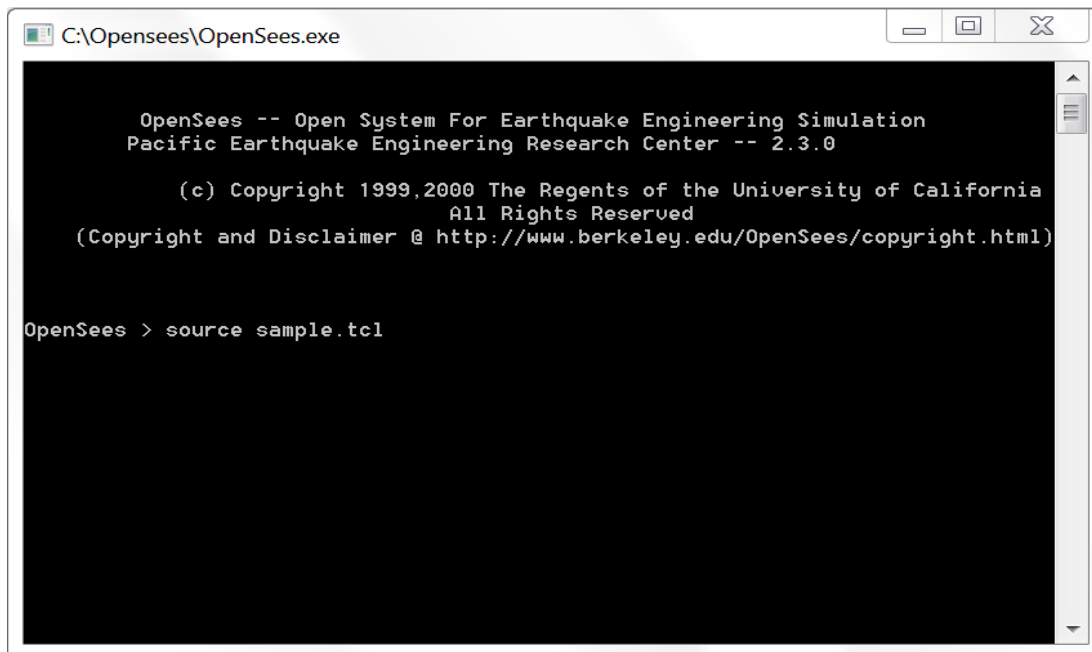
The second method is to write an input file separately in some word processing software such as notepad, Microsoft Word, etc. and save it with *.tcl* extension. This file consists of similar set of commands which are defined at the OpenSees command prompt in Method 1. A *sample.tcl* file is created to demonstrate this method is shown in Figure 5.4.



```
sample - Notepad
File Edit Format View Help
# SET UP -----
model basic -ndm 3 -ndf 6;           # Structure is in 3 dimesion and has 6 DOF per node
# define GEOMETRY -----
# nodal coordinates:
node 1 0 0 0;                       # node#, X Y Z
node 2 0 1 0;
node 3 1 0 0;
node 4 1 1 0;
node 5 0 0 1;
node 6 0 1 0;
node 7 1 0 1;
node 8 1 1 1;
# Single point constraints -- Boundary Conditions
fix 1 1 1 1 1 1;                     # node DX DY DZ RX RY RZ
fix 2 1 1 1 1 1;
fix 3 1 1 1 1 1;
fix 4 1 1 1 1 1;
# Define Material
uniaxialMaterial Steel01 36000 29000000 0.3; # Yield Stress, modulus of elasticity, poison ratio
```

Figure 5.4. Method 2: Input file 'sample.tcl' is created

The user can execute this input file from the OpenSees command prompt by calling the *source* command followed by the *filename.tcl*. Figure 5.5 shows the above procedure.



```
C:\Opensees\OpenSees.exe

OpenSees -- Open System For Earthquake Engineering Simulation
Pacific Earthquake Engineering Research Center -- 2.3.0

(c) Copyright 1999,2000 The Regents of the University of California
All Rights Reserved
(Copyright and Disclaimer @ http://www.berkeley.edu/OpenSees/copyright.html)

OpenSees > source sample.tcl
```

Figure 5.5. Calling input file ‘sample.tcl’ from OpenSees command prompt

Please note that the written input file should be saved in the same folder in which OpenSees.exe is present. This is done to make sure that when the user calls this input file at the OpenSees command prompt, it will be available to OpenSees.

The third method which is least comfortable, is calling the *.tcl* input file through the MS-DOS command prompt. This way of executing OpenSees commands is rarely used. In general, the second method is much easier than the other two. Some of the advantages of this method over the other two are following.

- For modeling large structural systems, one has to input many lines of data. Therefore, the possibility of inputting wrong data is always there. Using a separate editable file for this purpose makes work much easier and systematic as one can edit any mistake any time.
- A developer can save progress of the modeling and call the input file any time by using OpenSees command prompt.
- The most important and useful aspect of using the second method is that developers can use external programs like MATLAB to write their input files and execute them without even opening notepad or OpenSees.



### 5.3.2 Running OpenSees through MATLAB

As discussed in the previous section, there are three possible ways of executing commands in OpenSees. The second mode of running is most suitable for a developer. A Tcl scripted file has *.tcl* extension. It is nothing but a notepad or any word processor file which has been saved in *.tcl* format. A developer can make use of MATLAB to write a Tcl input file in notepad using specific commands. This Tcl script is written on a blank M file (default MATLAB file having *.m* extension) by using following commands.

```
file=fopen('fileName.tcl','w+');           (#13)
```

```
fclose(file);                               (#14)
```

Command #13 opens or creates a file having name as *fileName* and extension as *.tcl*. It also makes the reading and writing of this file possible. Command #14 closes the writing process and it is normally put at the end in M file. The OpenSees commands are printed in this Tcl file using the following generalised MATLAB command.

```
fprintf(file,'OpenSees command') ;         (#15)
```

Command #15 uses *fprintf* (a MATLAB command) to write the data in text format in the *file* variable which holds *fileName.tcl* (defined by Command #13). The user has to execute the above command each time when he or she wants to add an OpenSees command in the Tcl file. The process explained above will just create an input file and one still has to analyze the model in OpenSees. For accessing OpenSees from MATLAB, the following command is used.

```
!OpenSees fileName.tcl                     (#16)
```

Command #16 connects to OpenSees and runs *fileName.tcl*. It is written after the *fclose* command (Command #14) in the *.m* file. Please note that the Command #16 only executes when OpenSees.exe is present in the same folder in which the input file *fileName.tcl* is present. The analysis results are written in recorders, which are output files. Each file is specified with a path where it is to be saved. These recorders are defined with their paths inside the Tcl file. To process the analysis results stored in output files further, they are stored in variables in the MATLAB workspace by using following commands.

```
Disp1=load('Disp1.out')                    (#17)
```

```
Force=load('eleLocal.out')                 (#18)
```

Command #17 and #18 extract the data from the output files, which are *Displ.out* and *eleLocal.out* in this case and store that into variables *Displ* and *Force*, respectively. Now the user can process the information stored in these variables by performing usual operations of MATLAB. A sample file showing the use of above set of commands in MATLAB is shown in Figure 5.6.

```

1 - file=fopen('breakdown.tcl','w+');           % Open or creates a file name breakdown.tcl
2 - %Model for three dimensional truss element
3 - fprintf(file,'model basic -ndm 3 -ndf 3; \r\n'); % three dimension and three DOF per node
4 - fprintf(file,'file mkdir Data; \r\n');      % creates a directory 'data' for output
5 - fprintf(file,'# nodal coordinates: \r\n');
6 - % Add material for non linear springs
7 - fprintf(file,'uniaxialMaterial ElasticBilin 1 3030.30 3030.30 0.000001 1000000 1000000 -0.000001;\r\n');
8 - % Defining nodes
9 - fprintf(file,'node 1 0 0 0; \r\n');
10 - fprintf(file,'node 2 0 0 0.5; \r\n');
11 - fprintf(file,'fix 1 1 1 1; \r\n');
12 - fprintf(file,'equalDOF 1 2 1 2; \r\n');
13 - % Defining truss element
14 - fprintf(file,'element truss 1 1 2 0.5 1; \r\n');
15 - % Define analysis method
16 - fprintf(file,'recorder Node -file Data/example.out -time -node 2 -dof 1 2 3 disp; \r\n');
17 - fprintf(file,'pattern Plain 1 Linear {load 2 0.0 0.0 -50.0};\r\n');
18 - fprintf(file,'constraints Plain;\r\n');
19 - fprintf(file,'numberer Plain;\r\n');
20 - fprintf(file,'system ProfileSPD;\r\n');
21 - fprintf(file,'integrator LoadControl 0.1; \r\n');
22 - fprintf(file,'algorithm Linear; \r\n');
23 - fprintf(file,'analysis Static; \r\n');
24 - fprintf(file,'analyze 10;\r\n');
25 - fprintf(file,'puts "Done!";
26 - fclose(file);
27 - !OpenSees2.3.1 breakdown.tcl

```

Figure 5.6. MATLAB script executing modeling and analysis commands in OpenSees

This chapter described the ways in which SAP2000 and OpenSees can be accessed from MATLAB. The general commands which are written in MATLAB to model and analyze any structure in the above two programs are also outlined. The following chapter provides a comprehensive description of the GUI development process. Various parameters of a column base connection which are considered in designing of the GUI are also presented in a systematic manner.

## Chapter 6 Graphical User Interface (GUI) Development

### 6.1. Introduction

A Graphical User Interface (GUI) is a graphical or visual display with which a user can interact and perform a desired task without knowing the complexities of operations performed to fulfill that task. The research presented herein required the development of a GUI for calculating the rotational stiffness of any configuration of a column base connection. A GUI can be developed by using any GUI building software present in the market. The software used for developing the GUI for this research is MATLAB. A user can develop a GUI either by manually creating each component with a written code or by using a GUI builder provided in MATLAB, better known as GUIDE.

As far as the functionality of a GUI is concerned, each GUI consists of components such as text boxes, push buttons, radio buttons, menu bar, axes, list boxes, etc. In MATLAB, each of these components has a user defined routine or program known as Callback, which asks MATLAB to perform certain operation whenever the user triggers an action on that component. This action is called an event and can be different for different components. For instance, this action can be clicking for a pushbutton, entering text for a textbox, clicking on circular dots for a radio button, selecting an item from the list for a list box. It is the responsibility of the developer to keep in mind the needs of the users and define appropriate callback for each component. These callbacks decide how components are going to handle the events.

A sample GUI performing the operations of a simple calculator is shown in Figure 6.1. This GUI contains push buttons, edit text boxes and static text boxes. Each of these components has a callback function which performs a specific action displayed by the label situated on left side of every component. '1<sup>st</sup> No.' and '2<sup>nd</sup> No.' edit text boxes let the user enter a value for each number. Each push button on clicking performs the labeled action and shows the result in 'Answer:' edit text box.

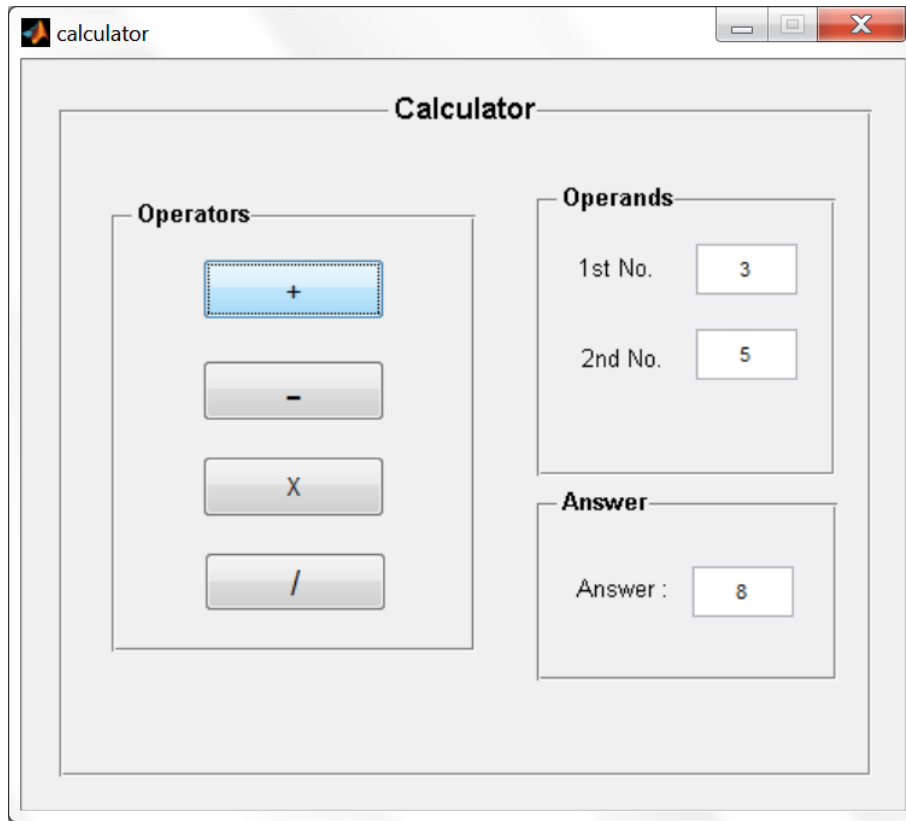


Figure 6.1. A GUI performing task of a simple calculator

As mentioned in Chapter 4, every preliminary model of the connection consisted of components such as base plate, anchor bolts, column and grout (in some models). This means that for modeling each component the values for various parameters must be provided. After a close review of all the parameters which are considered for different components, they are divided into inputs and outputs. In this wizard, the inputs are the parameters used for modeling and analyzing the column-base connection in OpenSees and SAP2000. The input parameters for modeling include dimensions of the base plate, grout and column section; material properties of the steel and concrete; Anchor bolt parameters like anchor bolt length, position of bolts and mesh information, etc. The input parameters for analyzing mainly include applied loads, namely gravity loads and wind loads in various directions and a processor desired for analyzing, either SAP2000 or OpenSees. Table 6-1 summarizes the modeling parameters which are considered for creating various input fields in the GUI. The output parameters in this wizard are estimate of the rotational stiffness, maximum bearing pressure on the base plate, moment-rotation and bearing pressure plots.

In essence, a user enters the input parameters for a desired structural model in the GUI. The GUI calls the *.m* file (described in Chapter 5) for either SAP2000 or OpenSees, based on the user's selection of analysis program. The *.m* file creates an input file for the chosen analysis program, which takes the input values of various parameters, models and analyzes the connection in

respective program. The results obtained from the analysis program are saved and processed in MATLAB to get the output parameters mentioned in previous paragraph.

Table 6-1. Input parameters considered while designing the GUI

Component Name	Geometrical Properties	Material Properties	Finite Element Used in Analysis
Base Plate	Dimensions: $l_{bp}$ : Length of base plate $b_{bp}$ : Width of base plate $t_{bp}$ : Thickness of base plate	$f_y$ : Yield strength of steel used for base plate $\nu_s$ : poison ratio of steel $E_s$ : Modulus of elasticity of steel	Thick Shell - SAP2000  ShellMITC4 - OpenSees
Grout	Dimensions: $t_g$ : Thickness of the grout	$f'_g$ : Compressive strength of concrete $E_c$ : Modulus of elasticity of concrete used for grout $\nu_c$ : poison ratio of concrete	Solid – SAP2000  8 node Brick - OpenSees
Column Section	Either choose a section from the list of W sections or customize a section.  $L_c$ : Length of the column stub	$f_y$ : Yield strength of steel used. $E_s$ : Modulus of elasticity of steel used for column $\nu_s$ : poison ratio of steel	Thick Shell - SAP2000  ShellMITC4 - OpenSees
Anchor Bolts	N: Number of bolts $X_b$ : Horizontal distance from origin in the plane of the base plate $Y_b$ : Vertical distance from origin in the plane of the base plate D: diameter of bolt L: Anchor bolt length	$f_y$ : Yield strength of steel used for anchor bolts $E_s$ : Modulus of elasticity of steel $\nu_s$ : poison ratio of steel	Nlink Element - SAP2000  2 node Truss Element - OpenSees

## 6.2 GUI Development Process in MATLAB

There are two ways to create a GUI in MATLAB. The first way is to use a GUI development environment better known as GUIDE. This is a tool kit provided by MathWorks, the developers of MATLAB software, to provide a user the ability to create a GUI with a layout editor. The user

can add any control from the list of controls provided. Note that the term 'controls' here refers to the components of a GUI such as text boxes, check boxes, list boxes, axes, pushbuttons, etc. The drawn GUI 'figure' (a 'figure' is a display window of any GUI such as Figure 6.1) and its components are saved as *.fig* file. This *.fig* file automatically creates a *.m* file, which is corresponding code file consists of created functions and callbacks of various components present in the *.fig* file. One can run the GUI from either FIG file or *.m* file. The second way of creating a GUI is programmatic approach in which one writes a *.m* file having codes to create the figure, its components and their callbacks. When this file is run then a figure is formed with various components added to it. The interaction of various components of the GUI with actions triggered by the user depends on the callbacks defined in *.m* file.

In this project the first way of creating a GUI is followed. The GUIDE lets the user to create a GUI without knowing the complex codes that one has to write explicitly if it is designed by the second way. Although the second approach is not explicitly used, it is followed without realization when the user modifies the codes generated by the GUIDE. The *.fig* file developed for this project is shown in Figure 6.2 and is referred as main figure in this thesis. Please note that a standalone version of the GUI or the wizard is created which enables the user to run the wizard on any computer without having MATLAB installed on it.

The main figure is divided into four major components namely panels, action buttons, menu bar and display windows (see Figure 6.3). The user can interact with first three components but the fourth component is just a visual interface to display the designed model of the connection and results. Note that each panel is a small section in the wizard which describes the materialistic and geometrical properties of a specific component of the connection. Some of the panels in this wizard are Column Properties, Grout Properties, Load Application, etc. All the panels have input fields for entering the values for various modeling parameters. Action buttons and menu bar items are objects which perform specific tasks on clicking them.

It should be noted that in addition to the above mentioned components, there are few elements which are not displayed on the main figure. Instead they are displayed in new figures over which the user has no control. They are just for displaying output plots, namely moment-rotation plot and bearing pressure plot. A comprehensive discussion of various features provided in this wizard is presented in the following sections.

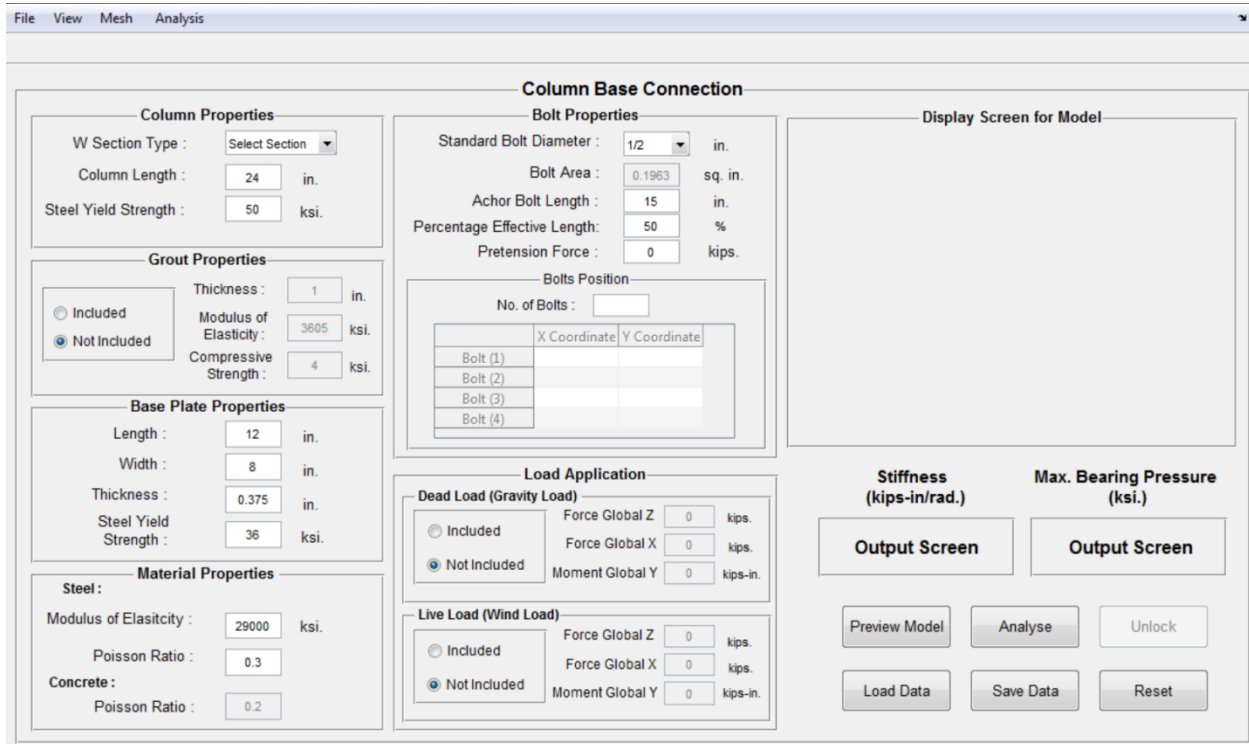


Figure 6.2. Display Window of Column Base Connection Wizard

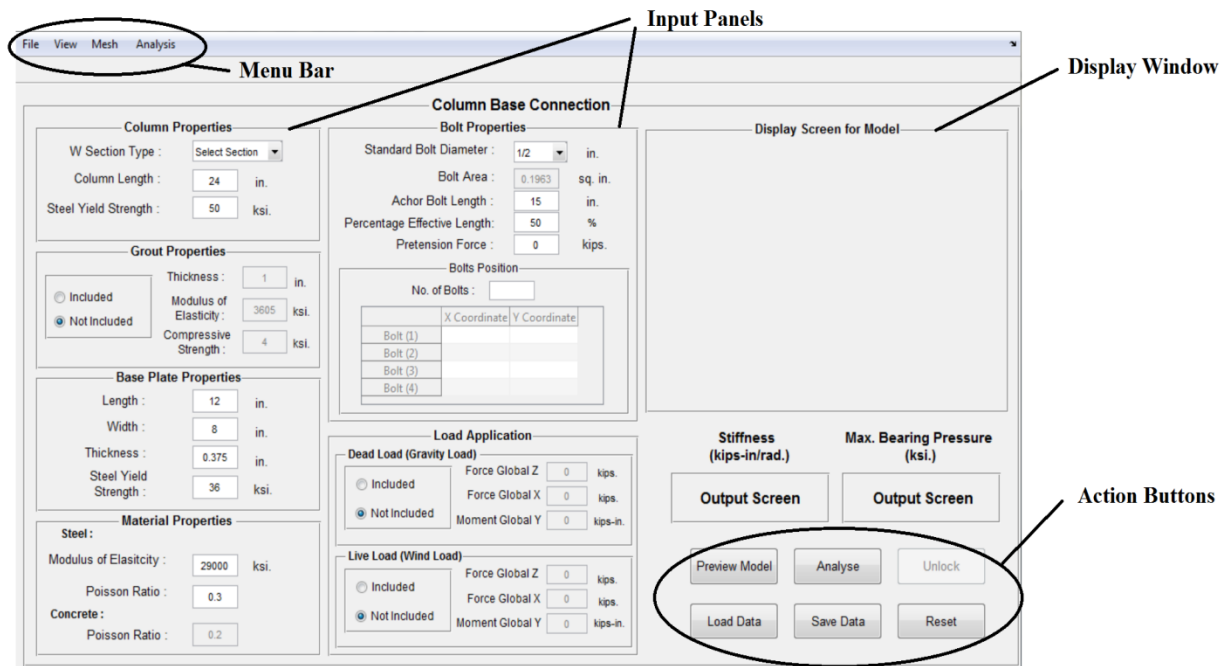


Figure 6.3. Major Components of Column Base Connection Wizard

## 6.2.1 Input Panels

An input panel is a small area assigned for each component in the GUI. Each panel contains geometrical and material properties pertaining to a specific component. There are six input panels provided in this wizard. The following is the list of panels provided.

- Column Properties
- Grout Properties
- Base Plate Properties
- Material Properties
- Bolt Properties
- Load Application

The following sections describe the input fields present in each of these panels.

### 6.2.1.1 Column Properties

It contains three input fields. W section type is a list box which contains standard AISC wide flange sections and an option to create tapered section. The item 'Tapered Section' from the list can be chosen to model a tapered section. The list box is shown in Figure 6.4. The length and yield strength of the column stub are entered in 'Column length' and 'Steel Yield Strength' text fields, respectively. The default values for the length and yield strength are provided as 24 in. and 36 ksi., respectively.

The screenshot displays the 'Column Base Connection' GUI. The 'Column Properties' panel is active, showing a list box for 'W Section Type' with options: 'Select Section', 'Tapered Section', and a list of standard AISC wide flange sections (W27 x 178, W27 x 161, W27 x 146, W27 x 114, W27 x 102, W27 x 94, W27 x 84, W24 x 162, W24 x 146, W24 x 131, W24 x 117, W24 x 104, W24 x 94, W24 x 84, W24 x 76, W24 x 68, W24 x 62, W24 x 55). Other panels include 'Bolt Properties' (Standard Bolt Diameter: 1/2 in., Bolt Area: 0.1963 sq. in., Anchor Bolt Length: 15 in., Percentage Effective Length: 50%, Pretension Force: 0 kips.), 'Load Application' (Dead Load and Live Load options), 'Material Properties' (Steel: Modulus of Elasticity: 29000 ksi., Poisson Ratio: 0.3; Concrete: Poisson Ratio: 0.2), and 'Display Screen for Model' (Stiffness and Max. Bearing Pressure output screens).

Figure 6.4. W Section Type list box in Column properties panel



### **6.2.1.2 Grout Properties**

This panel contains a choice box which provides the user an option to include or exclude the grout from the connection. If the grout is included then three input fields are enabled. The field 'Thickness' specifies the thickness of grout beneath the base plate; 'Modulus of Elasticity' and 'Compressive Strength' fields specify the modulus of elasticity and strength of the concrete used to build the grout, respectively. The default values for 'Thickness', 'Modulus of Elasticity' and 'Compressive Strength' are provided as 1 in., 3605 ksi. and 4 ksi. respectively. Please note that the value of compressive strength provided by the user is used by the wizard to provide warning when the maximum bearing pressure on the base plate exceeds the compressive strength.

### **6.2.1.3 Base Plate Properties**

This panel contains critical information about the geometry and material of the base plate. 'Length', 'Width' and 'Thickness' fields specify the geometry of the base plate and 'Steel Yield Strength' specifies the yield strength of steel used for making the base plate. The default values for 'Length', 'Width', 'Thickness' and 'Steel Yield Strength' are provided as 12 in., 8 in., 0.375 in. and 36 ksi. respectively. Please note that the value of yield strength provided by the user is used by the wizard to provide warning when the loads applied on the connection creates yielding of the base plate and exceeds the allowable strength of the connection.

### **6.2.1.4 Material Properties**

Mechanical properties of the materials used for building the connection are specified here. A default value of 29000 ksi is provided as modulus of elasticity of steel and default values of 0.3 and 0.2 are provided for the poisson ratio of the steel and concrete respectively.

### **6.2.1.5 Bolt Properties**

This panel contains several input fields related to anchor bolts. 'Standard Bolt Diameter' provides a drop down list containing standard bolt sizes given by AISC. The user can also model a desired bolt by clicking on 'other' option from the list. Anchor rod, which is used for providing anchorage in the connection, its length is provided in 'Anchor Bolt Length' input field. 'Percentage Effective Length' field contains percentage of anchor length effective in providing the stiffness. This effective length depends on the bond stress between concrete and steel (discussed in Section 4.2). The pretension force in each bolt is provided in 'Pretension Force' field. The number of bolts can be provided in 'Number of Bolts' field and their positions can be defined in terms of X and Y coordinates in the table provided. The default values for 'Standard Bolt Diameter', 'Anchor Bolt Length', 'Percentage Effective Length' and 'Pretension Force' are provided as 1/2 in., 15 in., 50 % and 0 kip respectively.

### **6.2.1.6 Load Application**

To estimate the stiffness of the connection, it is subjected to two load cases, namely gravity load and wind load. The gravity load case contains all the loads on the connection which are acting due to weight of other structural components sitting on top of the connection. The wind load case contains the loads which are acting due to wind. In load application panel, two small panels are given for gravity or dead loads and wind or live loads respectively. Each of these panels contains

three input fields to provide loads in three directions. Please note that the positive directions for the loads are shown in the display screen of the GUI. When the user wants to input a load in the opposite direction to that of shown in the display screen, he should provide the load with a negative sign. Also note that all axial loads in the direction of gravity should be provided with a negative sign and the wizard works fine with the application of uplift loads.

## 6.2.2 Menu Bar

There are four submenus in the menu bar of the GUI. The functionality of each submenu and its options are discussed below.

### 6.2.2.1 File

'File' submenu consists of options which affect file properties. The options present in 'File' submenu are shown in Figure 6.5. The function of 'New' is to replace the current GUI with a new GUI, which contains default values of all the parameters. 'Open' loads any previously saved file in the current wizard window. 'Save As' provides an interactive window to the user for saving the currently entered data to a file for future reference. The user can save the data at any location in the computer with a desired file name. 'Close' option closes the GUI. However before closing it asks the user whether he wants to save the data. If the user wants to save, then he can, otherwise the wizard gets closed.

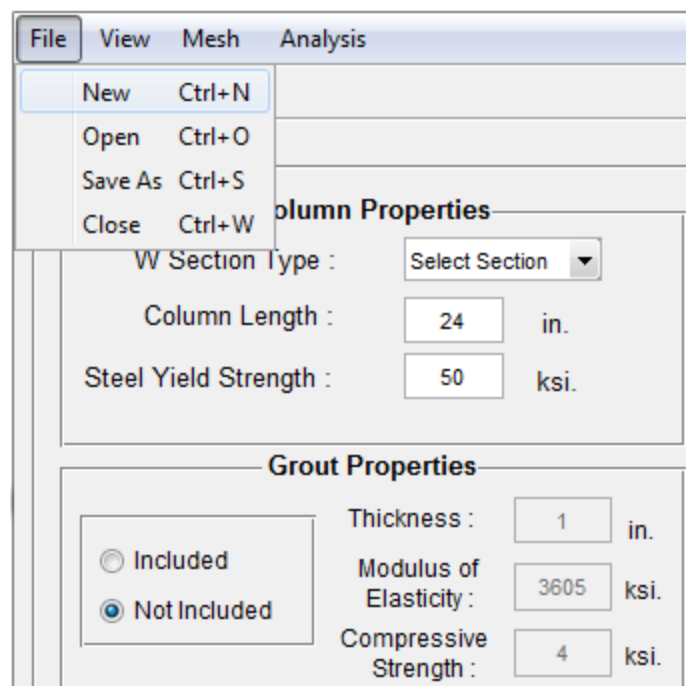


Figure 6.5. Options provided in File submenu

### 6.2.2.2 View

'View' submenu contains various options for viewing the created model from different angles. 'Original' View shows the three dimensional view of the model on the display screen. 'XY view', 'YZ view' and 'XZ view' options show the orthographic views of the model in three different orthogonal directions. Figure 6.6 shows the options provided by 'View' submenu.

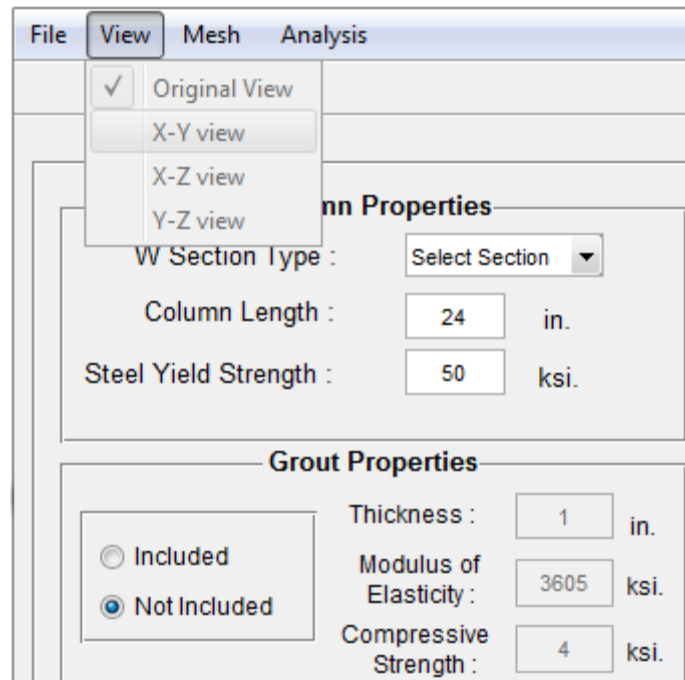


Figure 6.6. Options provided in View submenu

### 6.2.2.3 Mesh

'Mesh' submenu contains two options, namely coarse mesh or fine mesh. The type of meshing which is chosen by the user is considered in the analysis engine. When a coarse mesh is chosen, the model is meshed into small elements having aspect ratio less than 4:1 (aspect ratio is the ratio of any two dimensions of a finite element). On the other hand, when fine mesh is chosen then aspect ratio is less than 2:1. Although, the fine mesh analysis is more accurate, it takes more time to analyze the model. Clearly, there is a tradeoff between accuracy and time taken to analyze. A prudent call has to be made by the user on the basis of accuracy requirement. The default mesh type provided in this wizard is coarse mesh. Figure 6.7 shows the options provided by 'Mesh' submenu.

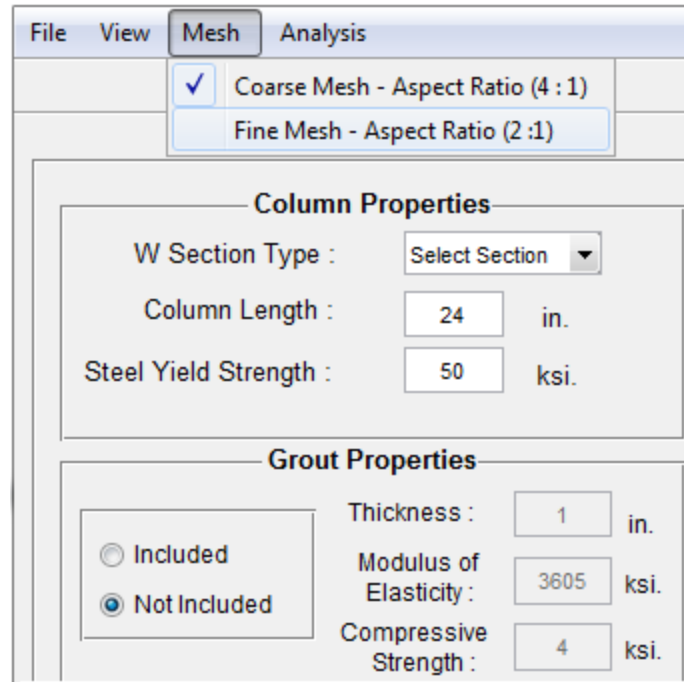


Figure 6.7. Options provided in Mesh submenu

#### 6.2.2.4 Analysis

This submenu provides the user with the options of two analysis engines for performing the analysis of the built model. The two analysis engines are SAP2000 and OpenSees. Note that the results obtained from both the programs are dependent on the finite elements used for modeling the connection. It can be seen from the Chapter 3 that the responses of the finite elements in both the programs under certain load cases are not identical, but fairly close. This means that the final values of stiffness and maximum bearing pressure from both the programs may not be identical, but they will be fairly close. The default analysis engine for this wizard is OpenSees. Figure 6.8 shows the options provided by Analysis submenu.

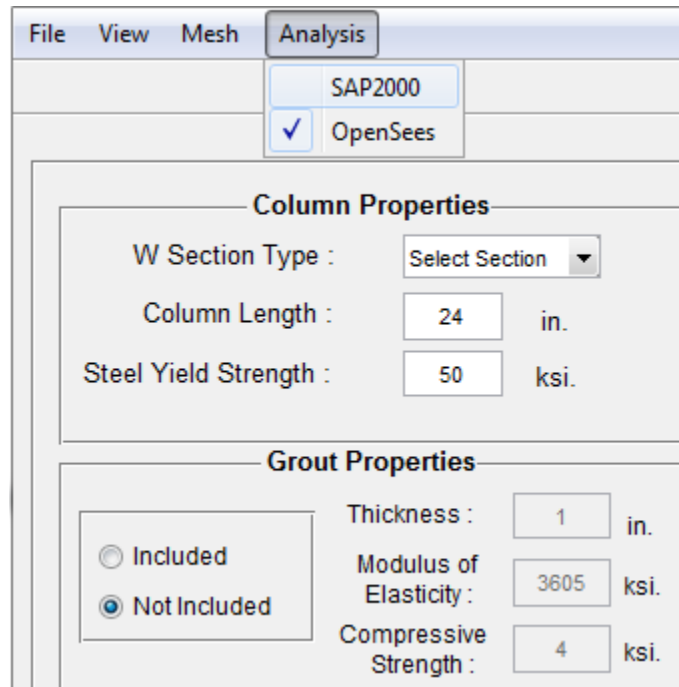


Figure 6.8. Options provided by Analysis submenu

### 6.2.3. Action Buttons

An action button is an object which on clicking triggers an event in the GUI. The functioning of each action button differs as per the requirements. Usually, the functioning of each button is programmed in a small code known as Callback. When the action button is clicked, the callback executes and performs a specific event. These events may be plotting data in another figure, saving data, resetting data, etc. In this wizard, six action buttons are provided and they are situated at bottom right of the GUI (see Figure 6.3). 'Preview Model' creates a preliminary model of the connection by taking the values of parameters entered by the user and displays it on the display screen. Once a preview is created, the model is all set for the analysis. 'Analyse' button executes specific analysis engine (chosen by the user from 'Analysis' submenu), creates a meshed model of the connection, and performs the analysis for provided loads. Once the analysis is complete then the results are retrieved from the chosen software, processed in the wizard and displayed on the display screen and output windows (discussed in next section). When the analysis is complete, the user can't perform another analysis unless he unlocks the model by clicking 'Unlock' button. The saving of unsaved data and loading of saved data in the GUI can be easily done by 'Save Data' and 'Load Data' buttons, respectively. Please note that 'Save As' and 'Open' options provided in 'File' submenu have the same functioning as of 'Save Data' and 'Load Data' buttons, respectively and the user can use either mode to save and load the data in the GUI. 'Reset' button cancels all the changes made in the default values of the parameters and reset the wizard to its default state.

## 6.2.4. Display Screen and Output Windows

As mentioned in Section 6.2 these components of the wizard don't allow the user to make any changes in them. In this wizard, these components display vital information such as preview of the modeled connection, values of the stiffness and maximum bearing pressure, moment rotation curve and bearing pressure distribution. The preview display of the model and the values of stiffness and maximum bearing pressure can be easily interpreted. However, the other two displays need some clear understanding as in how these plots are interpreted in this wizard.

### 6.2.4.1 Moment Rotation Plot

One of the main outputs of this wizard is moment rotation plot for any configuration of the connection under flexural loading or flexural-axial loading. In every moment rotation plot, the moment plotted is the total moment which includes moments applied in both gravity load case and wind load case and the rotation plotted is calculated according to the procedure explained in Section 1.4. Figure 6.9 shows a moment-rotation plot for some random configuration of the column base connection. The symbol ' $K_i$ ' is slope of the line which is tangent to the initial part of the moment rotation curve; it also represents the initial rotational stiffness of the connection which is also displayed on the output screen present beneath the label 'Stiffness' on bottom right of the main GUI. The labels 'Rotation' and 'Moment' below the plot display the values of rotation and moment corresponding to present position of the mouse cursor on the plot. The label 'Secant Stiffness' displays the secant stiffness of any point on the curve. The secant stiffness of any point on the curve represents the slope of a line joining the origin of the plot and the point of observation. As soon as the mouse cursor is pointed to a point on the curve, it turns into a red dot if there is a data point close by and shows corresponding moment, rotation and secant stiffness value. The secant stiffness for the data point shown in Figure 6.9 is represented by ' $K_s$ ', which is the slope of the line joining the point to the origin. Note that in this wizard, the rotation values are plotted after multiplying the actual rotation values by 1000. To get an actual value of rotation, the user must divide the values of rotation obtained from the plot by 1000.

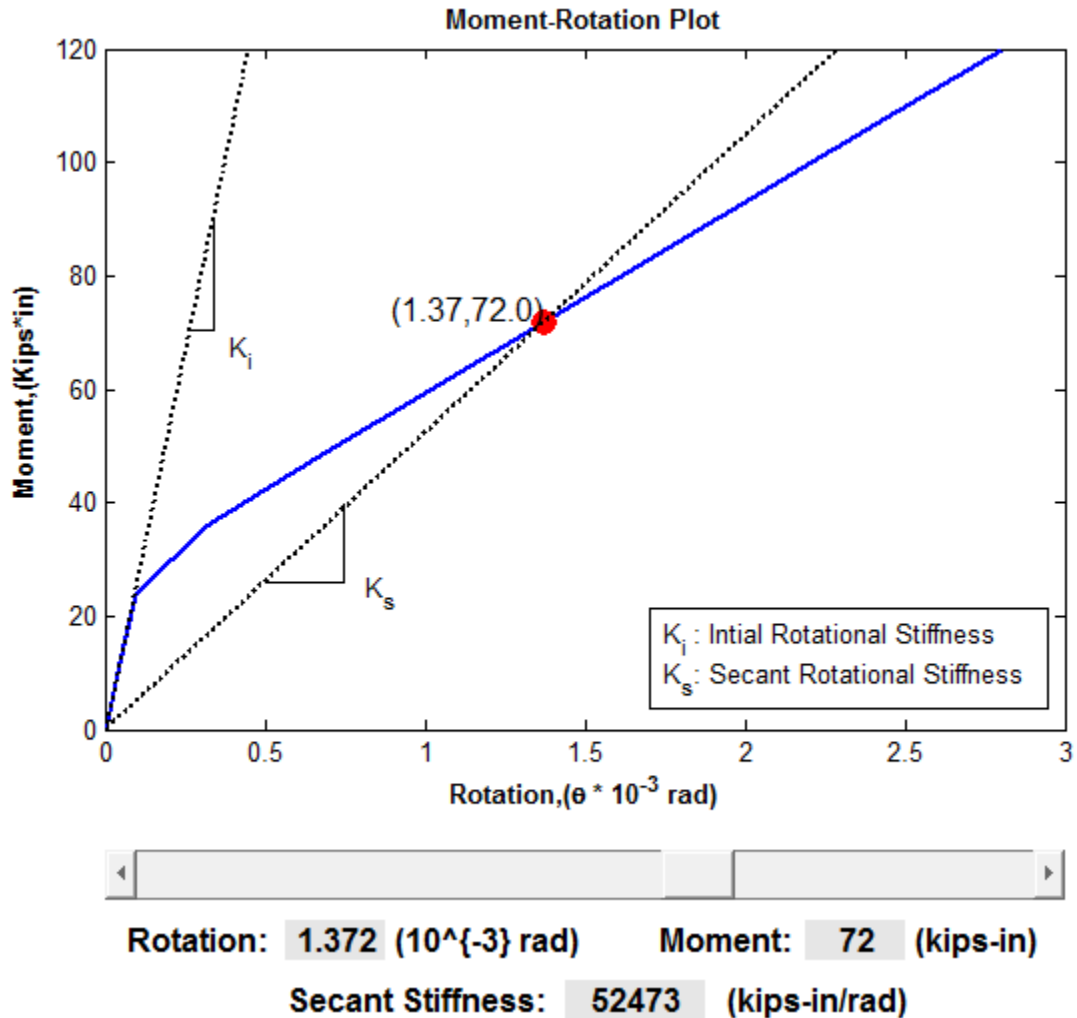


Figure 6.9. A typical moment-rotation plot obtained from the wizard

#### 6.2.4.2 Bearing Pressure Distribution Plot

The distribution of bearing pressure on the base plate obtained from the wizard for some configuration of the connection is illustrated in Figure 6.10. The whole copper color region represents the base plate and the numbers written orthogonally on the left and bottom sides of the plot represent the length and width of the base plate respectively. The rectangles and circles inside the colored region show the positions of the column and bolts over the base plate. The color bar on the right shows the bearing pressure values varying across the base plate. For example, dark red color represents a region of maximum bearing pressure in Figure 6.10. The maximum value of the pressure exists somewhere in the range of 1.8 to 2 ksi. The exact value of the maximum pressure can be read from the output screen present at the bottom right of the main GUI.

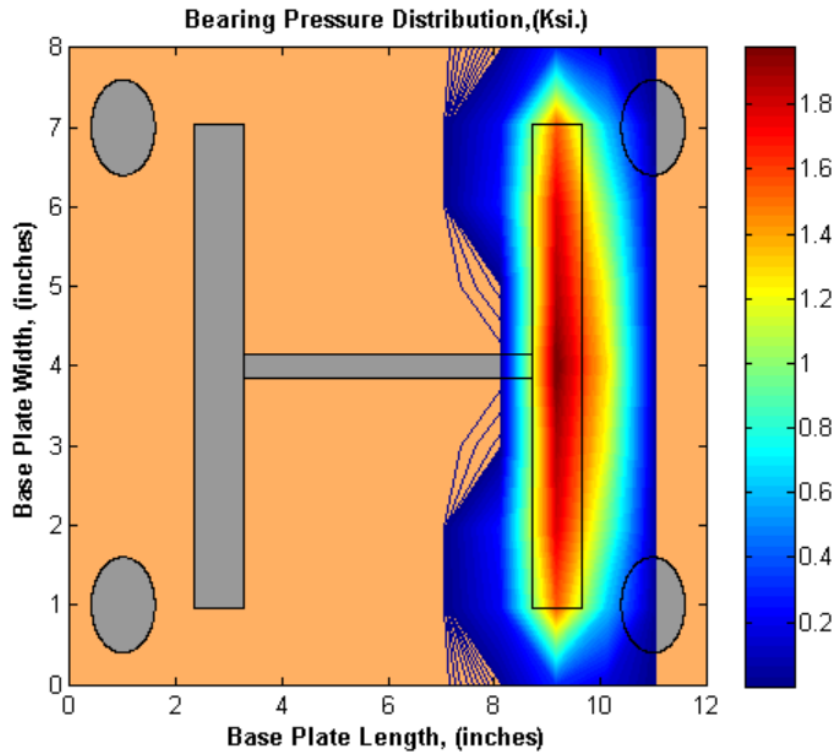


Figure 6.10. Typical layout of Bearing Pressure Distribution

### 6.2.4.3 Report

After each analysis the wizard displays a report which summarizes all the aspects of that analysis including the inputs and outputs. The user can save this report or export it to Microsoft (MS) Word. The report exported to MS word contains Figures 6.9, 6.10 and 6.11. Once the user obtains the complete report as a word document, he can save it in any format provided by MS Word or print it. Figure 6.11 shows a typical report which is given by the wizard. The values of the input parameters and outputs change with the configuration of the connection. As mentioned above, the user can save this report by clicking 'Save' icon or export it to MS word by clicking 'Print' icon, shown by the arrows in Figure 6.11.



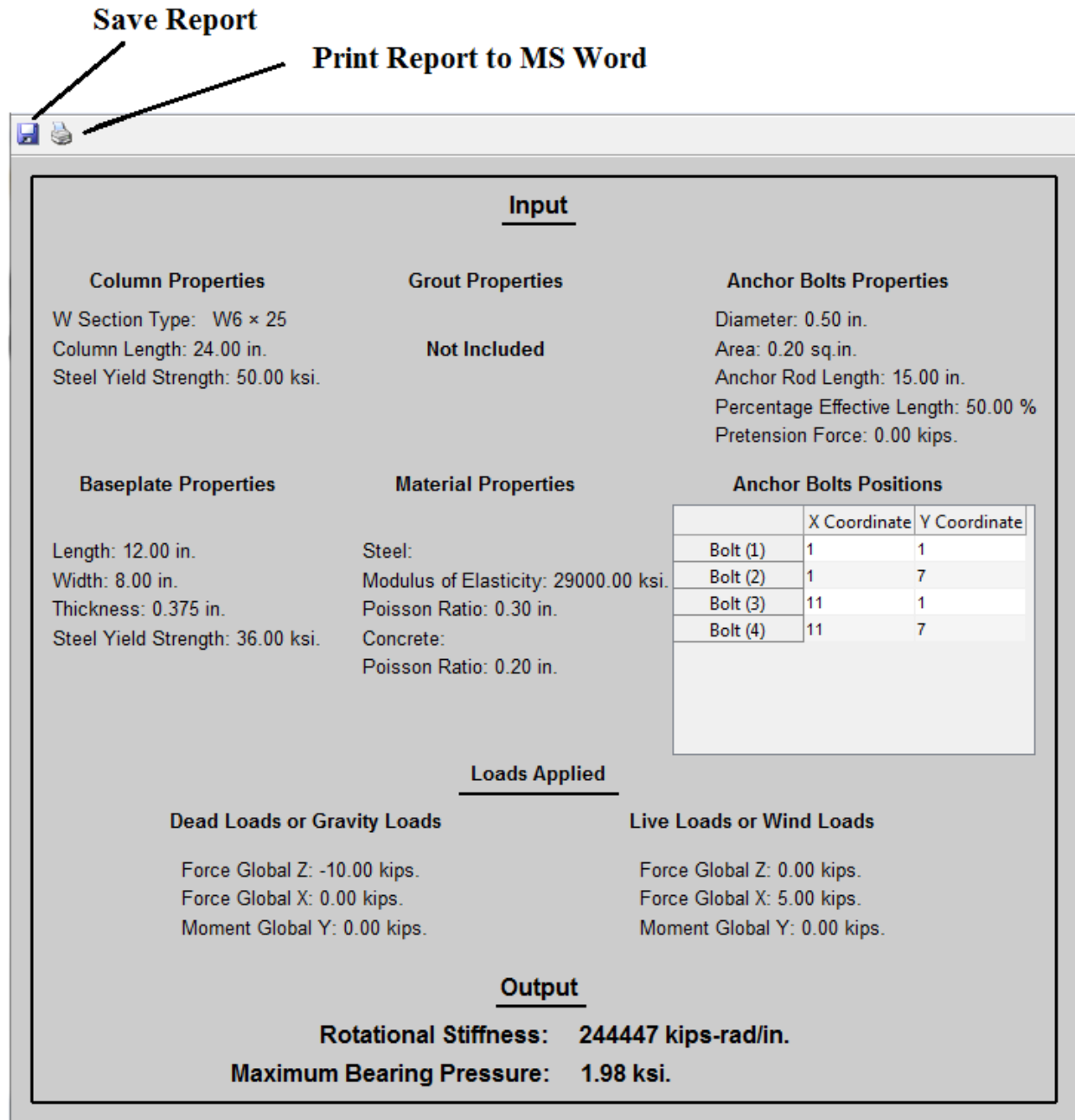


Figure 6.11. A typical report which is displayed by the wizard as one of the outputs

### 6.3 Batch Mode Operation

The batch mode operation lets a user to model the connection, perform the analysis and fetch the rotational stiffness value without opening the GUI. The values of all the parameters of the connection should be provided in a predefined sequence in a notepad text file. The notepad file should be named as '*Batch.txt*' and saved into the installation folder of the wizard. The batch mode input file '*Batch.txt*' is shown in Figure 6.12. An executable script specially written

for batch mode operation extracts the values of different parameters from the batch mode input file, models a connection, performs an analysis and calculates a rotational stiffness. Further details of the working of the batch mode operation can be found in the Appendix C.

