COMPUTER PROGRAM GROUP
Version 2022

A Program for the Analysis of a Group of Piles Subjected to Vertical and Lateral Loading
(User's Manual)

by

Lymon C. Reese
Shin Tower Wang
Jose A. Arrellaga
Joe Hendrix
Luis Vasquez

for

ENSOFT, INC
3003 W. Howard Lane
Austin, Texas 78728

February - 2022

PROGRAM LICENSE AGREEMENT

IMPORTANT NOTICE: *Please read the terms of the following license agreement carefully. You signify acceptance of the terms of this Agreement by using the product.*

This software is licensed only to the user (company or individual) whose name is registered with ENSOFT, INC., or to users at the registered location, on only one computer at a time. Additional installations of the software may be made by the user, as long as the number of installations is equal to the number of purchased and registered licenses.

Users that purchased a network version of this software product are entitled to install the software on all computers in the network at their registered location. However, the software can only be used simultaneously by as many users as the total number of purchased and registered licenses.

The user is not entitled to copy this software unless copying for backup purposes. The user may not loan, rent, lease, or transfer this software to any other person, company, or location. This software and documentation are copyrighted materials and should be treated like any other copyrighted material (e.g. a book or musical recording). This software is protected by United States copyright law and the International Copyright Treaty.

Copyright ® 1987, 1994, 1996, 2000, 2003, 2006, 2010, 2012, 2013, 2014, 2015, 2016, 2018, 2019, 2022 by ENSOFT, INC. All rights reserved.

Printed in the Unites States of America. Except as permitted under the United States Copyright Act of 1976, no part of this publication may be reproduced, translated, or distributed without the prior written approval of ENSOFT, INC.

Although the program has been used with apparent success in many analyses, new information is developed continuously and new or updated versions may be written from time to time. All users are requested to inform ENSOFT, INC. immediately of any possible errors that are found in the coding. As errors are detected, revised versions will be available for all who have purchased the program on the ENSOFT, INC website as a download.

No warranty, expressed or implied, is offered on the accuracy of the results from the program. The user relies on the program at his or her own risk. The program should not be used for design unless caution is exercised in interpreting the results and independent calculations are available to verify the general correctness of the results. The user is assumed to be knowledgeable of the information that is distributed with the digital media. The user is assumed to recognize that the input parameters, e.g., soil properties, increment length, tolerance on solution convergence, and many others, can have a significant effect on the solution and must be chosen carefully. The user should have a thorough understanding of the relevant theoretical criteria (appropriate references are suggested in the software documentation). The information presented herein is not intended or represented to be suitable for direct use in engineering design without additional engineering evaluation.

FOREWORD

Analyzing a group of piles that support a cap or a structure in a system subjected to axial and lateral loading is an important problem in soil-structure interaction. The piles may be of different sizes and may be installed vertically or on a batter. The piles are assumed to be far enough apart so that there is no significant pile-soil-pile interaction or, if the piles are spaced closely together, such interaction is reflected by an appropriate modification of soil resistance curves.

Computer program GROUP allows the user to analyze the behavior of pile groups using either a 2-dimensional model or a 3-dimensional model.

The procedures for analyzing a single pile subjected to lateral loading are included in GROUP and the user should be familiar with that technology. A complete discussion of the method of analysis of a single pile under lateral loading is given in the documentation for LPILE, another program distributed by ENSOFT, Inc.