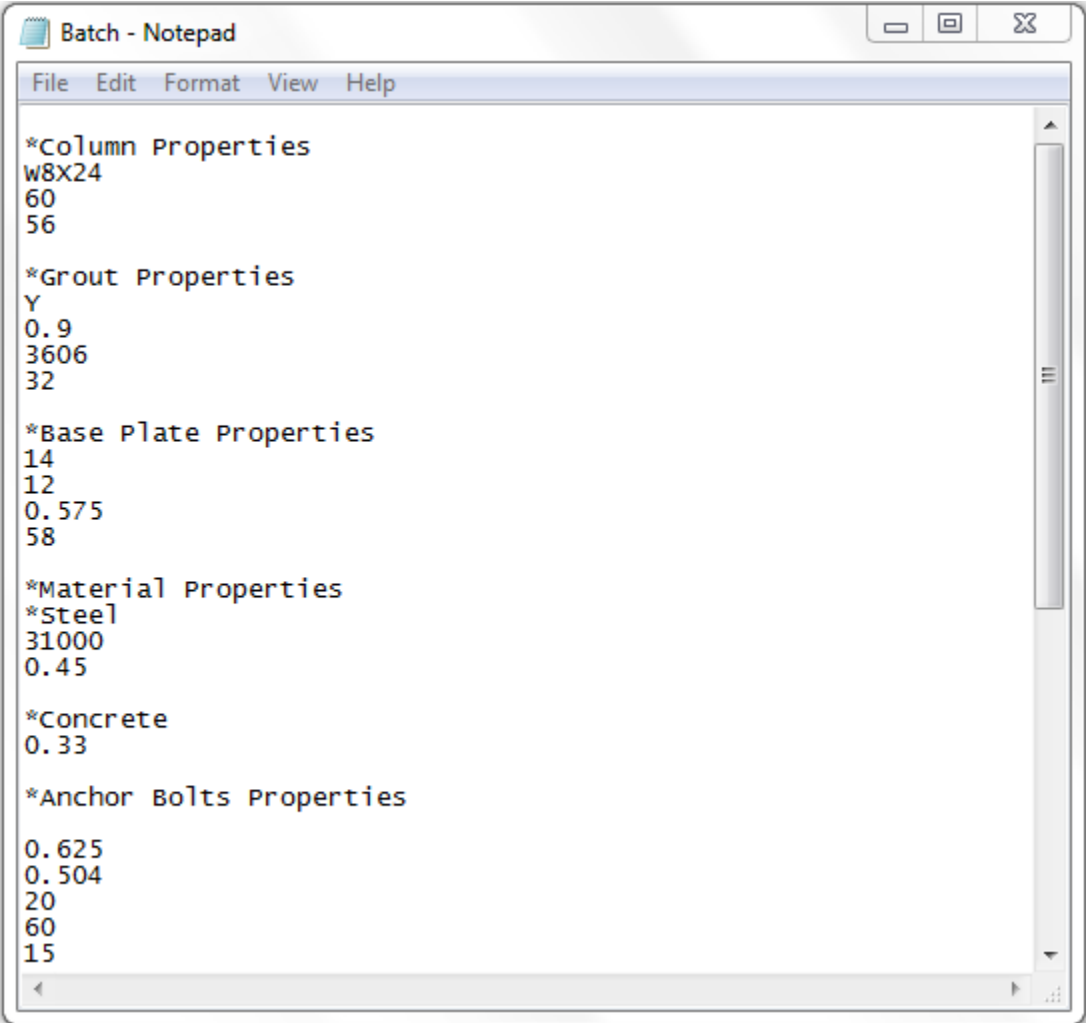


Figure 6.12 A batch mode input file

This chapter discussed the development process of the GUI or the wizard developed for this research and outlined its functioning. A detailed discussion of the working of the GUI can be found in the Appendix C. Please note again that a standalone version of the GUI or the wizard is created which enables the user to run the wizard on any computer without having MATLAB installed on it. The instructions for installing the standalone version of the wizard are also provided in the Appendix C. The next chapter presents some parametric studies that are done with the wizard.

## Chapter 7 Parametric Studies

### 7.1 Factors affecting the Stiffness of the Connection

This chapter presents some parametric studies that are done with the wizard. The last section of this chapter compares the results obtained from the wizard with some of the experimental results, which are presented in Section 2.6.

First objective should be to find the parameters which significantly affect the stiffness of the connection. Please note that the word 'stiffness' here and wherever used in this thesis means the initial tangent stiffness obtained from a moment rotation plot. By looking at the past researches done on the column base connection, a list of important parameters can be presented below.

- Gravity load prior to application of lateral load
- Base plate thickness
- Bolt Size
- Number of Bolts
- Effective length of bolt
- Position of Bolts
- Column Size
- Presence of Grout

Some other parameters like pretension in the bolts, degree of tapering, type of meshing and selected analysis engine are treated separately. For performing the parametric studies, a reference configuration of the connection was taken (see Figure 7.1). The reference configuration has a W6X25 column stub. The length of the stub is 2 ft. The base plate attached to the column stub has a length of 12 in, a width of 8 in and a thickness of 0.375 in. A standard diameter of 1/2 in is taken for the bolts and the anchor rod length is taken as 15 in. The coordinates of the anchor bolts on the base plate are shown in 'Bolts Position' panel in Figure 7.1. Material is steel for the column, base plate and bolts, which has a modulus of elasticity 29000 ksi and poisson ratio of 0.3. Yield strengths for the steel used for the column and base plate are 50 ksi and 36 ksi respectively. The percentage of length of the anchor rod which is effective in providing stiffness is taken as 50 %. A coarse mesh type is chosen from the 'Mesh' sub-menu and OpenSees is chosen as analysis engine from the 'Analysis' sub-menu. A lateral load of 5 kips and an axial compressive load of 10 kips are applied to the column stub. Note that compressive load must be entered with a negative sign.

Whenever the effect of a specific parameter had to be observed, the value of the considered parameter was varied and everything else was kept same in the reference configuration. The above mentioned parameters and their effects on the stiffness of the connection will be discussed one-by-one in subsequent sections. Note that each parametric study presented below contains a moment-rotation plot obtained from the wizard and a table which contains the values obtained

for the rotational stiffness and maximum bearing pressure. Every moment rotation plot is also followed by a zoomed in moment rotation plot which magnifies the initial part of the observed moment rotation curve. Some zoomed in plots may not show the origin and instead show a very small nonzero number at the bottom left of the plot. This may happen when the data points in the initial region are too close. A detailed explanation on interpreting the moment-rotation plot of this wizard can be found in Section 6.2.4.1.

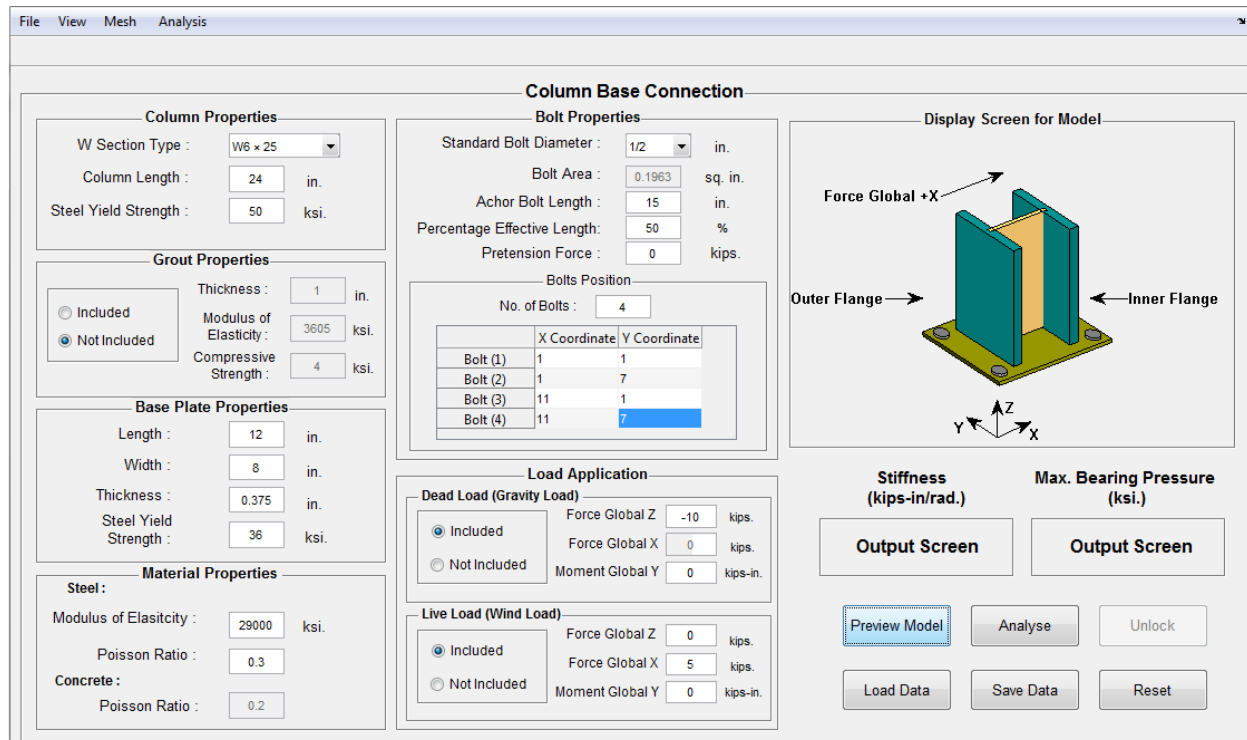


Figure 7.1. Test Configuration for observing effect of Axial Load on the Connection Stiffness

### 7.1.1 Axial Load

Picard and Beaulieu (1985) found out that the presence of axial loads has a very significant effect on the behavior of the connection under flexural loading. A connection was found out to be having higher stiffness under a flexure test when it was subjected to compressive axial load and vice versa. In actuality, the application of a compressive axial load delays the uplifting of the base plate under flexure. In the 'Load Application' input panel, an axial gravity load and a lateral wind load were applied on the connection. The compressive axial load was varied from 5 kips to 20 kips, keeping other connection details same as of the reference configuration (see Section 7.1). Figures 7.2 and 7.3 display a family of moment rotation curves obtained from the wizard, each curve corresponds to a unique value of axial load. It can be clearly observed from the Figures 7.2 and 7.3 that the stiffness of the connection slightly increases with the increase in axial compressive load. The values obtained for the stiffness and maximum bearing pressure obtained from the wizard at various compressive loads are summarized in Table 7-1. The possible reason for the irregular shift between the curves (see Figure 7.3) can be attributed to the nonlinear interaction between the flexure deformation and compressive deformation of the

connection. This may also be seen from the values of maximum bearing pressure. These values are not uniformly spaced but vary nonuniformly with a uniform increment of 5 kips of axial load. Please note that the initial part of any moment rotation curve for compression flexure loads corresponds to the response of the base plate under the effect of compressive axial load only. The first kink (or point of sudden slope change) in the moment rotation curve represents the point at which the base plate starts to lift up. At this stage, major part of the base plate remains in contact with the footing or grout. The second kink which can be subtle in some curves represents the point at which major part of the base plate lifts off and loses contact with the footing or grout.

Table 7-1. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Compressive Loads on the Connection

Compressive Load (kips.)	Rotational Stiffness (kips-in/rad.)	Max. Bearing pressure (ksi.)
5	244408	1.73
10	244447	1.98
15	244460	2.15
20	244474	2.29

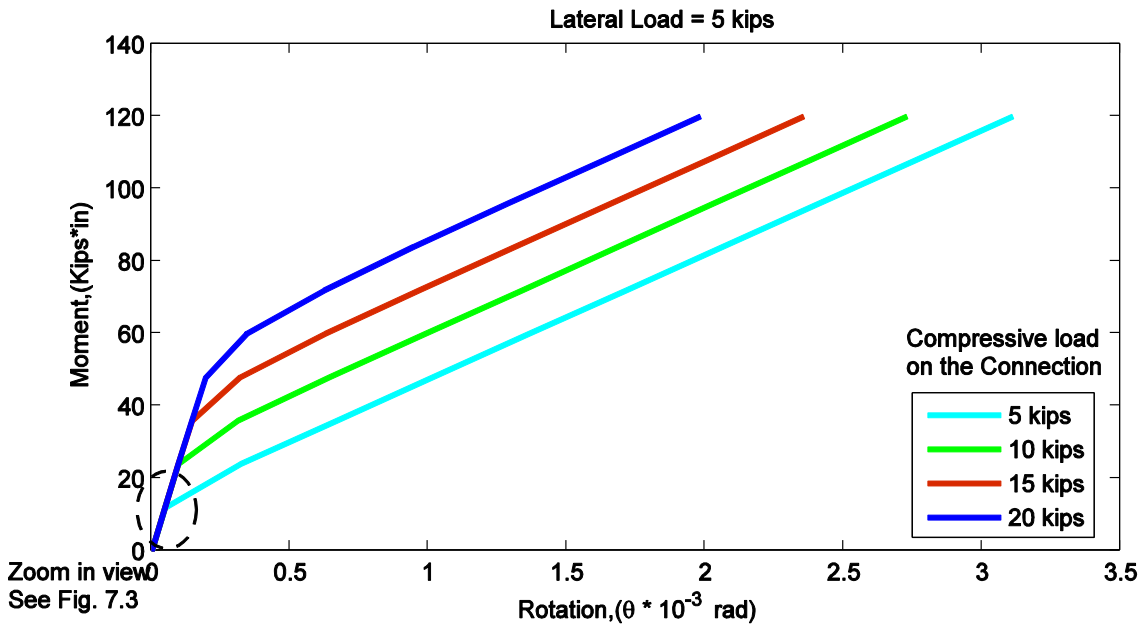


Figure 7.2. Moment Rotation Curves for different Compressive Loads on the Connection

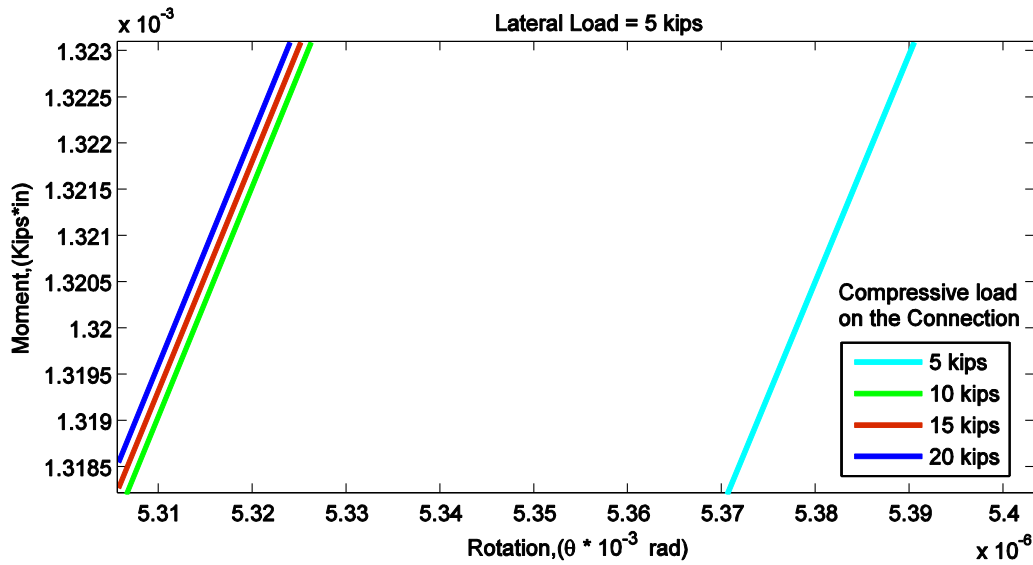


Figure 7.3. Zoom in view of Figure 7.2

### 7.1.2 Thickness of Base Plate

Hon and Melchers (1988) showed that thickness of the base plate has a significant effect on the stiffness of the connection, specifically when the base plate is thin. For thinner base plates, yielding of the base plate governs the failure mechanism of the connection under flexure test. For thicker plates, bolt failure governs the behavior. To observe the influence of the thickness of the base plate on the stiffness of the connection, the thickness was varied and everything else was kept same in the reference configuration (see Section 7.1). Figures 7.4 and 7.5 display a set of moment rotation curves obtained from the wizard, each curve corresponds to a unique value of thickness of the base plate. It can be clearly observed from the Figures 7.4 and 7.5 that the stiffness of the connection increases with increase in the thickness of the base plate. Thick plates are stiffer than thin plates, hence contribute more to the stiffness of the connection. The values obtained for the stiffness and maximum bearing pressure for different thicknesses of the base plate are presented in Table 7-2.

Table 7-2. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for different Base Plate Thicknesses

Base plate Thickness (in.)	Rotational Stiffness (kips-in/rad.)	Max. Bearing Pressure (ksi.)
0.375	244447	1.98
0.5	277935	1.67
0.625	310339	1.44
1	410241	1.01

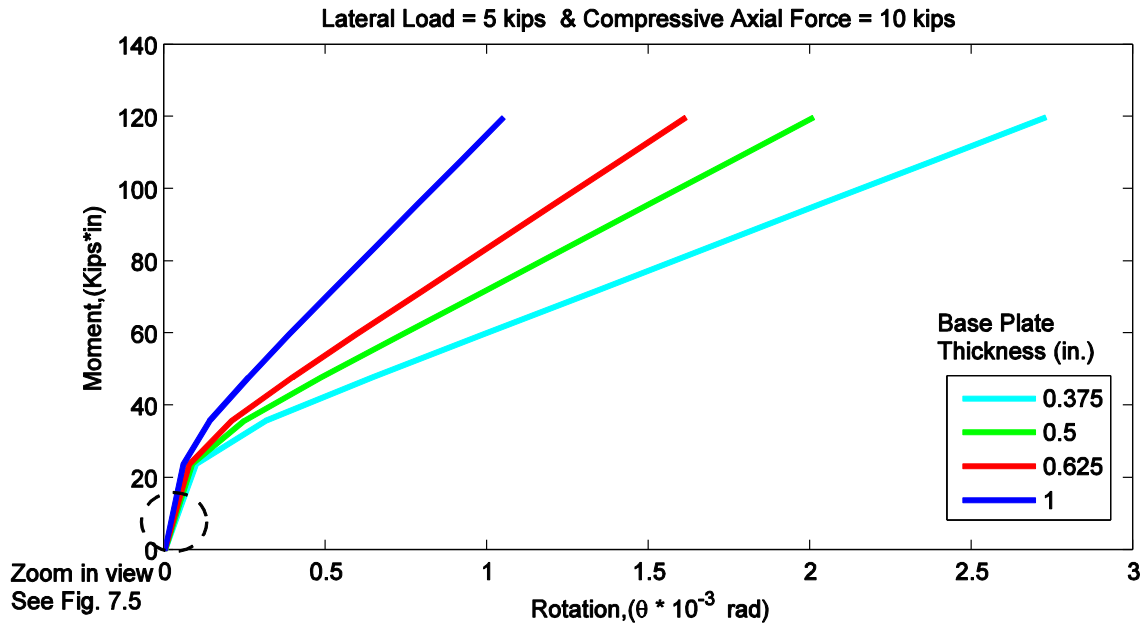


Figure 7.4. Moment Rotation Curves for different Base Plate Thicknesses

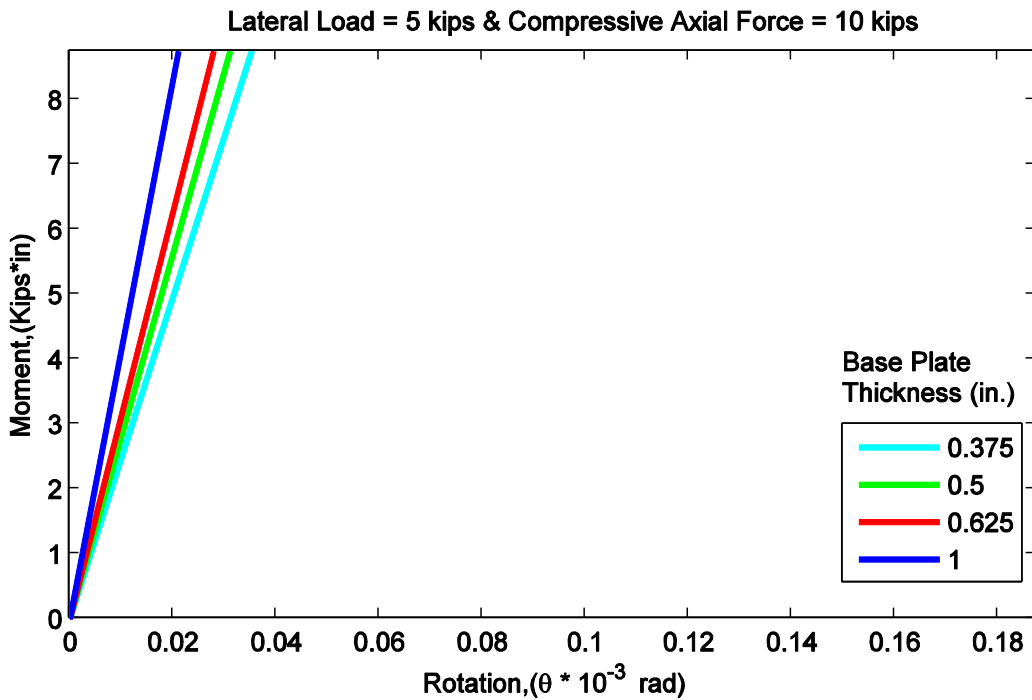


Figure 7.5. Zoom in view of Figure 7.4

### 7.1.3 Bolt Size

Hon and Melchers (1988) showed that as the bolt size increases, the stiffness of the connection also increases. To observe the influence of the size of the anchor bolts, the anchor bolt diameter was varied from 1/2 in. to 1 in., keeping other connection details same as of the reference

configuration (see Section 7.1). Figures 7.6 and 7.7 show a set of moment rotation curves obtained from the wizard. Each moment rotation curve corresponds to a unique bolt diameter. It can be observed from the Figures 7.6 and 7.7 that the stiffness of the connection increases slightly as the bolt size increases from 1/2 in. to 1 in. nominal diameter. This trend is observed because the stiffness of the anchor bolt is directly proportional to the area of the bolt (Equation 4-1, Section 4.2) or in other words the diameter of the bolt. Since the stiffness of a bolt contributes to the stiffness of the whole connection, hence with increase in bolt diameter, the stiffness of connection increases. The values of the stiffness and maximum bearing pressure obtained from the wizard for different bolt sizes are shown in table 7-3.

Table 7-3. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for different Bolt Sizes

Bolt Diameter (in.)	Rotational Stiffness (kips-in/rad.)	Max. Bearing Pressure (ksi.)
1/2	244447	1.98
5/8	244499	1.97
3/4	244537	1.97
7/8	244565	1.96
1	244586	1.96

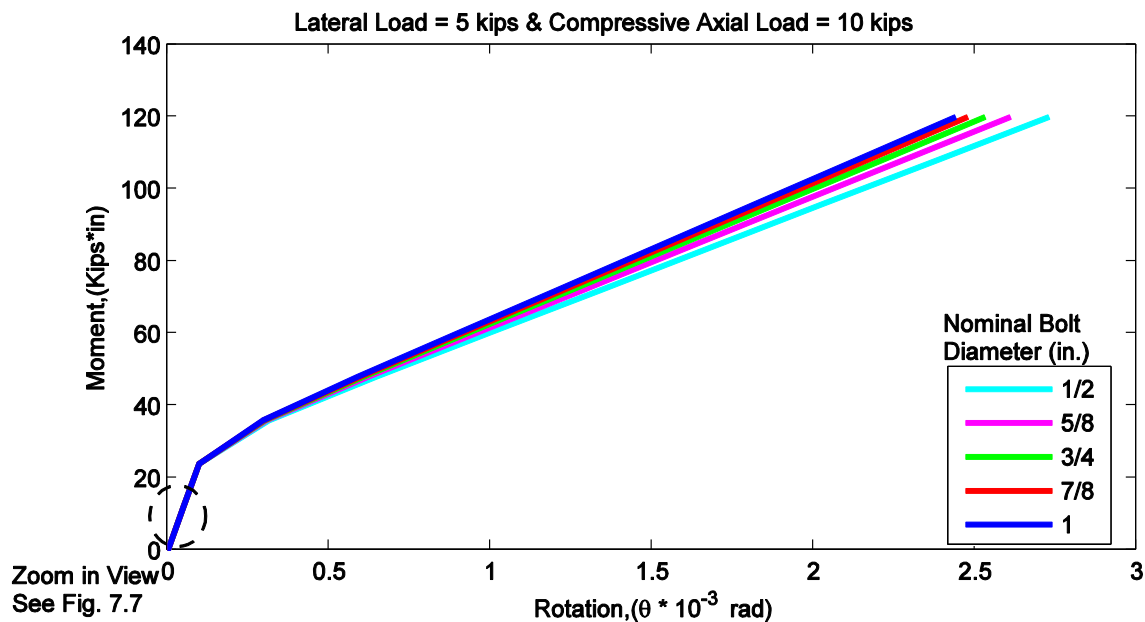


Figure 7.6. Moment Rotation Curves for different Bolt Sizes



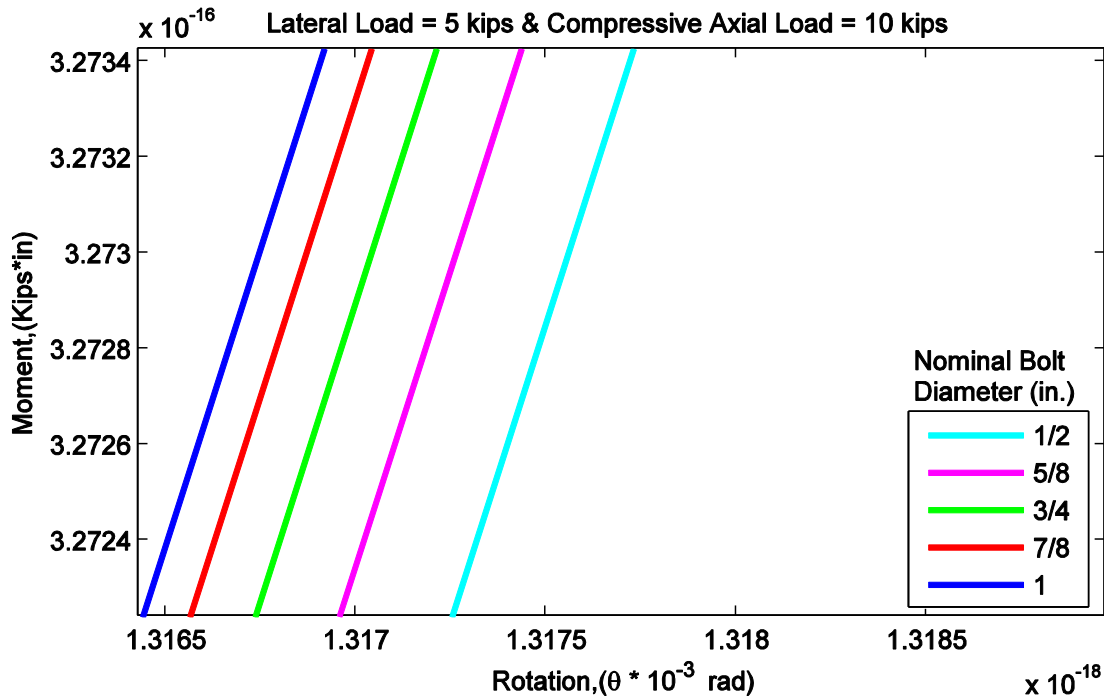


Figure 7.7. Zoom in view of Figure 7.6

#### 7.1.4 Number of Bolts

The anchor bolts resist the uplift of the base plate resulting due to bending. It is found that more the number of bolts attached in the connection, the greater is the stiffness of the connection. For performing this parametric study, the bolt configuration is changed every time a new analysis is performed, however other connection details are kept same as of the reference configuration (see Section 7.1). The considered bolt configurations are summarized in Table 7-4. Figures 7.8 and 7.9 display a set of moment rotation curves obtained from the wizard. Each curve corresponds to a unique number of anchor bolts. Table 7-5 presents the values of the stiffness and maximum bearing pressure obtained from the wizard for different number of anchor bolts in the connection.

Table 7-4. Different bolt configurations considered for observing the effect of number of bolts on the stiffness of the connection

<b>Bolt Configuration</b>	<b>Bolt Number</b>	<b>X Coordinate</b>	<b>Y Coordinate</b>
Two Bolts	1	6	1.5
	2	6	6.5
Four Bolts	1	1.5	1.5
	2	1.5	6.5
	3	10.5	1.5
	4	10.5	6.5
Six Bolts	1	1.5	1.5
	2	1.5	6.5
	3	6	1.5
	4	6	6.5
	5	10.5	1.5
	6	10.5	6.5
Eight Bolts	1	1.5	1.5
	2	1.5	4
	3	1.5	6.5
	4	6	1.5
	5	6	6.5
	6	10.5	1.5
	7	10.5	4
	8	10.5	6.5

Table 7-5. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for different number of Bolts

<b>Number of Bolts</b>	<b>Rotational Stiffness (kips-in/rad.)</b>	<b>Max. Bearing Pressure (ksi.)</b>
2	241096	2.04
4	244447	1.98
6	246022	1.96
8	249080	1.88

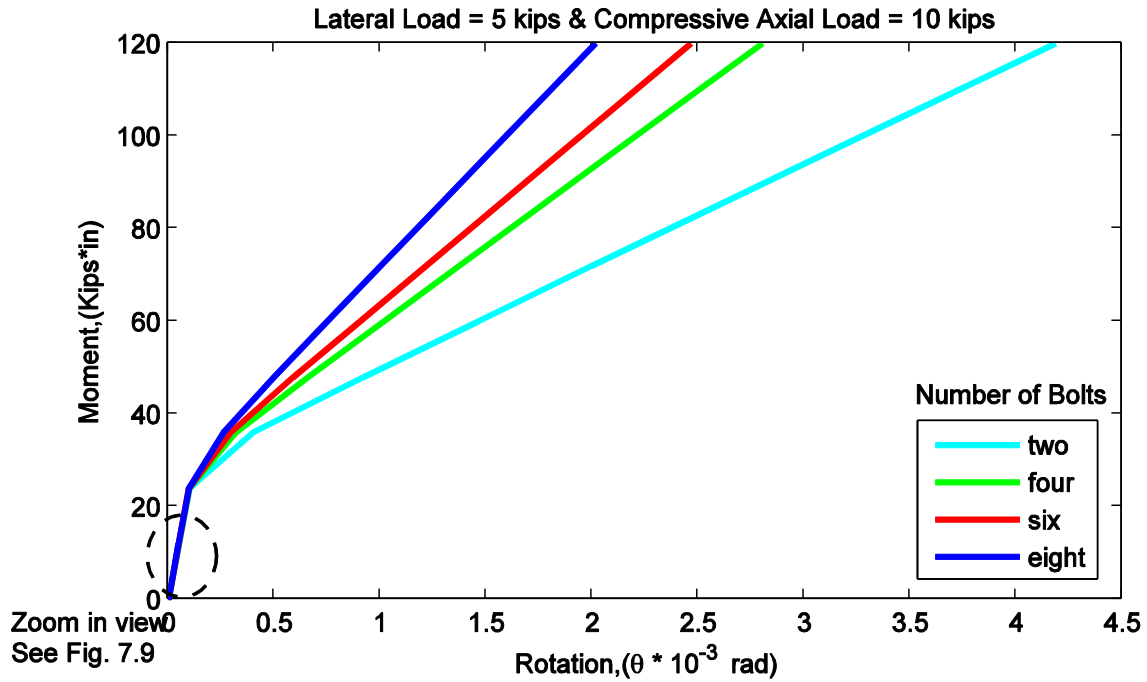


Figure 7.8. Moment Rotation Curves for different number of Bolts

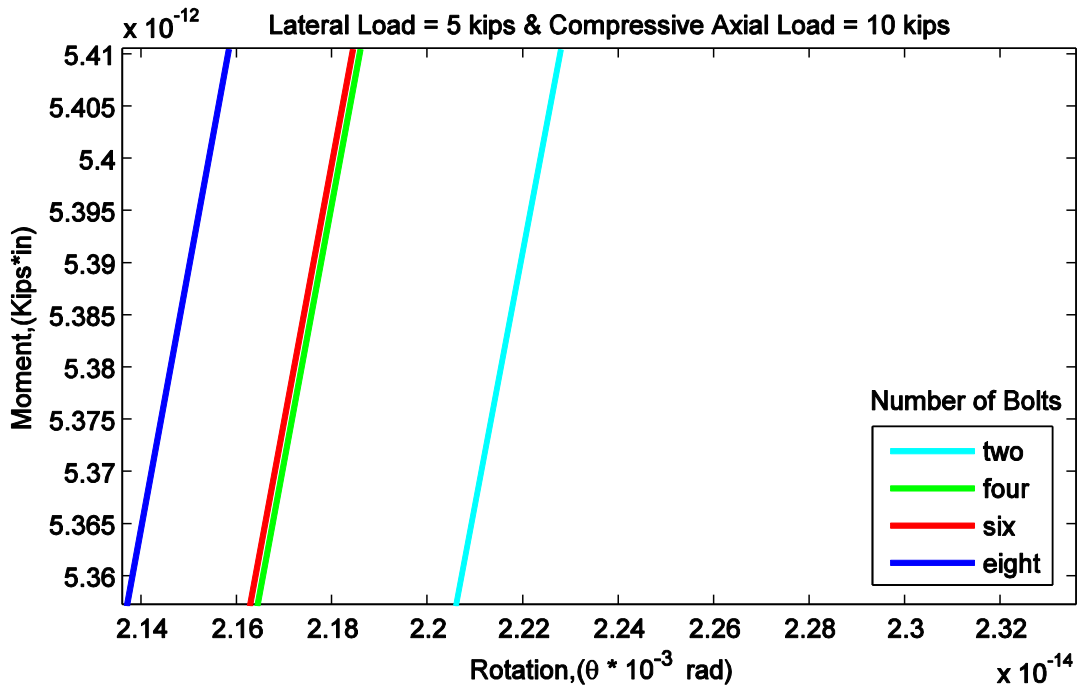


Figure 7.9. Zoom in view of Figure 7.8

### 7.1.5 Effective Length of Anchor Bolts

The effective length of the anchor bolts is defined as the percentage of anchor bolt length effective in resisting the uplift of the baseplate. It depends on the variation of bond stress

between the foundation and anchor bolts. Please refer to Section 4.2 for detailed explanation of the effective length. To observe the influence of effective length of the anchor bolts on the stiffness of the connection, the percentage effective length was varied from 40 % to 100%, keeping other connection details same as of the reference configuration (see Section 7.1). Figures 7.10 and 7.11 show a set of moment rotation curves obtained from the wizard. Each curve corresponds to a unique value of percentage effective length. It can be observed from the above mentioned figures that the stiffness decreases with increase in the percentage effective length. This trend was expected as per the definition of the anchor bolt stiffness in this wizard. According to Equation 4-1, the stiffness of an anchor bolt decreases with an increase in percentage of the anchor length which is effective. Since the stiffness of an anchor bolt contributes to the stiffness of the connection, hence the stiffness of the connection decreases with increase in percentage effective length. Table 7-6 presents the values of the stiffness and maximum bearing pressure obtained from the wizard for different percentages of effective length.

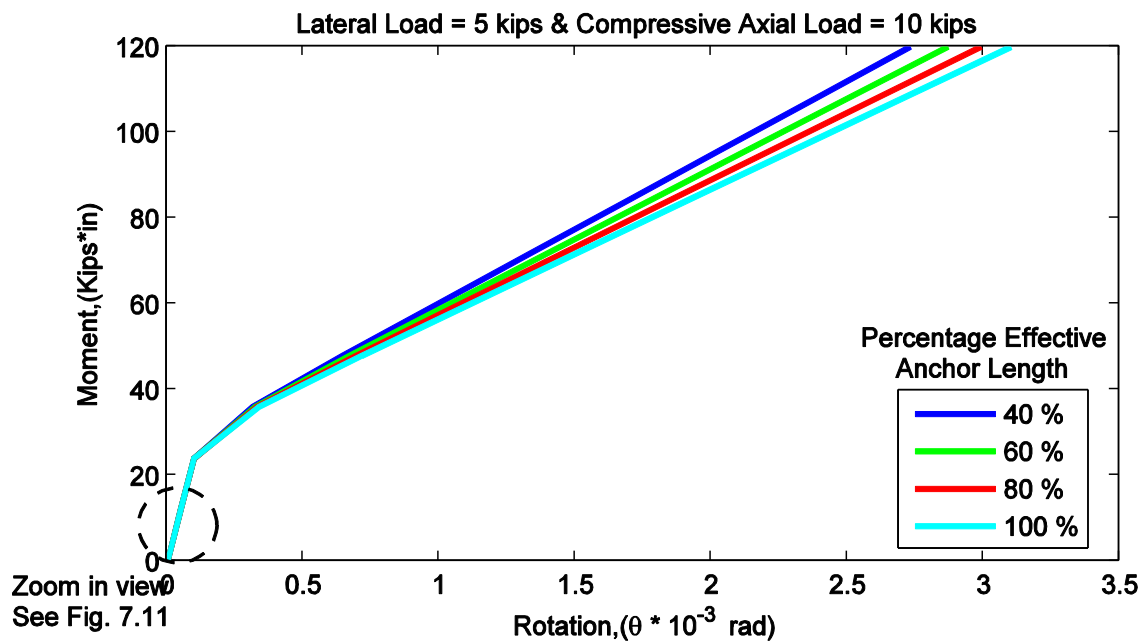


Figure 7.10. Moment Rotation Curves for different percentages of effective length

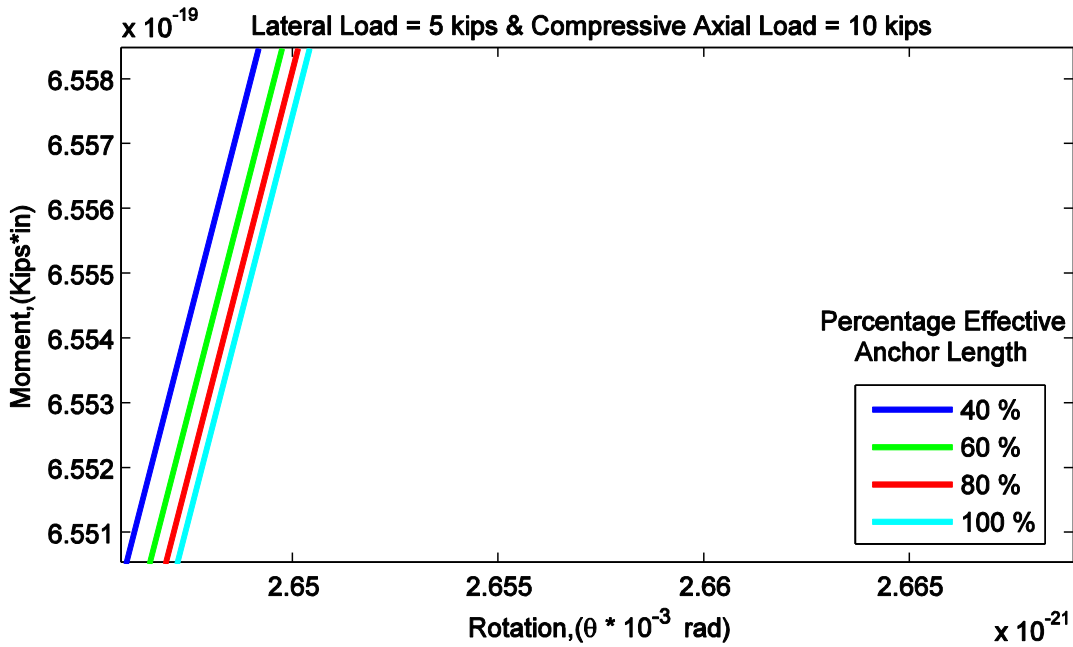


Figure 7.11. Zoom in view of Figure 7.10

Table 7-6. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for different percentages of effective length

Percentage Effective Length	Rotational Stiffness (kips-in/rad)	Maximum Bearing Pressure (ksi.)
40	244473	1.98
60	244426	1.98
80	244395	1.98
100	244371	1.98

### 7.1.6 Location of Bolts

To observe the effect of location of the bolts, two configurations were taken as shown in Figure 7.12 and Figure 7.13. It can be seen from the moment rotation plots obtained from the wizard for the two configurations (see Figures 7.14 and 7.15) that when the anchor bolts were present within the flange, the stiffness of the connection was found larger than when they were present outside the flanges. The above described trend is not always true. The stiffness and bearing pressure values also depend upon the size of the base plate and column section in addition to location of the bolts; hence one can't generalize anything reasonable from this particular study.

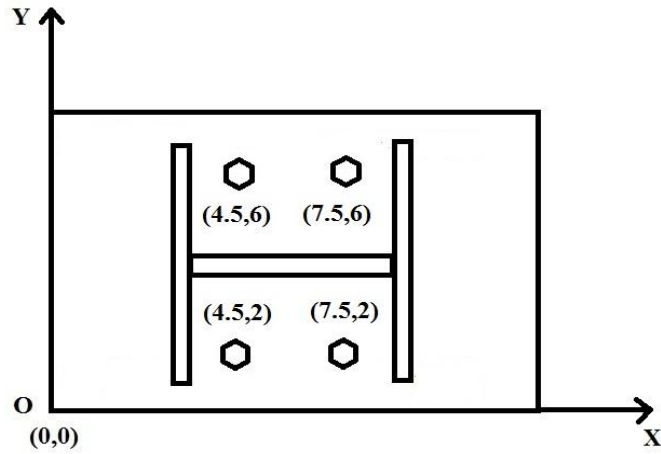


Figure 7.12. Positions of bolts inside the flanges of the Column

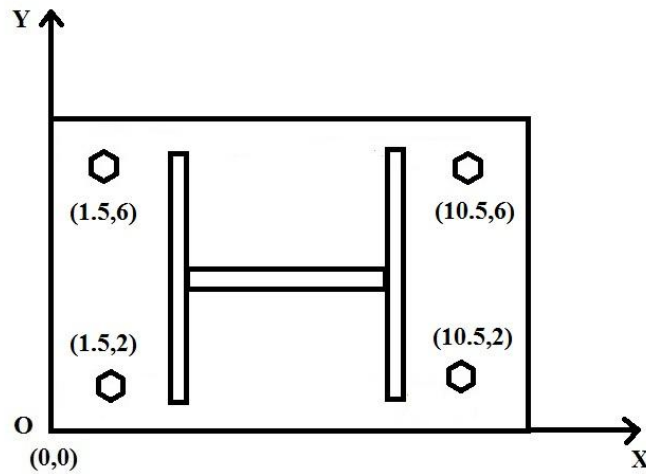


Figure 7.13. Positions of bolts outside the flanges of the Column

Table 7-7. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for two different configurations of Bolts

Location of Bolts	Rotational Stiffness (kips-in/rad)	Max. Bearing Pressure (ksi.)
Inside Flange	252385	1.91
Outside Flange	246436	1.91

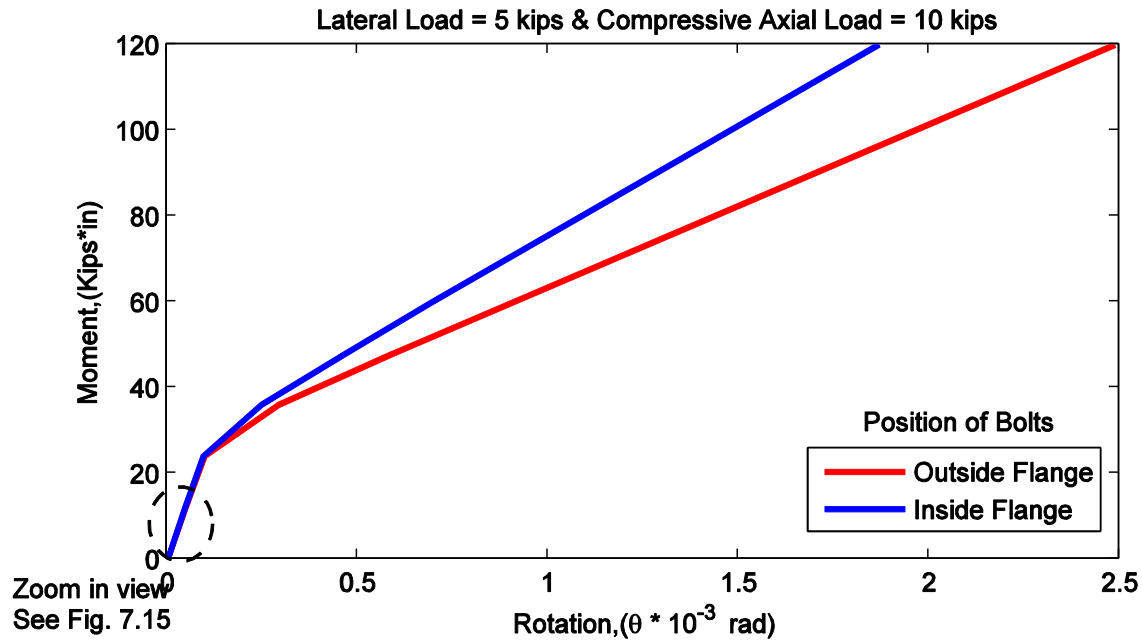


Figure 7.14. Moment Rotation Curves for two different configurations of Bolts

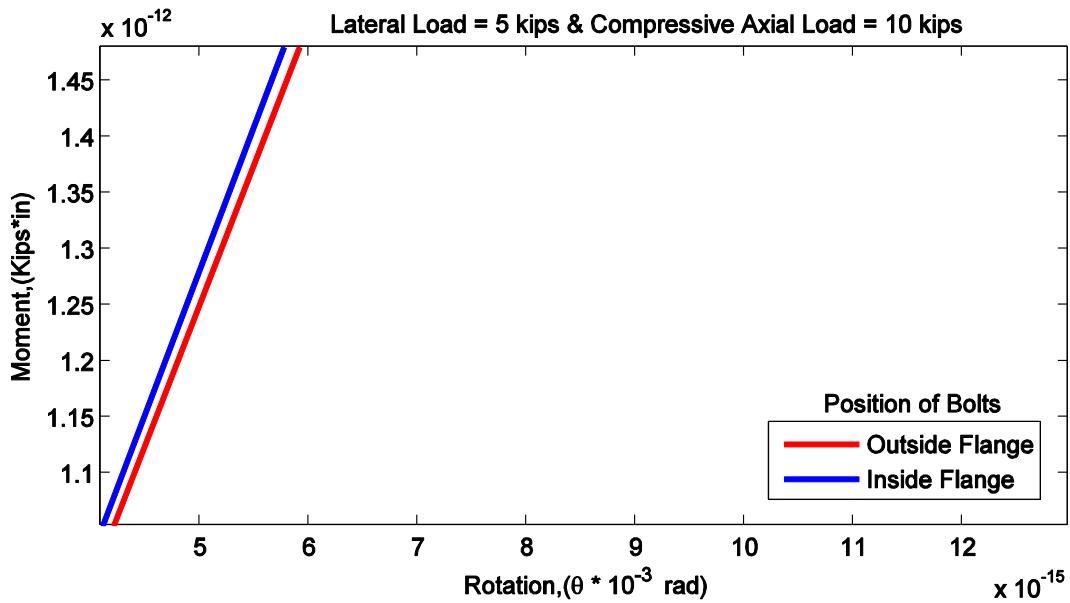


Figure 7.15. Zoom in view of Figure 7.14

### 7.1.7 Column Size

Hon and Melchers (1988) showed that as the column size increases, the stiffness of the connection also increases. Four different configurations of the connection were selected; each had a different wide flange section as a column section type. The other details were kept same as of the reference configuration (see Section 7.1). Figures 7.16 and 7.17 show a set of moment rotation curves obtained from the wizard. Each moment rotation curve corresponds to a unique

column section type. From the curves it can be seen that as the column size increases from W4X13 to W8X24, the stiffness also increases. This trend is observed as the lever arm ( $d_c$  in Figure 1.4) for calculating the rotation increases with increase in column size. The increase in lever arm decreases the measured rotation for a given flexure load or in other words increases the stiffness. The values for the stiffness and maximum bearing pressure obtained from the wizard for different column sections are shown in Table 7-8.

Table 7-8. Values of Stiffness and Maximum Bearing Pressure obtained from the Wizard for different Column Sections

Column Section	Rotational Stiffness (kips-in/rad.)	Max. Bearing Pressure (ksi.)
W4X13	72850	3.08
W5X19	107407	1.84
W6X25	244447	1.98
W8X24	391170	1.72

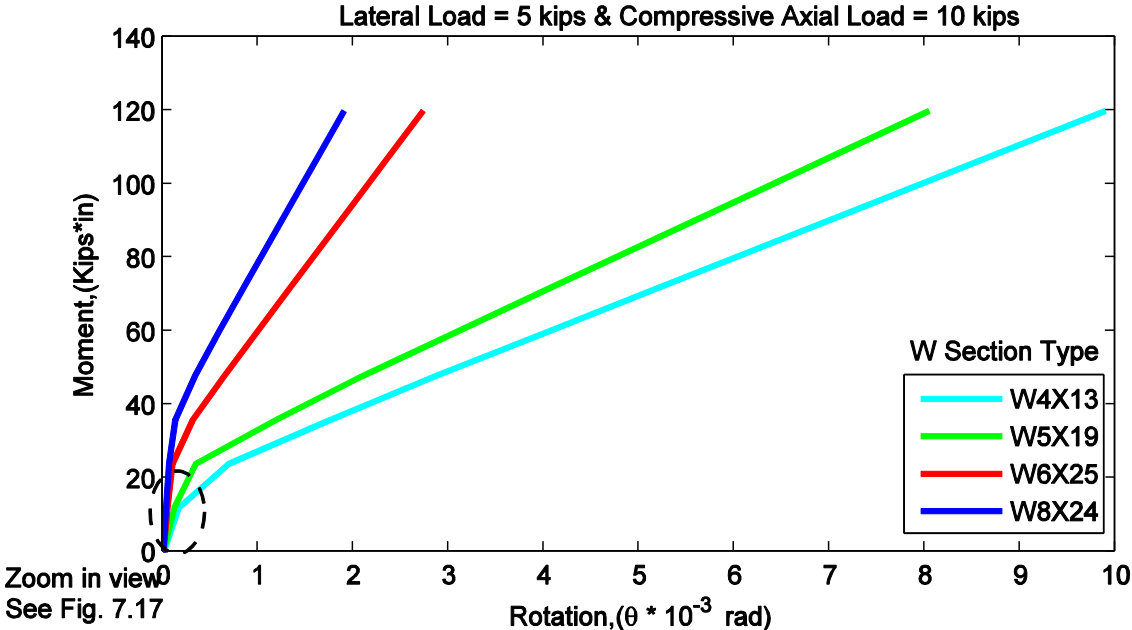


Figure 7.16. Moment Rotation Curves for different Column Sections



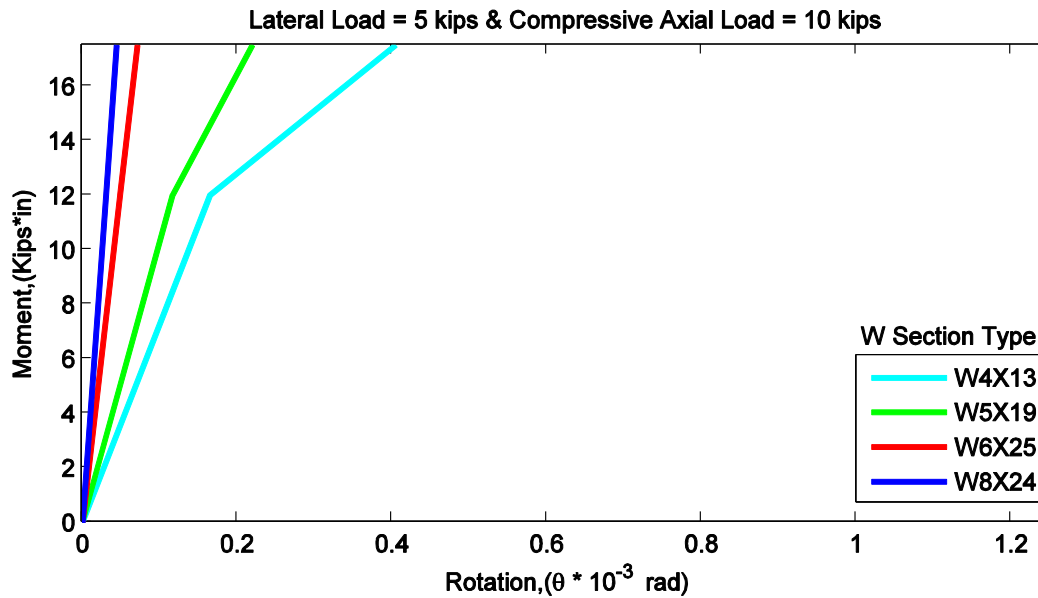


Figure 7.17. Zoom in View of Figure 7.16

### 7.1.8 Presence of Grout

The presence of the grout underneath the base plate does not affect the stiffness of the connection much but there is little decrease in the value. However, it has a drastic effect on the bearing pressure acting on the base plate. If the grout is present beneath the baseplate then compression in the base plate spreads more effectively over the grout surface area. This results in increase in bearing area and decrease in bearing pressure acting on the base plate. For performing this study, the default values of the grout parameters are taken to model the grout underneath the base plate. The above mentioned observations are validated from the moment rotation curves and maximum bearing pressure values obtained from the wizard. Table 7-9 shows the stiffness and maximum bearing pressure values obtained from the wizard for the connection with and without grout. Figures 7.18 and 7.19 show the moment rotation curves for the two types of connection.

Table 7-9. Values of Stiffness and Maximum Bearing Pressure obtained from Wizard for with and without Grout

Grout	Rotational Stiffness (kips-in/rad.)	Max. Bearing Pressure (ksi.)
Not Included	244447	1.98
Included	204906	1.81

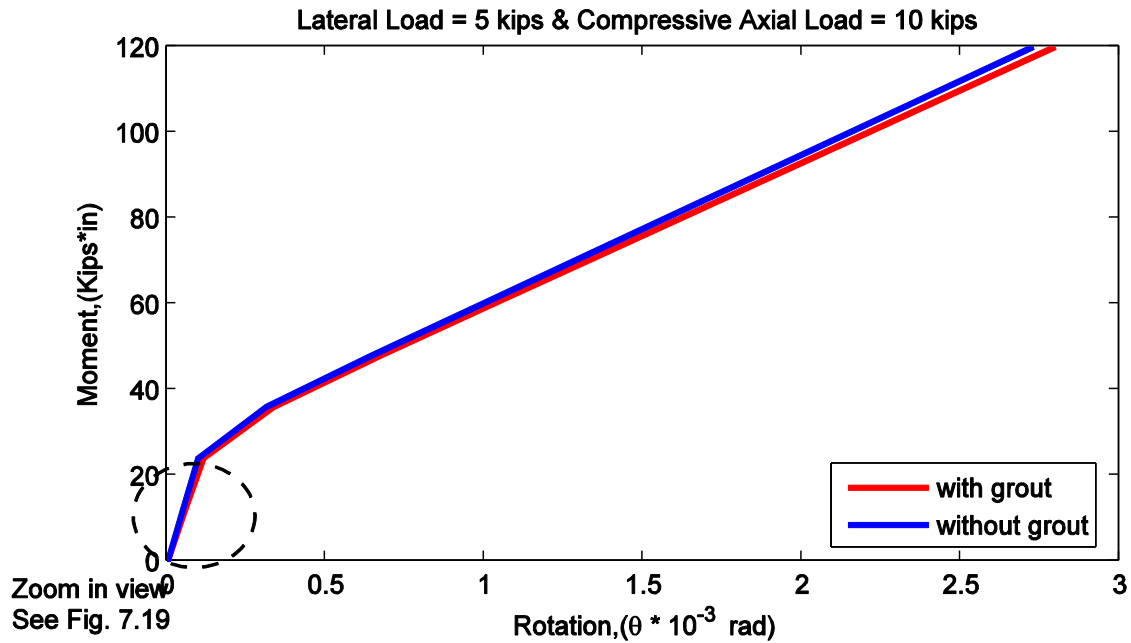


Figure 7.18. Moment Rotation Curves for the Connection with Grout and without Grout

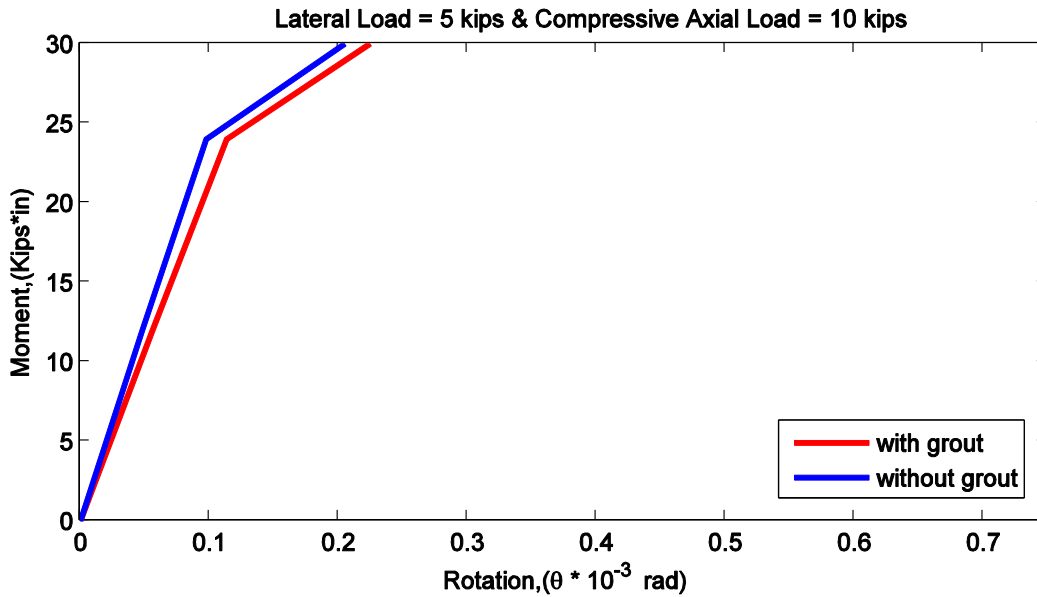


Figure 7.19. Zoom in view of Figure 7.18

The possible reason behind the decrement in the value of the stiffness in case of including grout is the flexibility provided by the grout due to its unconfined construction. In compression, the grout deforms through its thickness and cracks may develop on the edges, thereby allows the base plate to deform further. While in case of excluding the grout, the connection directly sits on the footing which has a lot more stiffness to resist any downward movement because it is confined from every side by the soil. The grout can be used very effectively wherever possible. It

is seen that the bearing pressure acting on the base plate can be reduced drastically by using grout without significantly compromising the stiffness of the connection.

## 7.2 Effects of Tapered Column Section on the Stiffness of the Connection

In light metal buildings, the maximum moment in each frame is present at the knee of the frame as shown in the Figure 7.20. To provide more flexural capacity at the knee joint, the column sections are tapered with increasing total depth from bottom to top. To observe the effect of the tapering on the stiffness of the connection, four different configurations of the connection were taken, each had a unique tapered section. Each configuration had all the details same as of the reference configuration (see Section 7.1) except the tapered column section properties. In tapered column section properties, a flange width of 6.08 in., a flange thickness of 0.45 in., a web thickness of 0.32 in. and a total thickness of 6.38 in. at the bottom of the column were taken. These dimensions were the common values in all the configurations. The only parameter which was varied was total depth at the top of the column. Table 7-10 presents the values for the stiffness and maximum bearing pressure obtained from the wizard for different values of total depth at the top of the column. Figures 7.22 and 7.23 show a set of moment rotation curves. Each curve corresponds to a unique total depth present at the top of the column stub. It can be observed that with the increase in tapering the stiffness of the connection decreases. This can be explained by the fact that the loads in this wizard are applied at centroid of the cross section at top of the column and the centroids of the top and bottom cross section do not lie on the same vertical line. Figure 7.21 display a general schematic of a tapered section configuration. The centroid at the top and bottom are represented by  $C_T$  and  $C_B$  respectively. The line of application of a gravity load applied at  $C_T$  does not pass through  $C_B$ . Since the stiffness in this wizard is measured at the base of the column stub. Hence any gravity load applied on the tapered section in this wizard would create an additional moment.

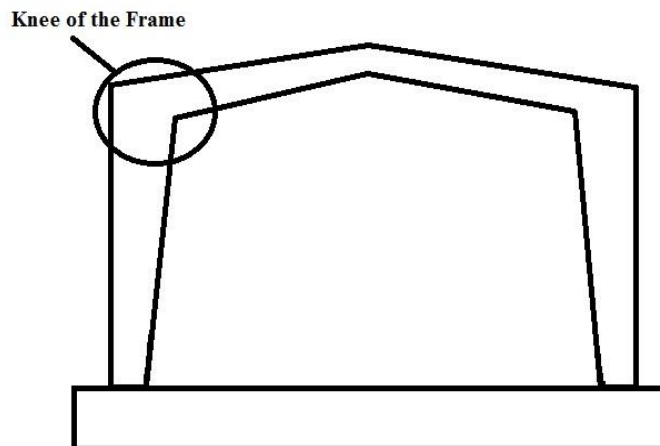


Figure 7.20. Knee of Gable Frame

Table 7-10. Values of Stiffness and Maximum Bearing Pressure for Tapered Sections

Web Depth at Knee Joint (in.)	Rotational Stiffness (kips-in/rad.)	Maximum Bearing Pressure (ksi.)
6.4	232732	1.98
8.4	231162	2.19
10.4	213281	2.49
12.4	163644	2.86

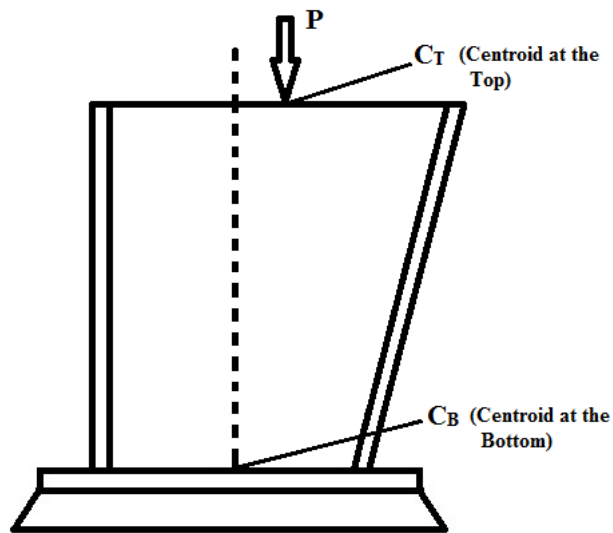


Figure 7.21. Point of application of load in the wizard in case of tapered section

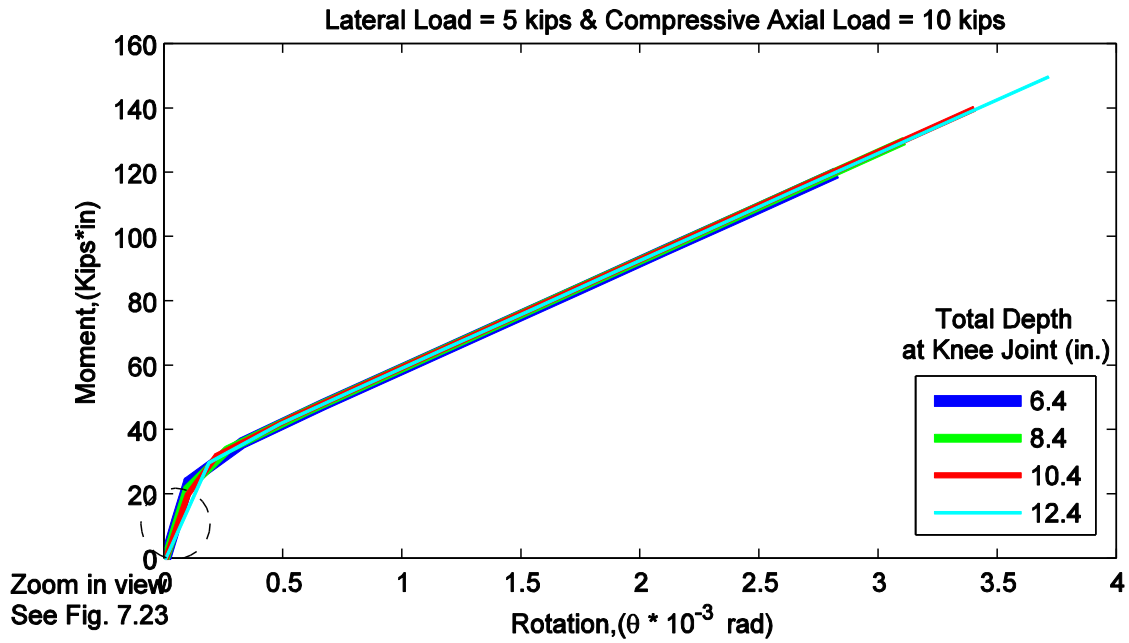


Figure 7.22. Moment Rotation Curves for Tapered Sections with different Web depths at the Knee Joint

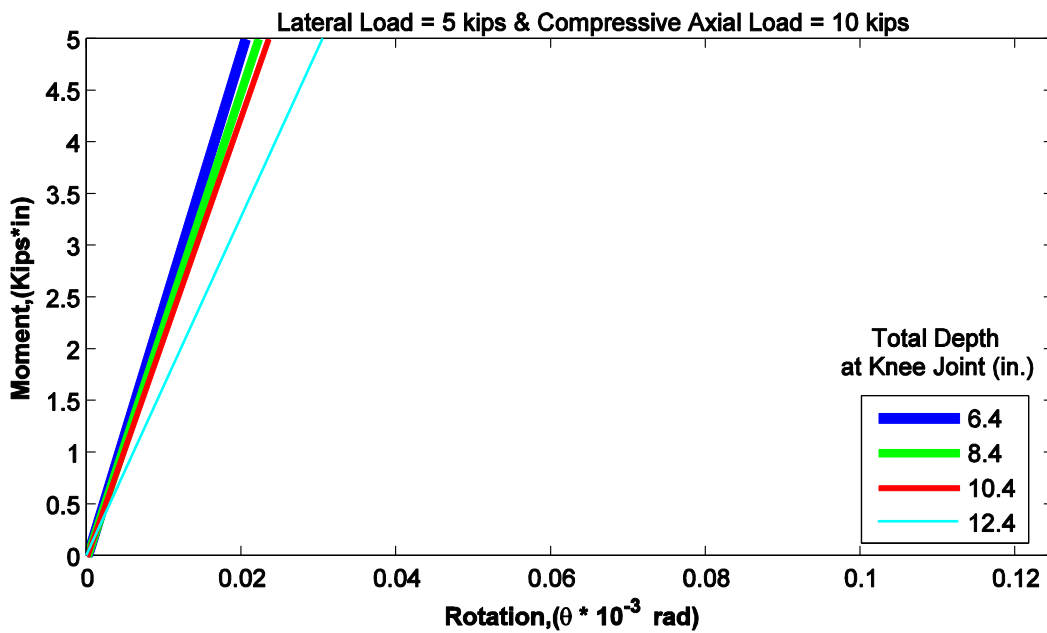


Figure 7.23. Zoom in view of Figure 7.22

When a lateral load of 5 kips and a gravity load of 10 kips were applied then with increase in tapering, the lever arm (the distance between  $C_T$  and  $C_B$ ) increased. As a result of which the section having the largest taper is subjected to maximum moment due to gravity load. Since the stiffness is load dependent, hence larger moment results in lesser stiffness of the connection.

### 7.3 Effect of Pretensioned Bolts on the Stiffness of the Connection

When a column base connection is subjected to a moment load or a lateral load, it results in uplift of the base plate from load application side of the connection (as shown in the Figure 7.24) Anchor bolts play their part by resisting that uplift. When anchor bolts are pretensioned, then their ability to resist deformation increases and the uplift is delayed. In another words connection becomes stiffer in general, however there is very little effect of pretensioning on the configuration of the connection before the uplifting of the base plate. Hence the initial rotational stiffness values with and without pretensioning are not that far apart. This wizard also provides the facility to model the connection with pretensioned bolts. The user can enter a value of pretensioning force for all the bolts in 'Bolt Properties' panel. Figures 7.25 and 7.26 validate the above explained behavior of the connection under pretensioned. The moment rotation curves with pretension and without pretension bolts are initially very close. The slightly lower initial stiffness shown by the curve having pretension bolts can be attributed to the local deformation in the base plate resulting due to pretensioning (or clamping). However after certain load level the curve without pretension (represented by blue line) overtakes the curve with pretension and at the end of the load application exhibits bigger final rotation. Table 7-11 show the values of stiffness and maximum bearing pressure obtained from the wizard.

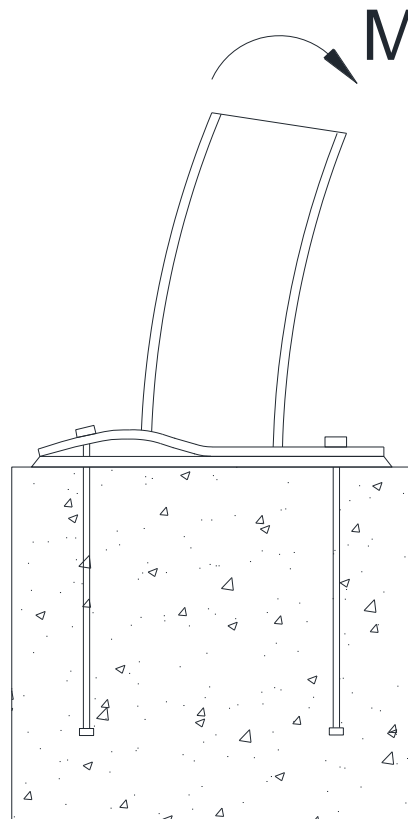


Figure 7.24. Uplift of the base plate of the connection under flexure load

Table 7-11. Values of Stiffness and Maximum Bearing Pressure for the Connection with or without Pretensioned Bolts

Pretensioned Bolts	Rotational Stiffness (kips-in/rad)	Max. Bearing Pressure (ksi.)
No	244447	1.98
Yes	221580	3.50

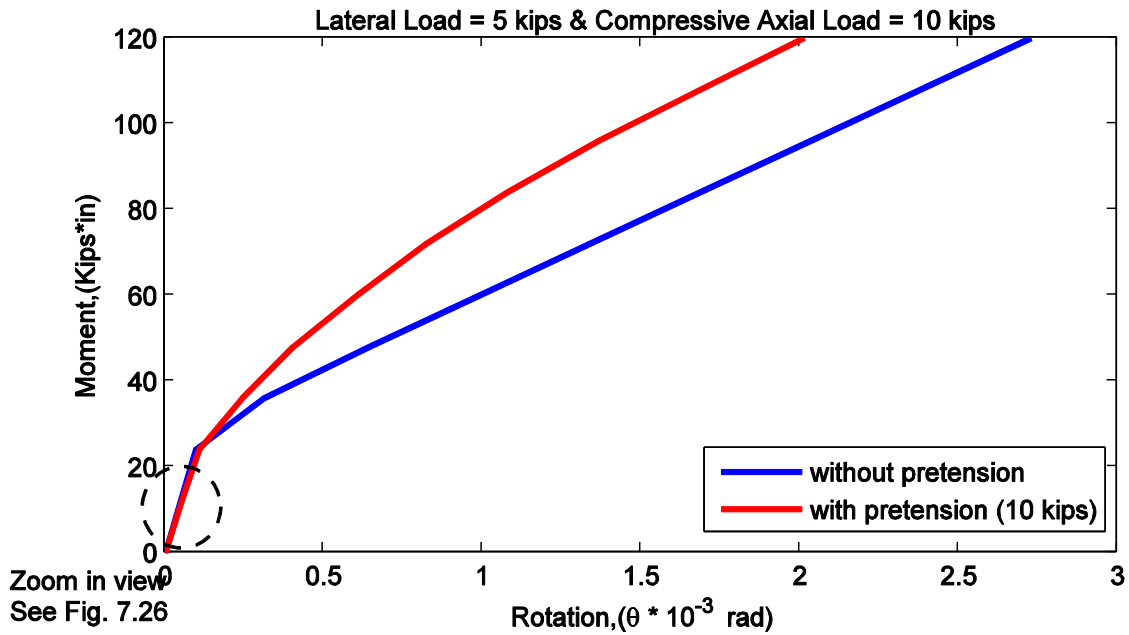


Figure 7.25. Moment Rotation Curves for connection with and without pretensioned bolts

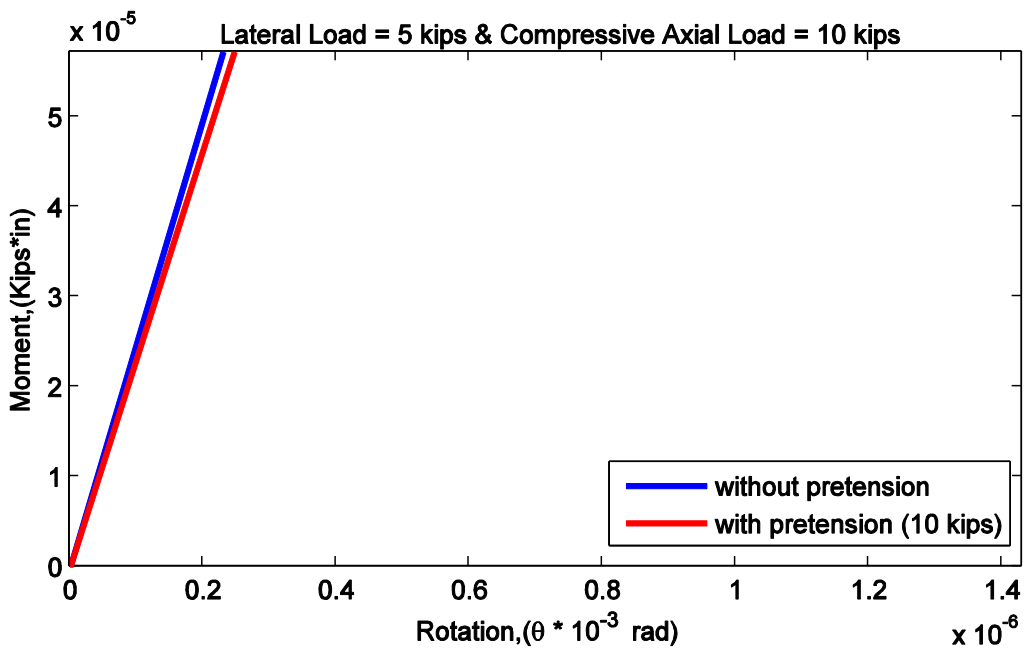


Figure 7.26. Zoom in view of Figure 7.25

## 7.4 Effect of Meshing on the Stiffness of the Connection

This wizard uses finite element analysis for estimating the stiffness. One of the most critical components of any finite element analysis is the size of the finite elements or mesh size. This wizard provides the user with the capability to change the meshing of the connection. The two meshing types that are provided in the wizard are listed below.

- **Coarse Mesh:** This type of meshing mode when selected, creates a mesh of aspect ratio 4:1. This means that the ratio of any two dimensions of a finite element in the model does not exceed the ratio 4:1.
- **Fine Mesh:** This type of meshing mode when selected, creates a mesh of aspect ratio 2:1. This means that the ratio of any two dimensions of a finite element in the model does not exceed the ratio 2:1.

In finite element analysis, a shape function is a mathematical expression which represents the variation field of any observed quantity. Some of the observed quantities are displacement, stresses, strains, etc. While performing analysis, usually smaller finite elements give more accurate results compared to coarser elements as the shape functions have to capture the observed quantity in a smaller region. Due to smaller domain of the observation, the error between the true behavior of the variable and the assumed shape function reduces, which results in more accurate analysis.

Everything is kept same as before to observe the difference in results when both types of meshing are used for the same configuration of the connection (see Section 7.1 for details of the configuration). Figures 7.27 and 7.28 shows the moment rotation curves obtained for the two types of meshing. Table 7-12 shows the values of stiffness and maximum bearing pressure obtained from the wizard.

Table 7-12. Values of Stiffness and Maximum Bearing Pressure for Coarse and Fine Meshing

<b>Meshing Type</b>	<b>Rotational Stiffness (kips-in/rad)</b>	<b>Max. Bearing Pressure (ksi.)</b>
Coarse	244447	1.98
Fine	219639	2.15



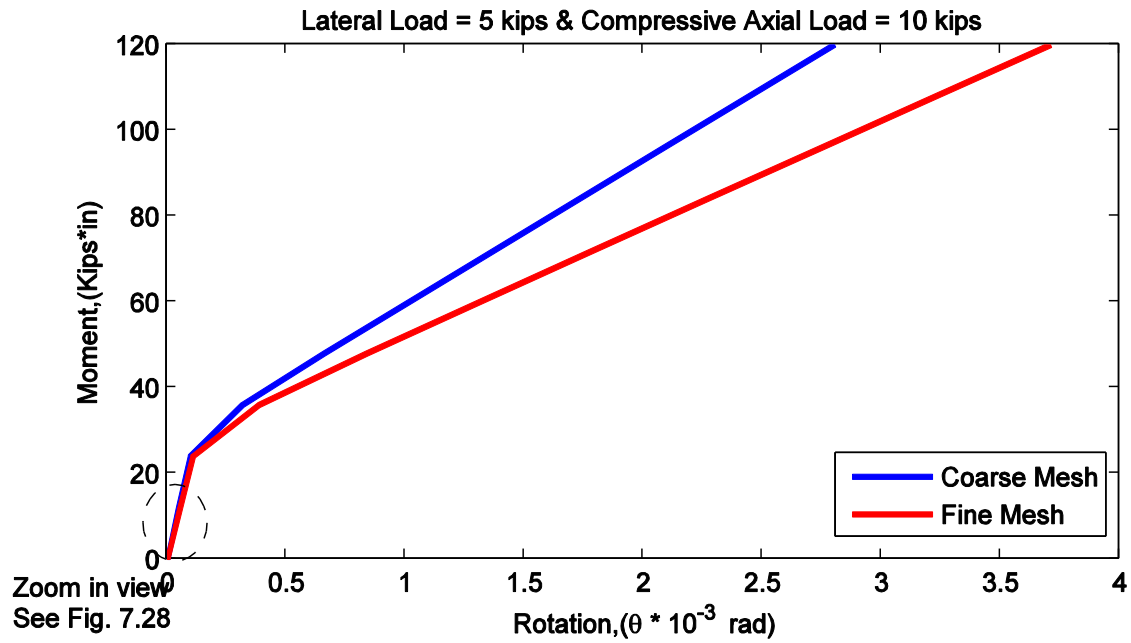


Figure 7.27. Moment Rotation Curves for Fine and Coarse Meshing

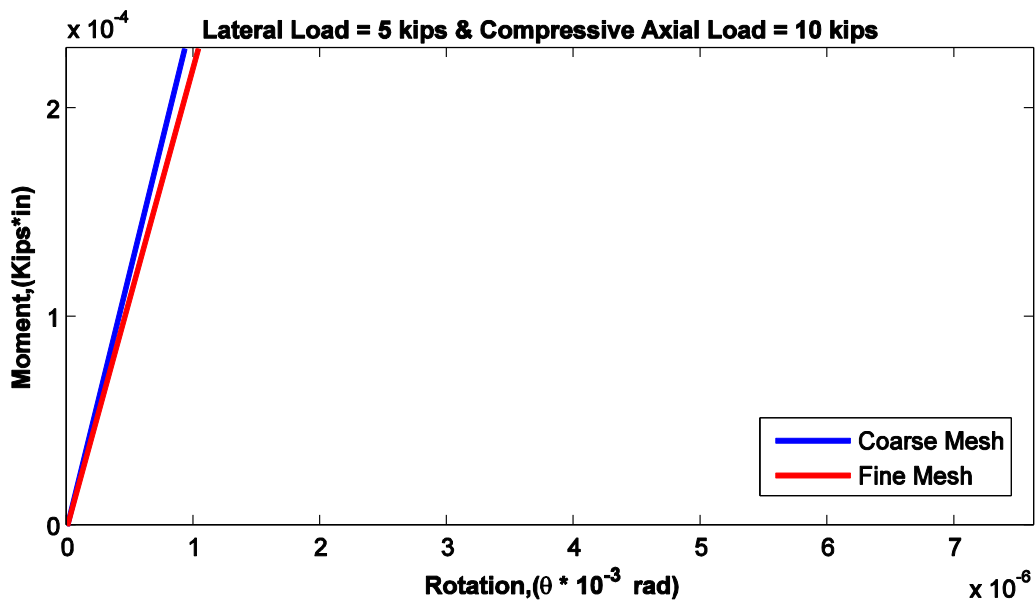


Figure 7.28. Zoom in view of Figure 7.27

From the Table 7-1 and Figures 7.27 and 7.28, it can be seen that there is not a considerable difference in the values of stiffness and maximum bearing pressure for the two meshing types. Although the consideration of fine mesh gives more accurate results, it takes a lot of time to analyze in comparison to coarse mesh type.

## 7.5 Analysis Engine: SAP2000 vs OpenSees

The wizard uses either OpenSees or SAP2000 as its analysis engine. To make sure that the user gets the same stiffness from the two programs for a configuration of the connection, the finite elements used for modeling the connection in both the programs were tested under different load cases. The results of these tests are discussed in Chapter 3. It is concluded from those tests that the responses of the finite elements used for modeling the connection in both the programs are not identical but they are close. Hence it is expected that the stiffness obtained from the two analysis engine will be different. Figures 7.29 and 7.30 show the moment rotation curves obtained from the wizard when both the programs were selected as analysis engines for the reference configuration of the connection (see Section 7.1). Table 7-13 summarizes the stiffness value calculated from each program. It can be observed that the moment rotation responses from the two programs are fairly close and the stiffness values are within 10% error. Again the possible reason for the difference in values can be attributed to the difference in responses of finite elements under nonlinear behavior of the connection.

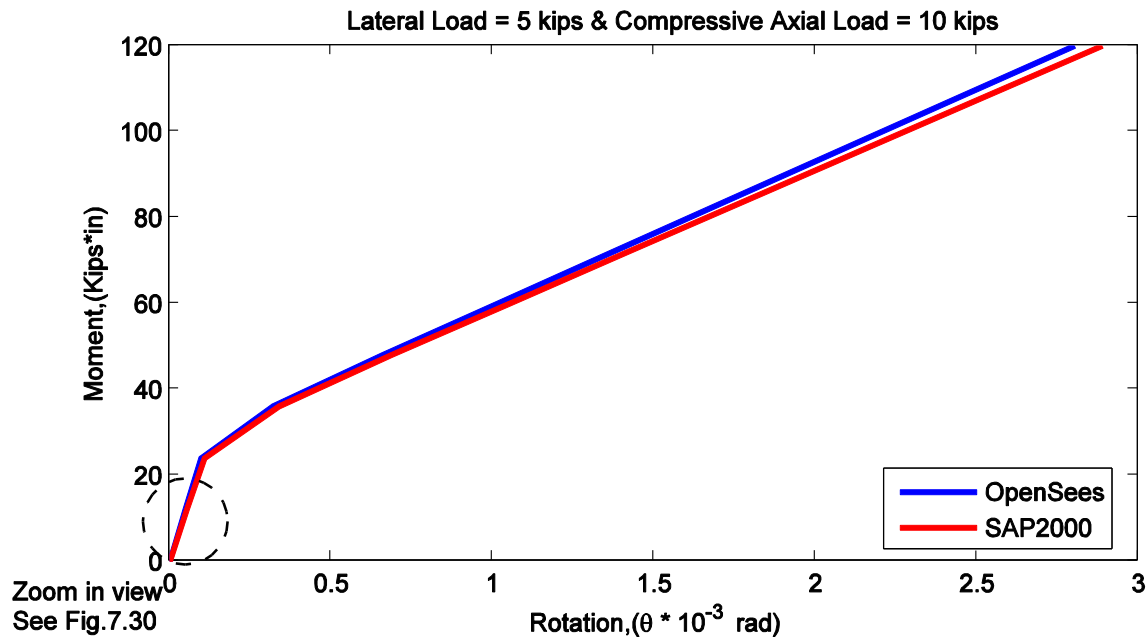


Figure 7.29. Moment Rotation Curves from OpenSees and SAP2000

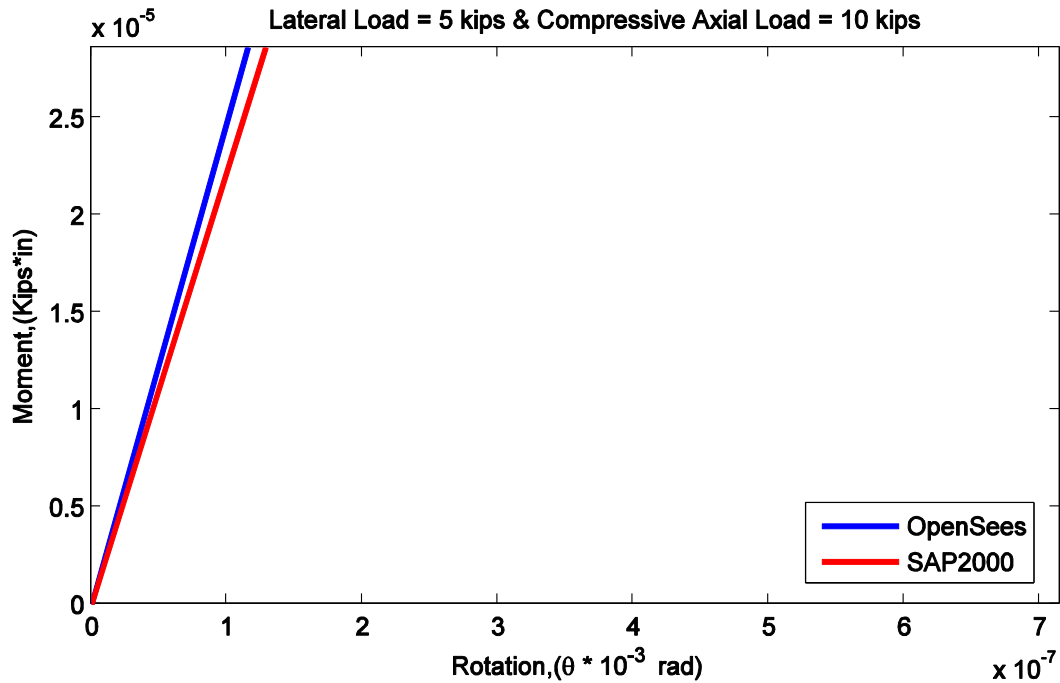


Figure 7.30. Zoom in view of Figure 7.29

Table 7-13. Values of Stiffness and Maximum Bearing Pressure from OpenSees and SAP2000

Analysis Engine	Rotational Stiffness (kips-in/rad)	Max. Bearing Pressure (ksi.)
OpenSees	244447	1.98
SAP2000	220496	2.15

## 7.6 Comparative Studies

To access the degree of accuracy of the results obtained from the column base connection wizard, its results are compared to the results obtained from the experimental tests done in the past. The typical methodology which was followed for doing this kind of study was to enter the values of different parameters in the wizard as taken in a particular experiment to model the connection. The modeled connection was then analyzed under load values which were taken in the experimental testing. The comparison was made between the moment-rotation curves obtained from the test and the wizard. The initial rotational stiffness values are calculated from the respective moment-rotation curve and plotted along with the moment rotation curves for the purpose of comparison. The following sections compare the moment rotation curves obtained from some selected analytical methods and experiments with the moment rotation curves obtained from the wizard. These selected experiments and analytical methods are also mentioned in Section 2.6. Note that this wizard is developed for predicting the initial rotational stiffness of the connection only and the plastic behavior of any component is not taken into account.

### **7.6.1 Comparison with Picard and Beaulieu (1985)**

Picard and Beaulieu (1985) performed a number of flexure and compression-flexure tests with different configurations of column base connection to observe its behavior. They used the moment rotation curves obtained from their tests to calculate a factor, referred as ‘fixity factor’ for each type of base connection. The fixity factor is equal to 1.0 for perfectly rigid connection and 0 for perfectly pinned connection. Each test was performed till the failure point of the base connection. The failure modes mainly consisted of either yielding of the base plate or tensile failure of the anchor bolts. Other than these, crushing of the concrete due to excessive compression and splitting failure of the concrete due to pull out forces were also seen in some of the tests. After observing the moment rotation curves of all the tests, one thing became clear that one can’t directly compare the results from the wizard to the results from the test as the wizard does not take account of these failure modes. In this comparative study the initial portion of the moment rotation curve obtained from a test was compared with the moment rotation curve obtained from the wizard. In any ultimate failure test, the load is increased till the failure point. In the initial part of any moment rotation curve, every component of the connection behaves elastically, since initially the loads remain under service load condition. Hence if we calculate the stiffness from the initial part of any ultimate failure moment rotation curve, we would get an elastic rotational stiffness for the connection. However when the load is increased beyond service load condition, some of the components start behaving inelastically and demonstrate plastic behavior and this wizard doesn’t have the capability to deal with that right now.

#### **7.6.1.1 Comparison with Flexure test of Picard and Beaulieu (1985)**

The values of various parameters which are entered in the wizard for the flexure test are shown in Figure 7.31. Figure 2.10 is shown again as Figure 7.32 with the wizard’s moment rotation curve and estimated stiffness plotted on it. In Figure 7.32, the curves represented by the black line and blue dotted line are the moment rotation curves corresponding to the flexure test and wizard respectively. In addition to the above mentioned moment rotation curves, the estimated stiffness by Picard and Beaulieu, wizard and Galambos (1960) are also plotted in this figure. Table 7-14 summarizes the values of the stiffness obtained from each source.

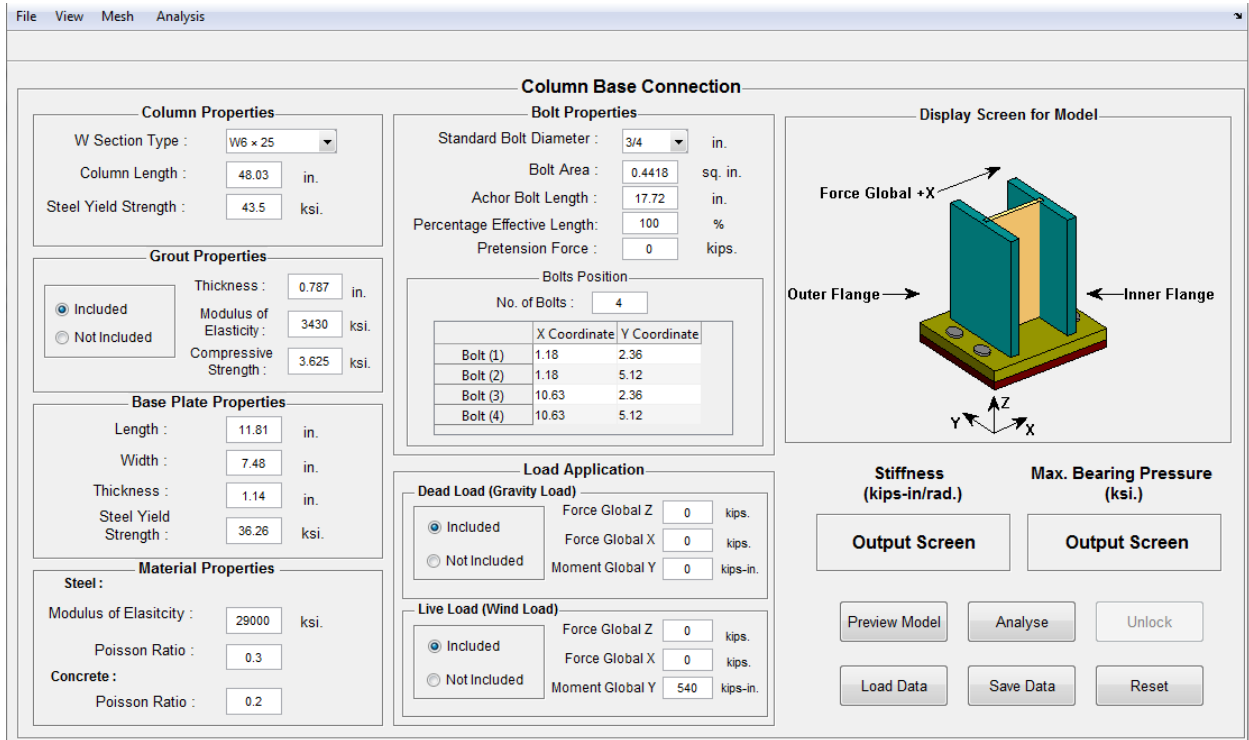


Figure 7.31. Values entered for different parameters as taken by Picard and Beaulieu for flexural test

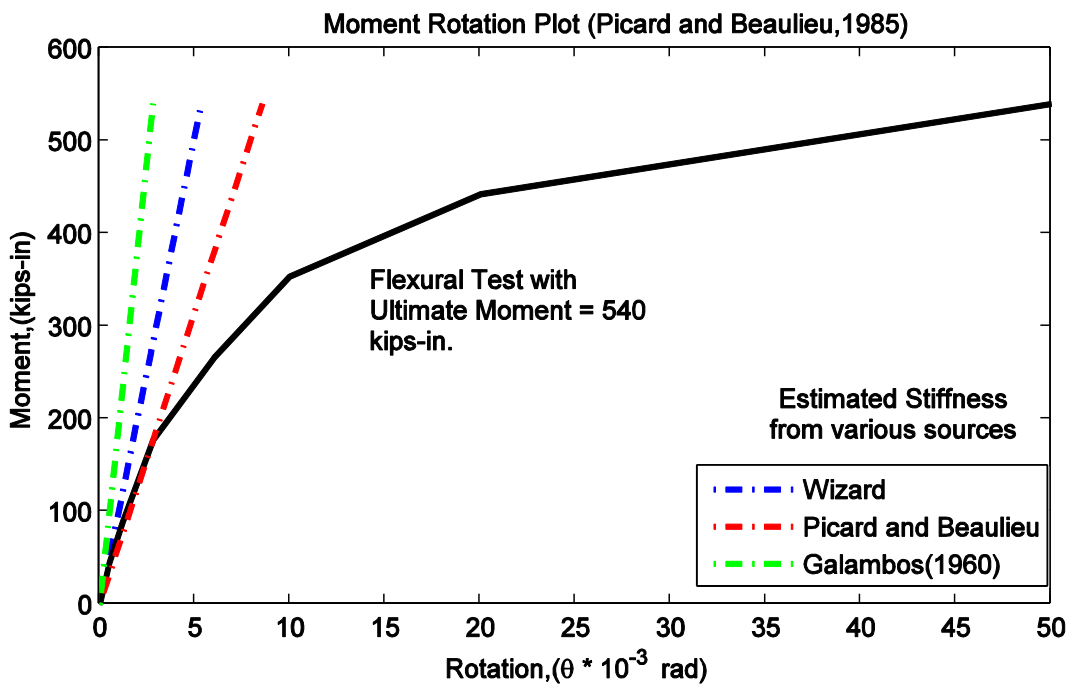


Figure 7.32. Values of stiffness obtained from different sources plotted on the moment-rotation plot of flexural test

Table 7-14. Values of Initial rotational stiffness obtained from different sources for the flexure test performed by Picard and Beaulieu (1985)

Source	Initial Rotational Stiffness (kips-in/rad.)
Picard and Beaulieu (1985)	58236 (*Actual)
Wizard	101263
Galambos (1960)	298205

From the Figure 7.32 and Table 7-14, it can be observed that the stiffness calculated by the wizard is higher than the actual stiffness and is lower than the stiffness calculated by Galambos.

### 7.6.1.2 Comparison with Compression-Flexure test of Picard and Beaulieu (1985)

The values of various parameters which are entered in the wizard for the Compression-flexure test are shown in Figure 7.33. Figure 2.11 is shown again as Figure 7.34 with the wizard's moment rotation curve and estimated stiffness plotted on it. In Figure 7.34, the curves represented by the black line and copper colored line are the moment rotation curves corresponding to the compression-flexure test and wizard respectively. In addition to the above mentioned moment rotation curves, the estimated stiffness by Picard and Beaulieu, wizard and Eurocode 3 are also plotted in this figure. Table 7-15 summarizes the values of the stiffness obtained from each source.

**Column Base Connection**

**Column Properties**

W Section Type : W6 x 25

Column Length : 48.03 in.

Steel Yield Strength : 43.5 ksi.

**Grout Properties**

Included  Not Included

Thickness : 0.787 in.

Modulus of Elasticity : 3430 ksi.

Compressive Strength : 3.625 ksi.

**Base Plate Properties**

Length : 11.81 in.

Width : 7.48 in.

Thickness : 1.14 in.

Steel Yield Strength : 36.26 ksi.

**Material Properties**

**Steel :**

Modulus of Elasticity : 29000 ksi.

Poisson Ratio : 0.3

**Concrete :**

Poisson Ratio : 0.2

**Bolt Properties**

Standard Bolt Diameter : 3/4 in.

Bolt Area : 0.4418 sq. in.

Achor Bolt Length : 17.72 in.

Percentage Effective Length : 50 %

Pretension Force : 0 kips.

**Bolts Position**

No. of Bolts : 4

	X Coordinate	Y Coordinate
Bolt (1)	1.18	2.36
Bolt (2)	1.18	5.12
Bolt (3)	10.63	2.36
Bolt (4)	10.63	5.12

**Load Application**

Included  Not Included

**Dead Load (Gravity Load)**

Force Global Z : -142.19 kips.

Force Global X : 0 kips.

Moment Global Y : 0 kips-in.

Included  Not Included

**Live Load (Wind Load)**

Force Global Z : 0 kips.

Force Global X : 0 kips.

Moment Global Y : 1707.31 kips-in.

**Display Screen for Model**

Force Global +X

Outer Flange

Inner Flange

Stiffness (kips-in/rad.)

Max. Bearing Pressure (ksi.)

Output Screen

Output Screen

Preview Model

Analyse

Unlock

Load Data

Save Data

Reset

Figure 7.33. Values entered for different parameters as taken by Picard and Beaulieu for compression-flexural test

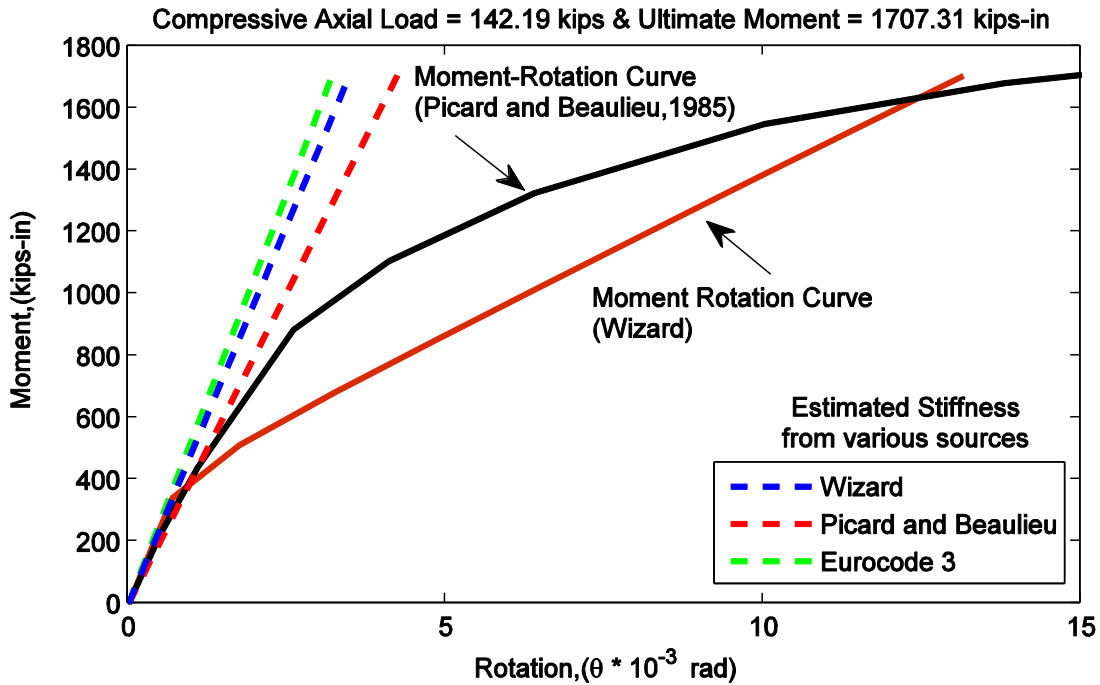


Figure 7.34. Moment Rotation Curves for compression-flexure test obtained from the Test and the Wizard

Table 7-15. Values of Initial rotational stiffness obtained from different sources for the compression-flexure test performed by Picard and Beaulieu (1985)

Source	Initial Rotational Stiffness (kips-in/rad.)
Picard and Beaulieu (1985)	402288 (*Actual)
Wizard	490021
Eurocode 3	529708

From the Figure 7.34 and Table 7-15, it can be observed that the stiffness calculated by the wizard is higher than the actual stiffness and is lower than the stiffness calculated by Eurocode.

It can be seen from the Figures 7.32 and 7.34 that the values of the stiffness predicted by the wizard are in good correlation with the two tests. The wizard performance can be further improved if it is calibrated more efficiently. For an efficient calibration, lots of experimental data is required. The experimental data can be acquired by performing tests. The tests performed in the past by various researches do not have the relevant data which can be used for the calibration or if it exists then it's not systematically presented. Hence, there is a need of experimental testing to improve the performance of this wizard. The next chapter concludes this research by summarizing each chapter and discusses some issues that are not currently included in this wizard.

## Chapter 8 Conclusions

The sole objective of this research project is to find the rotational stiffness of any configuration of the column-base connection. The research is mainly focused on finding the "initial" stiffness of the connection. To meet the objective, a systematic approach is followed, as described by Chapters 1 through 7. The final outcome of the project is a wizard that calculates the rotational stiffness of any configuration of the column-base connection using SAP2000 or OpenSees.

Chapter 1 introduces the objective of the research. The behavior of the connection under flexural loads is discussed. The procedure used in this research to calculate the stiffness of the connection is also presented.

Chapter 2 presents the past studies and experiments that are done on the column-base connection to understand its fixity behavior. The connection is classified as pinned, fixed and partially restrained on the basis of the amount of restraint provided. The methods currently used for designing of the base plate for axial loads are also summarized. Analytical procedures such as Galambos (1960) and Eurocode 3 (EN-1993-1-8) are described in detail and their applicability is shown in finding the stiffness of a connection. Finite element models of the connection investigated by various researchers are also presented in this literature review.

Chapter 3 investigates the finite elements present in SAP2000 and OpenSees which are used for modeling of the column base connection. The main elements selected for modeling the connection are shell elements and brick elements in both the programs. The elements are tested for three different load cases, namely gravity, moment and gravity plus moment. The observed response is the deformation of each finite element in both the programs. The following conclusions are obtained by the testing.