Table of Contents

Table of Contents v
List of Figures xviii
List of Tables xxxi
CHAPTER 1. Introduction 1-1
1.1 General Description 1-2
1.2 Program Features and Version History 1-2
1.2.1 GROUP 1.0 (1987) 1-2
1.2.2 GROUP 2.0 (1990) 1-2
1.2.3 GROUP 3.0 (1994) 1-3
1.2.4 GROUP 4.0 (1996) 1-5
1.2.5 GROUP 5.0 (2000) 1-5
1.2.6 GROUP 6.0 (2003) 1-6
1.2.7 GROUP 7.0 (2006) 1-6
1.2.8 GROUP 8.0 (2010) 1-7
1.2.9 GROUP 2013 (2013) 1-8
1.2.10 GROUP 2014 (2014) 1-8
1.2.11 GROUP 2016 (2016) 1-9
1.2.12 GROUP 2019 (2019) 1-9
1.2.13 GROUP 2022 (2022) 1-10
1.3 Technical Support 1-10
1.3.1 Support by Telephone 1-10
1.3.2 Upgrade Notifications and Internet Site 1-11
1.3.3 Software Customization Services 1-11
1.3.4 Changes to Support Policy 1-11
CHAPTER 2. Installation and Getting Started 2-1
2.1 Installation Procedures 2-2
2.1.1 Single User License 2-3

2.1.2 Network License 2-3
2.1.2.1 Installation to a Server 2-3
2.1.2.2 Installation to Clients 2-4
2.2 Software Updates 2-4
2.3 Versions Available 2-4
2.4 Getting Started 2-5
2.4.1 Program Start 2-7
2.4.2 File 2-9
2.4.3 Data 2-10
2.4.4 Edit 2-15
2.4.5 Options 2-19
2.4.6 View 2-23
2.4.7 Computation Options 2-25
2.4.8 Graphs Menu 2-28
2.4.9 Plot Menu 2-33
2.4.10 Show 3D View Menu 2-35
2.4.11 3D View Results 2-38
2.4.12 Window Menu 2-41
2.4.13 Help Menu 2-41
CHAPTER 3. References for Data Input 3-1
3.1 Numeric Data Entries 3-2
3.2 File Menu 3-3
3.2.1 File - New 3-4
3.2.2 File - New from Template 3-4
3.2.3 File - New from Template – Flexible Cap Version 3-11
3.2.4 File - Open 3-21
3.2.5 File - Save 3-23
3.2.6 File - Save As 3-23
3.2.7 File - Exit 3-25
3.2.8 File - Recent Files 3-25
3.2.9 File - Save Bitmap (3D View) 3-25
3.2.10 File - Page Setup (3D View) 3-26

3.2.11 File - Print (3D View) 3-27
3.3 Data Menu 3-28
3.3.1 Data - Title 3-31
3.3.2 Data - Pile Cross Sections 3-32
3.3.2.1 Cross Sections Definition Window 3-32
3.3.2.2 Edit Cross Section Window (Elastic Section) 3-34
3.3.2.3 Edit Cross Section Window (NonLinear Section) 3-46
3.3.3 Data - Pile Properties 3-63
3.3.4 Data - Pile Group Properties 3-65
3.3.4.1 2D Layout 3-65
3.3.4.2 3D Layout 3-69
3.3.5 Data - Pile-Head Coordinates 3-70
3.3.5.1 2D Layout 3-71
3.3.5.2 3D Layout 3-75
3.3.6 Data - Pile-Cap Options 3-81
3.3.7 Data – Node Data (Pile-Cap)– Flexible Cap Version 3-85
3.3.7.1 Data – Node Data (Pile-Cap) - Node Coordinates – Flexible Cap Version 3-86
3.3.8 Data – Plate Data (Pile-Cap) – Flexible Cap Version 3-87
3.3.8.1 Data – Plate Data (Pile-Cap) – Plate Connectivity – Flexible Cap Version 3-87
3.3.8.2 Data – Plate Data (Pile-Cap) – Plate Offsets – Flexible Cap Version 3-90
3.3.8.3 Data – Plate Data (Pile-Cap) – Plate Sections – Flexible Cap Version 3-92
3.3.9 Data – Beam Data (Pile-Cap) – Flexible Cap Version 3-93
3.3.9.1 Data – Beam Data (Pile-Cap) – Beam Connectivity – Flexible Cap Version 3-94
3.3.9.2 Data – Beam Data (Pile-Cap) – Beam Offsets – Flexible Cap Version 3-96
3.3.9.3 Data – Beam Data (Pile-Cap) – Beam End Zones – Flexible Cap Version 3-98
3.3.9.4 Data – Beam Data (Pile-Cap) – Beam Section Rotation – Flexible Cap Version 3-100
3.3.10 Data – Springs (Pile-Cap) – Flexible Cap Version 3-102
3.3.10.1 Data – Springs (Pile-Cap) – Concentrated Springs at Nodes – Flexible Cap Version 3-102
3.3.10.2 Data – Springs (Pile-Cap) – Distributed Springs on Plates – Flexible Cap Version . 3-104
3.3.10.3 Data – Springs (Pile-Cap) – Distributed Springs on Beams – Flexible Cap Version 3-105
3.3.11 Data - Loading 3-107