- The shell element in both the programs gives different deformations along the length of the base plate when a highly nonlinear force deformation relationship is assigned to the spring elements and the base plate is subjected to a moment load.
- It is also found out that the brick element in OpenSees deforms less in comparison to brick element of SAP2000 under gravity load.

As a result of the above testing conclusions, it is expected that the stiffness calculation from the two programs will not be exactly same, but it will be close enough (within 10%).

Chapter 4 describes the development process of the preliminary models of the connection in SAP2000 and OpenSees using various finite elements. A large number of models were created to obtain the best representative model of the actual connection. The models were tested under various load cases to understand the behavior of a real connection. The assumptions and methodology followed for the modeling are also described in detail in this chapter.



Chapter 5 describes the ways in which one can access SAP2000 and OpenSees from MATLAB. The MATLAB scripts which are used by the wizard to automatically create a model and perform an analysis in SAP2000 or OpenSees are also described briefly.

Chapter 6 presents the development process of the GUI developed for this wizard. The different methods for developing GUI's are discussed. The GUI for this wizard is classified into four major components, namely input panels, menu bar, action buttons and display screen and Output windows. Each of the components and its role in the GUI is discussed briefly. More details of the functioning of the wizard can be found in the Appendix C.

Chapter 7 lists some parameters which significantly affect the stiffness of the connection. A reference configuration is defined for performing a number of parametric studies. The effect of each parameter is observed by the moment rotation plot obtained from the wizard. For each parameter one plot is obtained, which contains a set of moment rotation curves. Each moment rotation curve in this plot corresponds to a unique value of the observed parameter which is being varied. While varying one parameter other details are kept same as of the reference configuration. The later part of this chapter compares the results obtained from the wizard with some of the tests done on column base connection. It is observed that the results obtained from the wizard correlated well with the tests used for comparison.

In conclusion, this research explores the behavior of column base connection by formulating a systematic approach to find its initial elastic rotational stiffness. To fulfill the objectives of this research, a wizard is created to predict the stiffness value for any typical column base connection configuration. Along with the value of stiffness, a moment rotation plot and a bearing pressure plot for each configuration can also be obtained from the wizard.

## 8.1 Scope for Future Research

- **Inclusion of plastic mechanism in the model:** Right now the wizard only predicts the initial stiffness of the connection, where most of the components of the connection behave elastically. Under high loads, various components start to behave nonlinearly and usually plastic behavior can be observed. Currently, the wizard only assigns linear elastic materials to the finite elements. Therefore, under high loads, the mechanism like plastic deformation is not observed in this wizard. To get the true behavior of the connection under the loads close to ultimate strength, a finite element model has to be developed to take account of mechanisms such as yielding of the base plate, crushing of the concrete, etc. This finite element model should contain nonlinear shell and brick elements to incorporate plastic mechanisms described above. Currently, OpenSees and SAP2000 do not have nonlinear shell or brick finite element, therefore the expansion of this research on current procedure is somewhat restricted.
- **Replacing OpenSees and SAP2000 with custom finite element program:** The restriction of the wizard that is mentioned in the above point can be removed by replacing OpenSees and

Sap2000 with the finite element program developed by Dr. Charney at Virginia Tech. This program can be customized to incorporate plastic mechanisms in the connection.

- **Modeling of whole gable frame to find equivalent stiffness of the frame:** The wizard calculates the stiffness of a single column base connection and it doesn't give an estimate of the stiffness of the frame. So essentially, if we think that stiffness of a gable frame is twice the stiffness of the connection, then that is not correct. In actuality, two connections on either side in gable frame contribute in different proportion to the stiffness of the frame. Out of these two connections, one connection is subjected to compression, therefore provides higher stiffness and the other one is subjected to tension, so provides lesser stiffness under lateral movement. Figure 6.1 shows that under application of a lateral load, the left column of the frame is subjected to compression force represented by force  $R_1$  while the right column is subjected to tension force represented by force  $R_2$ . Therefore, to get the true stiffness of the frame, the whole frame has to be modeled in the wizard.

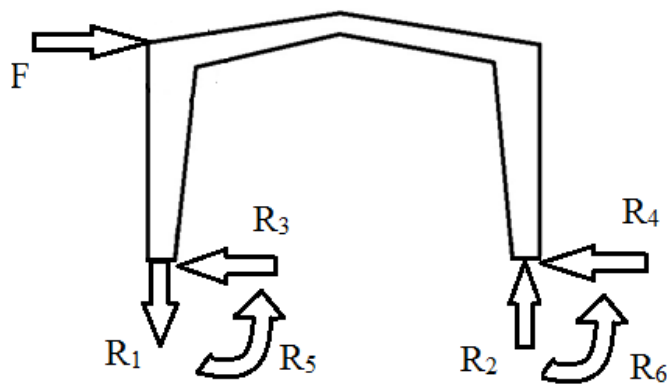


Figure 8.1. Reactions on the base of a gable frame subjected to a lateral load

- **Need for Experimental Testing:** The values of the stiffness calculated by the wizard are based on the analytical model explained in Chapter 4. Although the wizard estimates have been compared with some of the experiments in this thesis, there is room for improvement. The model can be further calibrated and brought closer to the actual behavior of the connection if there is sufficient experimental data available. This requires performing experiments on the column base connection under various load cases to gather more data to develop more accurate analytical and mechanical model of the connection.

Essentially, every column base connection has some restraint at the base and by not taking account of this fixity, one may end up getting larger drifts and larger moments at the knee of a gable frame. This research is an effort to quantify the amount of rigidity that is present in a column base connection. This research also investigates the behavior of the base connection

under various load cases. The effect of various parameters on the stiffness of the connection is also observed.

On final note, this research provides a way to calculate a stiffness value for any column base connection and then that value can be used for performing further analysis. The authors hope that the research will be really useful for those who want to use the characteristics of a partially restrained base connection instead of assuming it as pinned.

## REFERENCES

- AISC (2010). Steel Construction Manual. American Institute of Steel Construction, Chicago, IL.
- Bajwa, M. S. (2010). "Assessment of Analytical Procedures for designing metal buildings for Wind drift serviceability." M.S. thesis, Virginia Tech, Blacksburg, VA.
- DeWolf, J. T., and Ricker, D. T. (1990). AISC Design Guide No.1, *Column Base Plates*. AISC, Chicago, IL.
- Dumas, M., Beaulieu, D., and Picard, A. (2006). "Characterization equations for steel column base connections." *Canadian Journal of Civil Engineering*, 33(4), 409-420.
- Ermopoulos, J. C., and Stamatopoulos, G. N. (1996). "Mathematical modeling of column base plate connections." *Journal of Constructional Steel Research*, 36(2), 79-100.
- Eurocode 3 (1993). *Design of Joints* (ENV-1993-1-8). European Committee for Standardization, London, UK.
- Fling, R. S. (1970). "Design of Steel Bearing Plates." *Engineering Journal*, AISC, 7 (2), 37-40.
- Galambos, T.V. (1960). "Influence of partial base fixity on frame stability." *J. Struc. Div.*, 85(5), 85-117.
- Gresnigt, A., Romeijn, A., Wald, F., and Steenhuis, M. (2008). "Column bases in shear and normal force." *Heron*, 53(1-2), 87-108.
- Hamizi, M., and Hannachi, N. (2007). "Evaluation by a finite element method of the flexibility factor and fixity degree for the base plate connections commonly used." *Strength of Materials*, 39(6), 588-599.
- Hon, K., and Melchers, R. (1988). "Experimental behavior of steel column bases." *Journal of Constructional Steel Research*, 9(1), 35-50.
- Jaspart, J. P., and Vandegans, D. (1998). "Application of the component method to column bases." *Journal of Constructional Steel Research*, 48(2-3), 89-106.
- Krishnamurthy, N., and Thambiratnam, D. P. (1990). "Finite element analysis of column base plates." *Computers and Structures*, 34(2), 215-223.
- MATLAB (Version 7.12.0(R2011a)) [Computer Software]. Natick, Massachusetts : MathWorks.

Murray, T. M. (1983). "Design of Lightly Loaded Column Base Plates." *Engineering Journal*, AISC, 20 (4), 143-152.

OpenSees (Version 2.3.1) [Computer Software]. Berkley, CA : University of California

Picard, A., and Beaulieu, D. (1985). "Behavior of a simple column base connection." *Canadian Journal of Civil Engineering*, 12(1), 126-136.

SAP2000 (Version 14.0.0) [Computer Software]. Berkley, CA : Computer and Structures, Inc.

Stockwell, F. W. (1975). "Preliminary Base Plate Selection." *Engineering Journal*, AISC, 12(3), 92-99.

Thambiratnam, D. P., and Paramasivam, P. (1986). "Base plates under axial loads and moments." *Journal of Structural Engineering*, 112, 1166-1182.

Thornton, W.A. (1990). "Design of Small Base Plates for Wide-Flange Columns." *Engineering Journal*, AISC, 27(3), 108-110.

## APPENDIX A: Stiffness Calculations

1) Flexure Test - Picard and Beaulieu (1985)

(i) Stiffness calculated from the moment rotation plot

$$\text{Initial Rotational Stiffness (For any moment – rotation plot)} = \frac{\text{Initial Moment}}{\text{Initial Rotation}}$$

Initial Moment (within elastic range of the plot) = 180 kips-in.

Initial Rotation (within elastic range of the plot) = 0.0031 rad.

$$\text{Initial Rotational Stiffness} = \frac{180}{0.0031} = 58065 \text{ kips-in/rad}$$

(ii) Stiffness calculation by Galambos Analytical Procedure

According to Galambos (1960) , the rotational stiffness of a column base connection is given by following equation when the effect of the foundation is neglected (same as Equation 2-1 in this thesis)

$$K_{con} = \frac{M}{\theta} = \frac{b d^2 E_c}{12}$$

where  $M$  is the moment at the base of the column,  $\theta$  is the rotation at the base of the column,  $b$  is width of the base plate,  $d$  is length of the base plate and  $E_c$  is modulus of elasticity of the concrete.

Base plate width = 7.48 in.

Base plate length = 11.81 in.

Modulus of Elasticity of concrete used for the grout construction = 3430 ksi.

Putting above values in the above equation to get a value of  $K_{con}$

$$K_{con} = \frac{7.48 * 11.81^2 * 3430}{12} \approx 298,205 \text{ kips-in/rad.}$$

2) Compression-Flexure Test - Picard and Beaulieu (1985)

(i) Stiffness calculated from the moment rotation plot

$$\text{Initial Rotational Stiffness (For any moment – rotation plot)} = \frac{\text{Initial Moment}}{\text{Initial Rotation}}$$

Initial Moment (within elastic range of the plot) = 442.54 kips-in.

Initial Rotation (within elastic range of the plot) = 0.00011 rad.

$$\text{Initial Rotational Stiffness} = \frac{442.54}{0.0011} = 402309 \text{ kips-in/rad}$$

(ii) Stiffness calculated by Eurocode 3

Using the steps provided in Section 2.3.2 to find the rotational stiffness ( $S_j$ ) of the column base. Also see Section 2.3.2 for the definition of each term used or calculated below.

From Table 6-12, Eurocode 3, EN-1993-1-8

$$S_j = \frac{Ez^2}{\mu \left( \frac{1}{k_{T,l}} + \frac{1}{k_{C,r}} \right)} \frac{e}{e + e_k} \quad \text{where, } e_k = \frac{z_{C,r} k_{C,r} - z_{T,l} k_{T,l}}{k_{T,l} + k_{C,r}}$$

The configuration of the connection used for calculating different terms in the above equation is presented in Section 2.6

$z$  is the lever arm shown in Figure 2.5 calculated for the configuration of Section 2.6

$$z = \frac{11.81 - 6.38}{2} - 1.18 + 6.38 = 7.915 \text{ in.} = 201 \text{ mm}$$

$E$  is modulus of elasticity of the Steel = 29000 ksi. = 200000 N/mm<sup>2</sup>

$\mu$  (stiffness ratio) = 1 (for initial stiffness calculation)

$e$  is the eccentricity of the applied axial load from center of the connection

$$e = \frac{M}{N} = \frac{-1707.31}{142.2} = 12 \text{ in.} = 305 \text{ mm}$$

$Z_{C,r}$  is the distance from the center of the connection to the point on the base plate beneath the column flange in compression

$$z_{C,r} = 3.19 \text{ in.} = 81 \text{ mm}$$

$Z_{T,l}$  is the distance from the center of the connection to the point on the base plate intersecting with bolt position (Figure 2.5).

$$z_{T,l} = 4.725 \text{ in.} = 120 \text{ mm}$$

$k_{C,r}$  is the compression stiffness coefficient of the right side (compression side for this load configuration) of the connection and should be taken as equal to the stiffness coefficient  $k_{13}$  (Table 6.11, Eurocode 3, EN-1993-1-8)

$$k_{C,r} = k_{13} = \frac{E_c \sqrt{b_{eff} l_{eff}}}{1.275 E}$$

Effective length of T stub (Figure 2.6) = 7.48 in. = 189 mm

Effective width of T stub (Figure 2.6) = 3.3 in. = 83.82 mm (assumed on the basis of compression region of the baseplate)

$E_c$  is modulus of elasticity of concrete = 23661 N/mm<sup>2</sup>

Hence,

$$k_{C,r} = \frac{23661 \sqrt{189 * 83.82}}{1.275 * 200000} \approx 11.67$$

where  $k_{T,l}$  is the tension stiffness coefficient on the left side (tension side for this load configuration) of the connection and should be taken as equal to the sum of the stiffness coefficients  $k_{15}$  and  $k_{16}$  (Table 6.11, Eurocode 3, EN-1993-1-8)

The Stiffness coefficient ( $k_{15}$ ) for the base plate in bending in the absence of the prying force is given by the following relation.

\* Assuming no prying force

$$k_{15} = \frac{0.425 l_{eff} * t_p^3}{m^3}$$

Since,

$t_p$  is thickness of the base plate = 1.14 in. = 28.96 mm



$m$  is a dimension from the T stub model (Figure 2.6) = 1.38 in. = 35 mm

$$k_{15} = \frac{0.425 * 189 * (28.96)^3}{(35)^3} \approx 45.5$$

The Stiffness coefficient ( $k_{16}$ ) for the anchor bolts in tension in the absence of the prying force is given by the following relation.

$$k_{16} = \frac{2A_b}{L_b}$$

$A_b$  is area of the anchor bolt = 0.4418 in<sup>2</sup> = 285 mm<sup>2</sup>

$L_b$  is the length of the anchor bolt = 17.72 in. = 450 mm

$$k_{16} = \frac{2 * 285}{450} \approx 1.267$$

Since,

$$k_{T,l} = k_{1,5} + k_{1,6}$$

Hence,

$$k_{T,l} = 45.5 + 1.267 = 46.76$$

$$e_k = \frac{81 * 11.67 - 46.76 * 120}{11.67 + 46.76} = -79.83$$

Rotational Stiffness ( $S_j$ ) comes out to be following.

$$S_j = \frac{200000 * 201^2}{1 * \left( \frac{1}{11.8} + \frac{1}{46.76} \right)} \left( \frac{-305}{-305 - 79.83} \right) = 59849 \text{ KNm} = 529708 \text{ kips.in/rad}$$

## APPENDIX B: MBMA SURVEY

**Table B-1 Base Plate Dimensions**

<b>BASE PLATE DIMENSIONS</b>						
	<b>A &amp; S building system</b>	<b>Chief Buildings</b>	<b>United Structures of America, Inc.</b>	<b>BlueScope Buildings North America</b>	<b>NCI Group Inc.</b>	<b>Salsa Steel Corp.</b>
<b>Range of width</b>	6"-14"	6" - 14"	4"-12", rarely 24" or 36"	8"-18"	6 "-20"	6"-12"
<b>Common width</b>	6"-8"	8"	8"	8"	6"-8"	6"-8"
<b>Range of Length</b>	8"-24"	8" - 30+"	8" to 48", sometimes 72"	7"- 60"	8"-48"	10"-14"
<b>Common Length</b>	12"	10" - 12"	12"	14"	8"	12"
<b>Range of thickness</b>	3/8"-5/4"	3/8"-3/4" Pinned, up to 2" Fixed	3/8"-3/2", rarely 3"	0.375" - 2.0"	3/8"-2"	3/8"-1"
<b>Common thickness</b>	3/8" - 1/2"	3/8"	1/2"	0.375"	3/8"	1/2"

**Table B-2. Anchor Bolts Details**

<b>ANCHOR BOLTS</b>						
	<b>A &amp; S building system</b>	<b>Chief Buildings</b>	<b>United Structures of America, Inc.</b>	<b>BlueScope Buildings North America</b>	<b>NCI Group Inc.</b>	<b>Salsa Steel Corp.</b>
<b>Min. No. of bolts</b>	4	4	4	2	4	4
<b>Max. No. of bolts</b>	8	10	8 to 12	6	12	8

<b>Min. Dia of bolts</b>	5/8"	3/4"	1/2"	3/4"	3/4"	5/8"
<b>Max. Dia of bolts</b>	3/2"	3/2"	3/2"-2" or more	2"	3/2"	3/2"
<b>Common gauge dimension or spacing used</b>	<p>5/8" bolts: 3 x 3 square pattern, 2 1/2" from outside flange to first bolt row for exterior columns.</p> <p>5/8" &lt; bolt diameter &lt; 1": 4 x 4 square, 3" from outside flange to first bolt row.</p> <p>&gt;= 1" bolts: 5 x 5 square pattern, 4" from outside flange to first bolt row.</p>	<p>Typical 4" gage, 4" pitch for anchors less than or equal to 1.25" diameter.</p> <p>Anchor rod material is ASTM F15554, Fy = 36</p>	4" / 6" / 8"	g = 5"; p = 3.5" or 5"	4 - 5 when there are two columns of anchor rods, sometimes there are 4 columns of anchor rods	4"-4" most common, 3"-3" very few but is still used, 5"-5" used rarely but good spacing only on 8" wide base plates or greater

**Table B-3. Installation Details**

<b>INSTALLATION</b>						
	<b>A &amp; S building system</b>	<b>Chief Buildings</b>	<b>United Structures of America, Inc.</b>	<b>BlueScope Buildings North America</b>	<b>NCI Group Inc.</b>	<b>Salsa Steel Corp.</b>
<b>Method-tightening bolts</b>	No comment	No comment	Mostly socket wrenches - sometimes impact wrenches.	snug tight	snug tight	Hand tightened

<b>Embedment length- anchor bolts</b>	No comment	No comment	12" to 24" Hooked anchors - rarely used. AISC recommends headed or nutted anchors to resist uplift forces.	No comment	No comment	min 12" embed., min 3-4" hook
<b>Thickness of grout</b>	Not aware	1" typically, only 10-20% buildings require	3/2"	2"	the thickness will be about 1/2" large than nut thickness	0"

**Table B-4. Column Design Details**

<b>COLUMN DESIGN</b>						
	<b>A &amp; S building system</b>	<b>Chief Buildings</b>	<b>United Structures of America, Inc.</b>	<b>BlueScope Buildings North America</b>	<b>NCI Group Inc.</b>	<b>Salsa Steel Corp.</b>
<b>Typical Dimensions</b>	<p>Straight (parallel flanges) or taper from the base.</p> <p>The flanges 5" - 14" wide, with typically 6" or 8" .</p> <p>The column depth - 8"-24" with typically 10"-12".</p> <p>Web thicknesses</p>	Column flush with the outside edges of the base plate, both in depth and width	8" wide X 12" deep	design driven, varies, commonly $d_{min}=12"$	No comments	No comments

	range from 10 ga-3/8", with typically 10 ga or 3/16"					
<b>Connection between base plate and column</b>	The base plates welded with fillet welds on one side of the web, on the inside of each flange. When tapering columns (sloping inside flange) the fillet weld located on the outer side of the flange instead of the inside, or else a C.J.P. weld is used.	single or double fillet welds	fillet welds	fillet welds, 1-side or 2-sided	fillet weld on the inside of the flanges and along one side of the web. When there is a fixed base plate the flange and web are CJP welded	No comments
<b>Location of column relative to base plate</b>	Columns centered in the width of the base plate.  In the length of the base plates, the base plate is typically flush with the column's flanges or overhanging a small amount (not more than 1/4" typically).  This base plate extension allows the welder to locate the fillet weld on the outside of the flange.	Column is centered on base plate.  The first Anchor bolt is a specific distance in from the outside flange, so as the column depth varies and the quantity of bolts vary the bolt patterns will not be centered on the column or baseplate.	centered on base plate.	approx. centered; but anchor rods are often closer to one flange	centered plus or minus 1/2"	No comments

**Table B-5. Foundation Design Details**

<b>FOUNDATION DESIGN</b>						
	<b>A &amp; S building system</b>	<b>Chief Buildings</b>	<b>United Structures of America, Inc.</b>	<b>BlueScope Buildings North America</b>	<b>NCI Group Inc.</b>	<b>Salsa Steel Corp.</b>
<b>Slab thickness in vicinity of base plate</b>	No comment	No comment	4" to 8"	No comment	refer to next cell	18" min, (includes 12" min. embedment footing into soil + 6" min top of slab above grade) up to 54" Thick
<b>Common Footing Details</b>	No comment	we specify minimum 3000 psi concrete for bearing	No comment	No comment	refer to next cell	spread or pad footing centered below column baseplate
<b>Common Soil Details</b>	No comment	No comment	No comment	standard design based on $f'_c=2000$ psi - 3000 psi	refer to next cell	1,000-1,500 psf soil bearing capacity, Sd soil profile, unless soils report is present

## APPENDIX C: USER'S MANUAL

### C.1.1 Introduction

The column base connection wizard is a program which can be used to predict the rotational stiffness of any configuration of a column base connection. A typical configuration of the column base connection consists of column, base plate, anchor bolts, grout and footing as its components (see Figure C.1). Under an application of external loads the above mentioned components interact with each other and contribute to the stiffness of the connection in different proportion. The wizard works in two modes, Graphical User Interface (or GUI) mode and Batch mode. The GUI mode provides a visual interface by which a user can interact with the wizard and see a visual display of the modeled connection. While the Batch mode is a non-visual interface of interacting with the wizard. The user can obtain the rotational stiffness for any configuration of connection very quickly. The external loads that can be applied to the connection in this wizard are gravity loads and wind loads. This user manual provides the installation instructions (Section 1.3), functioning of various features of GUI mode (Chapter 2), functioning of various features of Batch mode (Chapter 3) and three comprehensive examples (Chapter 4). Note that the stiffness in this wizard simply means the rotational stiffness of the connection.

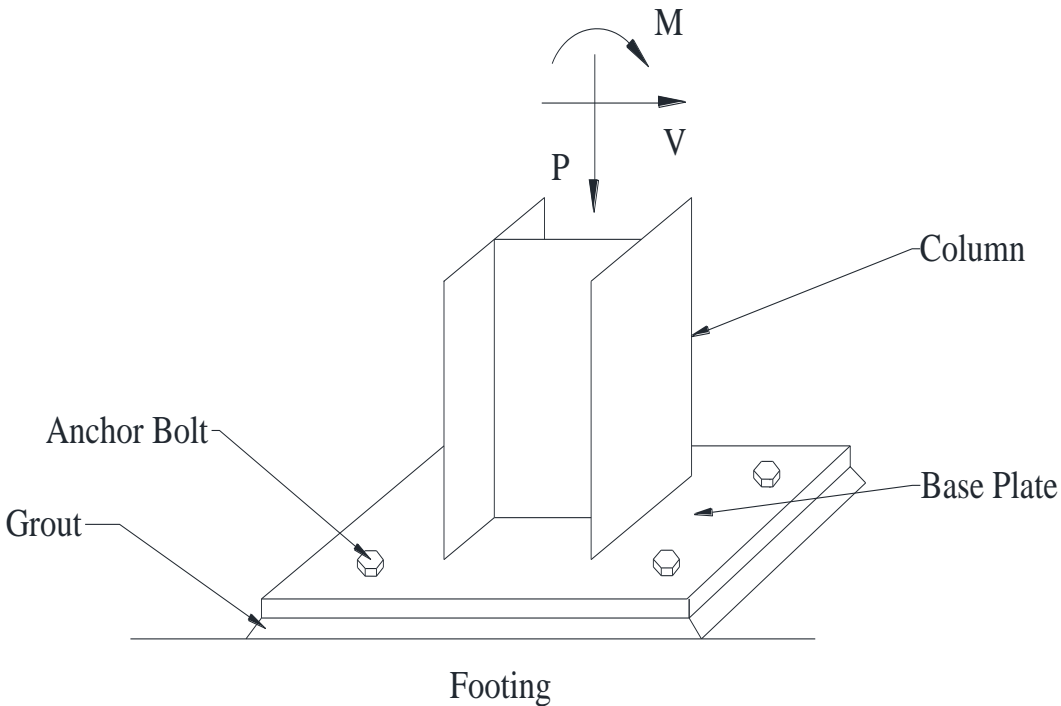


Figure C.1. A typical configuration of column base connection

## C.1.2 Computer System Requirements

Table C-1 lists the computer system requirements for the column base connection wizard.

Table C-1. Computer System Requirements

Component	Requirement
Computer and Processor	1.5-gigahertz (GHz) processor or higher.
Memory	2048 megabytes (MB) RAM or higher.
Hard disk	1.5 gigabytes (GB) available disk space for a typical installation.
Display	1024 × 768 or higher resolution monitor.
Operating system	Windows 7 operating systems, Windows XP with SP 3 (32-bit), Windows XP x64 Edition with SP 2. Mac OS with Windows Platform
Others	SAP2000 must be installed on the computer system to use it as one of the analysis engines

## C.1.3 Instructions for Installing the Wizard in Windows 7

As described in Section 1.1, this wizard has two modes of operation, GUI mode and Batch mode. The instructions for installing the GUI mode are provided in the section below. The instructions for installing the Batch mode are provided in Section 1.3.2. The standalone version of the column base connection wizard can be found in a folder labeled as 'Connection\_Wizard'. Copy this folder to any location on your computer. Open the folder by clicking on it. The folder contains two folders, namely 'Batch\_Mode' and 'GUI\_Mode' (see Figure 1.2).



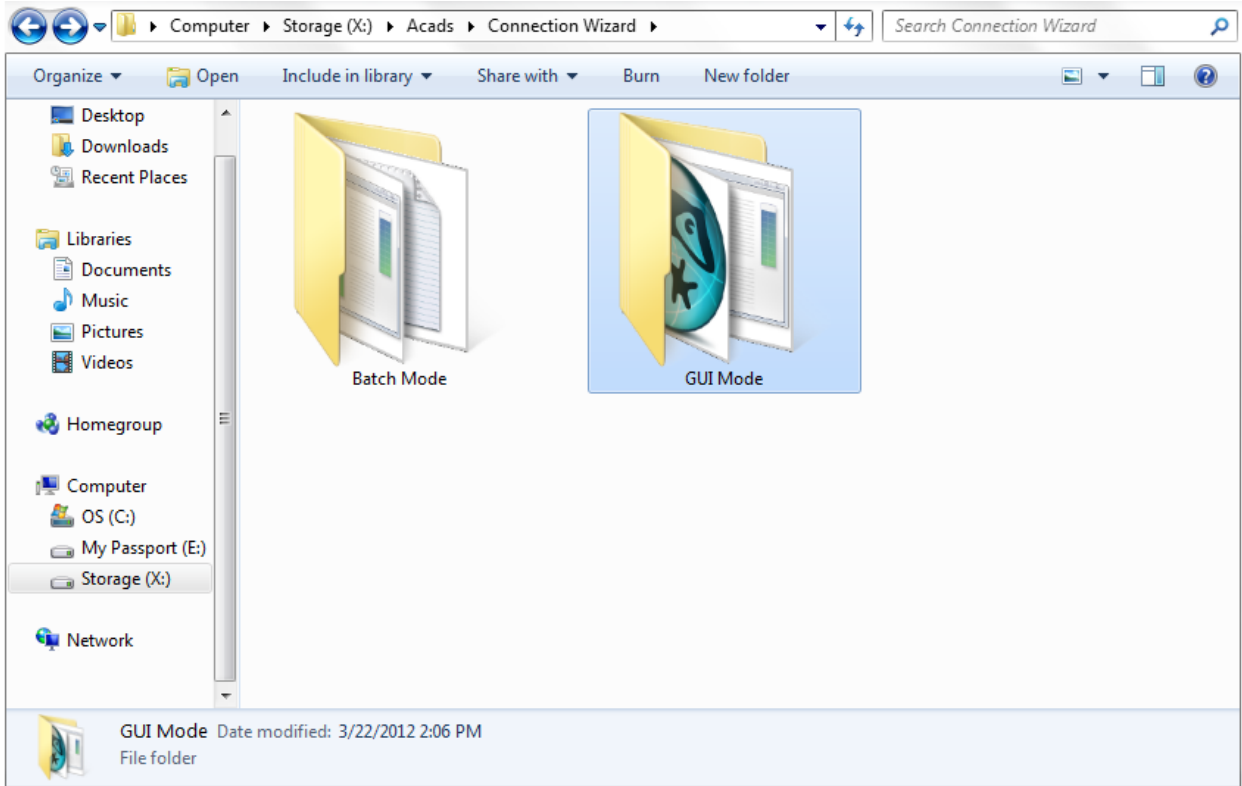


Figure C.2. 'Connection\_Wizard' folder

### C.3.1 Instructions for Installing the GUI Mode

1. Open the folder 'GUI\_Mode' by clicking on it (see Figure C.2). The folder typically contains two folder and three files as shown in Figure C.3. Right click the file labeled as 'GUI\_pkg' and click on 'Run as Administrator' option (if available) as shown in Figure C.4. A user account control warning may or may not appear on the screen. If it pops up on the screen, then click 'Yes'. This whole process will expand the files compressed inside this package. Note that if the 'Run as Administrator' option is not available, then simply double click the 'GUI\_pkg' file and proceed ahead by following the steps given below.

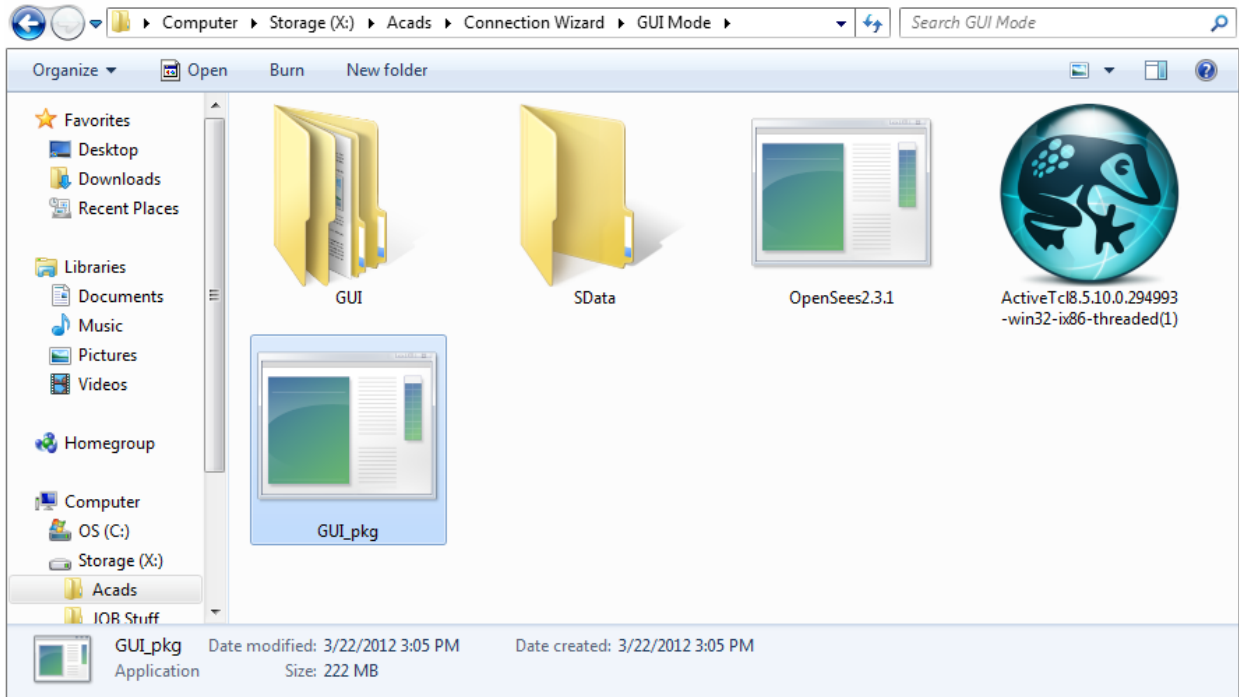


Figure C.3. Folder containing installation package for GUI Mode

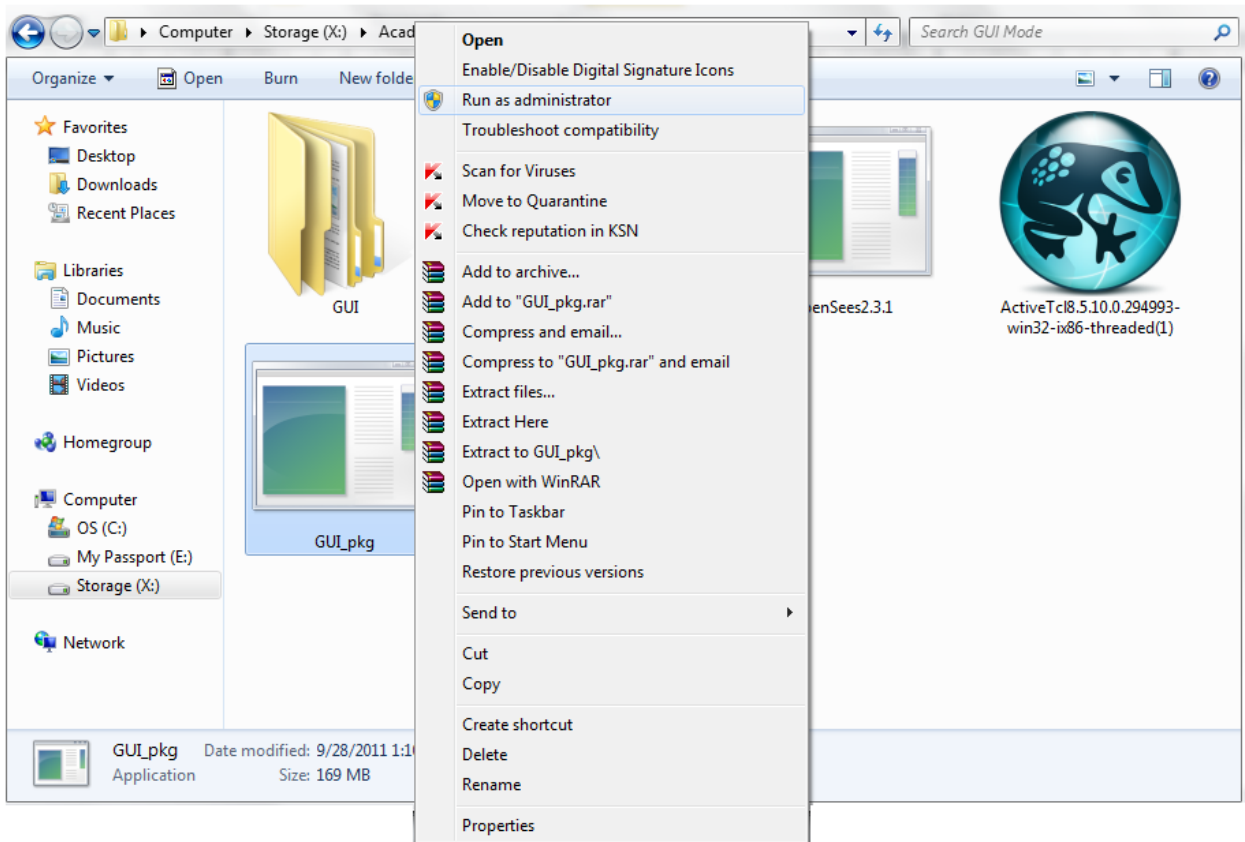


Figure C.4. 'GUI\_pkg' is executed by right clicking and choosing 'Run as administrator'

1. A number of files and a black window (shown in Figure C.5) automatically appear in the same 'GUI\_Mode' folder after executing the file 'GUI\_pkg' as per step 1.
2. This black window automatically executes a wizard for installing MATLAB Compiler Runtime (MCR). Note that the MCR must be installed on any computer machine to run the column base connection wizard in a standalone format.
3. Follow the instruction on the screen as shown in the Figure C.5 and choose a language for the installation.

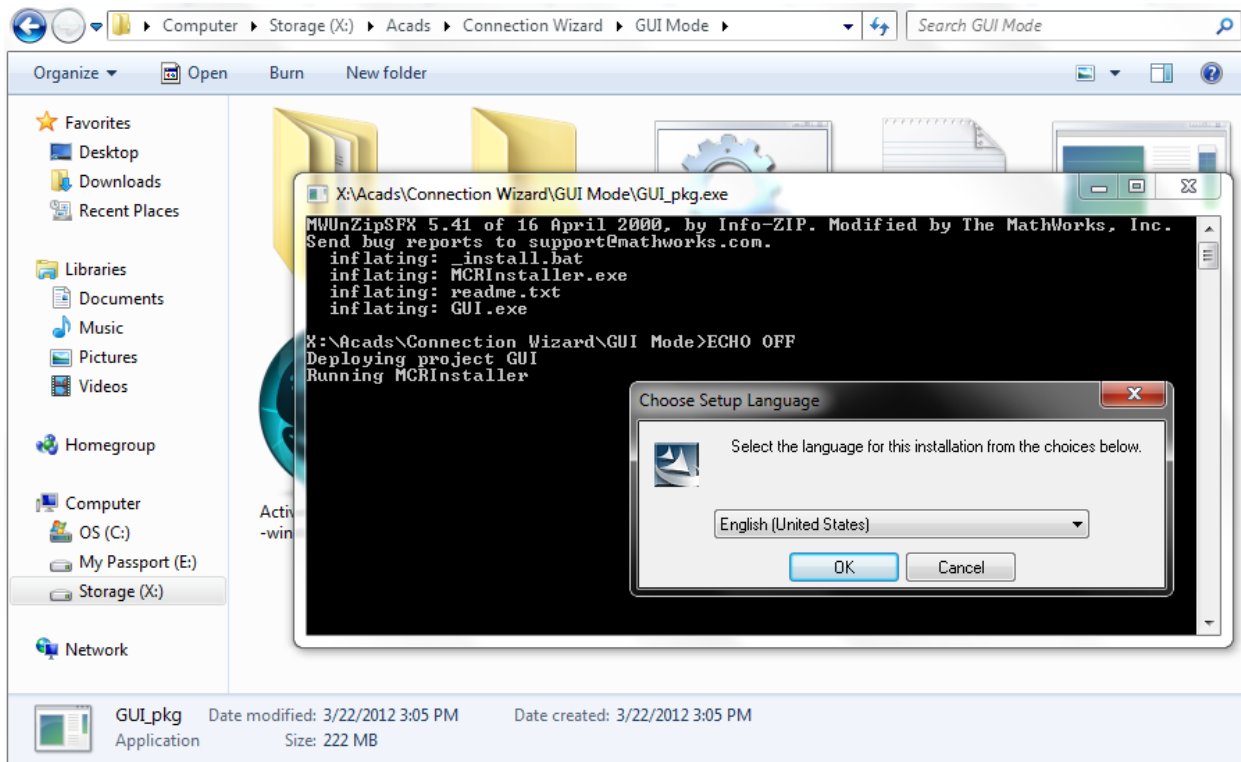


Figure C.5. MCR installation instruction described by Step 4

4. Click on 'Install' as shown in Figure C.6.

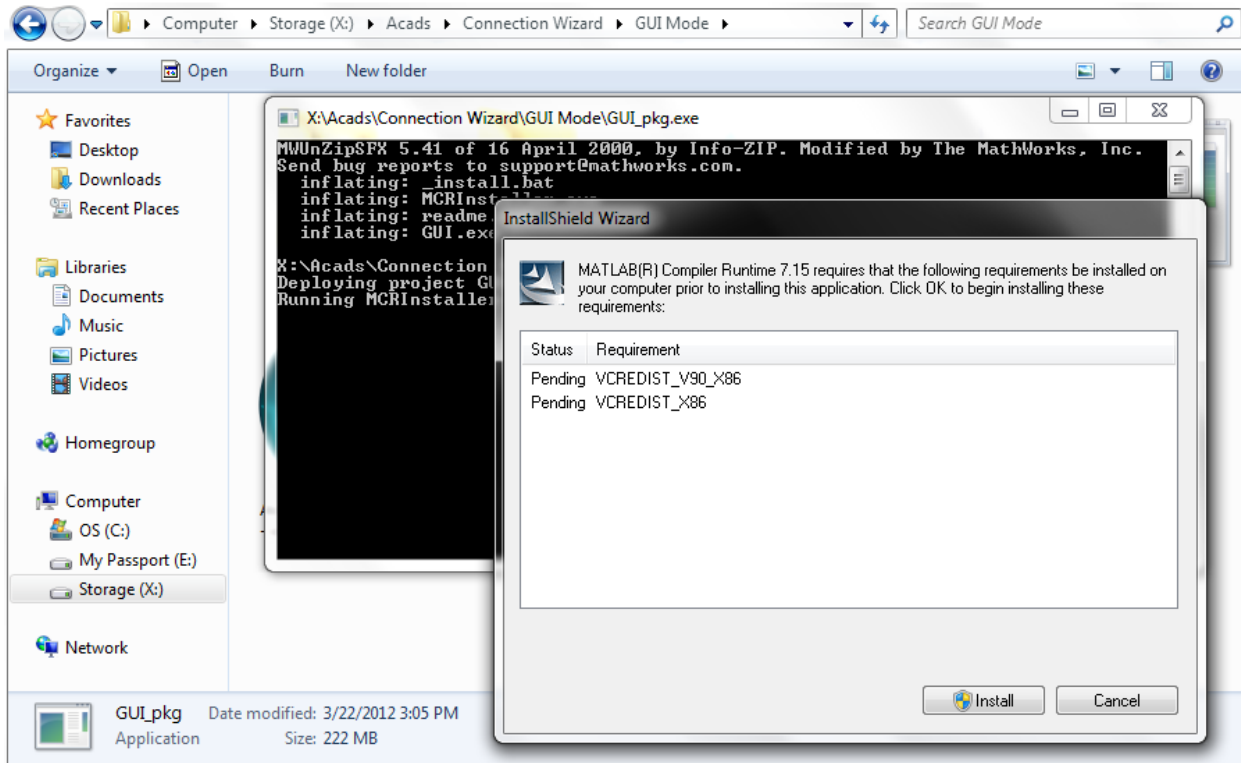


Figure C.6. MCR installation instruction described by Step 5

5. Click 'Next' as shown in Figure C.7.

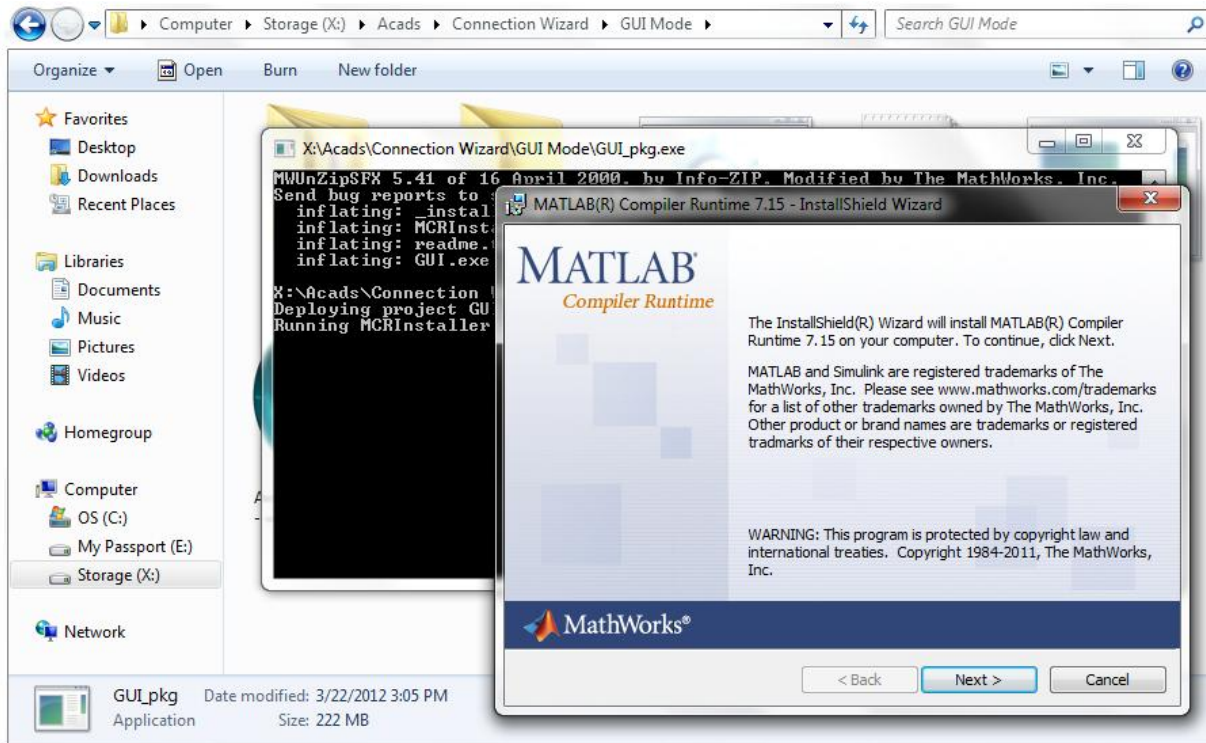


Figure C.7. MCR installation instruction described by Step 6

6. Fill in the appropriate fields; otherwise click 'Next' as shown in Figure C.8.

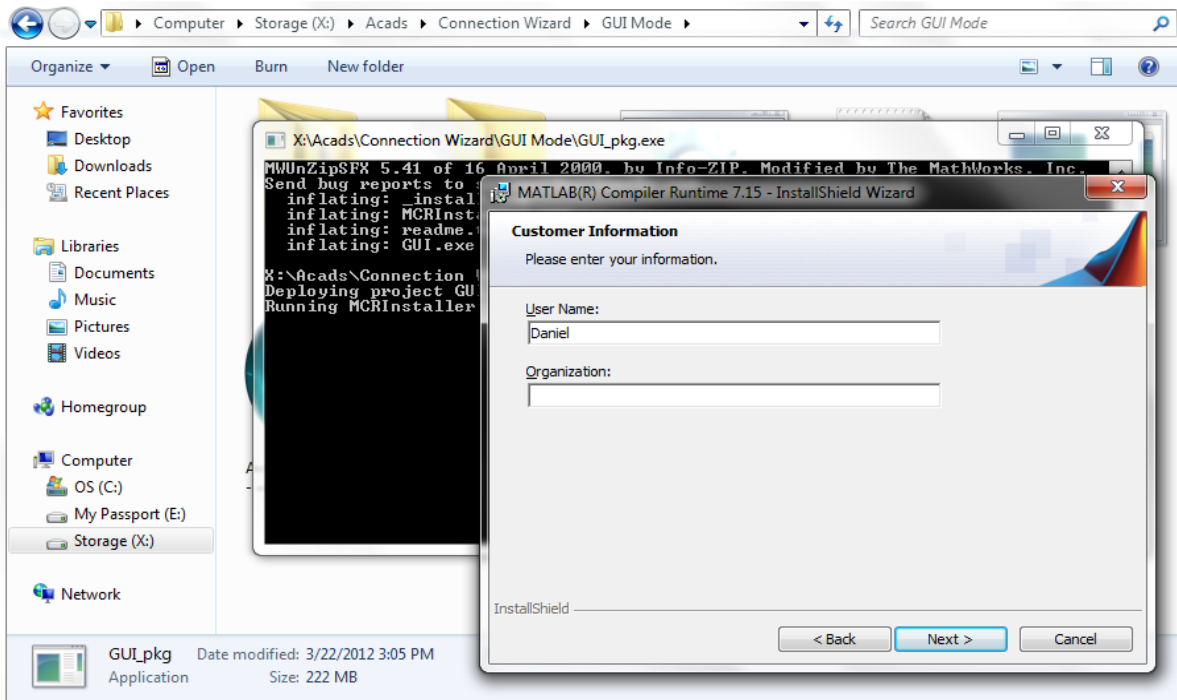


Figure C.8. MCR installation instruction described by Step 7

7. Change the destination folder by clicking 'Change' for storing installation files at any location other than default; otherwise click 'Next' and then click 'Install'. (Figure C.9, C.10 & C.11)

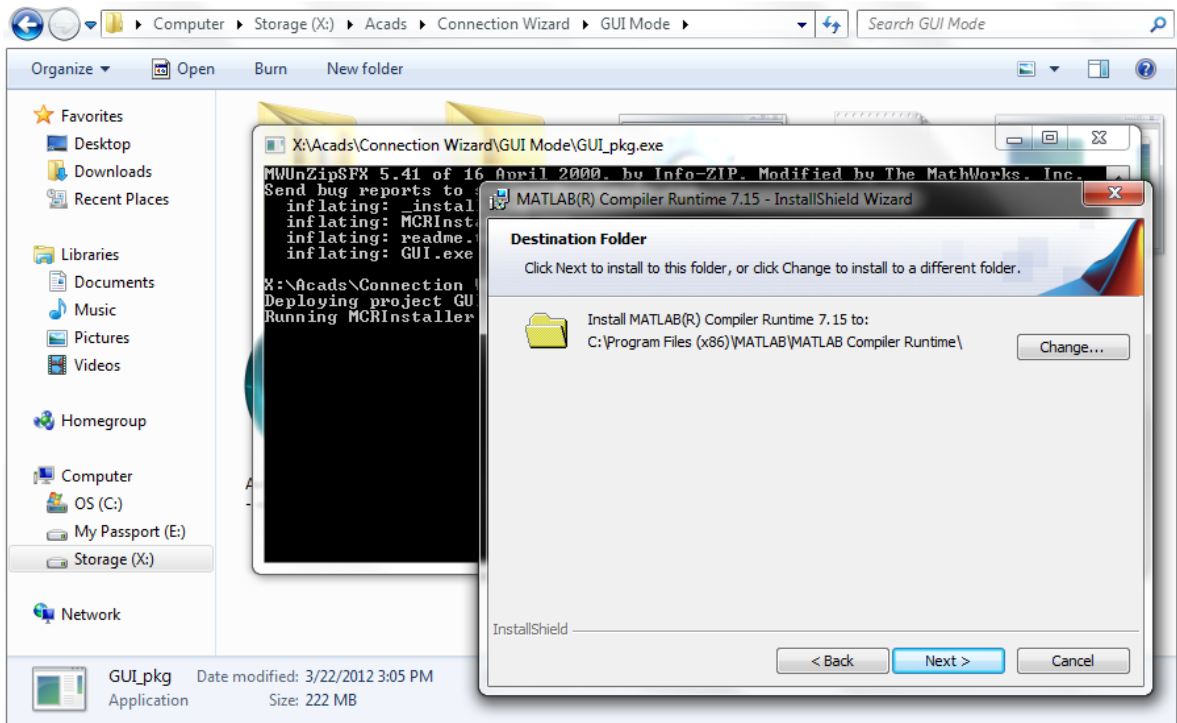


Figure C.9. MCR installation instruction described by Step 8

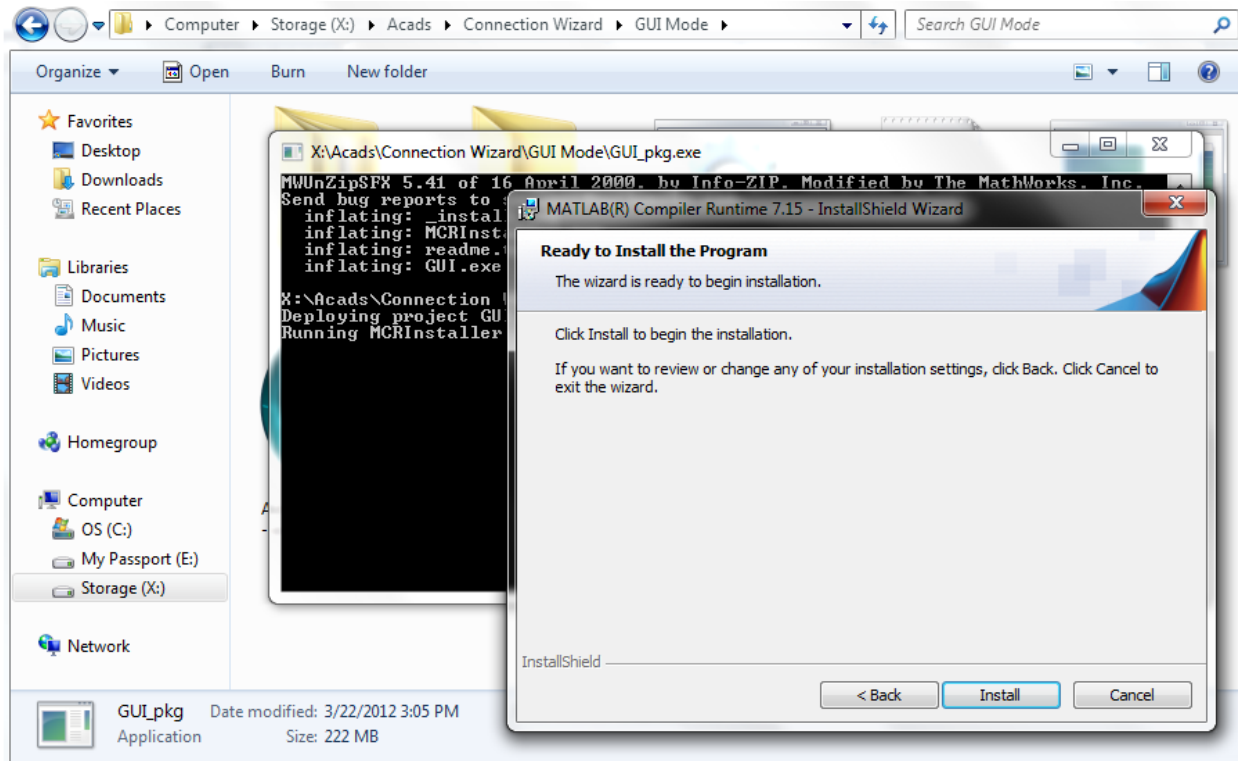


Figure C.10. MCR installation instruction described by Step 8

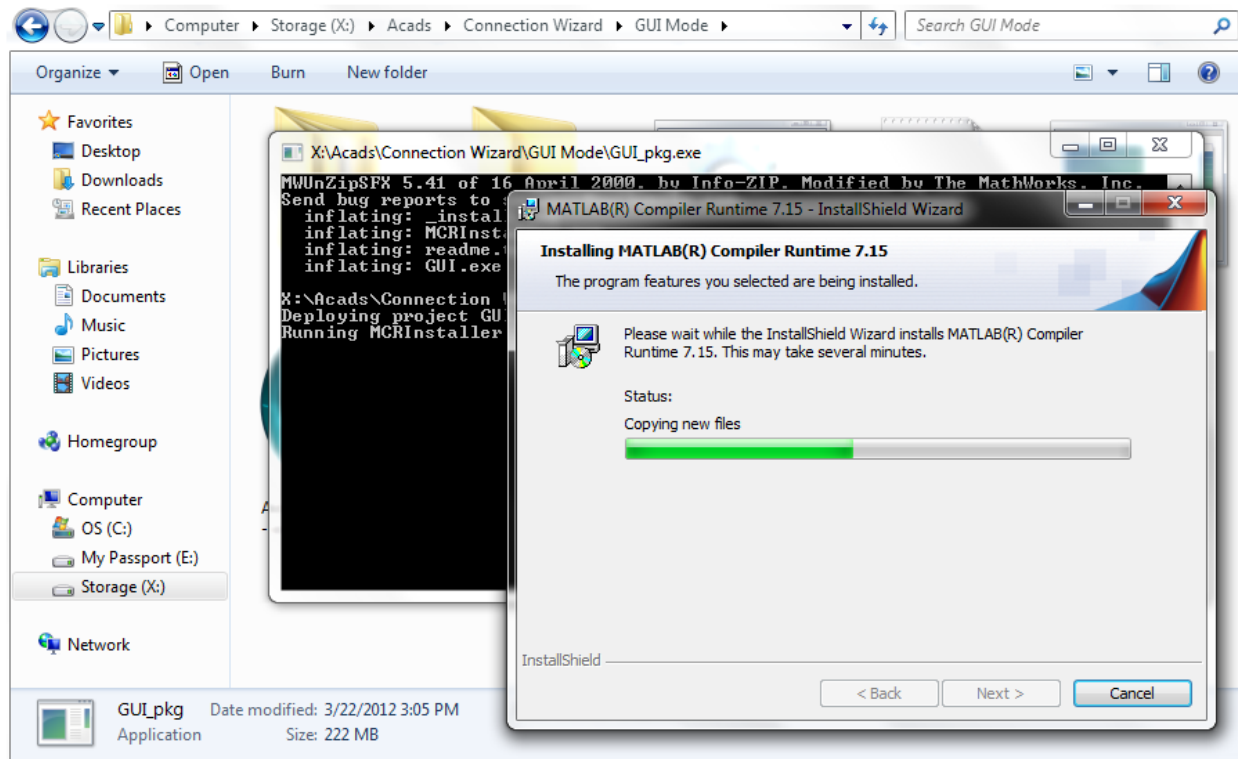


Figure C.11. MCR installation instruction described by Step 8

8. Click 'Finish' to exit the MCR installation wizard (Figure C.12).

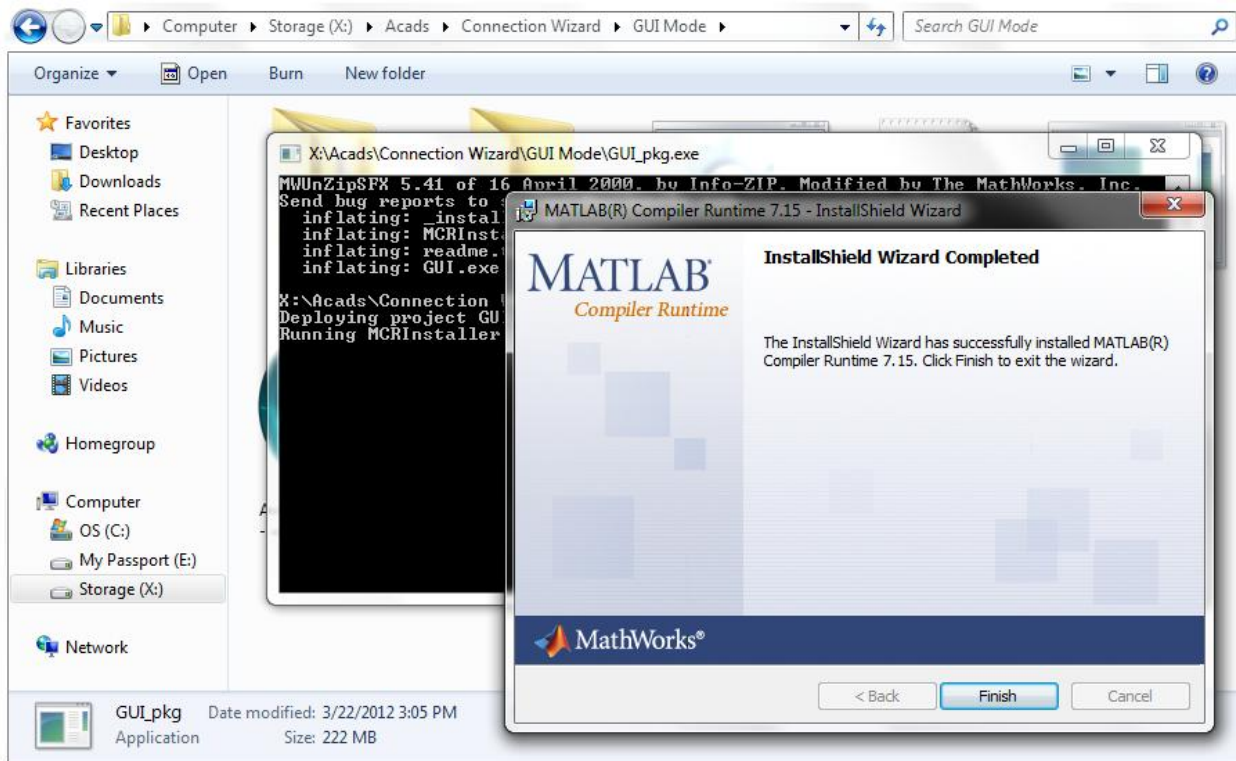


Figure C.12. MCR installation instruction described by Step 9

This wizard uses OpenSees as one of the analysis engines. OpenSees uses Tcl/Tk, a scripting language to interact with the user. Therefore, installation of Tcl/Tk is essential to run OpenSees on any system. The wizard operates on Tcl/Tk version 8.5.10.0, which is provided in the package (Figure 1.3). Below are the instructions for installing the Tcl/Tk version 8.5.10.0. It should be noted that if the version 8.5.10.0 is already installed on a system, then the user can skip step 10 through step 13. For any other installed version of Tcl/Tk, the user should first uninstall that version and install the above mentioned version by following the instructions given below.

9. Right click the 'ActiveTcl' logo present in the same folder labeled as 'GUI\_Mode' (shown in Figure C.13) and click 'Run as administrator' option. Note that if the 'Run as Administrator' option is not available, then simply double click the 'ActiveTcl' logo and proceed ahead by following the steps given below.

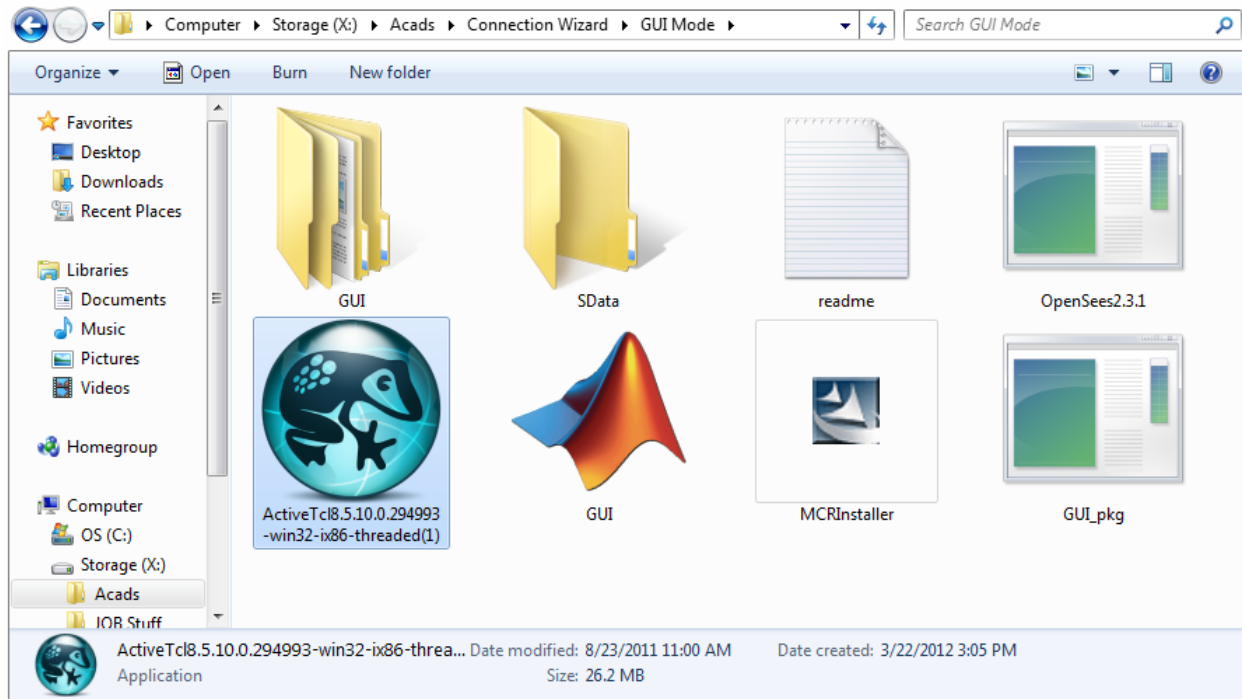


Figure C.13. Tcl/Tk installation instruction described by Step 10

10. The next two screens show some information and the license agreement, respectively (Figure C.14 and Figure C.15). Accept the license and click next.
11. Next screen specifies the installation mode and location. Here the user must change the default installation location from *C:/Tcl* to *C:/Program Files/Tcl* (there is a space between Program and Files), as shown in Figure C.16 and Figure C.17.



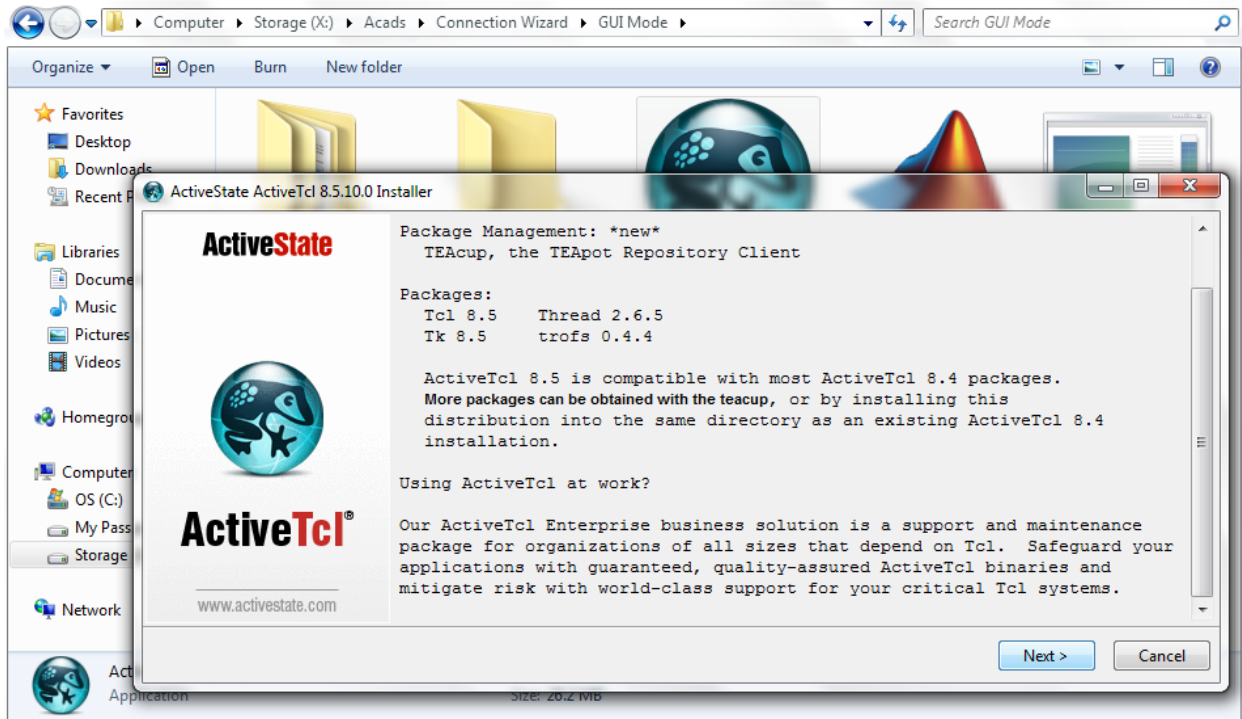


Figure C.14. Tcl/Tk installation instruction described by Step 11

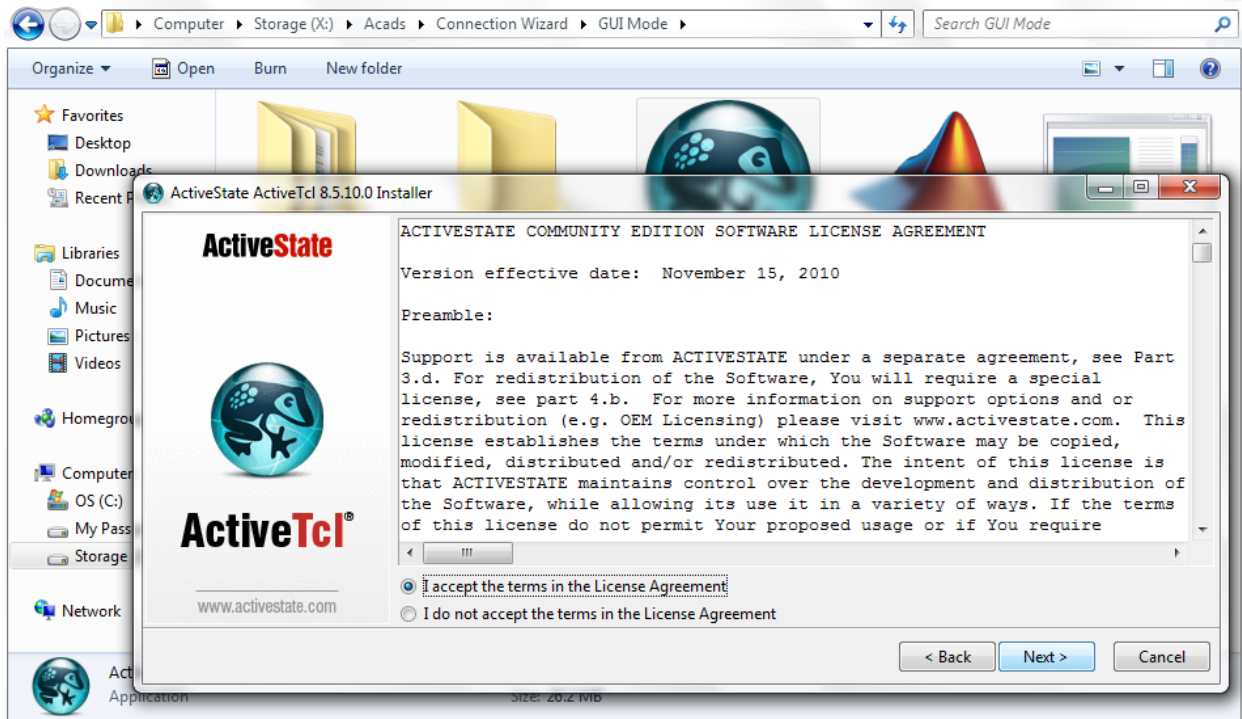


Figure C.15. Tcl/Tk installation instruction described by Step 11

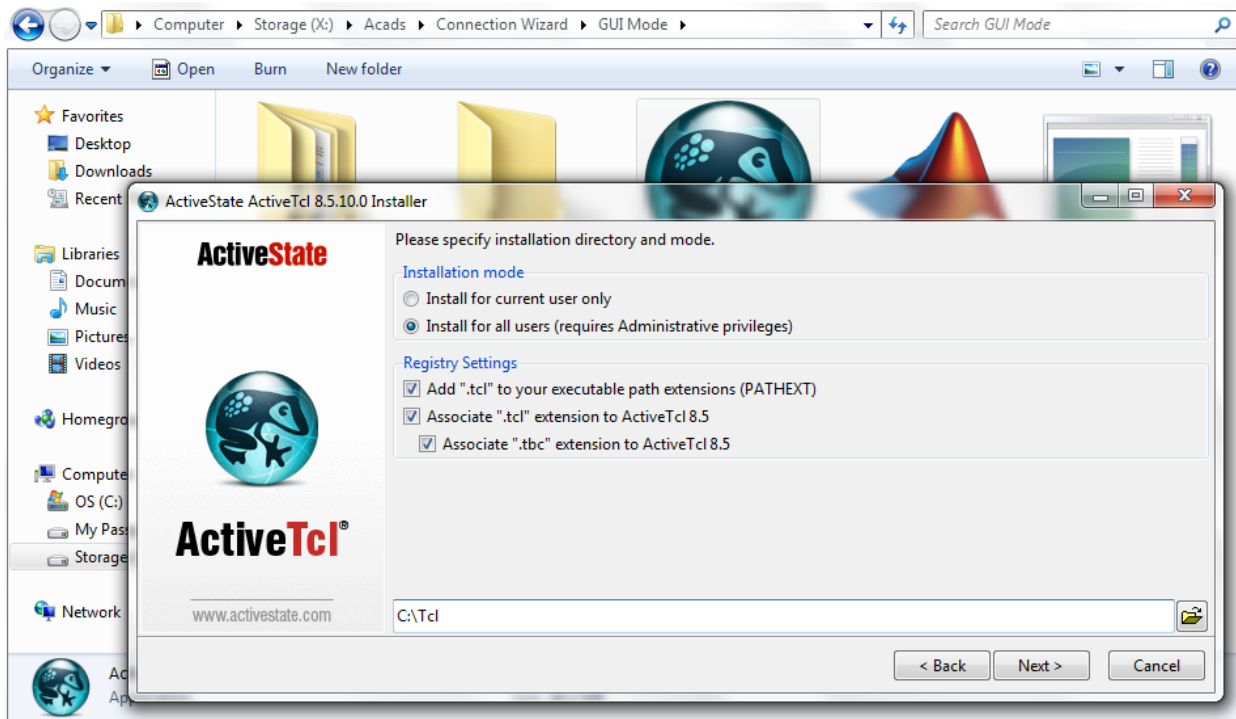


Figure C.16. Tcl/Tk installation instruction described by Step 12

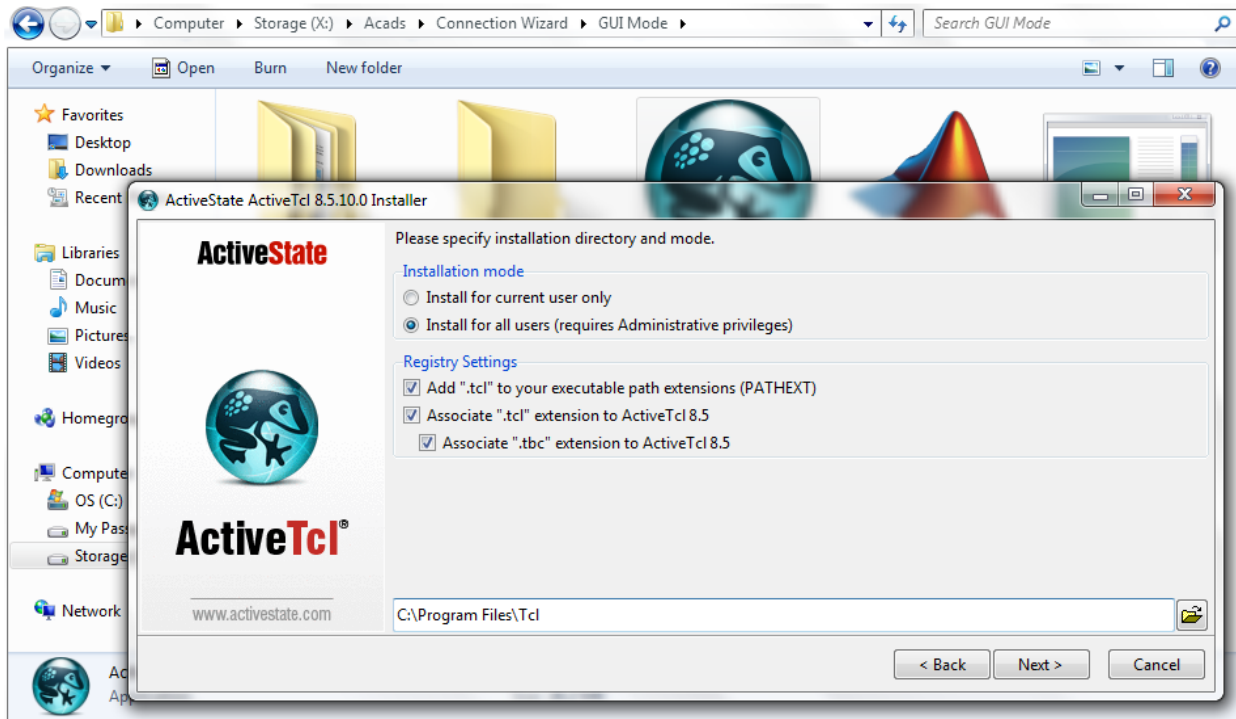


Figure C.171. Tcl/Tk installation instruction described by Step 12

12. Keep clicking 'Next' button until it finishes the installation and exit by clicking on 'Finish' button.
13. Now that when the user has already installed MCR and Tcl/Tk interpreter, he or she can click on 'GUI.exe' present in the same folder (Figure C.18) and the column base connection wizard

appears on the screen (Figure C.19). The user is advised to be patient after clicking on 'GUI.exe', since the wizard takes 1-2 minutes (depending on speed of the processor) to start up for the first time. This time is taken by the processor to load the application from hard disc to temporary memory (RAM).

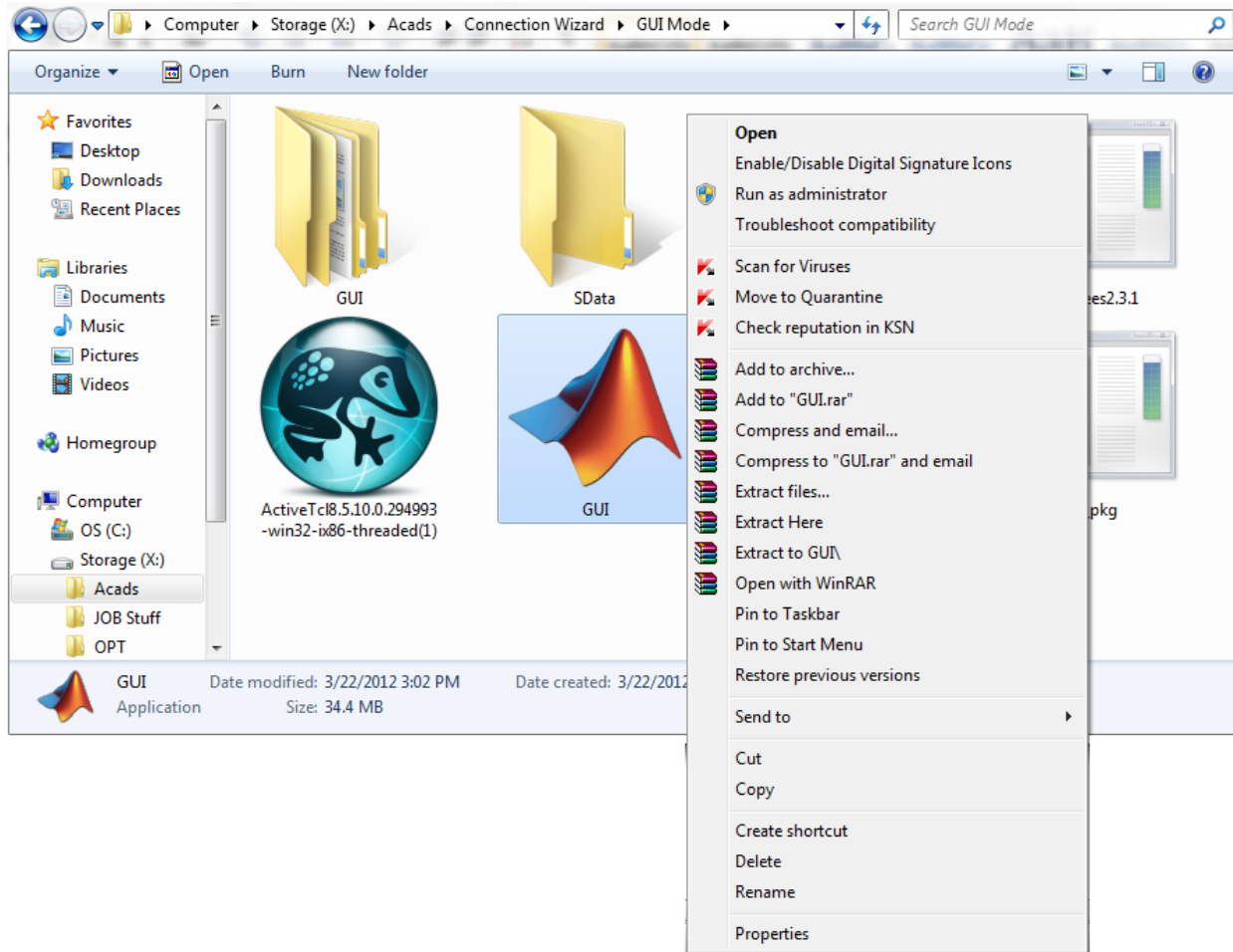


Figure C.18. Execution of 'GUI.exe' to run the wizard

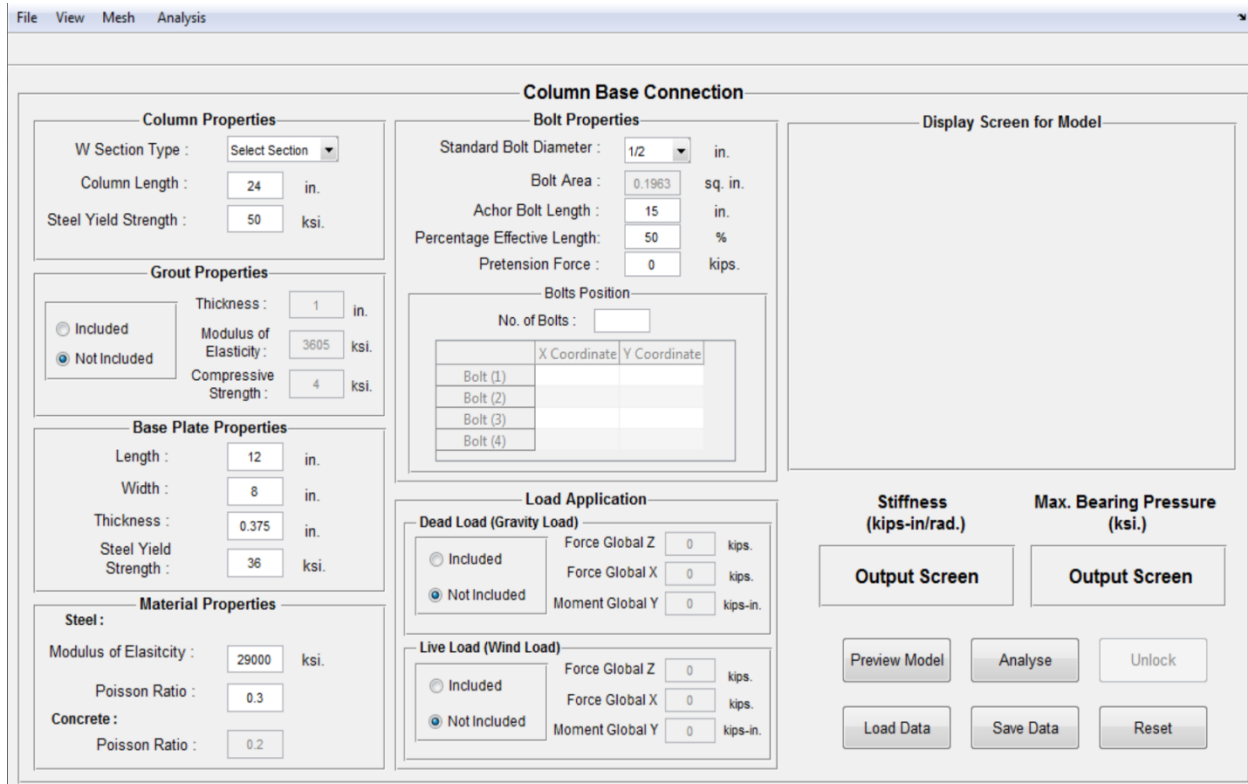


Figure C.192. Column Base Connection Wizard pops up as user clicks on GUI.exe

- At this stage the user can start entering the values for different parameters by referring to the Chapter 2 which describe the functioning of various features provided in this wizard.

### C.1.3.2 Instructions for installing the Batch Mode

- Open the folder 'Batch\_Mode' by clicking on it (see Figure C.20). The folder typically contains two folders and three files. Right click the file labeled as 'batch\_pkg' and click on 'Run as Administrator' option (if available). A user account control warning may or may not appear on the screen. If it pops up on the screen, then click 'Yes'. This whole process will expand the files compressed inside this package. Note that if the 'Run as Administrator' option is not available, then simply double click the 'batch\_pkg' file and proceed ahead by following the steps given below.

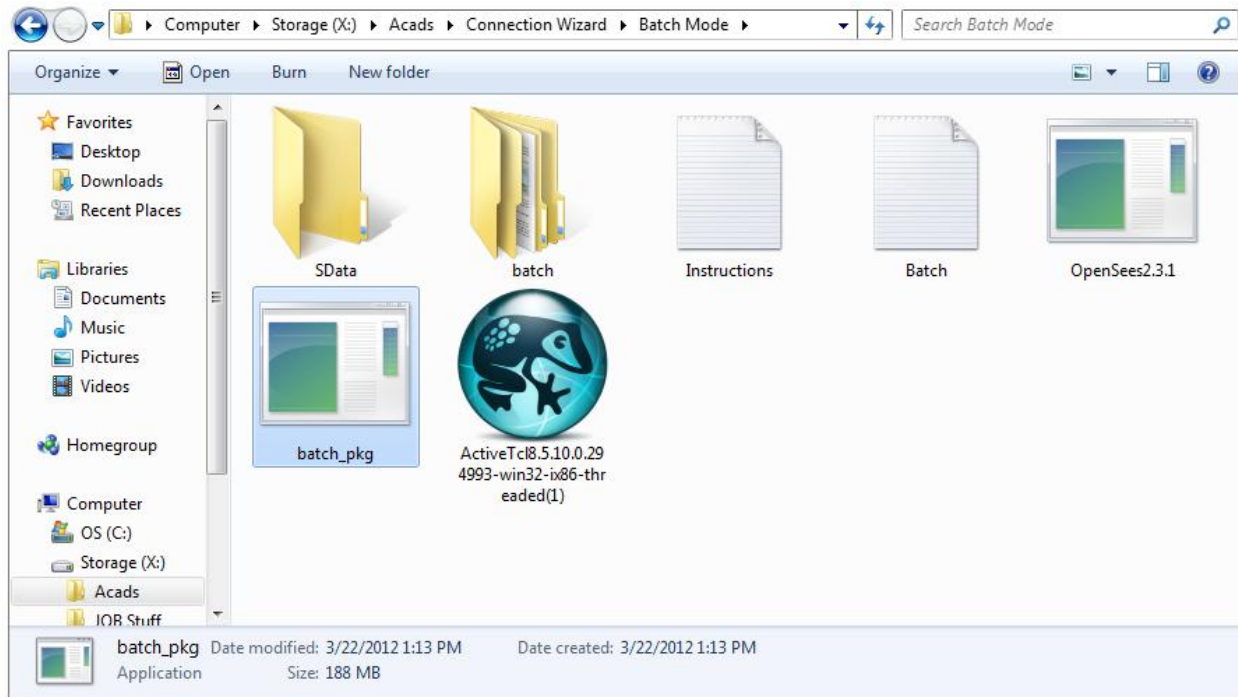


Figure C.20. Folder containing installation package for Batch Mode

2. A number of files and a black window (shown in Figure C.21) automatically appear in the same 'Batch\_Mode' folder after executing the file 'batch\_pkg' as per step 1.

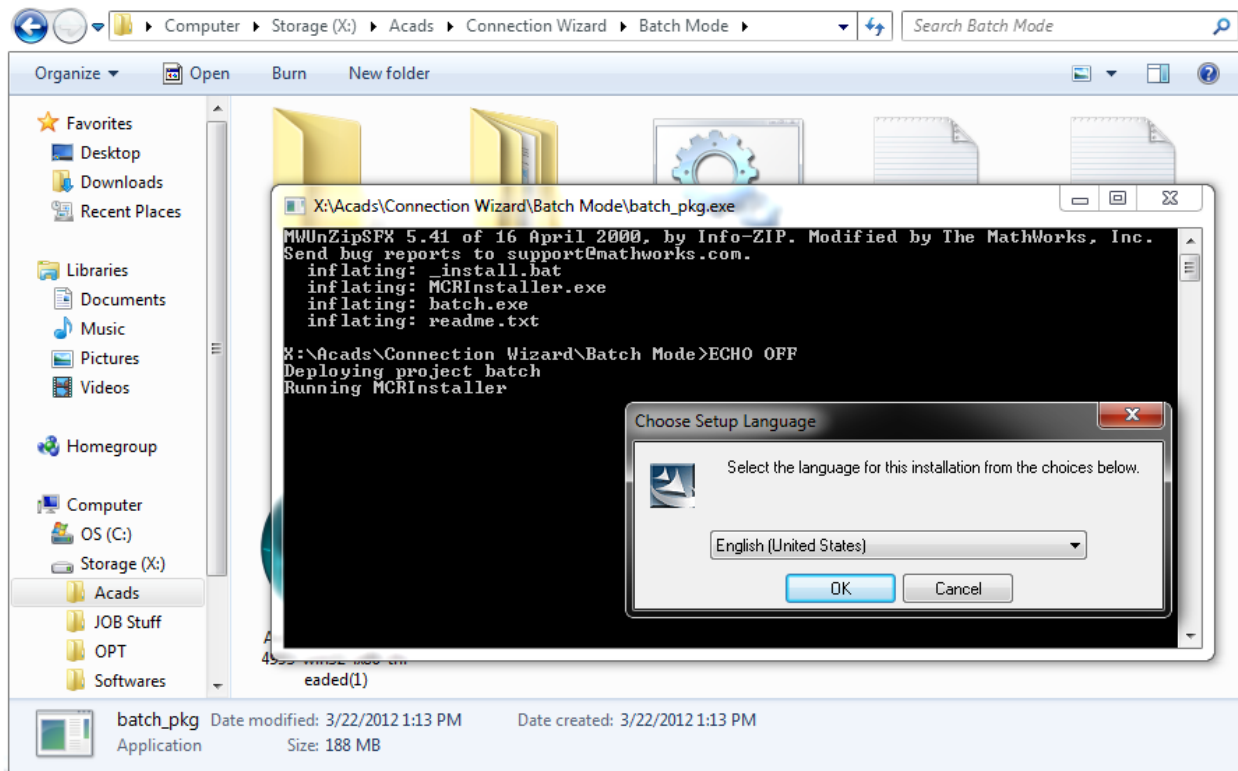


Figure C.21. MCR installation screen

3. The black window (see Figure C.21) automatically executes a wizard for installing MATLAB Compiler Runtime (MCR). Note that the MCR must be installed on any computer machine to run the column base connection wizard in a standalone format.
4. At this point if the user has already installed MCR by following Steps 4 through 9 of Section 1.3.1 then he doesn't need to proceed further and click 'Cancel' to return to the 'Batch Mode folder'. However if the MCR is not installed before, please follow Step 4 through Step 9 of Section 1.3.1 as these steps provide instructions for installing MCR on any computer machine.
5. After following the previous step instruction the window looks like Figure C.22.

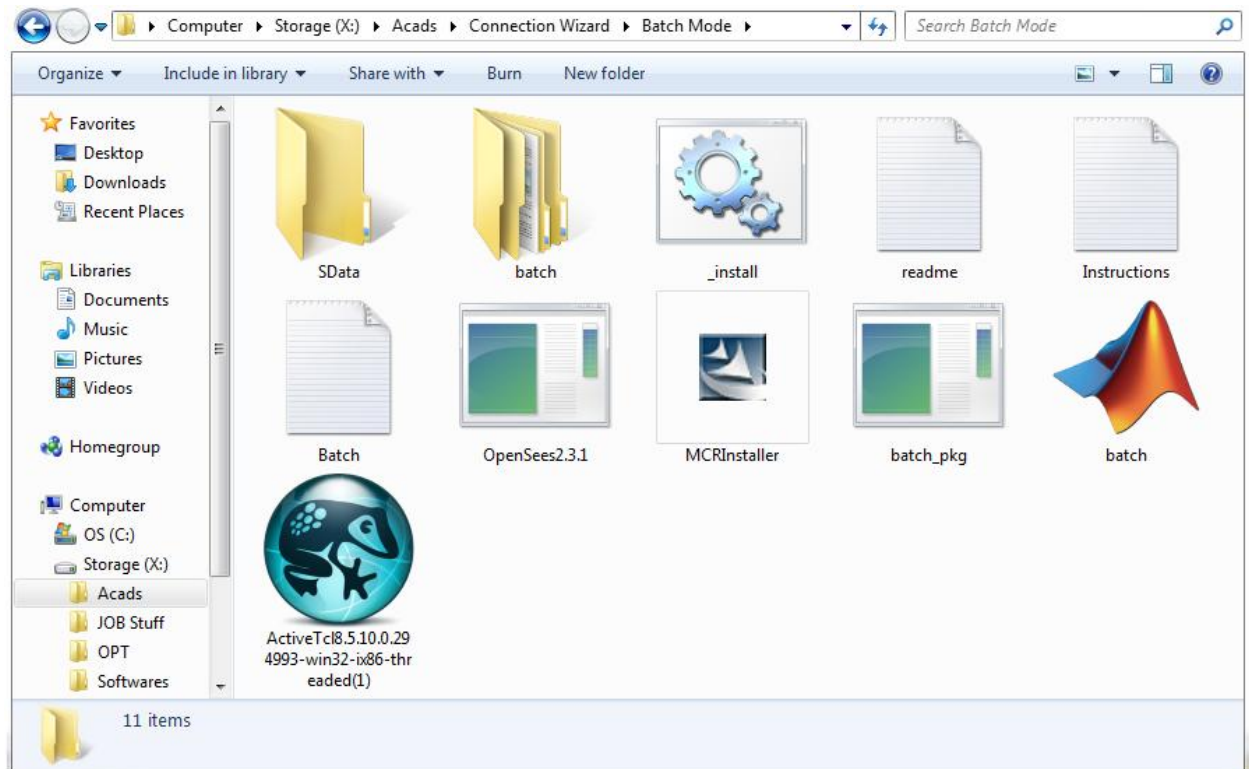


Figure C.22. 'Batch Mode' folder after the expansion of batch\_pkd

This wizard uses OpenSees as one of the analysis engines. OpenSees uses Tcl/Tk, a scripting language to interact with the user. Therefore, installation of Tcl/Tk is essential to run OpenSees on any system. The wizard operates on Tcl/Tk version 8.5.10.0, which is provided in the package (Figure C.3). The user should follow the instructions provided in Step 10 through Step 13 of Section 1.3.1 for installing the Tcl/Tk version 8.5.10.0. It should be noted that if the version 8.5.10.0 is already installed on a system, then the user can skip step 10 through step 13. For any other installed version of Tcl/Tk, the user should first uninstall that version and install the above mentioned version.

6. At this stage the user can start entering the values of all the parameters of the connection in input file 'Batch.txt'. Please see Chapter 3 for understanding the functioning of the Wizard in Batch Mode.

### **C.1.4 Instructions for Uninstalling the Wizard from Windows 7**

To uninstall the wizard from the computer, the user must go to 'Control Panel' and click on 'Uninstall a program' present under 'Programs' logo. Find 'MATLAB(R) Compiler Runtime 7.15' from the list of installed programs; select it and click uninstall. Follow the screen instructions until it finishes the uninstallation process. The user must also find 'ActiveState ActiveTcl 8.5.10' program from the above mentioned list and unistall it by following screen instructions. Finally the user can delete the 'Connection Wizard' folder, if he wants to remove the wizard completely from the computer.

## C.2.1 Functioning of the GUI Mode

This section discusses the functioning of various features provided in this wizard. The wizard only models wide flange or tapered column section (made out of wide flange sections). The material provided for the column sections is steel. Figure C.23 shows the orientation of the connection with respect to the global axes X,Y and Z defined for the connection. The 'O' represents the origin of this coordinate system, which is defined at the front left corner of the base plate. In this wizard the web of the column always orients parallel to X-Z plane and the centroid of the column always coincides with the centroid of the base plate. The external loads which can be applied to the connection in the GUI are shown by P,V and M in the Figure C.23, where P is an axial force applied in global Z direction, V is a shear force applied in global X direction and M is a bending moment applied about global Y direction. The next few sections discuss the functioning of each field provided in the GUI. Note that this wizard does not take account of the contribution of the footing in calculation of the rotational stiffness, hence no footing properties can be entered in the wizard.

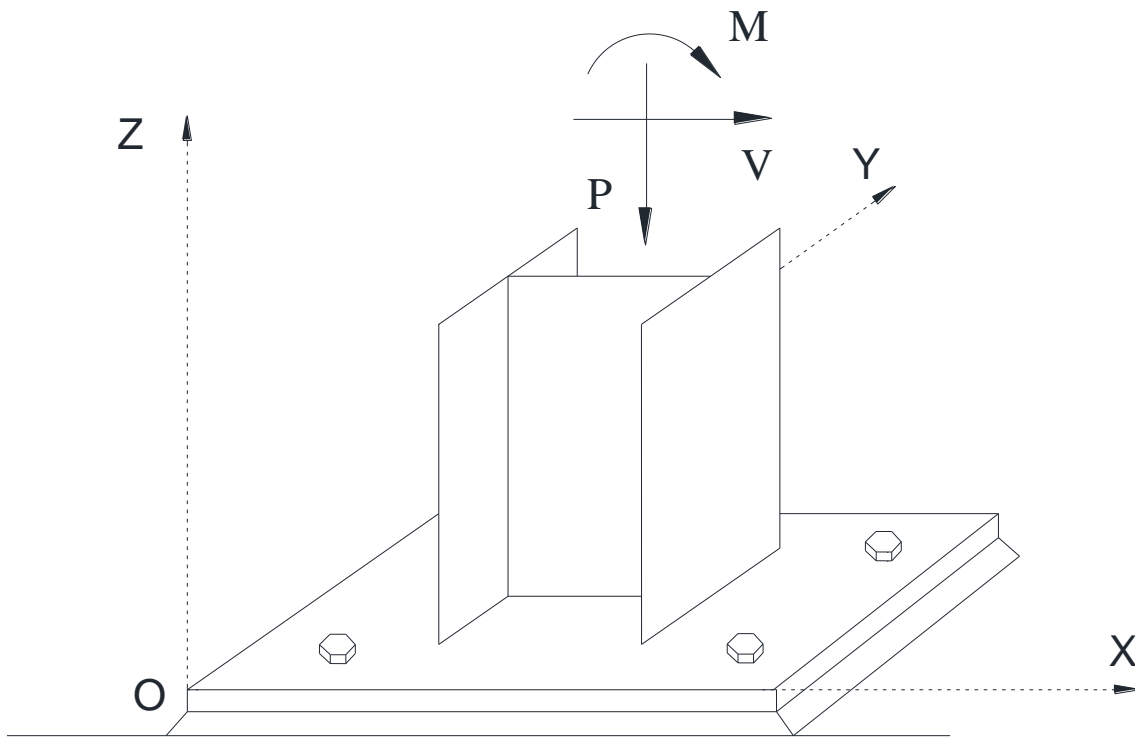


Figure C.23. Orientation of the base plate and column section in the wizard



## C.2.2 Major Components in GUI

The main graphical user interface (GUI) for this wizard is shown in Figure C.24. It is divided into four major components namely input panels, action buttons, menu bar and display screen (Figure C.25). The user can interact with first three components but the fourth component is just a visual interface to display the modeled connection. It should be noted that in addition to above mentioned components, there are few elements which are not displayed on the main figure or GUI. These elements are displayed in new figures over which the user has no control. They are just for displaying output plots, namely moment-rotation plot and bearing pressure plot. A comprehensive discussion of various features provided in this wizard is presented in the following sections.

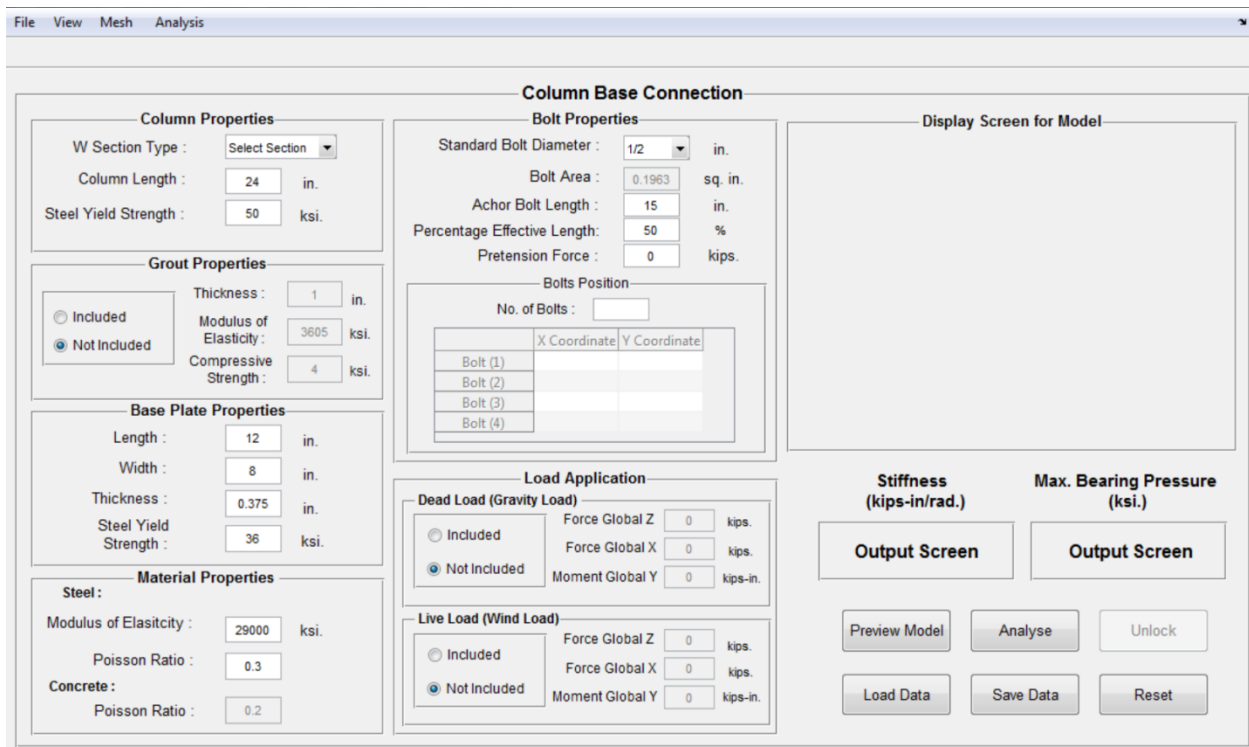


Figure C.24. Main GUI window for Column-Base Connection Wizard

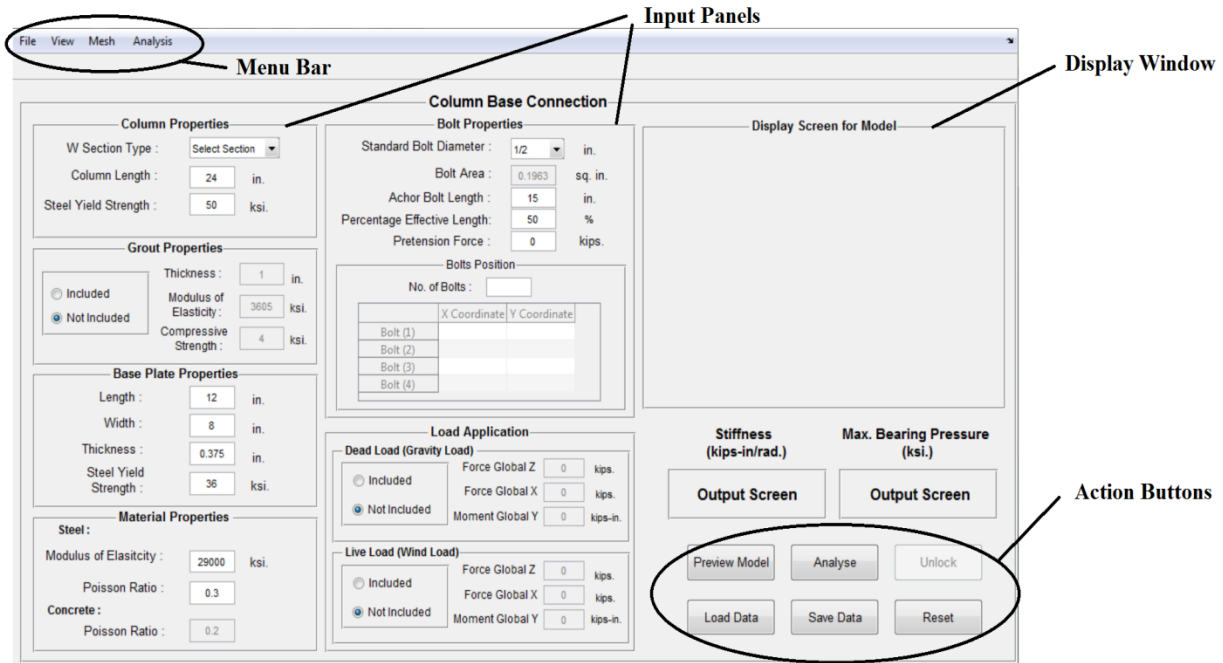


Figure C.25. Components of main GUI

### C.2.2.1 Input Panels

Each panel is a small section in the wizard which describes materialistic and geometrical properties of a specific component of the connection. Some of the panels in this wizard are 'Column Properties', 'Grout Properties', 'Load Application', etc. All the panels have input fields for entering values of various modeling parameters. Note that values in all the input fields should be greater than zero.