3.3.11.1 Loading Definition Window 3-108
3.3.11.2 Edit Load Case Window....3-110
3.3.11.3 Edit Concentrated Loads at Pile Cap Window....3-116
3.3.11.4 Concentrated Load Window 3-117
3.3.11.5 Specified Displacements/Loads Window....3-120
3.3.11.6 Displacement at Nodes Window -Flexible Cap Version....3-122
3.3.11.7 Concentrated Loads at Nodes Window - Flexible Cap Version....3-124
3.3.11.8 Distributed Loads on Plates Window - Flexible Cap Version 3-126
3.3.11.9 Distributed Loads on Beams Window - Flexible Cap Version 3-127
3.3.11.10 Concentrated Loads on Beams Window - Flexible Cap Version 3-129
3.3.11.11 Distributed Lateral Loads Window....3-131
3.3.11.12 Soil Movement Window....3-135
3.3.11.13 Incremental/PushOver Analysis Options Window....3-137
3.3.11.14 Foundation Stiffness Options Window 3-139
3.3.11.15 Analysis Options Window 3-154
3.3.11.16 Output Variables Window....3-156
3.3.11.17 Load Combinations Window....3-158
3.3.11.18 Edit Load Combination Window 3-162
3.3.12 Data - Soil Layers....3-164
3.3.13 Data - Set of Soil Resistance (p-y) Curves 3-4
3.3.14 Data - Sets of Axial Load-Displacement Curves 3-7
3.3.15 Data - Sets of Torque-Rotation Curves 3-10
3.3.16 Data - Sets of Nonlinear EI 3-12
3.3.17 Data - Sets of Red. Factor Lat. Load-Along Depth....3-15
3.3.18 Data - Sets of Soil Movement....3-18
3.3.19 Data - Sets of Distributed Loads....3-20
3.3.20 Data- Sets of Data to Print p-y Curves 3-22
3.3.21 Data – Import Soil Layers from an EnCPT Soil File 3-25
3.3.22 Data – Import Soil and Section Data from LPILE Model....3-26
3.3.23 Data – Import p-y Curves Generated by LPILE....3-28
3.3.24 Data - Import Nonlinear EI Data Generated by LPILE 3-30
3.3.25 Data - Import Group Layout from Excel File 3-32