#### C.2.2.1.1 Column Properties

The 'Column Properties' input panel contains the parameters which are used to model a column. The specific parameters that are provided in this panel are 'W Section Type', 'Column Length' and 'Steel Yield Strength'. Each parameter and its functioning are discussed very briefly below. It should be noted that this wizard can only model steel W sections and steel tapered sections; it cannot be used for creating any other types of sections such as HSS sections, S sections, etc.

##### C.2.2.1.1.1 W Section Type

This is a list box which contains a list of standard W sections provided by the AISC Steel Construction Manual. When a W section is selected from the list, all the geometrical properties of that section are automatically made available to the wizard and the user doesn't have to enter these cross section properties explicitly. For instance, W8x35 is chosen from the list which is shown in Figure C.26.

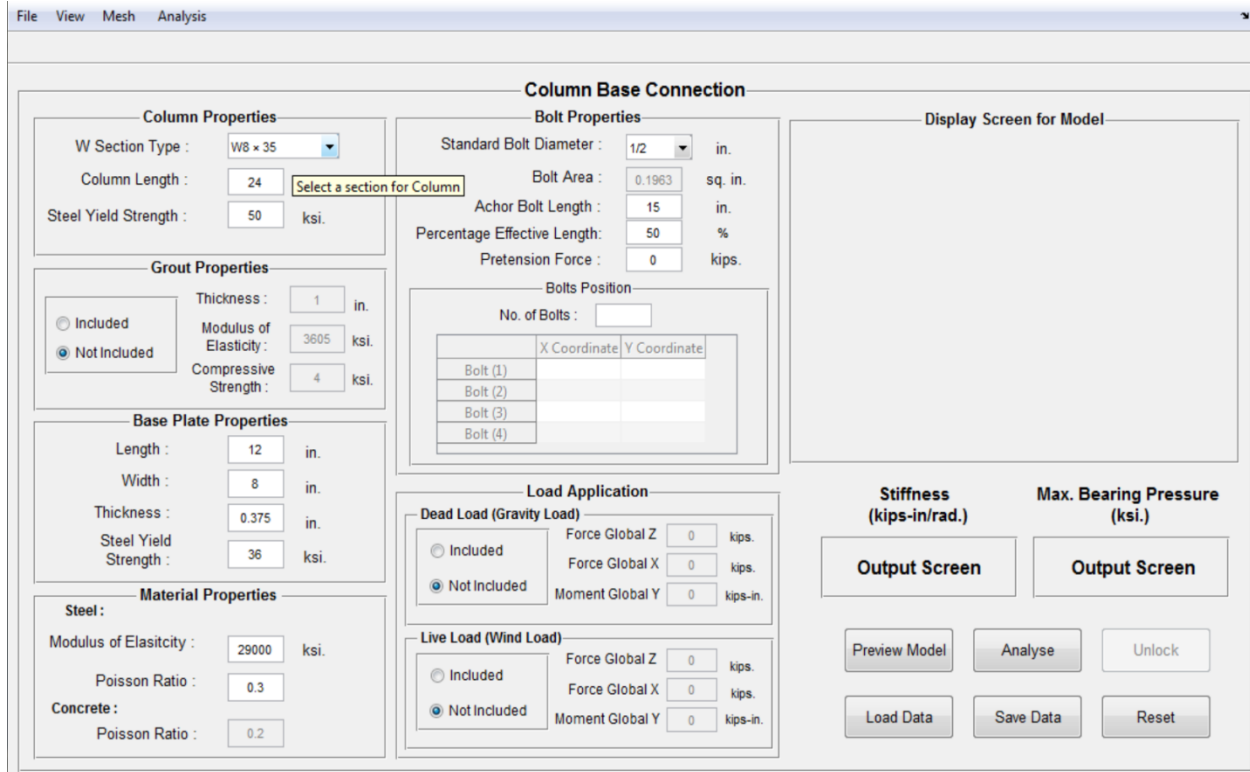


Figure C.26. GUI window showing selection of W8x35 section from the list of W sections

The user can also create a tapered column by defining desired cross sections at two ends of the column. This can be done by choosing 'Tapered Section' option provided in the W section list box (Figure C.27). On choosing this 'Tapered Section' option, a new GUI window labeled as 'Tapered Section' appears as shown in Figure C.28. This GUI asks the user to enter the cross section dimensions at the two ends (top and bottom) of the column. One can easily create a tapered column, which is very common in frame structures.

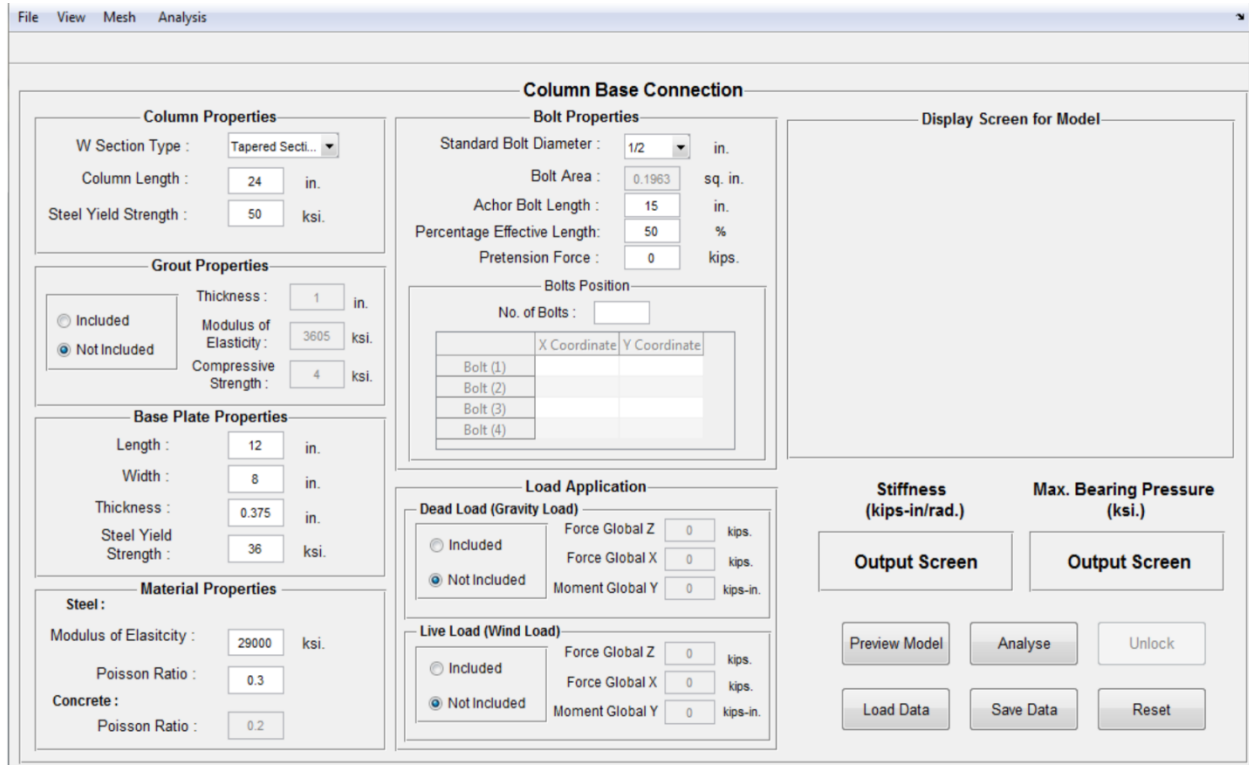


Figure C.27. GUI showing the option provided in W section type list box for creating a Tapered Section

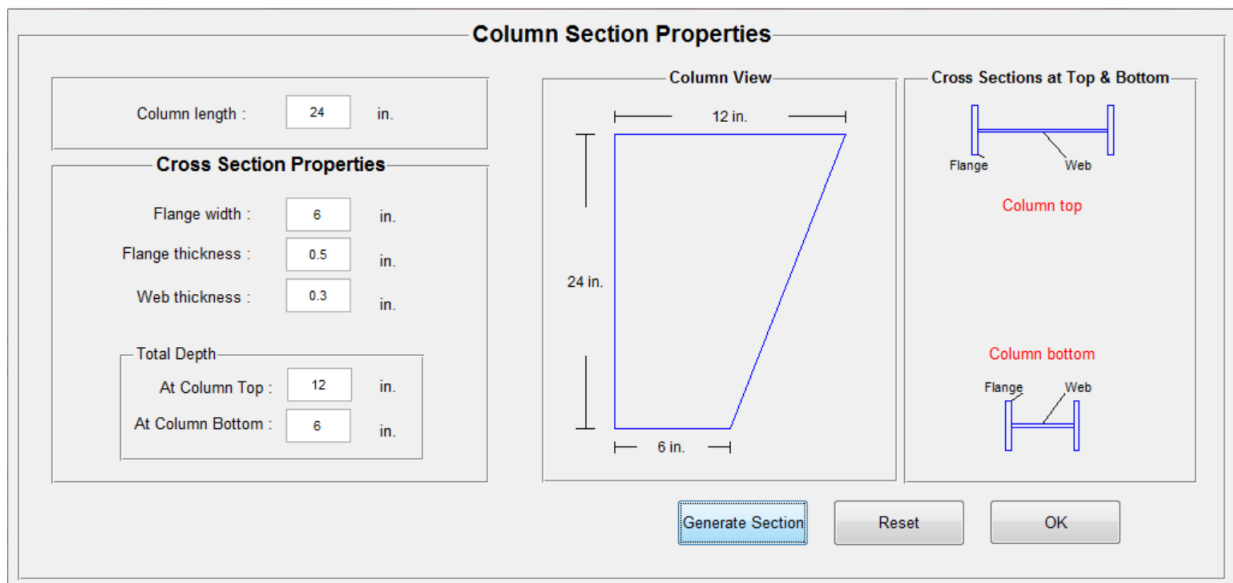


Figure C.28. GUI window for creating a Tapered Section

This 'Tapered Section' GUI window provides the following text fields to the user for entering desired dimensions of cross sections at two ends of the column.

1. Column Length: It takes a numeric value for length of the column stub used.
2. Flange Width: It takes a numeric value for width of the column flange.
3. Flange Thickness: It takes a numeric value for thickness of the column flange.
4. Web Thickness: It takes a numeric value for thickness of the column web.
5. Total Depth: For creating a tapered section, total depth is usually varied linearly with length of the column. Hence, the depth is entered at the top and bottom of the column.

There are three action buttons (refer Section C.2.2.2 for more explanation on action buttons) also provided on this GUI window. Each action button performs a specific task on clicking, which is discussed below.

1. Generate Section: This button when clicked creates a section by using dimensions entered by the user and displays it on the two axes labeled as 'Column View' and 'Cross Sections at Top & Bottom'. This button also performs an error check to verify that dimensions entered are reasonable.
2. Reset: This button allows the user to clear all the input fields in the GUI window. The user can re-enter all the values as desired.
3. Ok: This button closes the 'Tapered Section' GUI and takes the user back to the main GUI, if all the input fields are filled up with appropriate values.

#### **C.2.2.1.1.2 Column Length**

This input field takes a numeric value of column stub length. If the user does not input any value, then software takes 24 in. as a default value of column stub length. If the user created his own section by choosing 'Tapered Section' option from the list box of W section type, then he does not need to enter the column length again in the main GUI as he already entered that value in 'Tapered Section' GUI. Whatever the value user enters in the Tapered Section GUI, it is automatically updated in the main GUI.

#### **C.2.2.1.1.3 Steel Yield Strength**

This input field allows the user to enter yield strength of the steel used for the column section. This value is used in the analysis. Default value for yield strength of steel is provided as 50 ksi. This value can be changed as per user's requirements.

#### **C.2.2.1.2 Grout Properties**

The grout is provided underneath the base plate to make transfer of the loads from the base plate to the footing smoother and more effective. The important parameters which affect this transfer are thickness of the grout, modulus of elasticity and the compressive strength of the concrete (used for constructing grout), which can be entered in 'Grout Properties' input panel.

#### **C.2.2.1.2.1 Thickness**

This field takes a numeric value for thickness of the grout used under the base plate. A default value of 1 in. is provided in the wizard, which can be changed as per requirements.

#### **C.2.2.1.2.2 Modulus of Elasticity**

This field allows the user to input a positive numeric value of modulus of elasticity of concrete used for the grout. A default value of 3615 ksi. is provided in the wizard, which can be changed as per requirements. Note that the concrete used for the grout does not necessarily follow the equation shown below because the concrete used for constructing grout contains fine aggregates in majority and the following equation does not specifically capture the effect of fine aggregates on the modulus of elasticity; hence the user has to input both the modulus of elasticity and compressive strength of the concrete separately.

$$E_G = 57000 * \sqrt{f_c'} \quad (C-1)$$

where  $f_c'$  is compressive strength of the concrete which is used for the grout  
 $E_G$  is modulus of elasticity of the concrete which is used for the grout

#### **C.2.2.1.2.3 Compressive Strength**

This field takes a numerical value for compressive strength of concrete used for the grout. A default value of 4 ksi is provided. The user can modify this value as desired.

### **C.2.2.1.3 Base Plate Properties**

The 'Base Plate Properties' input panel provides various input fields which help in modeling the base plate. The user can enter an appropriate value in each of the given fields. The fields provided in this panel are following.

#### **C.2.2.1.3.1 Length**

The user should enter a numeric value for length of the base plate in this input field. Default value provided in this field is 12 in.

#### **C.2.2.1.3.2 Width**

This input field allows the user to enter a desired value for width of the base plate. Default value provided in this field is 8 in.

#### **C.2.2.1.3.3 Thickness**

This input field allows the user to enter a desired value for thickness of the base plate. Default value provided in this field is 0.375 in.

#### **C.2.2.1.3.4 Steel Yield Strength**

This input field allows the user to enter yield strength of the steel used for the base plate. Default value for yield strength of steel is provided as 36 ksi. This value can be changed as per user's requirements.

#### **C.2.2.1.4 Material Properties**

The 'Material Properties' panel allows the user to enter the values for modulus of elasticity and poisson ratio of the steel and concrete used for modeling the components of the connection such as column, base plate, anchor bolts and the grout.

##### **C.2.2.1.4.1 Steel: Modulus of Elasticity**

This input field allows user to enter any value of modulus of elasticity for the steel used in the connection. A default value of 29000 ksi is provided, which can be changed as per user's requirements. Note that the quantity entered should be positive.

##### **C.2.2.1.4.2 Steel: Poisson Ratio**

The user can enter a numeric value for poisson ratio of the steel used. A default value of 0.3 is provided, which is most common for steel material. Note that the value entered must satisfy the following inequality.

$$0 < \mu_s < \frac{1}{2} \quad (C-2)$$

where  $\mu_s$  is poisson ratio of steel

##### **C.2.2.1.4.3 Concrete: Poisson Ratio**

The user can enter a numeric value for poisson ratio of the concrete used for constructing grout pad. A default value of 0.2 is provided, which is most common for concrete material. Note that the value entered must satisfy the following inequality.

$$0 < \mu_c < \frac{1}{2} \quad (C-3)$$

where  $\mu_c$  is poisson ratio of concrete

#### **C.2.2.1.5 Bolt Properties**

The 'Bolt Properties' input panel provides various input fields which help in modeling of the anchor bolts. The user can enter an appropriate value in each of the given fields. The fields provided in this panel are following.

### C.2.2.1.5.1 Standard Bolt Diameter

This field provides the user with a list box containing a list of standard bolt diameters given in AISC steel construction manual. When user selects a diameter for the bolt from this list, it automatically displays the corresponding gross area of that bolt in 'Bolt Area' field.

### C.2.2.1.5.2 Bolt Area

This input field remains disabled and shows the bolt area corresponding to a chosen diameter in standard bolt diameter list box. When user selects 'Other' option from the list box of standard bolt diameter, it gets enabled and allows the user to design his own bolt by entering a desired value for the bolt area.

### C.2.2.1.5.3 Anchor bolt length

This input field allows the user to enter the anchor rod length. Anchor rods are used to anchor the base plate to the foundation. A default value of 15 in is provided, which can be modified by the user.

### C.2.2.1.5.4 Percentage Effective length

This input field provides the user the ability to use an effective length of the anchor rod for the analysis, by entering the percentage of whole length which is effective. The effective length of an anchor rod depends on the variation of bond stress (between the concrete and the steel) with length of the rod. A default value of 50% is provided, which depicts that the stress distribution is linearly varying with length of the rod and effective length of the anchor rod becomes half of the anchor bolt length provided (Figure C.29). The following equation shows the relation between anchor rod length and effective anchor rod length.

$$L_{eff} = (eff_{per}) * L \quad (C-4)$$

where  $L_{eff}$  is effective embedded length of an anchor rod

$L$  is embedded length of an anchor rod

$eff_{per}$  is percentage of an anchor rod length which is effective.

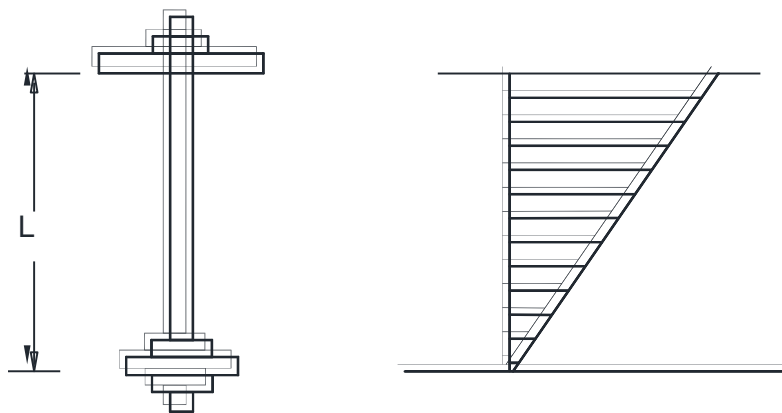


Figure C.29. Linear variation of bond stress between the concrete and the steel



### C.2.2.1.5.5 Pretension Force

The wizard provides the facility to pretension the anchor bolts. The user can specify a value of pretension force in the text field and this value will be used as a pretension force in each bolt.

The default value provided is 0 kip. Note that the pretension force delays the uplifting of the base plate under lateral loads. Hence, the connection does not rotate as much as it does without pretension force.

### C.2.2.1.5.6 Bolt Position

The Bolt position subpanel provides input fields for entering the number of bolts and the positions of bolts in terms of coordinates. The minimum number of bolts for the wizard is one and there is no upper limit on the maximum number of bolts that can be provided in the wizard. The user can define the position of each anchor bolt on the base plate by providing coordinates in a pair of  $(X,Y)$ , where  $X$  and  $Y$  are the distance in inches from the Origin 'O', which is shown in Figure C.30. For the four bolts configuration, the coordinate pairs  $(X_1, Y_1)$ ,  $(X_2, Y_2)$ ,  $(X_3, Y_3)$ ,  $(X_4, Y_4)$  are  $X$  and  $Y$  coordinates of Bolts 1, 2, 3 and 4, respectively. It should be noted that there is no particular sequence in which the bolts positions have to be entered; the user can provide the positions in any order.

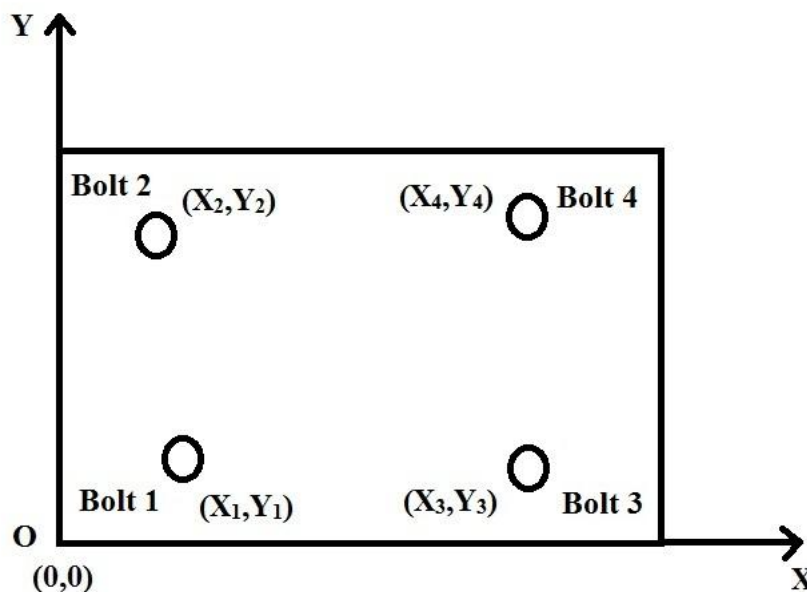


Figure C.30. Position of Bolts on the Base Plate

### C.2.2.1.6 Load Application

The 'Load Application' panel is provided for applying loads on the connection. The wizard takes unfactored loads as load inputs. When the load values are provided in this panel, it should be kept in mind that these loads are acting on the centroid (point 'A') of top cross section of the column having length 'L', as shown in Figure C.31. Hence, apply the resulting concentrated loads (axial force, shear force and moment) coming at top of the column. Also note that the wizard does not take account of the self-weight of the connection and the user is advised to calculate the

self-weight of the connection and apply it as a nodal load (a force in Global Z and a Moment in Y, if the self-weight is coming eccentrically on the connection) in the 'Dead Load' subpanel.

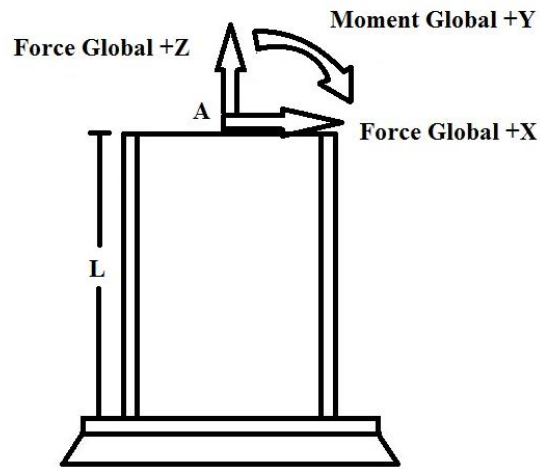


Figure C.31. The directions of loads applied at point A of the Connection

#### C.2.2.1.6.1 Dead Load (Gravity Load)

This panel provides three input fields to apply loads on the connection. Following three types of dead loads can be given.

- 1) Force Global Z - this represents the force applied in global Z direction (as shown in Figure C.31). It usually represents the gravity loads.
- 2) Force Global X - this represents the force applied in global X direction. It usually includes lateral or shear loads present in static condition of the connection.
- 3) Moment Global Y- this represents the moment applied in global Y direction. It usually includes moments generated due to any eccentric loads applied on the connection.

This input panel should also include the self-weight of the structural members which is supported by the connection and the self-weight of the connection (i.e. the weight of the column and base plate). These loads are time independent and act constantly throughout the service life of the structure. Hence, the connection develops some internal forces by resisting these loads and the resulting stiffness is treated as initial stiffness for the analysis performed under live loads or wind loads. Please note that 'dead load' in this manual means 'gravity load' and vice versa, hence the user should not confuse between the two terms.

#### C.2.2.1.6.2 Live load (Wind Load)

This panel provides the same input fields as provided by the 'Dead Load (Gravity Load)' panel. Live loads are the loads which act on the structure variably during its lifetime. Here variably means that these loads are time dependent and vary in magnitude at different instants. For this wizard the live loads are same as the wind loads. The main difference between this panel and 'Dead Load' panel is the state of the structure. When dead loads are applied on the connection,

there are no prior loads on the connection, so we start with a stiffness value as if the connection has no loads on it. However on application of live loads we take initial stiffness as stiffness of the connection resulted from the application of dead loads and then analyse the connection to get the final stiffness of the connection under the action of both dead and live loads.

In the GUI, there are certain options for the user to choose in load application panel. If the user only wants to include dead loads, he can click on 'not included' option in live load panel. If the user wants to include only live loads, he can click on 'not included' option in dead load panel. If he wants both load cases, he may choose 'included' from both the panels and enter the desired load values.

### C.2.2.2 Action Buttons

Action buttons are objects that when clicked; perform a specific event or action in the wizard. There are six action buttons provided in the main GUI. Following is the list of action buttons and their functionality.

#### C.2.2.2.1 Preview Model

It creates a visual model of the column-base connection using dimensions entered by the user and displays it on the GUI. The result of pressing this button after entering values of all the parameters is shown in Figure C.32.

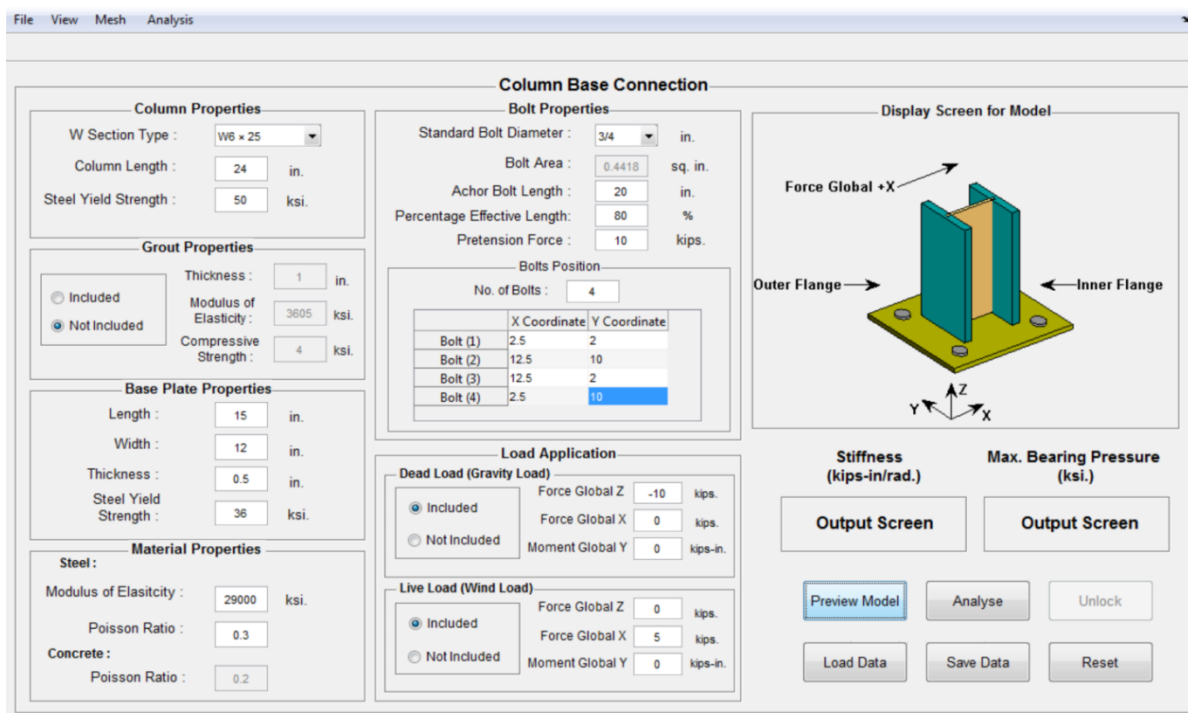


Figure C.32. Model display on execution of 'Preview Model' button

### C.2.2.2.2 Analyse

This button first triggers the processor used for analysis either SAP2000 or OpenSees, based on user's selection from the menu item 'Analysis'. Then it displays a meshed model of the modeled column base connection on the display screen of the main GUI. This model is analyzed in the selected processor. Figure C.33 shows the meshed model which is analyzed by SAP2000 or OpenSees that appears on the screen when the analysis is running on. Note that as soon as this button is pressed, everything in the GUI gets disabled or inactive. The user cannot change anything in the GUI during this processing time. Once the analysis is done, the unlock button becomes active.

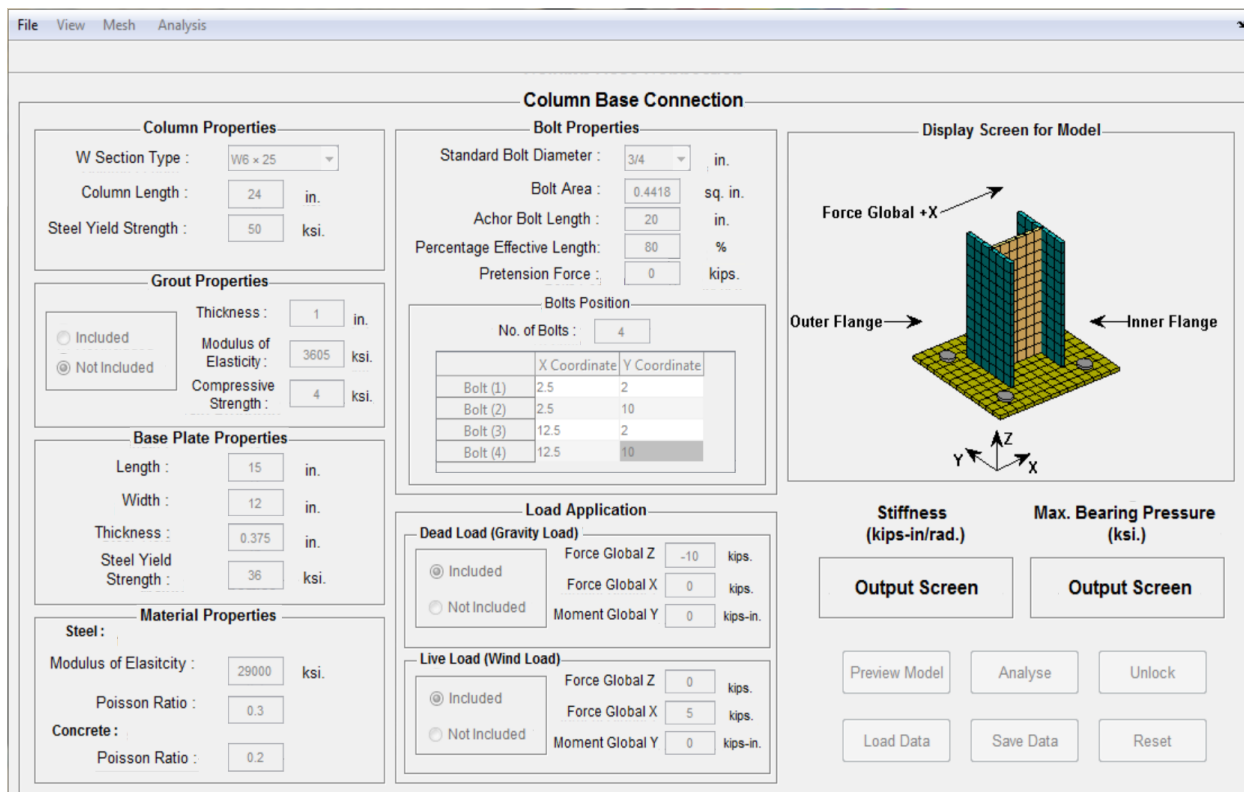


Figure C.33. Meshed Model of designed connection which is analyzed in SAP2000 and OpenSees

### C.2.2.2.3 Unlock

This button clears all the current analysis results (which are stiffness and maximum bearing pressure values in our case) and allows the user to make suitable changes and perform another analysis if desired. Note that this button remains inactive by default. As soon as an analysis is performed, it becomes active and 'Preview Model' and 'Analyse' buttons become inactive. If one wants to perform another analysis then he or she has to press this button to make the above two buttons active again.

#### C.2.2.2.4 Reset

This button allows the user to clear all the input fields and analysis results and reset the wizard to its default state.

#### C.2.2.2.5 Save Data

This button allows the user to save the data entered in various input fields for particular configuration of the column-base connection. On clicking 'Save Data' button, the user interacts with a dialogue box shown in Figure C.34. He can enter a desired name for the data file and save it through this dialogue box.

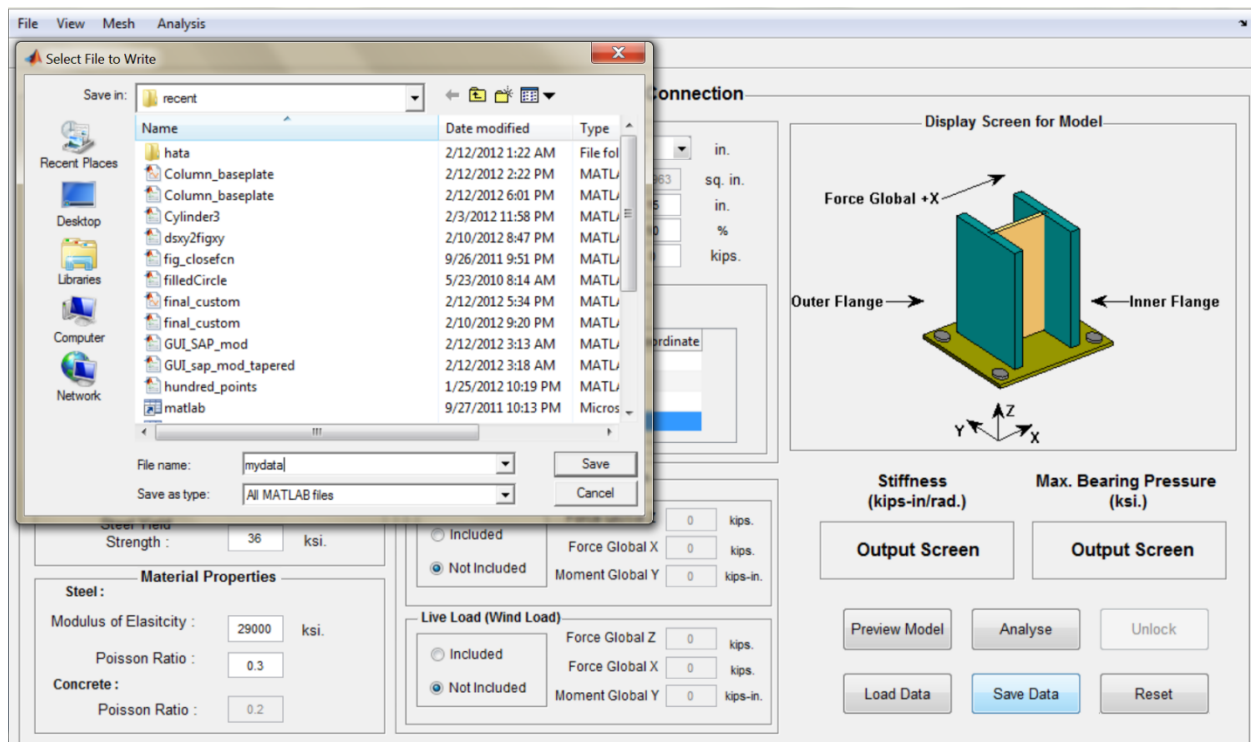


Figure C.34. A dialogue box appears on clicking 'Save Data' button

#### C.2.2.2.6. Load Data

This button allows the user to load any previously saved data file. It works like the menu item 'Open' which is discussed later. On clicking 'Load Data' button, a window pops up and asks for the file to be loaded. The window looks like Figure C.35.

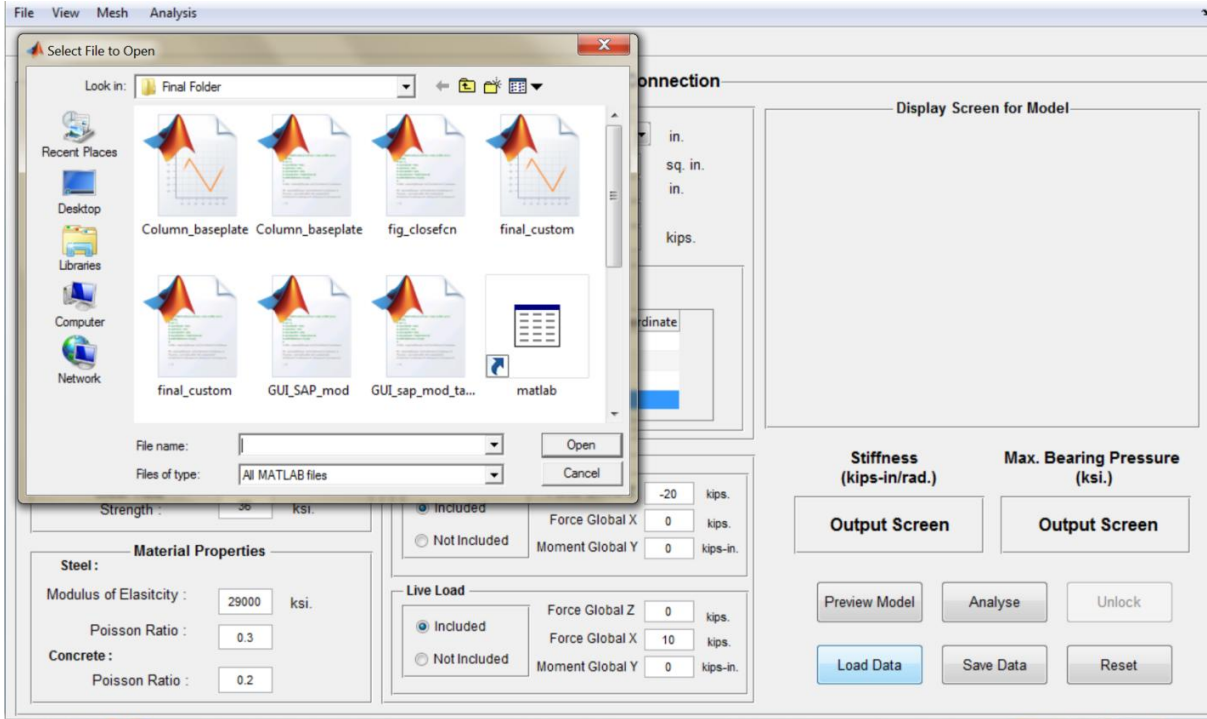


Figure C.35. A dialogue box appears on clicking 'Load Data' button

### C.2.2.3 Menu Bar

There are four sub-menus items in the menu bar of the main GUI. Each of the submenus has some specific options. Functionality of each menu and its sub-menus are discussed below.

#### C.2.2.3.1 File

File submenu provides the user with following sub menus. The options for file are also shown in Figure C.36.

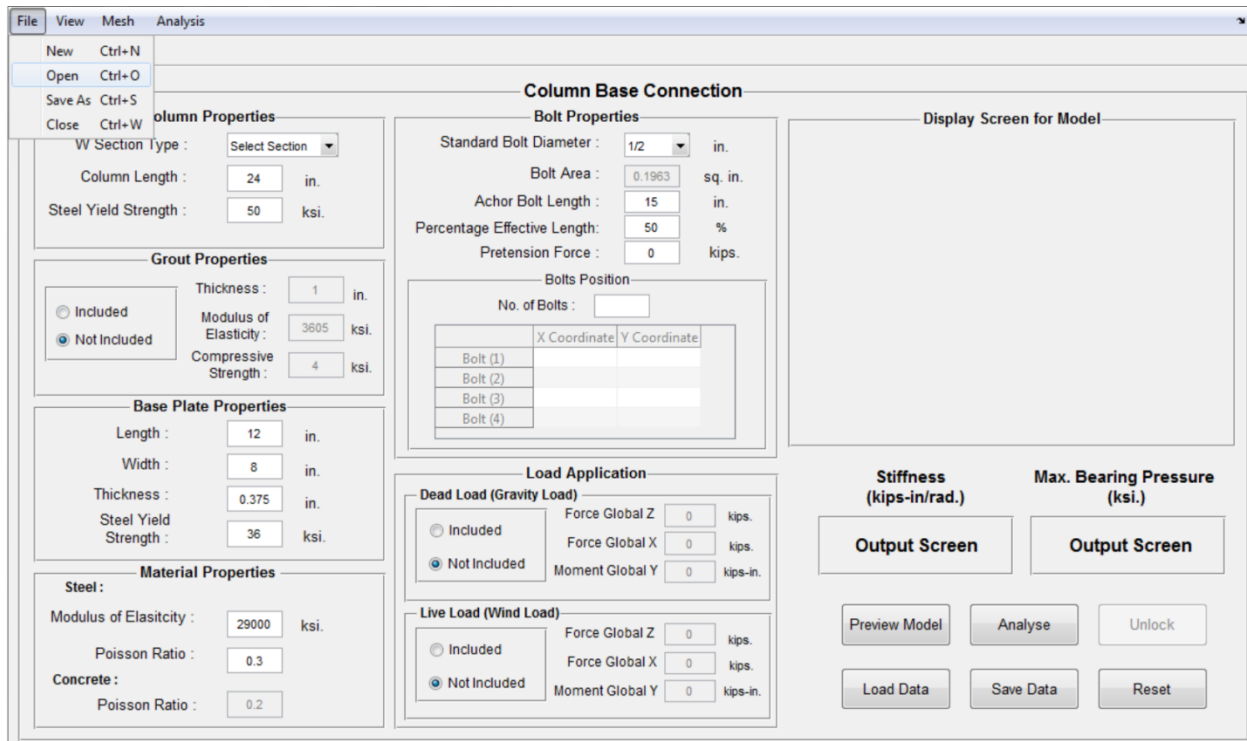


Figure C.36. Sub-menu options under File menu

#### C.2.2.3.1.1 New

'New' replaces the current main GUI window having data entries with another main GUI window which have all the default values. Actually its objective is almost like 'reset' action button. However, instead of clearing all the data fields in the current main GUI, it just replaces the current GUI with another main GUI having default values in all input fields.

#### C.2.2.3.1.2 Open

'Open' brings a dialogue box on clicking, which allows the user to load any previously saved data file created in this wizard.

#### C.2.2.3.1.3 Save As

'Save As' allows the user to save current data entries present in the input fields into a file. On clicking this, a dialogue box appears on the screen which asks the user to enter desired file name under which data is to be saved.

#### C.2.2.3.1.4 Close

'Close' closes the main GUI.

#### C.2.2.3.2 View

View submenu provides user the capability to view the designed model from different angles. It contains four options, shown in Figure C.37.

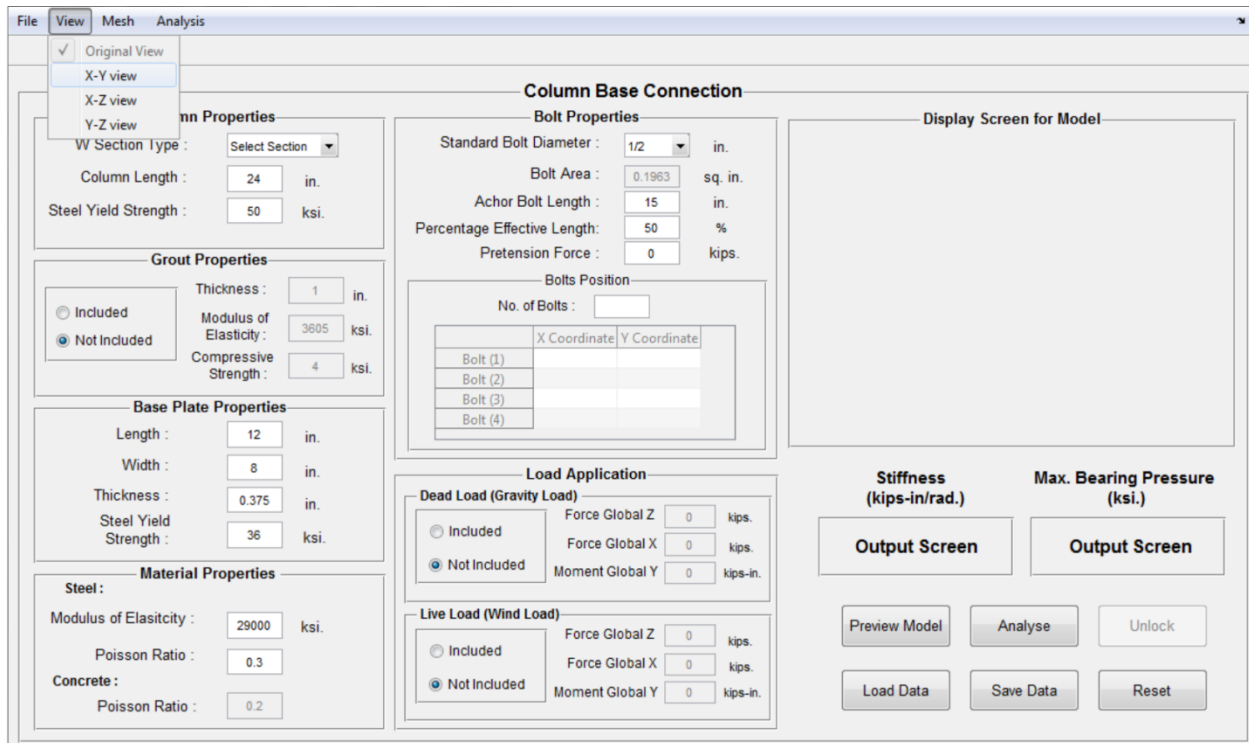


Figure C.37. Sub-menu options under View menu

### C.2.2.3.2.1 Original View

'Original View' option lets the user to see the preliminary and meshed model in three dimensional space, which is represented by global axes X,Y and Z (Figure C.23).

### C.2.2.3.2.2 X-Y view

This option allows the user to see the preliminary and meshed model in X-Y plane.

### C.2.2.3.2.3 Y-Z view

This option allows the user to see the preliminary and meshed model in Y-Z plane.

### C.2.2.3.2.4 X-Z view

This option allows the user to see the preliminary and meshed model in X-Z plane.

### C.2.2.3.3 Mesh

This submenu provides two options to create mesh sizes for analysing the modeled connection. Meshing is done for performing accurate analysis by dividing large elements into small elements. The two options provided for this submenu item are shown in Figure C.38.



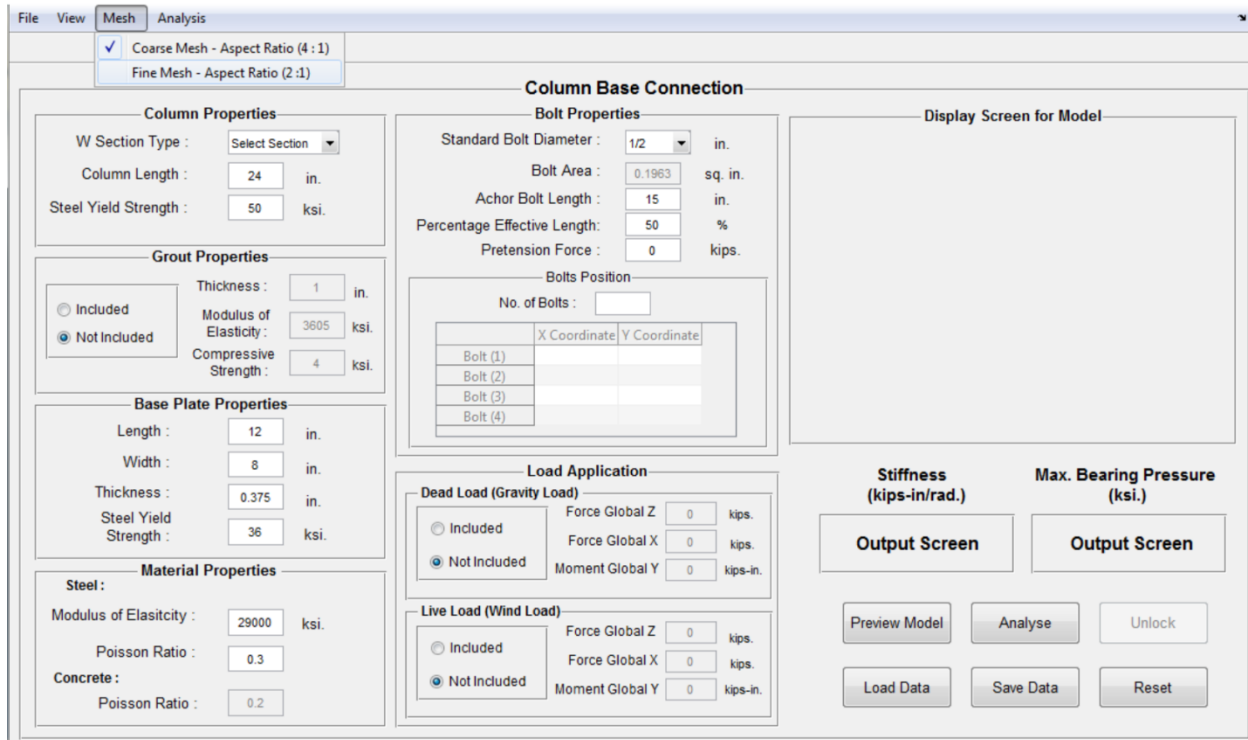


Figure C.38. Sub-menu options under Mesh menu

### C.2.2.3.3.1 Coarse Mesh

This option creates a coarse mesh in the connection model maintaining the aspect ratio (ratio of maximum dimension to minimum dimension of a finite element) within 4:1; and processors SAP2000 and OpenSees analyze the model by taking this mesh size. Here, this 4:1 aspect ratio is considered as maximum limit to get good results. Otherwise this limit varies from software to software which is not mentioned explicitly. Default mesh option is coarse mesh but the user can change this to fine mesh as per need.

### C.2.2.3.3.2 Fine Mesh

This option creates a fine mesh in the connection model maintaining the aspect ratio within 2:1; and processors SAP2000 and OpenSees analyze the model by taking this mesh size. Note that 2:1 aspect ratio creates very fine mesh in the connection model and results in increase in analysis time taken by the above mentioned programs. Essentially, if the user decides to use smaller aspect ratio then he has to wait more to get the stiffness value but off course it increases the accuracy of the results.

### C.2.2.3.4 Analysis

This sub-menu has two options for choosing analysis engine for getting an estimate of the rotational stiffness. The two sub-menu options are shown in Figure C.39.

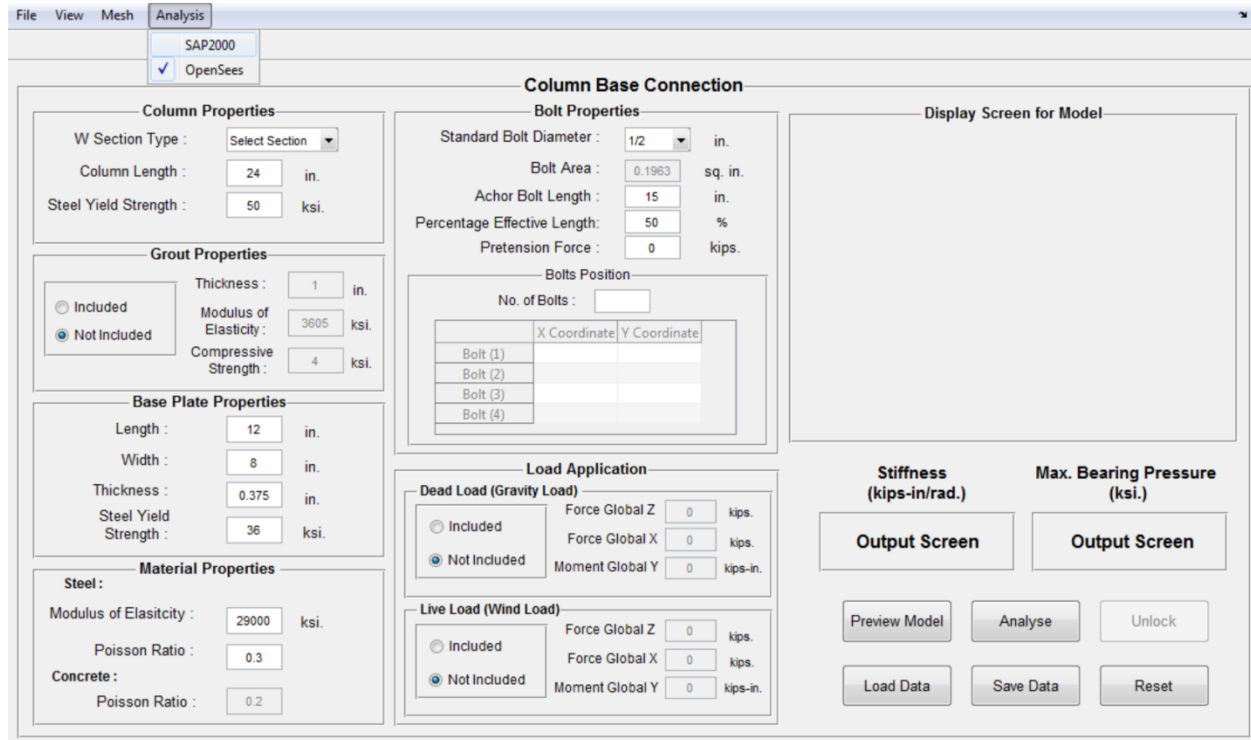


Figure C.39. Sub-menu options under Analysis menu

#### C.2.2.3.4.1 SAP2000

By clicking this submenu option, the user can use SAP2000 as analysis software. On clicking 'Analyse' action button, SAP2000 application is triggered and it performs an analysis based on the input data and gives a stiffness value and a bearing pressure distribution as results.

#### C.2.2.3.4.2 OpenSees

OpenSees is an open source earthquake engineering analysis software and it is used as one of the analysis engine for this wizard. For users who don't have access to SAP2000, can select this option from Analysis menu and get an estimate of stiffness of the connection. Note that the default analysis engine for this wizard is OpenSees, however the users can switch to SAP2000 anytime.

**Note** that the stiffness estimates for the same connection configuration from the two softwares SAP2000 and OpenSees may not come exactly same, but they will be close. The reason behind the unequal estimates is the use of different finite elements in SAP2000 and OpenSees for modeling each component of the connection.

#### C.2.2.4. Display Screen and Output Windows

These components display vital information such as preview of the modeled connection, values of the stiffness and maximum bearing pressure, moment rotation curve and bearing pressure distribution plot. Moreover, these components don't allow the user to make any changes in them.

The preview display of the model and the values of the stiffness and maximum bearing pressure are quite explicable. However, the other two displays need some clear understanding as in how these plots are interpreted in this wizard.

#### **C.2.2.4.1 Moment Rotation Plot**

One of the main outputs of this wizard is moment rotation plot for any configuration of the connection under flexural loading or flexural-axial loading. In every moment rotation plot, the moment plotted is the total moment which includes moments applied in both gravity load case and wind load case. Figure C.40 shows a moment-rotation plot for some random configuration of the column base connection. The symbol 'K' is slope of the line which is tangent to the initial part of the moment rotation curve; it also represents the initial rotational stiffness of the connection which is also displayed on the output screen present beneath the label 'Stiffness' on bottom right of the main GUI. The labels 'Rotation' and 'Moment' below the plot display the values of rotation and moment corresponding to present position of the mouse cursor on the plot. The label 'Secant Stiffness' displays the secant stiffness of any point on the curve. The secant stiffness of any point on the curve represents the slope of a line joining the origin of the plot and the point of observation. As soon as the mouse cursor is pointed to a point on the curve, it turns into a red dot if there is a data point close by and shows corresponding moment, rotation and secant stiffness value. Note that in this wizard, the rotation values are plotted after multiplying the actual rotation values by 1000. To get an actual value of rotation, the user must divide the values of rotation obtained from the plot by 1000.

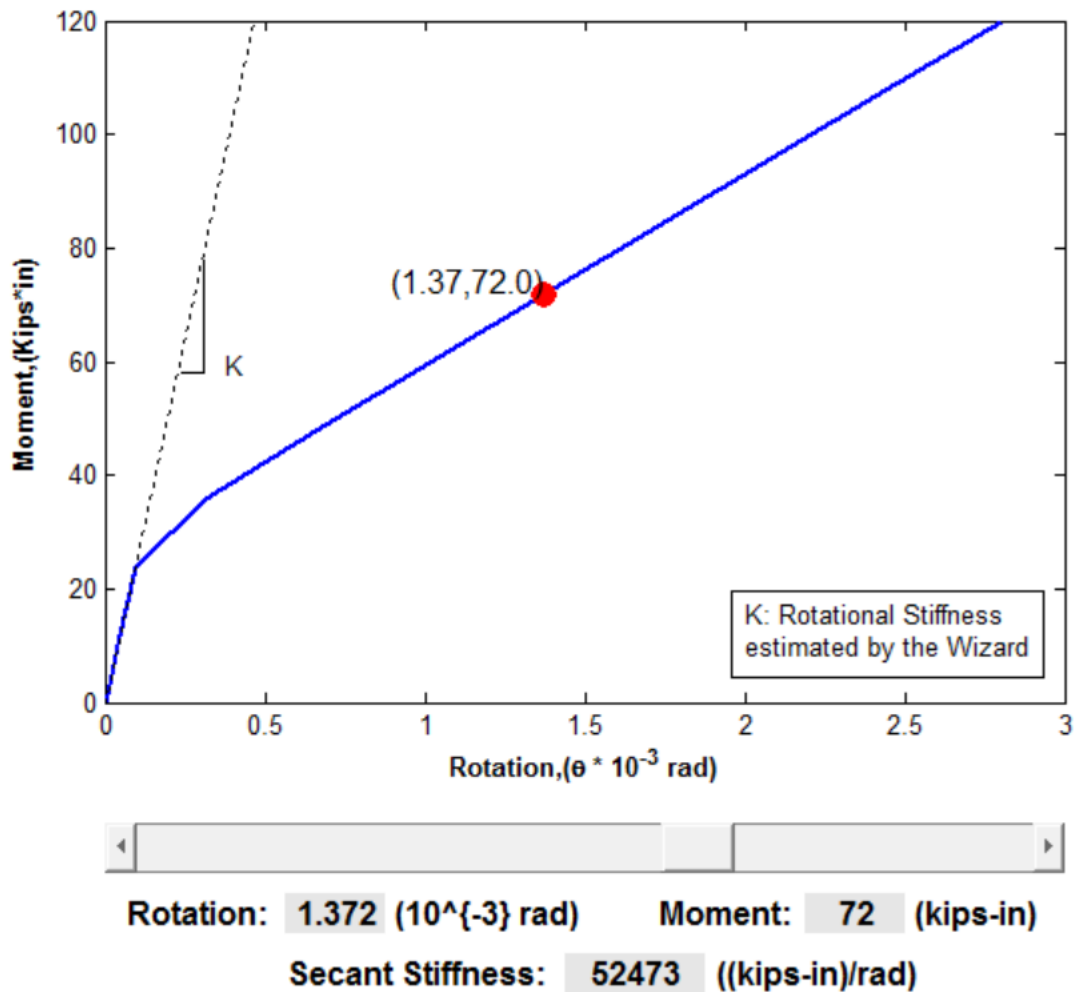


Figure C.40. Moment Rotation Curve obtained from the wizard

#### C.2.2.4.1 Bearing Pressure Distribution

The distribution of bearing pressure on the base plate looks something like Figure C.41. The whole copper color region represents the base plate and the numbers written orthogonally on left and bottom of the plot represent the length and the width of the base plate. The rectangles and circles inside the colored region show the positions of column and bolts over the base plate. And the color bar on the right shows the bearing pressure values varying across the base plate. For example, dark red color represents a region of maximum bearing pressure in Figure C.41. The maximum value of pressure exists somewhere in the range of 1.8 to 2 ksi.

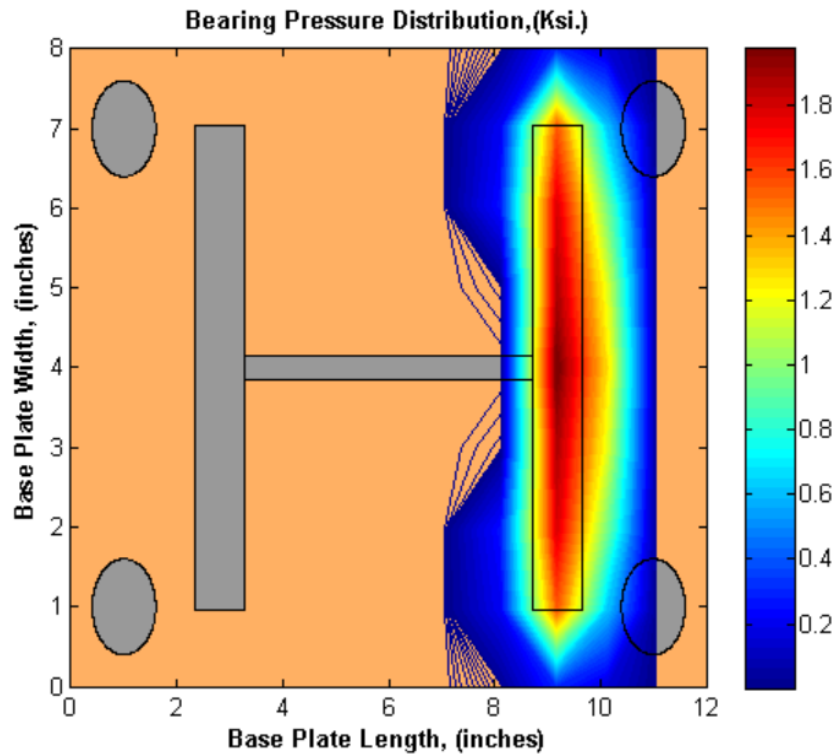


Figure C.41. Typical layout of Bearing Pressure Distribution

#### C.2.2.4.3 Report

After each analysis the wizard displays a report which summarizes all the aspects of that analysis including the inputs and outputs. The user can save this report or export it to Microsoft (MS) Word. Once the user obtains the report as a word document, he can save it in any format provided by the MS Word or print it. Figure C.42 shows a typical report which is given by the wizard. The values of the input parameters and outputs change with the configuration of the connection. As mentioned above, the user can save this report by clicking 'Save' icon or export it to MS word by clicking 'Print' icon, shown by the arrows in Figure C.42.

Save Report

Print Report to MS Word

**Input**

Section	Property	Value	
Column Properties	W Section Type	W6 × 25	
	Column Length	24.00 in.	
	Steel Yield Strength	50.00 ksi.	
Grout Properties		Not Included	
Anchor Bolts Properties	Diameter	0.50 in.	
	Area	0.20 sq.in.	
	Anchor Rod Length	15.00 in.	
	Percentage Effective Length	50.00 %	
	Pretension Force	0.00 kips.	
Baseplate Properties	Length	12.00 in.	
	Width	8.00 in.	
	Thickness	0.375 in.	
	Steel Yield Strength	36.00 ksi.	
Material Properties	Steel:		
	Modulus of Elasticity	29000.00 ksi.	
	Poisson Ratio	0.30 in.	
	Concrete:		
	Poisson Ratio	0.20 in.	
Anchor Bolts Positions		X Coordinate	Y Coordinate
	Bolt (1)	1	1
	Bolt (2)	1	7
	Bolt (3)	11	1
	Bolt (4)	11	7

**Loads Applied**

Category	Property	Value
Dead Loads or Gravity Loads	Force Global Z	-10.00 kips.
	Force Global X	0.00 kips.
	Moment Global Y	0.00 kips.
Live Loads or Wind Loads	Force Global Z	0.00 kips.
	Force Global X	5.00 kips.
	Moment Global Y	0.00 kips.

**Output**

**Rotational Stiffness: 244447 kips-rad/in.**

**Maximum Bearing Pressure: 1.98 ksi.**

Figure C.42. A typical report which is displayed by the wizard

### C.3.1 Functioning of the Batch Mode

The column base connection wizard provides the capability to model the connection, perform the analysis and calculate the rotational stiffness without using the GUI mode. This mode of operation of this wizard is referred as batch mode operation. The installation instruction for Batch Mode is provided in Section C.1.3.2. After the installation, the user can access the batch mode operation of the wizard by going to 'Batch Mode' folder present in 'Connection Wizard' folder. The following sections discuss the functionality of the wizard in batch mode operation.

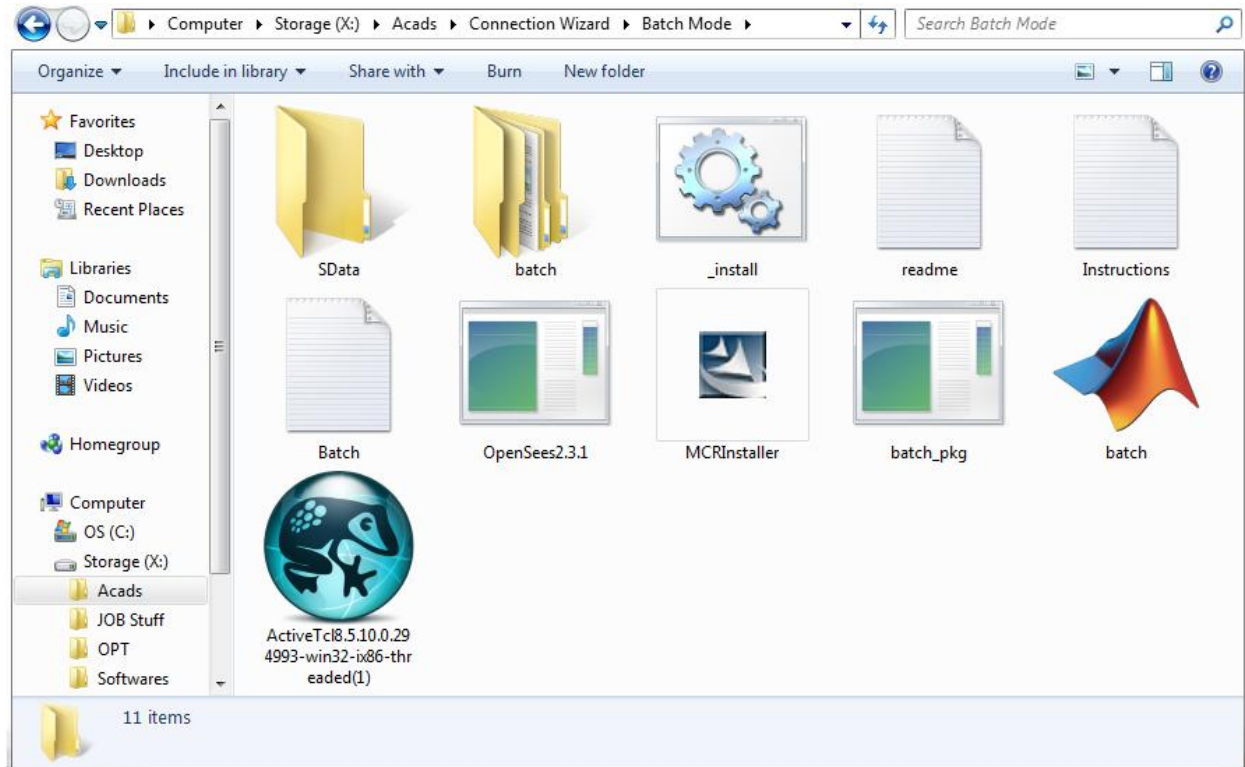


Figure C.43. 'Batch Mode' folder

### C.3.2 Batch Mode Operation

A Notepad file named as *Batch.txt* is given in 'Batch Mode' folder (see Figure C.43). This is an input file provided for the batch mode operation of the wizard. Figure C.44 shows the file 'Batch.txt'. The values of all the parameters of the connection are provided in this input file. The user can modify the default values of the parameters provided in the file and replace them with the desired values. The user can also write a new input file, however it should be noted that while writing a new input file, the sequence of the values of the parameters must be same as provided in 'Batch.txt'. In other words, the format of the data entered should exactly look like Figure C.44 except the values, which can be different. Moreover, the written file must be saved with the name *Batch.txt*. This is because the wizard in batch mode operation expects the presence of *Batch.txt* as its input data file. Any name other than *Batch.txt* will not work and if

*Batch.txt* file is not present in 'Batch Mode' folder then execution of wizard in batch mode may give an error on the screen.

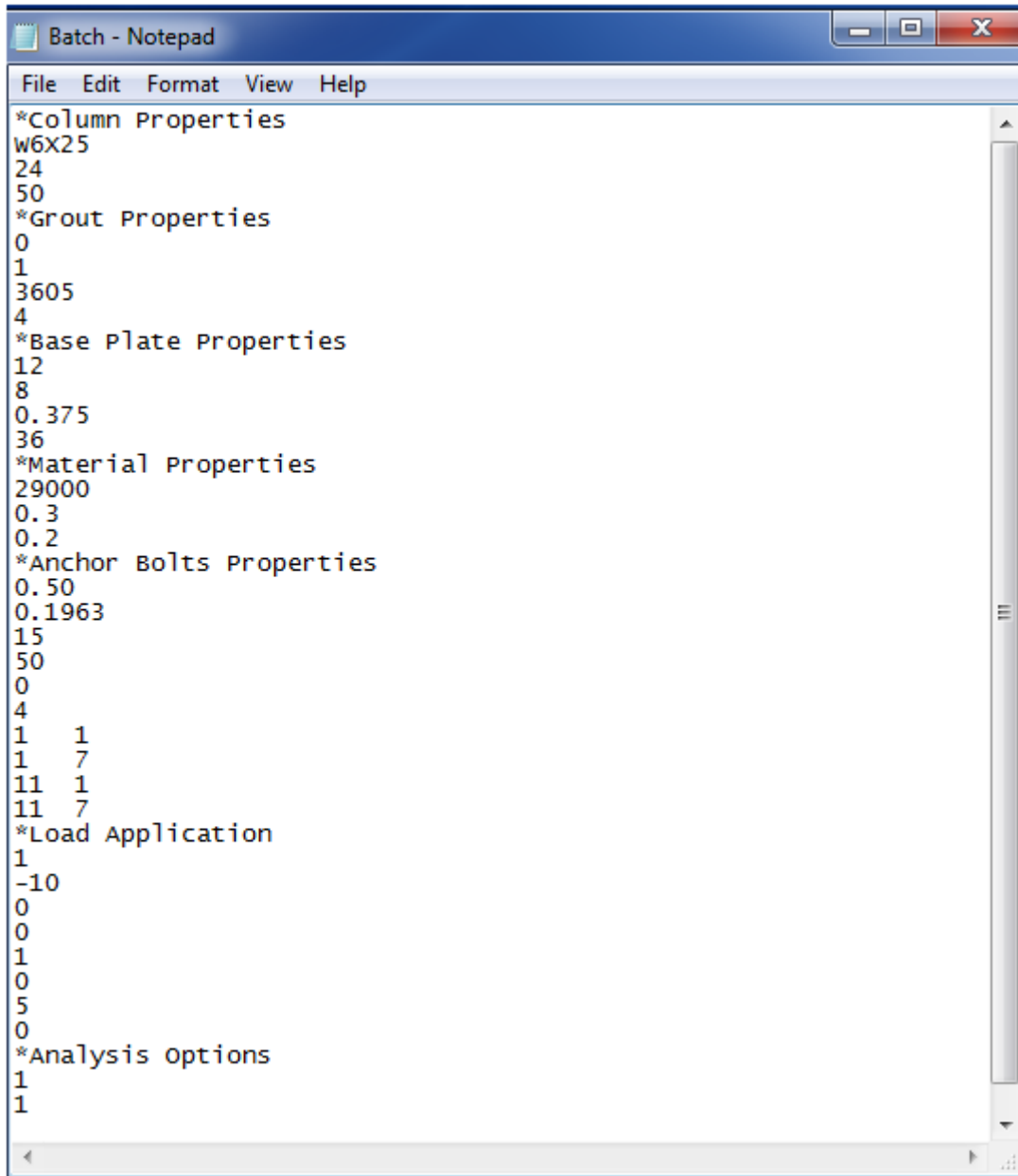


Figure C.44. *Batch.txt*, Input file for the batch mode operation of the wizard

As *Batch.txt* seems to have data without annotation, the user will become confused, as in what value represents what. For the convenience of the user, a notepad file labeled as *Instructions.txt* is provided in 'Batch Mode' folder. Figure C.45 shows *Instructions.txt* file. This file provides an annotative explanation of each value that is entered in the input file. In other words, it explains what each value entered in the input file represents.



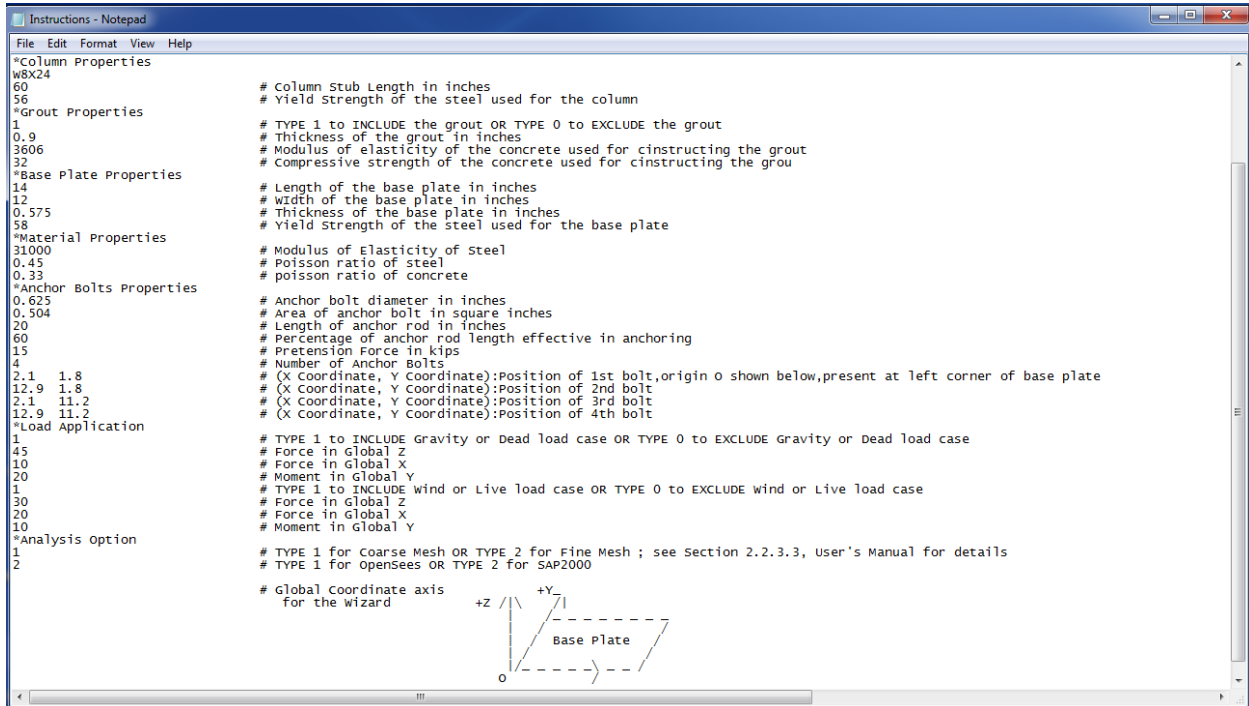


Figure C.45. *Instructions.txt* file provides annotative explanation for *Batch.txt*

As mentioned above that after entering the desired values for all the parameters, the file is saved again. At this stage the user just has to click on *batch.exe* shown in Figure C.46 for getting the values for the initial rotational stiffness and maximum bearing pressure.

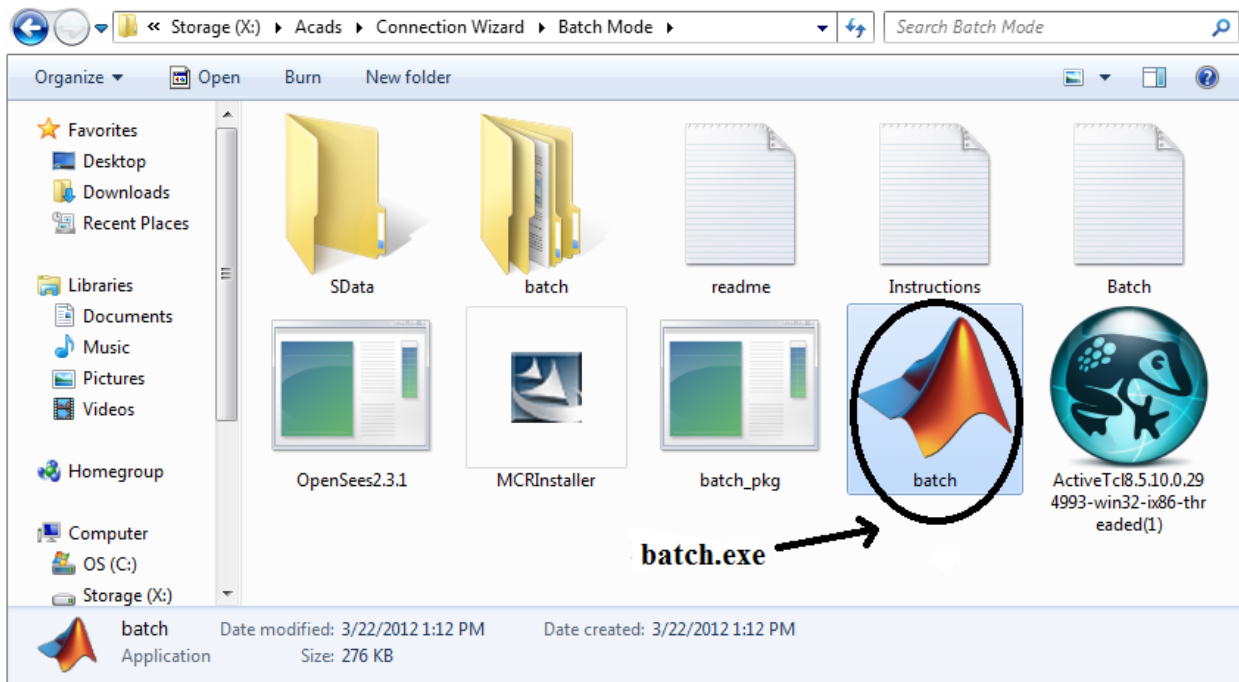


Figure C.46. *batch.exe* is an executable file, which on clicking execute analysis

*batch.exe* is an executable file which first reads the numerical data from the input file *Batch.txt* then accesses the compiled MATLAB scripts, which were written to model and

analyze any configuration of the column base connection. The scripts are compiled with the standalone version and are not visible to the user. They perform analysis and prints the results in an output file labeled as *Results.txt*. This output file displays the values of initial rotational stiffness of the connection and maximum bearing pressure on the base plate (Figure C.47). The output file can be saved at some other location in the computer for future references. Once done with recording of the results this output file must be deleted manually or it will be overwritten each time *batch.exe* is clicked.

Please note that if the user doesn't want to perform the whole process explained above and instead he has got his own program, which has a capability of running an .exe file, then he can write a small routine in his program which can write an input file labeled as *Batch.txt* which has the same format as instructed by *Instructions.txt*. Sequentially, he can call *batch.exe* from his program and get the values of the initial rotational stiffness of the connection and maximum bearing pressure on the base plate. The only thing he has to be careful about is the availability of *batch.exe* to his program. Moreover, *batch.exe* and the input file written from his program should be in the same folder or path.

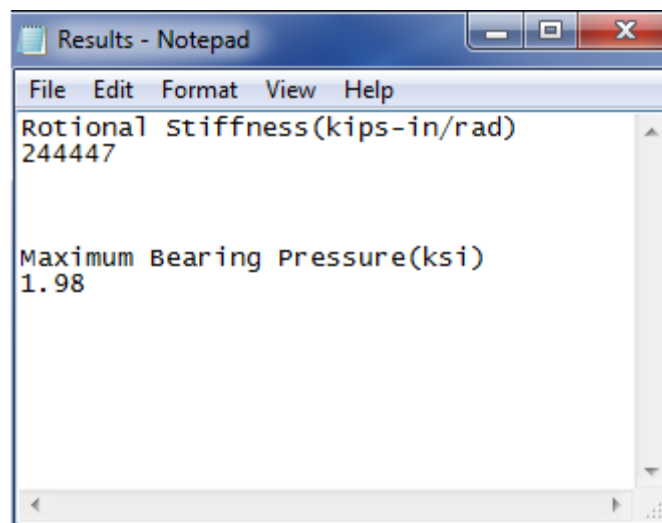


Figure C.47. An output file, *Results.txt* from batch mode operation of the wizard

The procedure explained above executes batch mode operation for any wide flange section upto W27X178. If the user wants to analyze a connection which has a tapered column section, then he has to enter values of all the parameters in an input file *Batch\_Tapered.txt* (see Figure C.48), which is also provided in 'Batch\_Mode' folder (see Figure C.49). The file defines various parameters to model a tapered section. The instructions for writing or modifying the input file for the tapered section batch mode operation is provided in *Instruction\_tapered.txt*. The user can modify the default values of these parameters as per his requirement by following the instructions and save the file again. An executable file *batch\_tapered.exe* is also provided along with the input file in 'Batch\_Mode' folder. This executable file performs a similar function which *batch.exe* performs for connections having wide flange sections. When the user clicks

this .exe file, it performs analysis for the configuration defined in *Batch\_Tapered.txt* and publishes the results in *Results.txt*. If the user has a program which is capable of executing .exe file then he can use the batch mode operation for the tapered column in a similar fashion to that of the batch mode operation for the wide flange section which is explained above.

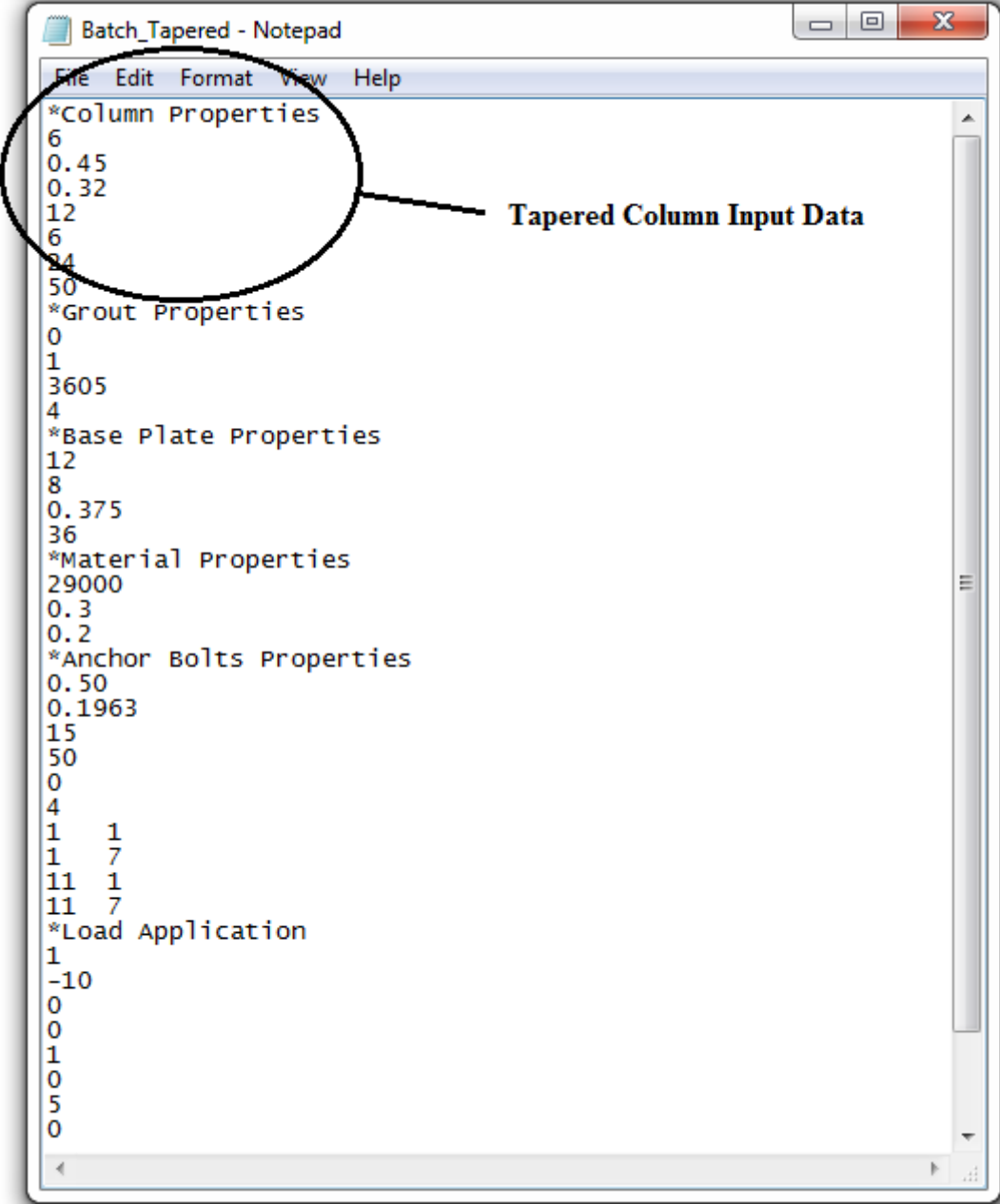


Figure C.48. *Batch\_Tapered.txt*, the input file for tapered column batch mode operation

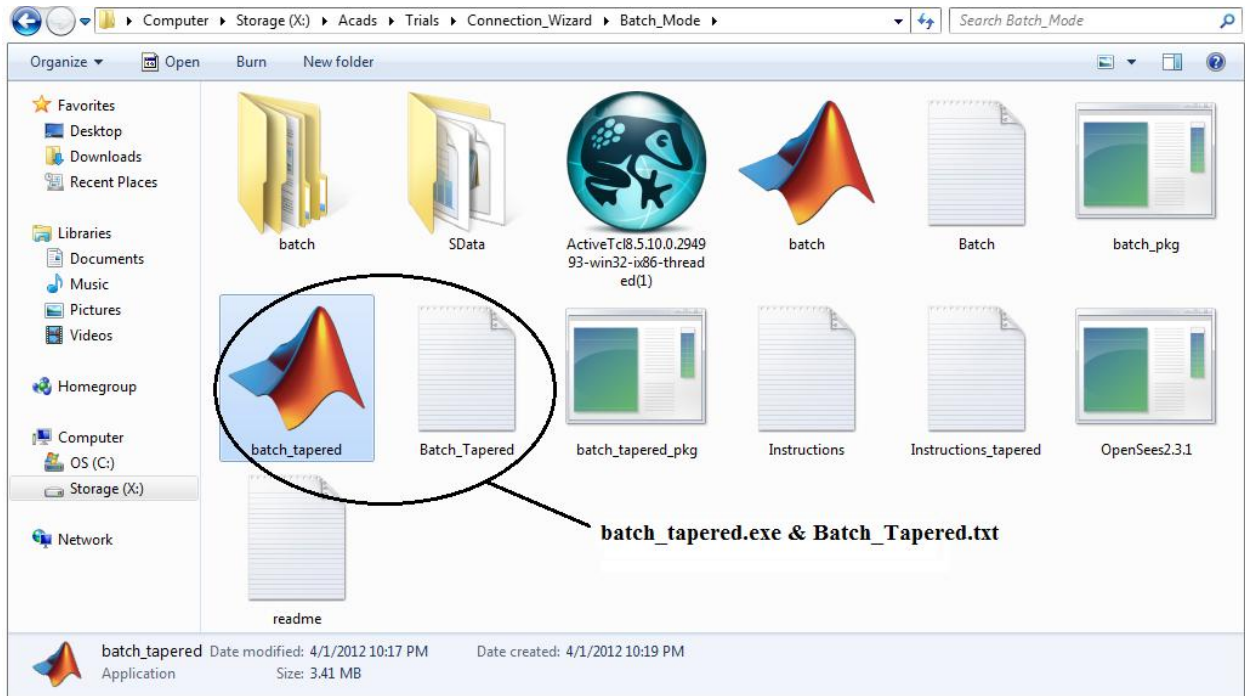


Figure C.49. *Batch\_Tapered.txt* and *batch\_tapered.exe*

## **C.4 Solved Examples**

### **C.4.1. Example 1**

This example shows how to calculate initial rotational stiffness of a modeled column base connection and the maximum bearing pressure on the base plate by using the GUI mode of this wizard for any applied loads. The various features of the wizard that are demonstrated in this example are following.

- Wide Flange Section
- Pretension Force in Bolts
- Fine Meshing
- OpenSees
- Percentage Effective Length

#### **Geometry and Material Used**

In this example, a W6X25 column stub having a length of 2 feet is chosen. The base plate connected to the column stub has a length of 15 inches, a width of 12 inches and a thickness of 0.5 inch. A standard diameter of 3/4 inch is taken for the bolts and the anchor rod length is taken as 20 inches. There is 10 kips of pretension force provided in each anchor bolt. Positions of anchor bolts on the base plate are shown in Figure C.50. Material is steel for the column, base plate and bolts, which has a modulus of elasticity 29000 ksi and poisson ratio of 0.3. Yield strengths for the steel used for the column and base plate are 50 ksi and 36 ksi respectively. The percentage of length of anchor rod which is effective in providing stiffness is taken as 80 %.

#### **Applied Loads**

The Figure C.51 shows the wizard with all the properties entered. For this example fine meshing type is chosen from the 'Mesh' sub-menu and OpenSees as analysis engine from 'Analysis' sub-menu. A lateral load of 5 kips and an axial compressive load of 10 kips are applied to the column stub. Note that compressive load must be entered with a negative sign.

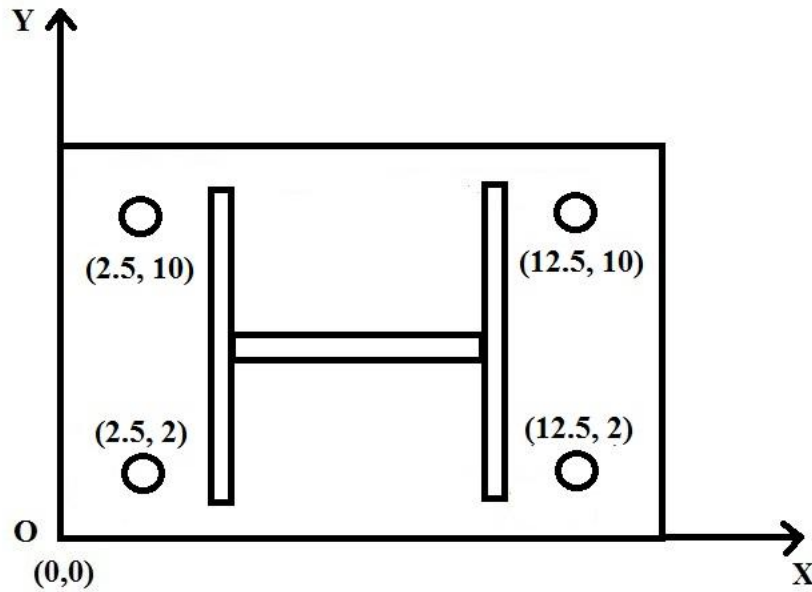


Figure C.50. Positions of Bolts on the Base-Plate

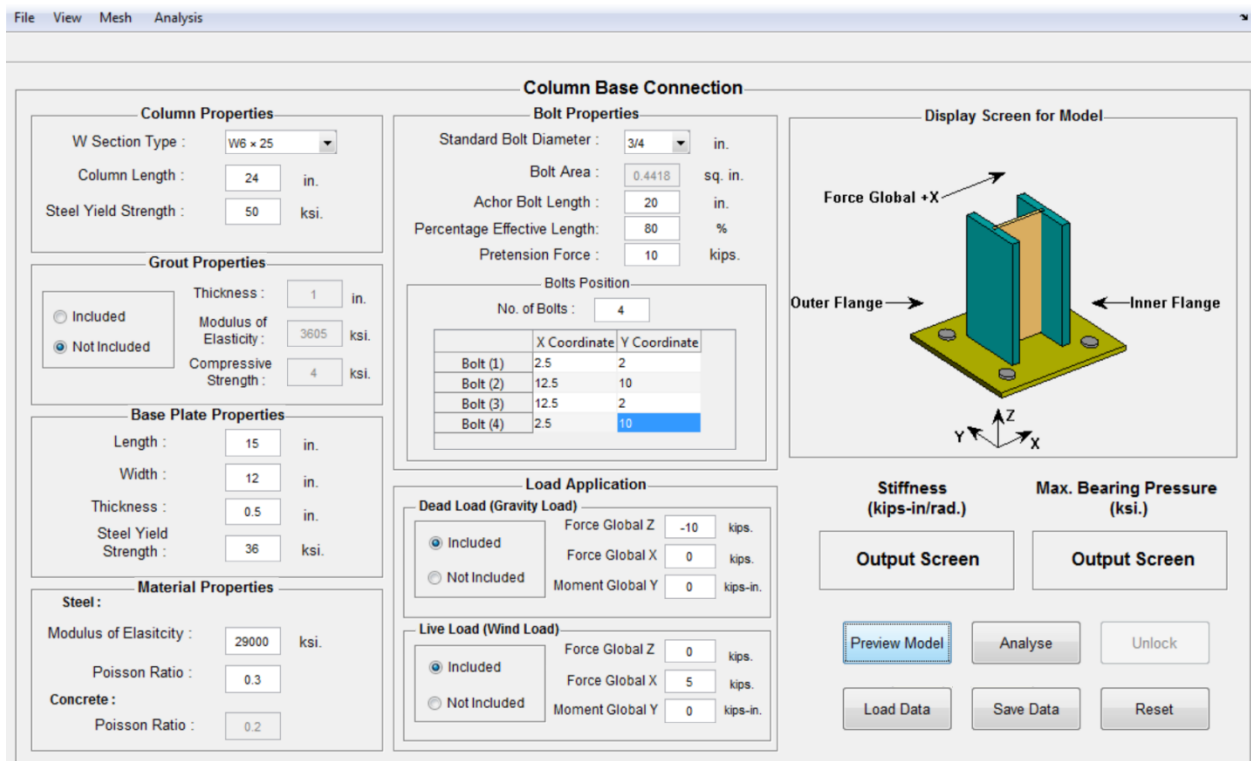


Figure C.51. Wizard window with all the properties entered

By clicking on 'Preview Model' button, the user can view the model of the connection on display screen. User can also switch between different views to check model by selecting the desired view angle from the view submenu. Note that the model showed on the display screen is the one

which is analyzed by either SAP2000 or OpenSees on clicking analysis button. User can always change the configuration of the connection before analysis by changing the values in appropriate fields of GUI. However if the analysis is performed then user has to click the unlock button to make any changes in the model. Following are the step by step instructions to use this wizard to get the stiffness and bearing pressure of above defined configuration of the connection.

- Choose W6X25 section from the W Section type drop down list box present in column properties input panel as shown in Figure C.52.

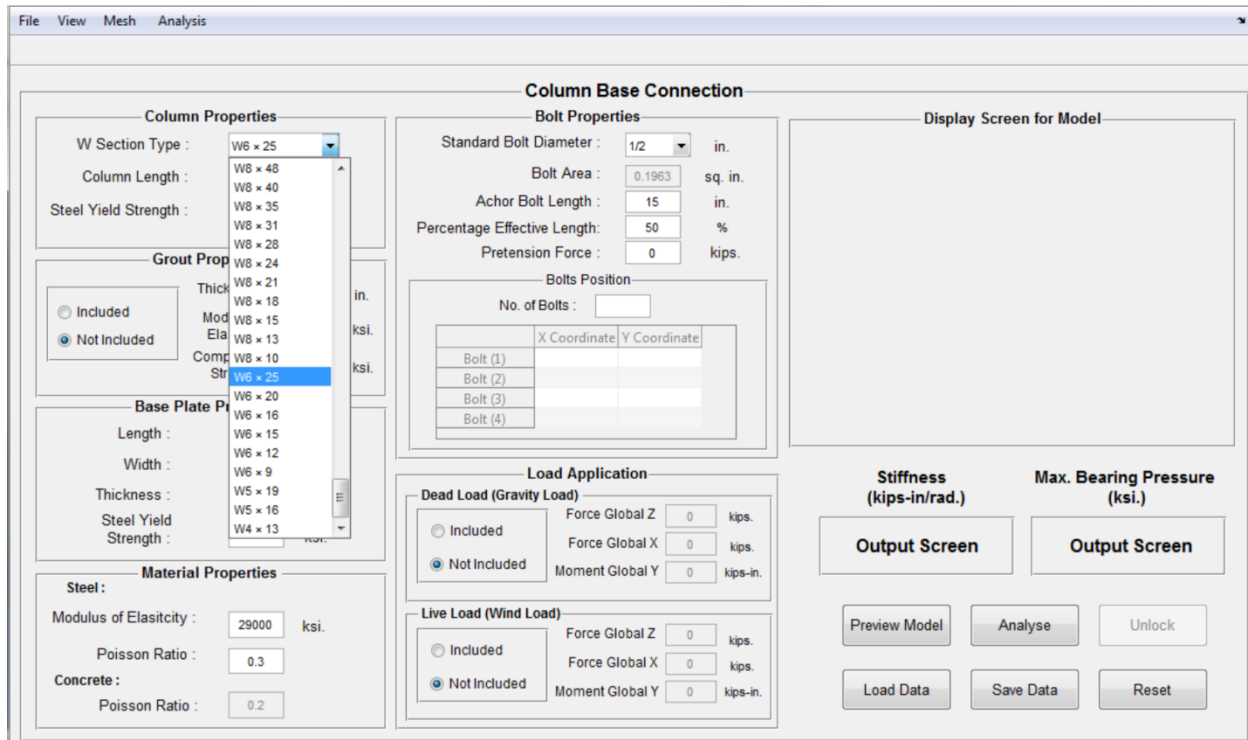


Figure C.52. W6X25 selected from W section type drop down list box

- Input the column stub length as 24 inches and yield strength of steel as 50 ksi in the 'Column Properties' input panel.
- Check on 'Included' radio button in 'Grout Properties' panel if the grout is to be modeled. In this example the grout is not modeled, hence user can skip this panel.
- Enter values for the parameters present in 'Base Plate Properties' input panel. For this example, a length of 15 inches, a width of 12 inches, a thickness of 0.5 inch and a yield strength of 36 ksi are entered.
- Material Properties panel is provided with default properties for the steel and concrete. User can change these properties as desired. Note that poisson ratio for concrete will remain disabled unless the user decides to include grout in the model.
- Bolts have lots of parameters, which can be entered by the user as desired in 'Bolt Properties' panel. From 'Standard Bolt Diameter' drop down list box, a desired standard bolt diameter

can be selected. If the bolts used are not standard sized AISC bolts, then the user can enter any other bolt diameter by providing its area in 'Bolt Area' field. A length for anchor rod and an effective percentage for that embedded length can also be entered. In this example, a standard diameter of 3/4 inch, an anchor bolt length of 20 inches and effective length percentage of 80 % are considered.

- Pretension in bolts can be provided in the pretension force input field present in 'Bolt Properties' input panel. In this example, there is 10 kips of pretension force is provided in each bolt.
- Locations of bolts on the base plate can be provided in bolt position table. Input the number of bolts and press enter; it will enable the table and coordinates of the bolts can be entered as shown in Figure C.53. The modeling of the connection is over with this step.

**Column Base Connection**

**Column Properties**  
W Section Type : W6 x 25  
Column Length : 24 in.  
Steel Yield Strength : 50 ksi.

**Bolt Properties**  
Standard Bolt Diameter : 3/4 in.  
Bolt Area : 0.4418 sq. in.  
Anchor Bolt Length : 20 in.  
Percentage Effective Length : 80 %  
Pretension Force : 10 kips.

**Grout Properties**  
 Included  
 Not Included  
Thickness : 1 in.  
Modulus of Elasticity : 3605 ksi.  
Compressive Strength : 4 ksi.

**Base Plate Properties**  
Length : 15 in.  
Width : 12 in.  
Thickness : 0.5 in.  
Steel Yield Strength : 36 ksi.

**Material Properties**  
**Steel :**  
Modulus of Elasticity : 29000 ksi.  
Poisson Ratio : 0.3  
**Concrete :**  
Poisson Ratio : 0.2

**Bolts Position**  
No. of Bolts : 4

	X Coordinate	Y Coordinate
Bolt (1)	2.5	2
Bolt (2)	2.5	10
Bolt (3)	12.5	2
Bolt (4)	12.5	

**Load Application**  
**Dead Load (Gravity Load)**  
 Included  
 Not Included  
Force Global Z : 0 kips.  
Force Global X : 0 kips.  
Moment Global Y : 0 kips-in.  
**Live Load (Wind Load)**  
 Included  
 Not Included  
Force Global Z : 0 kips.  
Force Global X : 0 kips.  
Moment Global Y : 0 kips-in.

**Display Screen for Model**  
Stiffness (kips-in/rad.)  
Max. Bearing Pressure (ksi.)  
Output Screen  
Output Screen  
Preview Model Analyse Unlock  
Load Data Save Data Reset

Figure C.53. Coordinates of bolts entered in Bolts Position Panel

- In load application input panel, two load cases are provided. Dead loads signify the loads on the connection which are due to gravity loads of rafters and other structural members. Live loads signify the loads acting mostly due to wind in this wizard. In this example, enter 10 kips of gravity load in Global Z direction of 'Dead Load' panel with a negative sign and 5 kips of lateral load due to wind in Global X direction of 'Live Load' panel.
- After entering values for all the parameters described in the above steps, the model is all set to be analyzed. Click on 'Preview Model' button to see the preliminary model, which is going to be analyzed in selected analysis engine. If something needs to be changed, user can change it and again click on 'Preview Model' to see the changes. For this example, the model looks like Figure C.54.