3.3.26 Data - Import Loading from Excel File....3-32
3.3.27 Data - Export Group Layout to Excel File....3-32
3.3.28 Data - Export Loading to Excel File....3-33
3.4 Edit Menu (Editing Objects Using Graphics Tools)....3-35
3.4.1 Edit - Select Piles....3-37
3.4.2 Edit - Select Mode - Flexible Cap Version....3-39
3.4.3 Edit - Select Plates - Flexible Cap Version....3-39
3.4.4 Edit - Select Beams - Flexible Cap Version....3-41
3.4.5 Edit - Select Nodes - Flexible Cap Version....3-42
3.4.6 Edit – Pause Select....3-45
3.4.7 Edit -Add Pile....3-45
3.4.8 Edit - Pile Layout....3-48
3.4.9 Edit - Delete Piles....3-50
3.4.10 Edit - Move Piles....3-50
3.4.11 Edit - Copy Piles....3-52
3.4.12 Edit - Paste Pile....3-52
3.4.13 Edit - Paste Array....3-53
3.4.14 Edit -Add Plate - Flexible Cap Version....3-56
3.4.15 Edit – Edit Plate....3-57
3.4.16 Edit - Delete Plates (Flexible Cap Version)....3-58
3.4.17 Edit - Divide Plates - Flexible Cap Version....3-59
3.4.18 Edit -Add Beam - Flexible Cap Version....3-60
3.4.19 Edit – Edit Beam....3-61
3.4.20 Edit - Delete Beams (Flexible Cap Version)....3-63
3.4.21 Edit - Divide Beams - Flexible Cap Version....3-64
3.4.22 Edit -Add Node - Flexible Cap Version....3-65
3.4.23 Edit – Edit Node....3-66
3.4.24 Edit - Delete Nodes - Flexible Cap Version....3-67
3.4.25 Edit - Delete Unconnected Nodes - Flexible Cap Version....3-68
3.4.26 Edit – Move All Selected – Flexible Cap Version....3-69
3.4.27 Edit – Move All Selected – Flexible Cap Version....3-70
3.4.28 Edit - Delete All Selected - Flexible Cap Version....3-71

3.4.29 Edit - Pile Sets....3-71
3.4.29.1 Pile Sets Definition Window....3-71
3.4.29.2 Edit Pile Set Window....3-73
3.4.30 Edit - Plate Sets - Flexible Cap Version....3-74
3.4.30.1 Plate Sets Definition Window....3-74
3.4.30.2 Edit Plate Set Window....3-76
3.4.31 Edit - Beam Sets - Flexible Cap Version....3-77
3.4.31.1 Beam Sets Definition Window....3-77
3.4.31.2 Edit Beam Set Window....3-79
3.4.32 Edit - Node Sets - Flexible Cap Version....3-80
3.4.32.1 Node Sets Definition Window....3-80
3.4.32.2 Edit Node Set Window....3-82
3.5 Options Menu....3-84
3.5.1 Options - Units....3-86
3.5.2 Options – Use Kips as Unit of Force....3-87
3.5.3 Options - Analysis Mode....3-87
3.5.4 Options – Pile Cap Type - Flexible Cap Version....3-88
3.5.5 Options – Adjust Depth for Batter Piles....3-89
3.5.6 Options – Generate Axial L-D (and T-R) curves on Soil Profile....3-89
3.5.7 Extend Interpolation for L-D (and T-R) curves....3-89
3.5.8 Use Layering Correction (Method of Georgadis)....3-89
3.5.9 Do not use Layering Correction is soil above is of same type....3-90
3.5.10 Options - Modify Group Layout using Template....3-90
3.5.11 Options - Modify Group Layout using Template – Flexible Cap Version....3-91
3.5.12 Options - Modify Flexible Cap Layout using Template - Flexible Cap Version....3-93
3.5.13 Options – Shift Piles and Soil Elevations....3-94
3.5.14 Options - Enter User-Defined Soil-Resistance (p-y) Curves....3-96
3.5.15 Options - Enter User-Defined Load-Displacement Curves....3-96
3.5.16 Options - Enter User-Defined Torque-Rotation Curves....3-96
3.5.17 Options – Consider Non-Symmetric Sections for Nonlinear EI....3-96
3.5.18 Options - Reduction Factors for Lateral Loads....3-97
3.5.19 Options - Reduction Factors for Lateral Loads Along Depth....3-98