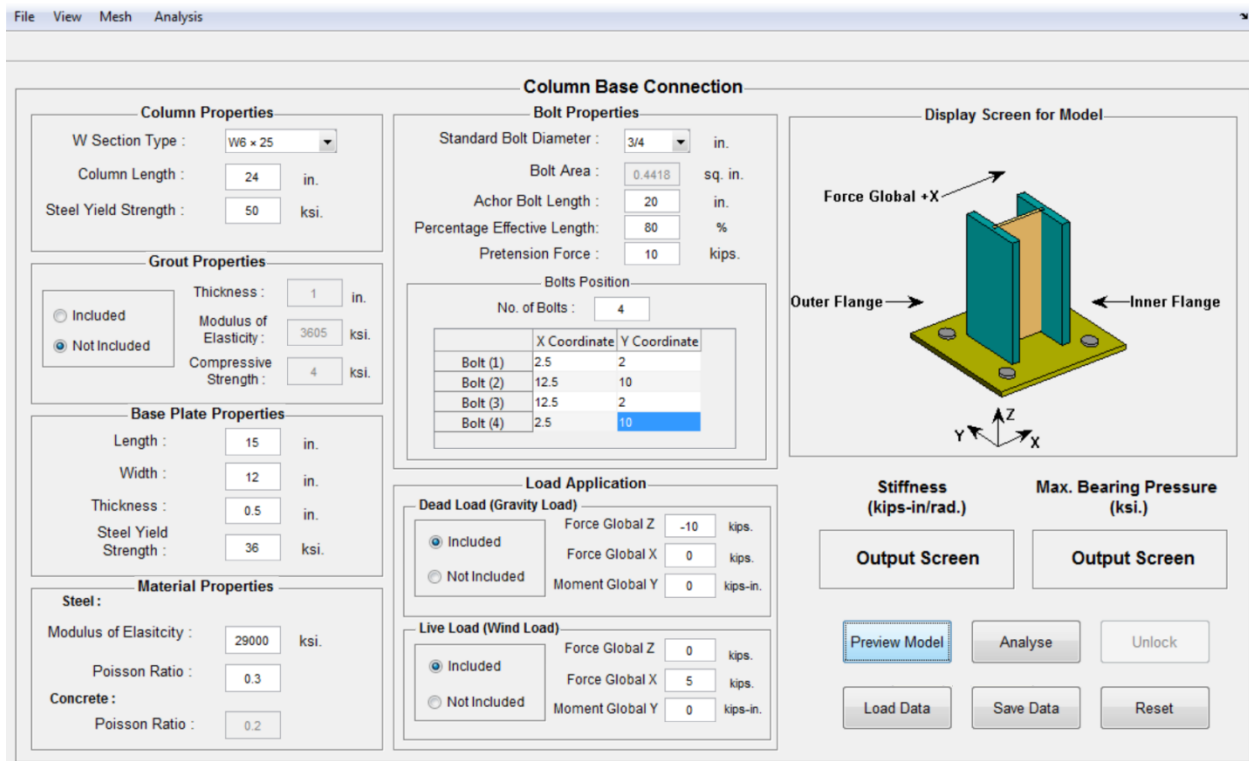


Figure C.54. Display Screen showing the preview of the Modeled Connection

- Before proceeding further, user can choose between specific analysis options. First option is the kind of meshing desired, either coarse or fine. The second option is analysis engine desired, either SAP2000 or OpenSees. The meshing type can be changed from the 'Mesh' submenu and a particular analysis engine can be chosen from the 'Analysis' submenu. For this example, meshing type is chosen as 'Fine Mesh' and analysis engine as 'OpenSees'.
- Model can also be viewed from different angles by exploring options provided in 'View' submenu.
- At this step, we are all set to perform the analysis. Click on 'Analyse' button, it takes few seconds to connect to the selected analysis engine and perform the analysis. The results or value for the rotational stiffness of the connection and maximum bearing pressure are displayed on the output screens present just below the display screen (Figure C.55).
- Two additional figures along with a report appear in addition to the obtained results, one shows the moment-rotation curve (Figure C.56) and the other shows the bearing pressure distribution on the base plate (Figure C.57).
- The report which appears on the screen summarizes all the inputs and outputs of a particular analysis. The report can be printed or saved as per the details provided in Section 2.2.4.3.

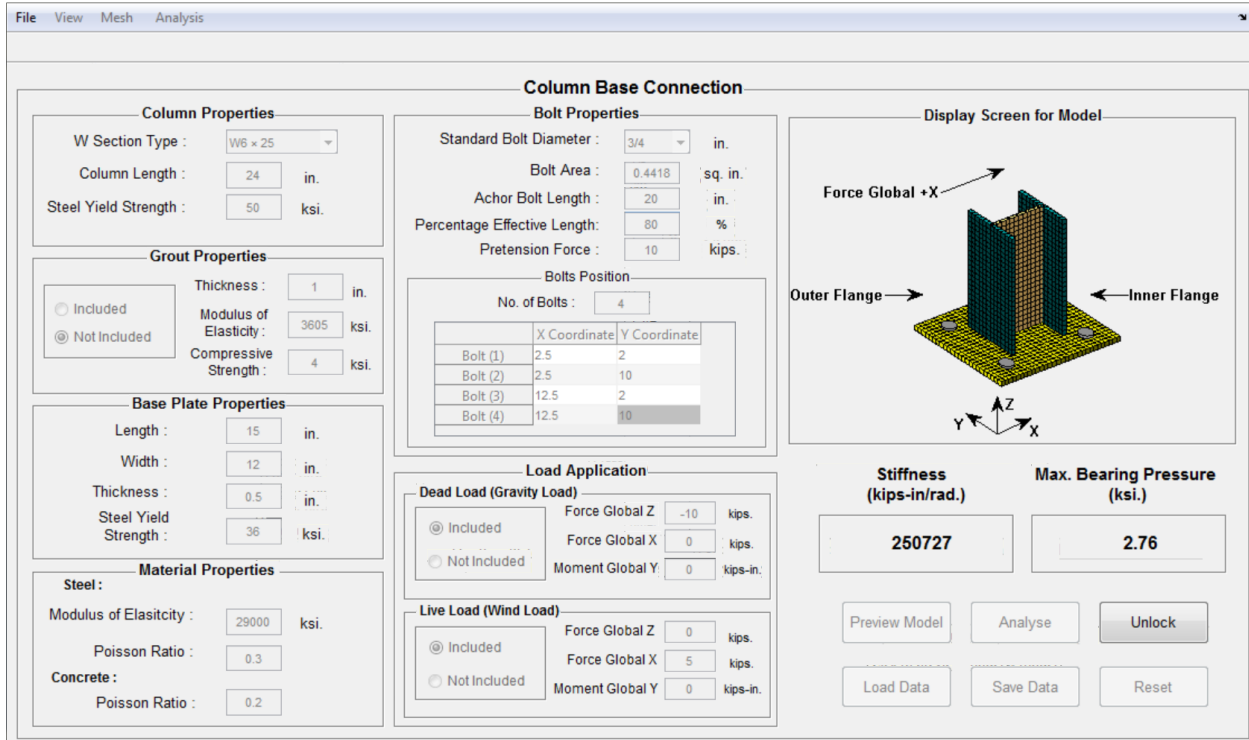


Figure C.55. Wizard Window after performing the analysis and giving back the results

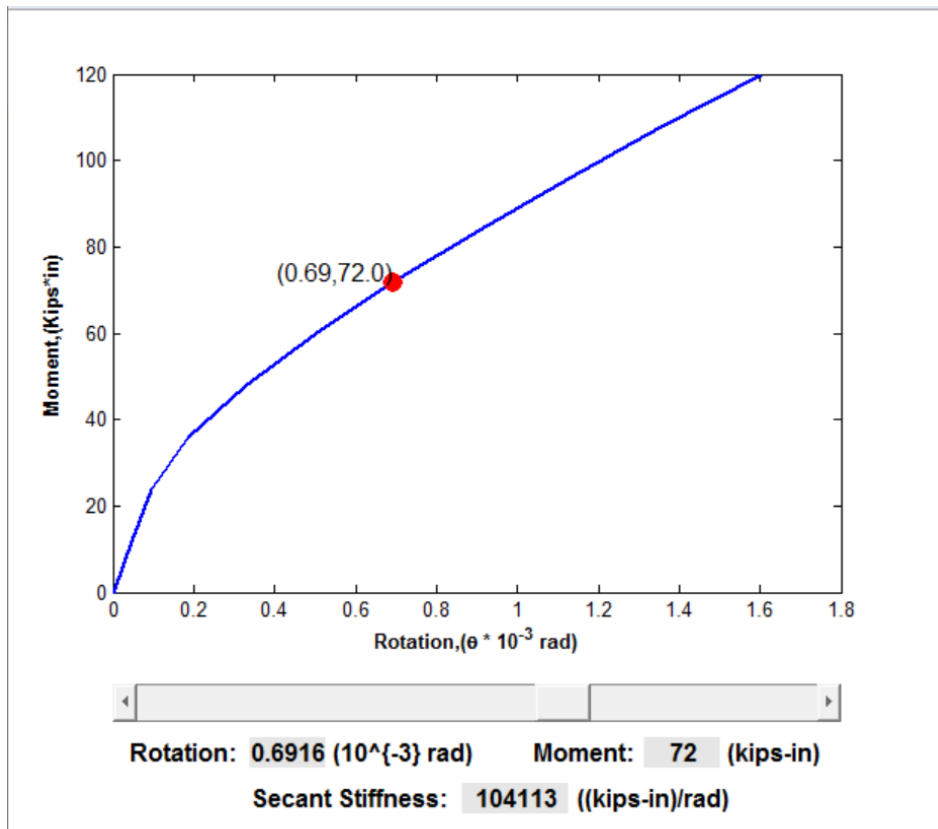


Figure C.56. Moment-Rotation plotted by the Wizard

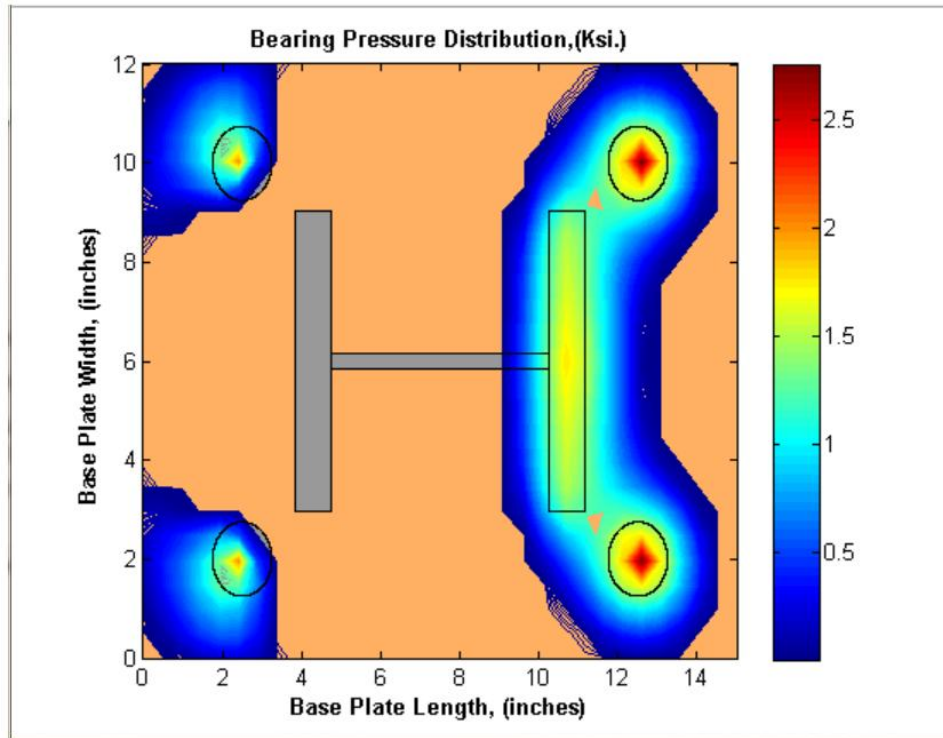


Figure C.57. Bearing Pressure Distribution on the Base Plate plotted by the Wizard

- The user can save the plots for future references. At this point, the user can't perform another analysis by making some changes. The model has to be unlocked by clicking 'Unlock' button to enable analysis button again.
- The user can save the values entered by either clicking on 'Save Data' button or by going to 'File' submenu and clicking on 'Save As' (as shown in Figure C.58). The user can also load the saved data by either clicking the 'Load Data' button or by going into 'File' submenu and clicking on 'Open', as shown in Figure C.59.
- The 'Reset' button provided in the wizard resets the value in each input field to its default value, which was provided by the programmer.

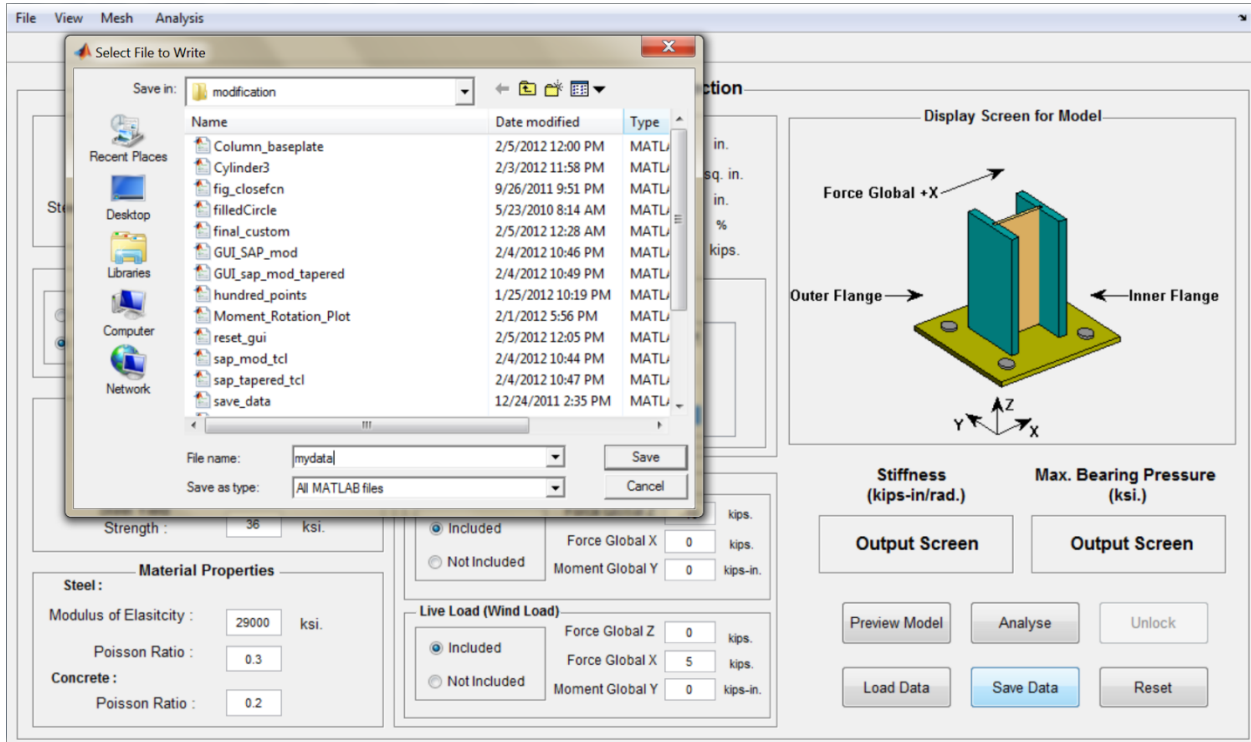


Figure C.58. Saving entered data in a file

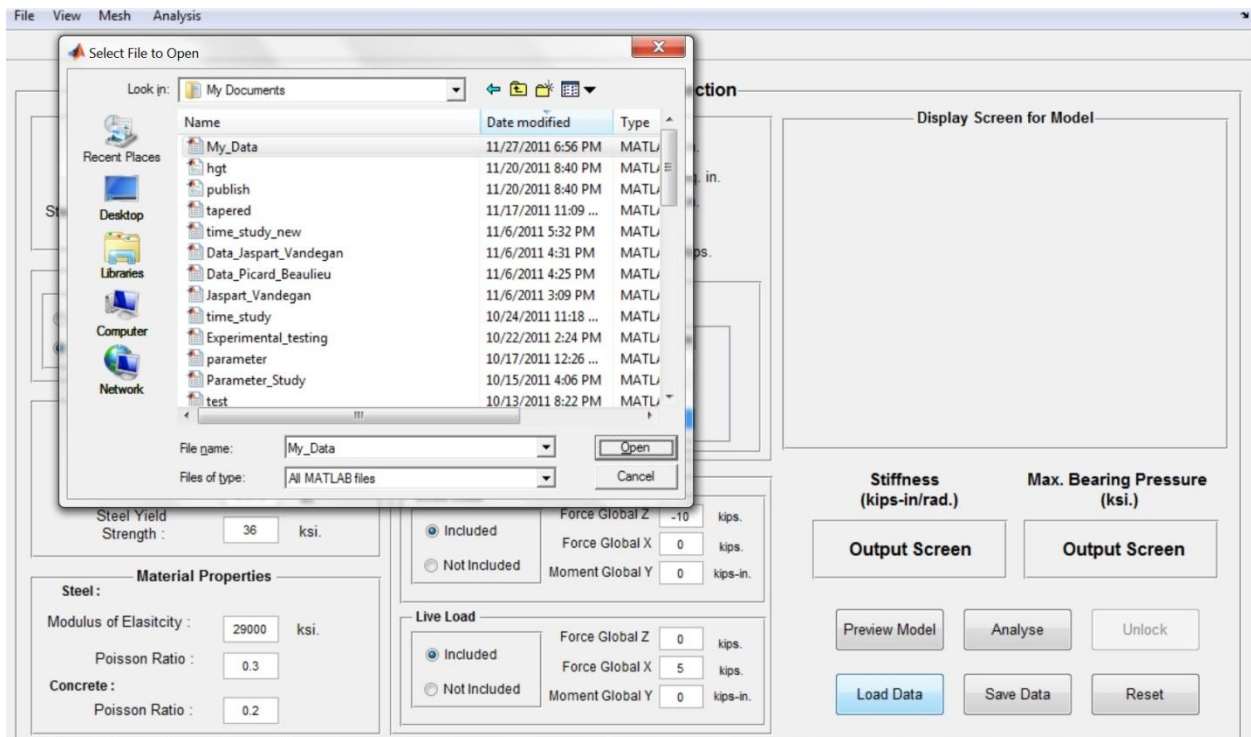


Figure C.59. Loading earlier saved data in the Wizard

## **C.4.2. Example 2**

This example shows how to calculate the rotational stiffness of a tapered column-base plate connection and maximum bearing pressure on the base plate by using the GUI mode of this wizard for any applied loads. The various features of the wizard that are demonstrated in this example are following.

- Tapered Section Modeling
- Grout Modeling
- Coarse Meshing
- SAP2000

### **Geometry and Material Used**

In this example, a tapered column section is modeled. The column section has a flange width of 6 inches, a flange thickness of 0.5 inch, a web thickness of 0.3 inches and top and bottom total depth of 12 inches and 6 inches respectively. Length of the column stub is taken as 2 feet. The base plate connected to column stub has a length of 14 inches, a width of 10 inches and a thickness of 0.375 inch. The grout provided underneath the base plate has a thickness of 0.8 inch and a compressive strength of 4 ksi. The bolts have a diameter of 1/2 inch and the anchor rod length is taken as 15 inches; which are the default values of these parameters. The positions of anchor bolts on the base plate are shown in Figure C.60. Material is steel for the column, base plate and bolts, which has a modulus of elasticity 29000 ksi and a poisson ratio of 0.3. Yield strengths for the steel used for the column and base plate are 50 ksi and 36 ksi respectively.

### **Applied Loads**

For this example, coarse meshing is chosen as meshing type from 'Mesh' sub-menu and SAP2000 is chosen to be the analysis engine from 'Analysis' sub-menu. A lateral load of 5 kips and an axial compressive load of 10 kips is applied at the column top. Note that compressive load must be entered with a negative sign.

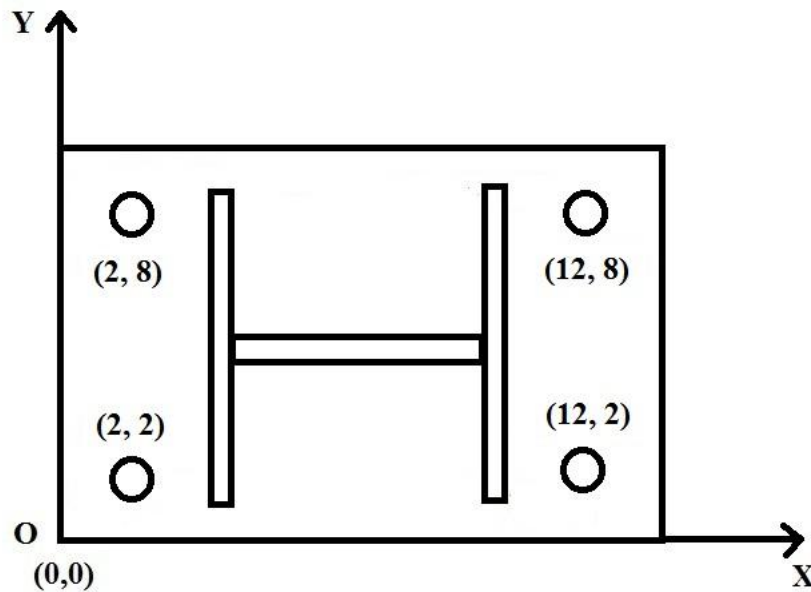


Figure C.60. Positions of Bolts on the Base-Plate

By clicking on 'Preview Model' button, user can view the model of the connection on display screen. User can also switch between different views to check model by selecting the desired view angle from the 'View' submenu. Note that the model showed on the display screen is the one which is analyzed by either SAP2000 or OpenSees on clicking 'Analysis' button. User can always change the configuration of the connection before analysis by changing the values in appropriate fields of GUI. However if the analysis is already performed, then user has to click the 'Unlock' button to make any changes in the model. Following are the step by step instructions to use this wizard to get the stiffness and bearing pressure of the above defined configuration of the connection.

- Choose 'Tapered section' from the W Section type drop down list box present in 'Column Properties' input panel as shown in Figure C.61.
- In 'Tapered Section' GUI which is shown in Figure C.62, input the section properties as described above and click on generate section to see the preview of the section. Press 'OK' button to return to the main wizard window to enter rest of the connection details.

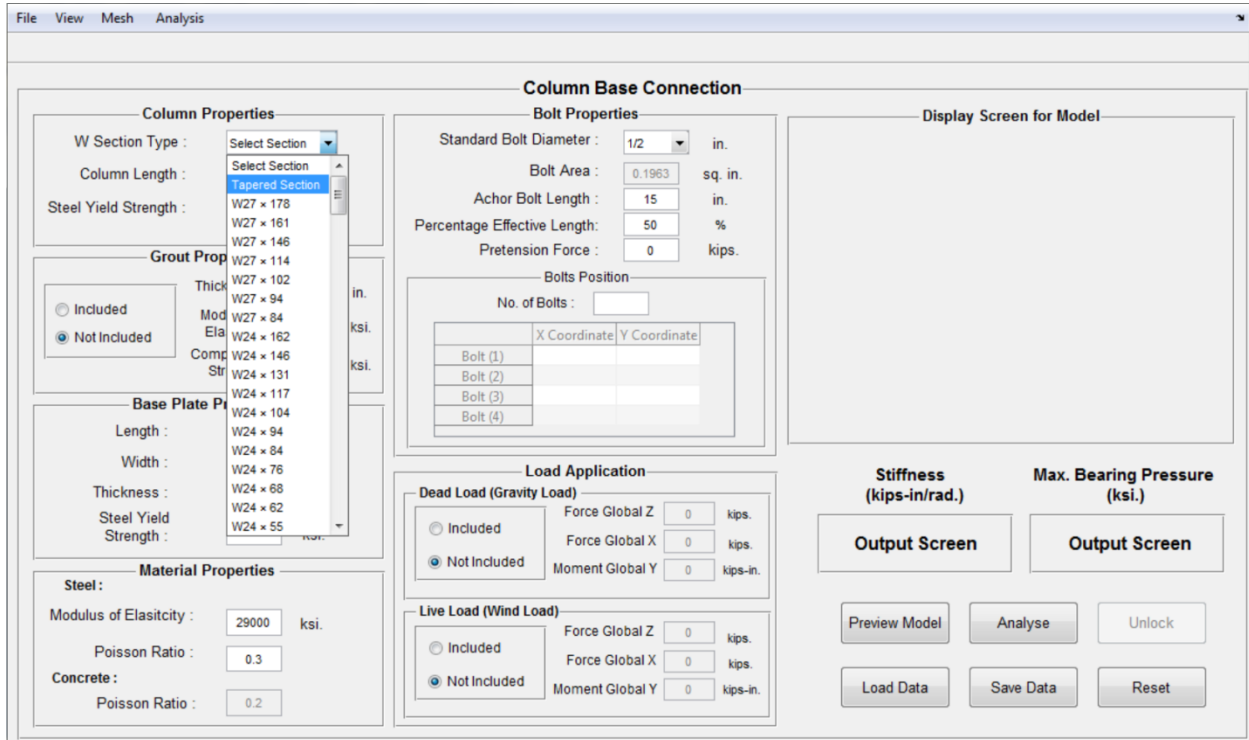


Figure C.61. Tapered Section is chosen from W Section Type list box to model Tapered Section

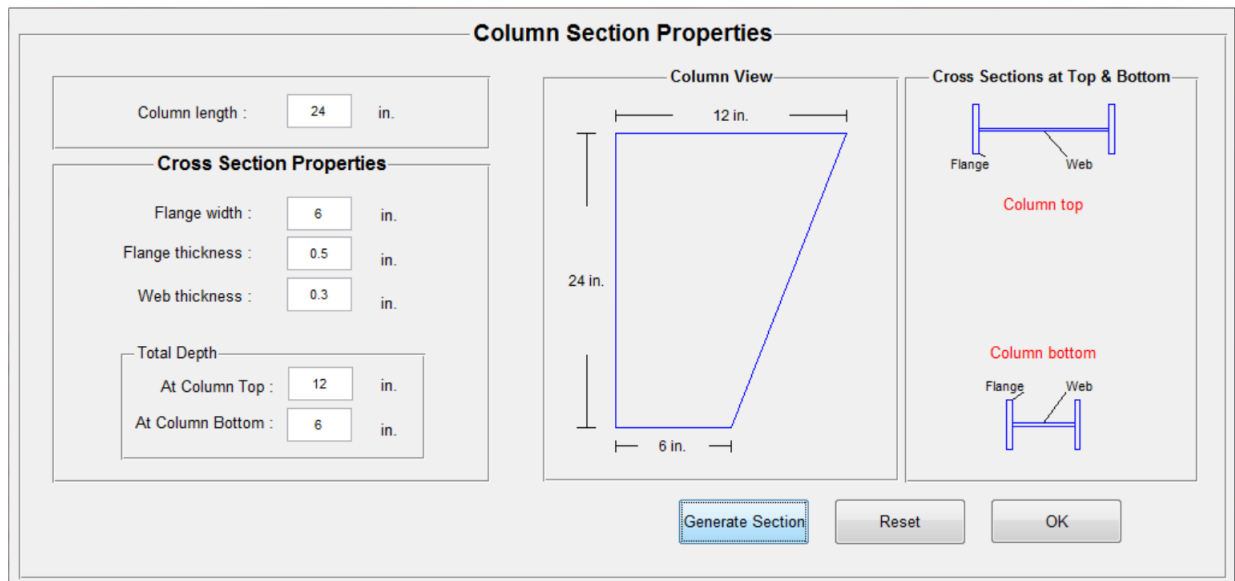


Figure C.62. GUI for creating desired tapered section for the Column Stub

- Input the column stub length and yield strength of steel desired in the column properties input panel. In this example, default values of 24 inches and 50 ksi are chosen for both the quantities.

- Check on 'Included' radio button present in 'Grout Properties' panel to include the grout in the connection. Enter a thickness of 0.8 inch in the 'Thickness' input field. For this example the modulus of elasticity and compressive strength of concrete used for the grout are taken as default values, 3615 ksi. and 4 ksi., respectively.
- Enter values for the parameters present in 'Base Plate Properties' panel. For this example, a length of 12 inches, a width of 8 inches, a thickness of 0.375 inch and a yield strength of 36 ksi are taken, which are default values of these parameters in the wizard.
- 'Material Properties' panel is provided with default properties for the steel and concrete. User can change these properties as desired. Note that 'Poisson Ratio' field for concrete will remain disabled unless the user decides to include the grout in the model.
- The anchor bolts have lots of parameters, which can be entered by the user as desired. From 'Standard Bolt Diameter' drop down list box, a desired standard bolt diameter can be selected. If the bolts used are not standard sized AISC bolts, then the user can also enter any other bolt diameter in the form of its area. A length for anchor rod and an effective percentage for that embedded length can also be entered. In this example, a standard diameter of 1/2 inch, an anchor bolt length of 15 inches and percentage effective length of 50 % are considered, which are default values of these parameters.
- Pretension in bolts can be provided in the pretension force input field present in 'Bolt Properties' input panel. In this example, there is no pretension provided in the bolts.
- The positions of bolts on the base plate can be provided in bolt position table. Input the number of bolts and press enter; it will enable the table and coordinates of bolts can be entered as shown in Figure C.63. Note that the minimum number of bolts that can be entered is one; however there is no such limitation on the maximum number of bolts. The modeling process of the connection is completed with this step.
- In load application input panel, two load cases are provided. Dead loads signify the loads on the connection which are due to gravity loads of rafters and other structural members. Live loads signify the loads acting mostly due to wind in this wizard. In this example, enter 10 kips of gravity load in Global Z direction of 'Dead Load' panel with a negative sign and 5 kips of lateral load due to wind in Global X direction of 'Live Load' panel.
- After entering values for all the parameters described in the above steps, model is all set to be analyzed. Click on 'Preview Model' button to see the preliminary model, which is going to be analyzed in selected analysis engine. If something needs to be changed, user can change it and again click on 'Preview Model' to see the changes. For this example, the model looks like Figure C.63.
- Before proceeding further, user can choose between specific analysis options. First option is the kind of meshing desired, either coarse or fine. The second option is analysis engine desired, either SAP2000 or OpenSees. The meshing type can be changed from the 'Mesh' submenu and a particular analysis engine can be chosen from the 'Analysis' submenu. For this example, the meshing type is chosen as 'Coarse Mesh' and the analysis engine is chosen as SAP2000.



- Model can also be viewed from different angles by exploring options provided in 'View' submenu.
- At this step, we are all set to perform the analysis. Click on 'Analyse' button, it takes few seconds to connect to the selected analysis engine and perform the analysis. The results or value for the stiffness of the connection and the maximum bearing pressure are displayed on the output screens present just below the display screen (Figure C.64).

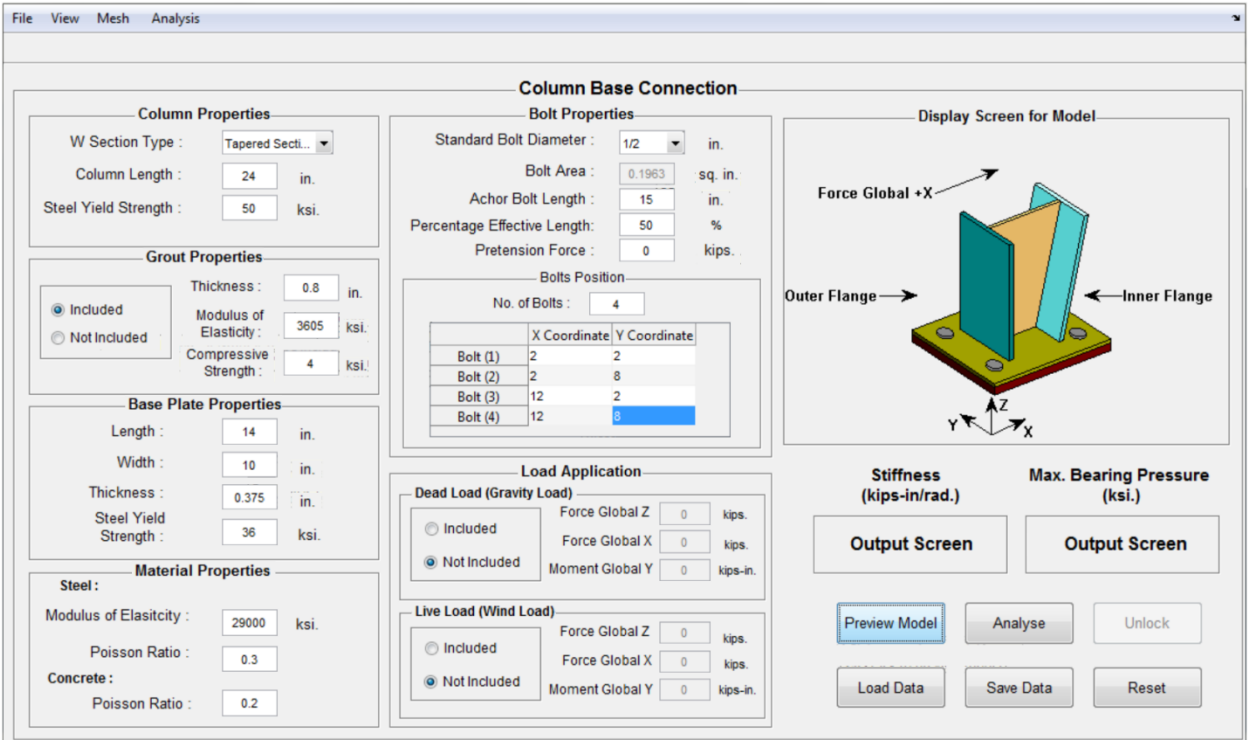


Figure C.63. Display Screen showing the preview of the Modeled Connection

- Two additional figures along with a report appear in addition to the obtained results, one shows the moment-rotation curve (Figure C.65) and the other shows the bearing pressure distribution on the base plate (Figure C.66).
- The report which appears on the screen summarizes all the inputs and outputs of a particular analysis. The report can be printed or saved as per the details provided in Section C.2.2.4.3.

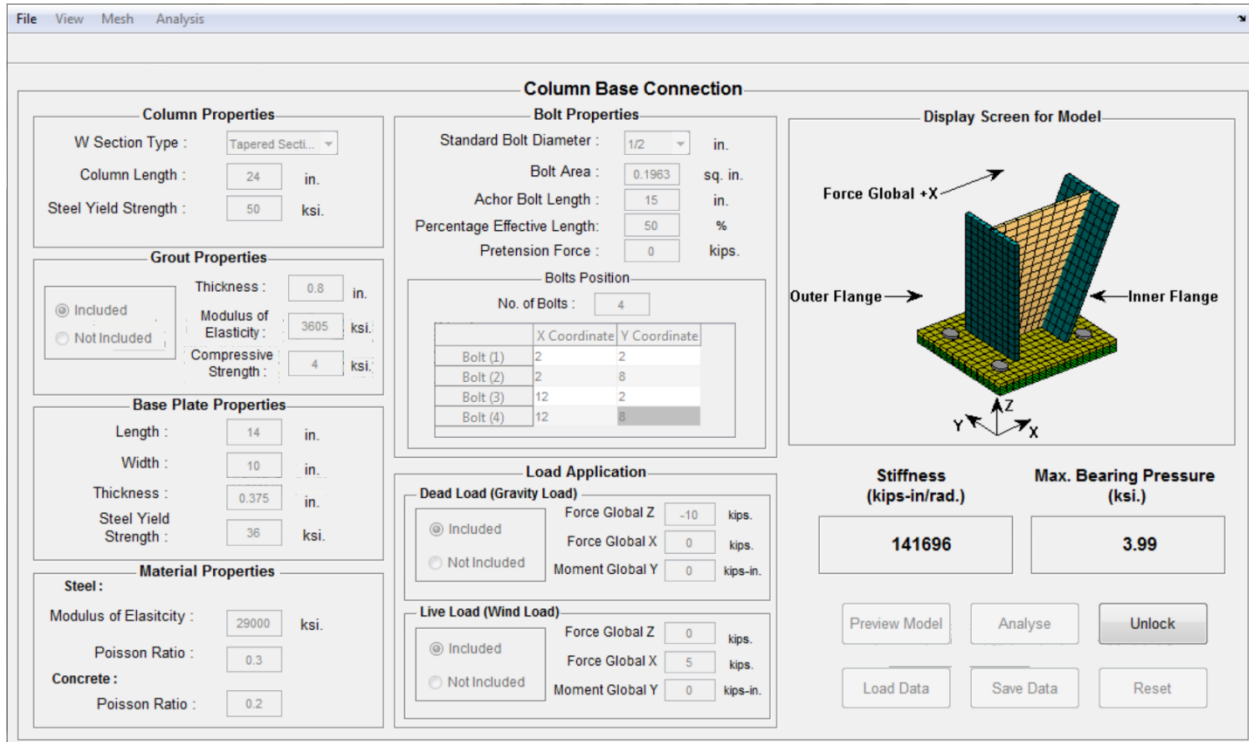


Figure C.64. Wizard Window after performing the analysis and giving back the results

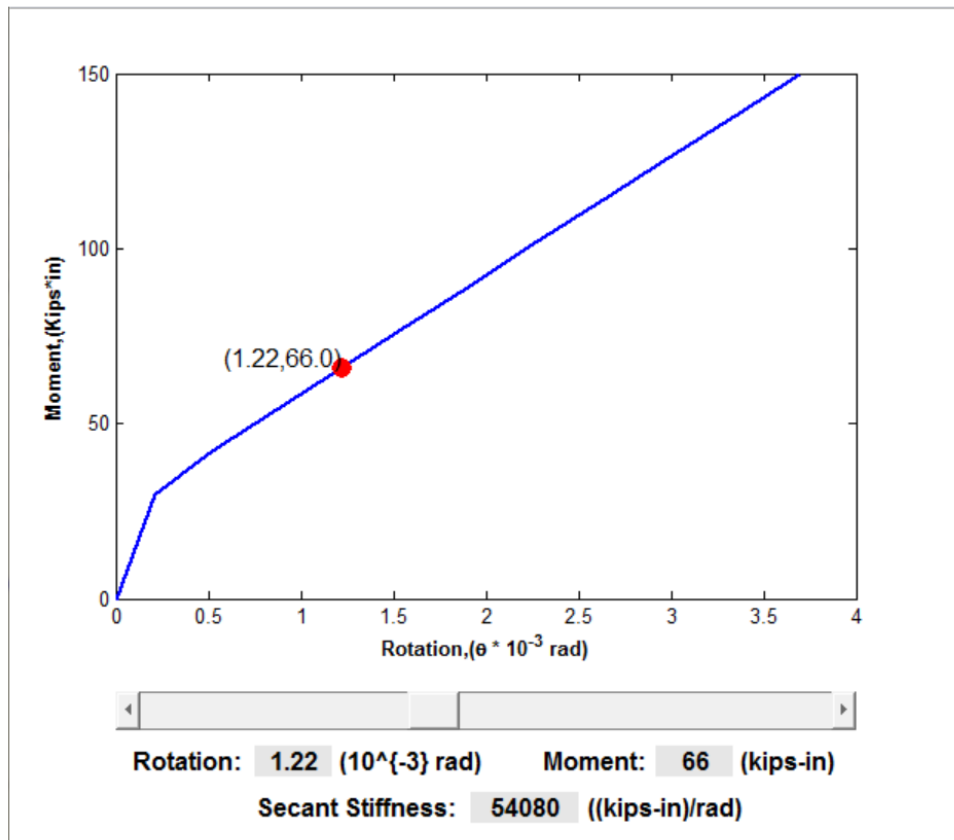


Figure C.65. Moment-Rotation for Tapered Column Stub plotted by the Wizard

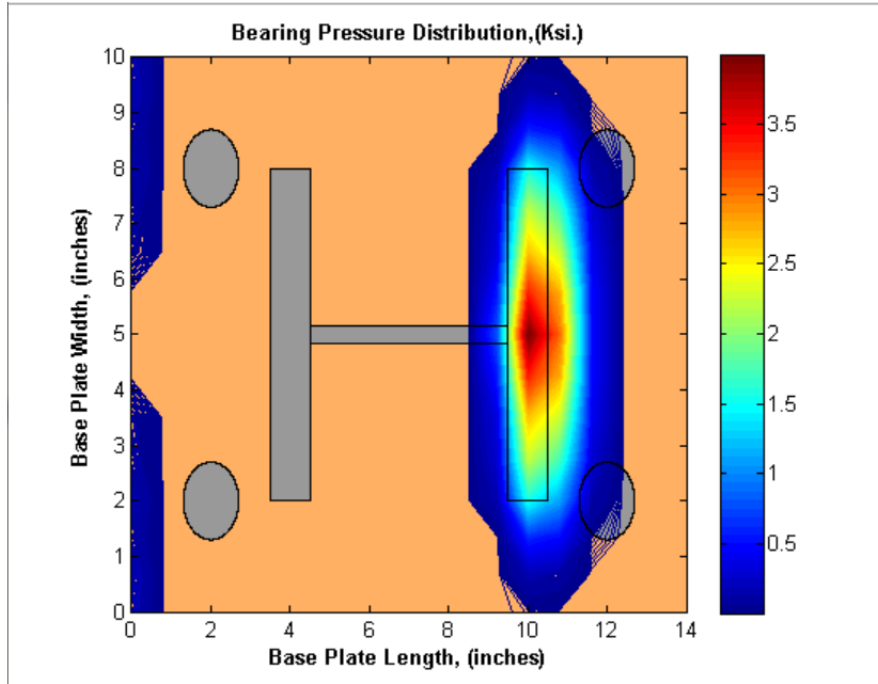


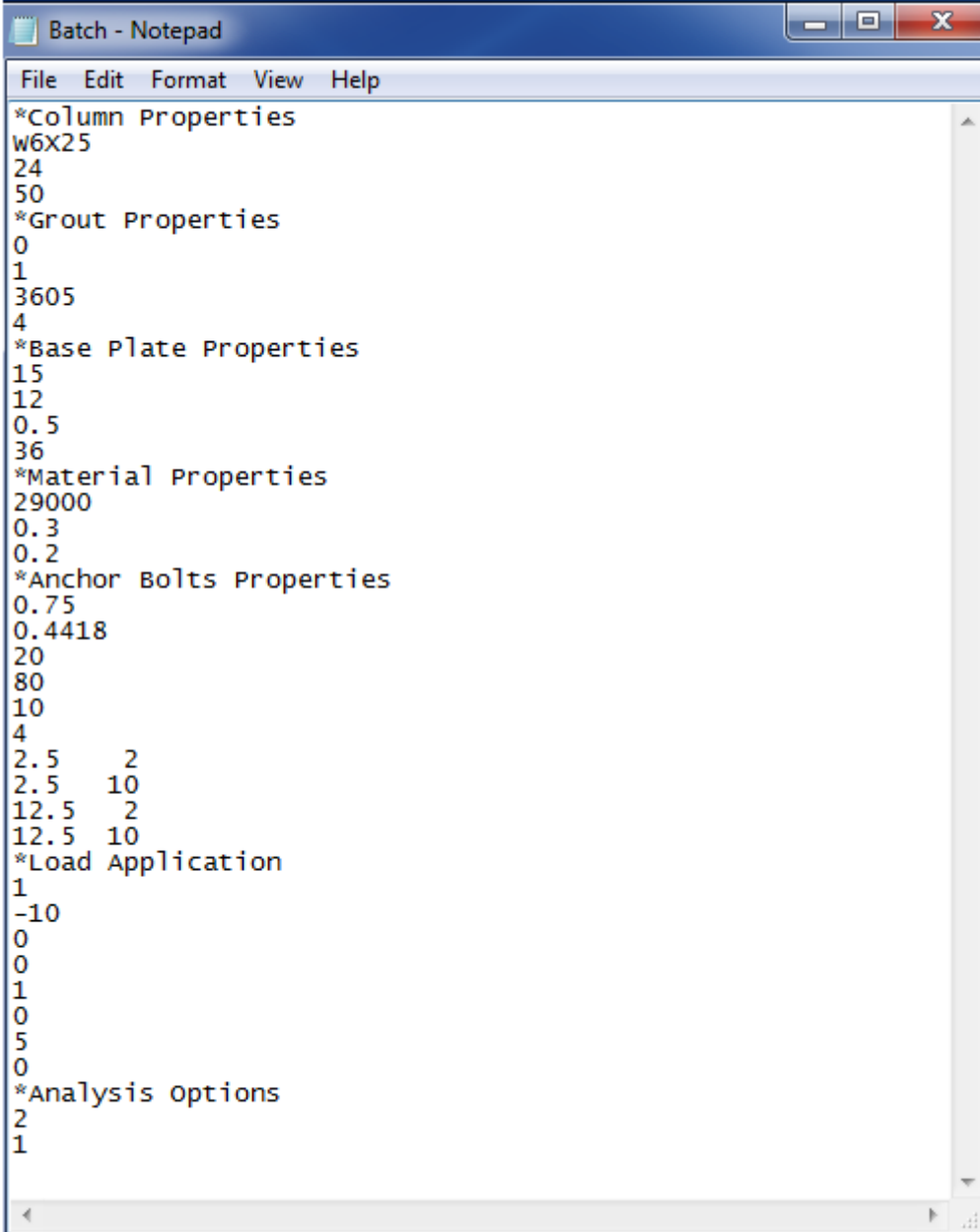
Figure C.66. Bearing Pressure Distribution on the Base Plate plotted by the Wizard

- The user can save the above mentioned plots for future references. At this point, the user can't perform another analysis by making some changes. At this point, the user can't perform another analysis by making some changes. The model has to be unlocked by clicking 'Unlock' button to enable analysis button again.
- The user can save the values entered by either clicking on 'Save Data' button or by going into 'File' submenu and clicking on 'Save As'. The process of saving the data is shown in Figure C.58. The user can also load the saved data by either clicking the 'Load Data' button or by going into 'File' submenu and clicking on 'Open'. The process of accessing the saved data is shown in Figure C.59.
- The 'Reset' button provided on the wizard resets the value in each input field to its default value which is provided by the programmer.

### C.4.3. Example 3

The objective and configuration of the connection selected for this example is exactly same as of the Example 1 (refer Section C.4.1). However, it uses the Batch mode of this wizard for performing the analysis and getting back the results. The various steps that should be followed are mentioned below.

1. The input file *Batch.txt* which is present in the folder 'Batch Mode' is modified and the values of the parameters used in Example 1 are entered (see Figure C.67). After entering all the values, the file is again saved.



```
Batch - Notepad
File Edit Format View Help
*Column Properties
W6X25
24
50
*Grout Properties
0
1
3605
4
*Base Plate Properties
15
12
0.5
36
*Material Properties
29000
0.3
0.2
*Anchor Bolts Properties
0.75
0.4418
20
80
10
4
2.5 2
2.5 10
12.5 2
12.5 10
*Load Application
1
-10
0
0
1
0
5
0
*Analysis Options
2
1
```

Figure C.67. Input file for running batch mode is labeled as *Batch.txt*

2. Click on *batch.exe* to model the connection, perform the analysis for the configuration provided in the input file (see Figure C.68).

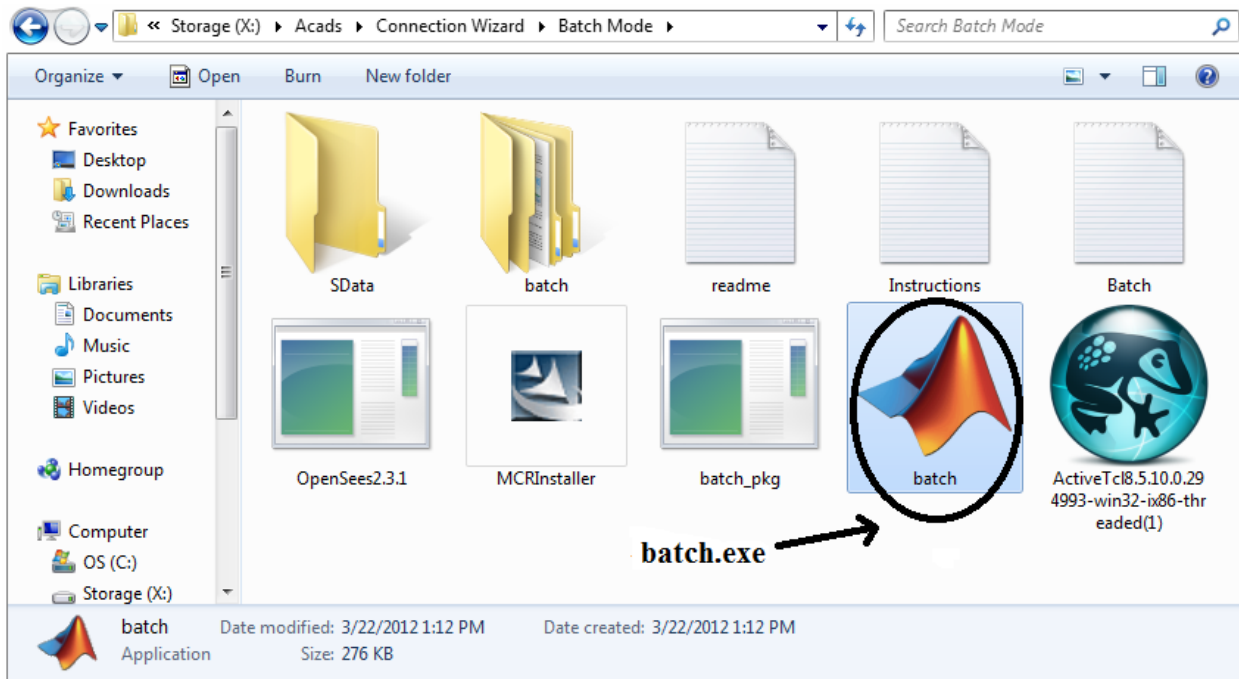


Figure C.68. *batch.exe* is an executable file, which on clicking execute analysis

3. As a result of clicking *batch.exe*, an output file labeled as *Results.txt* appears in 'Batch Mode' folder. Click on the file to check the results (see Figure C.70).

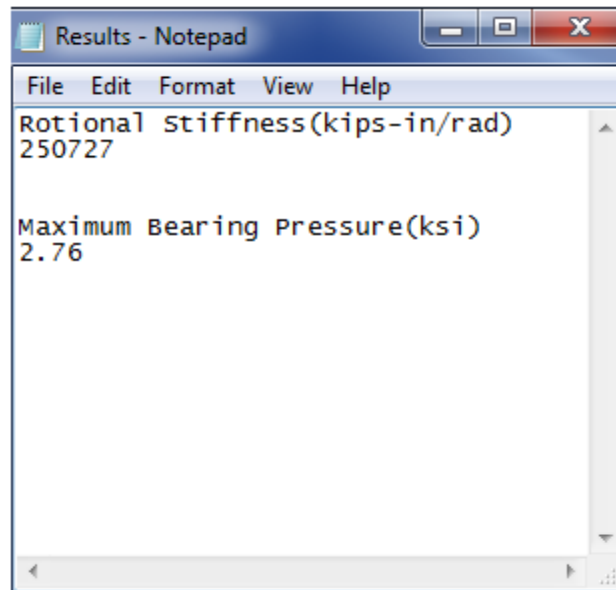


Figure C.69. Output file, *Results.txt*

4. The results are exactly same as obtained in Example 1.