3.5.20 Options - Reduction Factors for Axial Loads 3-98
3.5.21 Options - Print Generated p-y Curves 3-99
3.5.22 Options - Default Control Options 3-99
3.5.23 Options - Default Output Variables 3-102
3.5.24 Options - Text Editor Options 3-104
3.5.25 Options - Check for Updates as the Program Starts 3-105
3.5.26 Options - Include Load Case/Comb Number on Graphs 3-105
3.5.27 Options - Limit Active Curves on Graphs 3-106
3.6 View Menu 3-107
3.6.1 View - Pile Top 3-109
3.6.2 View - 3D View 3-110
3.6.3 View - XY View (Front) 3-111
3.6.4 View - XY View (Back) 3-111
3.6.5 View - XZ View (Right) 3-112
3.6.6 View - XZ View (Left) 3-112
3.6.7 View - YZ View (Top) 3-113
3.6.8 View - YZ View (Bottom) 3-113
3.6.9 View - XYZ View (Iso) 3-114
3.6.10 View - XYZ View (Reversed) 3-114
3.6.11 View - Panning 3-115
3.6.12 View - Zooming 3-115
3.6.13 View - Zoom Window 3-116
3.6.14 View - Rotating 3-116
3.6.15 View - Rotation Values 3-117
3.6.16 View - Zoom In 3-117
3.6.17 View - Zoom Out 3-118
3.6.18 View – Previous View 3-118
3.6.19 View – Next View 3-118
3.6.20 View - Restart View 3-119
3.6.21 View - Refresh View 3-119
3.6.22 View - General Toolbar 3-120
3.6.23 View - Data Input Toolbar 3-120

3.6.24 View - Edit Toolbar 3-121
3.6.25 View - Options Toolbar 3-122
3.6.26 View - View Toolbar 3-123
3.6.27 View - Transform Toolbar 3-123
3.6.28 View - Computations Toolbar 3-124
3.6.29 View - Output Toolbar 3-124
3.6.30 View - Plots Toolbar 3-126
3.6.31 View - Help Toolbar 3-127
3.7 Show 3D View Menu 3-127
3.7.1 Show 3D View -Piles 3-130
3.7.2 Show 3D View -Show Only 1 Row 3-130
3.7.3 Show 3D View -Pile Numbers 3-131
3.7.4 Show 3D View -Pile Properties 3-132
3.7.5 Show 3D View -Pile Wire/Pile Shade 3-133
3.7.6 Show 3D View -Pile Local Axes 3-134
3.7.7 Show 3D View –Pile Head Connections 3-135
3.7.8 Show 3D View -Pile Cap 3-136
3.7.9 Show 3D View – FEM Nodes - Flexible Cap Version 3-137
3.7.10 Show 3D View – Node Numbers - Flexible Cap Version 3-138
3.7.11 Show 3D View – FEM Edges - Flexible Cap Version) 3-139
3.7.12 Show 3D View – FEM Plates - Flexible Cap Version 3-140
3.7.13 Show 3D View – Plate Thickness - Flexible Cap Version 3-141
3.7.14 Show 3D View – Plate Numbers - Flexible Cap Version 3-142
3.7.15 Show 3D View – FEM Beams - Flexible Cap Version 3-143
3.7.16 Show 3D View – Shade beams - Flexible Cap Version 3-144
3.7.17 Show 3D View – Beam Numbers - Flexible Cap Version 3-145
3.7.18 Show 3D View – Beam Local Axes - Flexible Cap Version 3-146
3.7.19 Show 3D View – Plate/Beam Offsets - Flexible Cap Version 3-147
3.7.20 Show 3D View – Beam Local Axes - Flexible Cap Version 3-148
3.7.21 Show 3D View – Concentrated Springs at Nodes (Flexible Pile Cap) 3-149
3.7.22 Show 3D View - Distributed Springs on Plates (Flexible Pile Cap) 3-150
3.7.23 Show 3D View - Distributed Springs on Beams (Flexible Pile Cap) 3-151

3.7.24 Show 3D View -Soil Layers 3-152
3.7.25 Show 3D View -Layer Labels 3-153
3.7.26 Show 3D View -Layer Depth Labels 3-154
3.7.27 Show 3D View - Concentrated Loads at Pile Cap 3-155
3.7.28 Show 3D View - Distributed Loads on Piles 3-156
3.7.29 Show 3D View – Specified Displacements at Nodes (Flexible Pile Cap) 3-157
3.7.30 Show 3D View – Concentrated Loads at Nodes (Flexible Pile Cap) 3-158
3.7.31 Show 3D View - Distributed Loads on Plates (Flexible Pile Cap) 3-159
3.7.32 Show 3D View - Distributed Loads on Beams (Flexible Pile Cap) 3-160
3.7.33 Show 3D View - Concentrated Loads on Beams (Flexible Pile Cap) 3-161
3.7.34 Show 3D View -Global Axes 3-162
3.7.35 Show 3D View -Vertical Lines 3-163
3.7.36 Show 3D View -Show Loading Options 3-164
3.7.37 Show 3D View -Show Springs Options – Flexible Cap Version 3-166
3.7.38 Show 3D View –Display Values Options 3-167
3.7.39 Show 3D View -Show Grid 3-168
3.7.40 Show 3D View - Grid Options 3-169
3.7.41 Show 3D View – Pile Color 3-170
3.7.42 Show 3D View - Highlight Piles 3-171
3.7.43 Show 3D View - Highlight Plates - Flexible Cap Version) 3-172
3.7.44 Show 3D View - Highlight Beams - Flexible Cap Version) 3-173
3.7.45 Show 3D View - Highlight Nodes - Flexible Cap Version 3-174
3.7.46 Show 3D View - Select Piles 3-175
3.7.47 Show 3D View - Select Plates - Flexible Cap Version 3-176
3.7.48 Show 3D View - Select Beams - Flexible Cap Version 3-177
3.7.49 Show 3D View - Select Nodes - Flexible Cap Version 3-178
3.7.50 Show 3D View -Hide/Show Piles 3-179
3.7.51 Show 3D View -Hide/Show Piles, Plates, Beams - Flexible Cap Version 3-180
3.8 Window Menu 3-181
3.8.1 Window - Cascade 3-181
3.8.2 Window - Arrange Icons 3-181
3.8.3 Window - Windows 3-182

3.9 Help Menu....3-182
3.9.1 Help- Documentation User's Manual....3-183
3.9.2 Help- Documentation Example Manual....3-184
3.9.3 Help- Documentation Technical Manual....3-185
3.9.4 Help- About....3-186
3.9.5 Help- Check for Updates....3-187
3.9.6 Help- Ensoft Key Inquirer....3-188
3.9.7 Help- Check Network Dongle....3-189
CHAPTER 4. References for Program Execution and Output Reviews....4-1
4.1 Introduction....4-2
4.2 Computation Menu....4-2
4.2.1 Computation- Run Analysis....4-4
4.2.2 Computation- Run Moment Curvature Calculations....4-7
4.2.3 Computation- Select Load Case or Combination....4-8
4.2.4 Computation- View Input File....4-9
4.2.5 Computation- View Echo Print File....4-10
4.2.6 Computation- View Processor Run Notes....4-11
4.2.7 Computation- View Output Results File....4-12
4.2.8 Computation- View Summary Results File....4-14
4.2.9 Computation- View Moment Curvature File....4-15
4.2.10 Computation- View Echo File (Flexible Cap) – Flexible Cap Version....4-16
4.2.11 Computation- View Output Results File (Flexible Cap) – Flexible Cap Version....4-17
4.2.12 Computation- View Summary Results File (Flexible Cap) – Flexible Cap Version....4-18
4.2.13 Computation- Special Plots....4-20
4.2.14 Computation- Incremental/PushOver Plots....4-21
4.3 General Comments for Graphics Mode....4-22
4.4 Plot Menu....4-28
4.4.1 Plot - Show Legend....4-29
4.4.2 Plot - Show Markers....4-31
4.4.3 Plot - Font Size....4-33
4.4.4 Plot - Line Width....4-34
4.4.5 Plot - Graph Title....4-35

4.4.6 Plot - Edit Legend 4-36
4.4.7 Plot - Active Curves 4-37
4.4.8 Plot - Print 4-38
4.4.9 Plot - Page Setup 4-39
4.4.10 Plot - Save To Disk 4-40
4.5 Graphs Menu 4-41
4.5.1 Sign Convention for results along the piles 4-43
4.5.2 Graphs- Generated p-y Curves 4-45
4.5.3 Graphs- Generated Axial Load vs. Displacement 4-47
4.5.4 Graphs- Generated Torque vs. Rotation 4-48
4.5.5 Graphs- User Defined p-y Curves 4-49
4.5.6 Graphs- User Defined Axial Load vs. Displacement 4-50
4.5.7 Graphs- User Defined Torque vs. Rotation 4-51
4.5.8 Graphs - Deflection vs. Distance 4-52
4.5.9 Graphs - Moment vs. Distance 4-53
4.5.10 Graphs - Shear vs. Distance 4-54
4.5.11 Graphs - Soil Reaction vs. Distance 4-55
4.5.12 Graphs – Bending Stiffness vs. Distance 4-56
4.5.13 Graphs – Def., Mom., Shear vs. Distance 4-57
4.5.14 Graphs - Deflection y dir vs. Distance 4-58
4.5.15 Graphs - Deflection z dir vs. Distance 4-59
4.5.16 Graphs - Moment z dir vs. Distance 4-60
4.5.17 Graphs - Moment y dir vs. Distance 4-61
4.5.18 Graphs - Shear y dir vs. Distance 4-62
4.5.19 Graphs - Shear z dir vs. Distance 4-63
4.5.20 Graphs - Soil Reaction y dir vs. Distance 4-64
4.5.21 Graphs - Soil Reaction z dir vs. Distance 4-65
4.5.22 Graphs – Bending Stiffness z dir vs. Distance 4-66
4.5.23 Graphs – Bending Stiffness y dir vs. Distance 4-67
4.5.24 Graphs – Def. y, Mom. y, Shear z vs. Distance 4-68
4.5.25 Graphs – Def. z, Mom. y, Shear z vs. Distance 4-69
4.5.26 Graphs – Deflection x Dir vs. Distance 4-70

4.5.27 Graphs – Axial Force vs. Distance 4-71
4.5.28 Graphs - Resultant Deflection vs. Distance 4-72
4.5.29 Graphs - Resultant Moment vs. Distance 4-73
4.5.30 Graphs - Resultant Shear vs. Distance 4-74
4.5.31 Graphs - Resultant Soil Reaction vs. Distance 4-75
4.5.32 Graphs – Res. Def., Mom., Shear vs. Distance 4-76
4.5.33 Graphs - Pile Cap Vertical Load vs. Deflection 4-77
4.5.34 Graphs - Pile Cap Load vs. Lateral Deflection 4-78
4.5.35 Graphs - Pile Cap Load vs. Rotation 4-79
4.5.36 Graphs - Pile Cap Moment vs. Deflection 4-80
4.5.37 Graphs - Pile Cap Moment vs. Rotation 4-81
4.5.38 Graphs - Pile Cap Vertical Load vs. Deflection X 4-82
4.5.39 Graphs - Pile Cap Lateral Load vs. Deflection Y 4-83
4.5.40 Graphs - Pile Cap Lateral Load vs. Deflection Z 4-84
4.5.41 Graphs - Pile Cap Moment vs. Rotation X 4-85
4.5.42 Graphs - Pile Cap Moment vs. Rotation Y 4-86
4.5.43 Graphs - Pile Cap Moment vs. Rotation Z 4-87
4.5.44 Graphs - Pile Cap K11 vs. Deflection X 4-88
4.5.45 Graphs - Pile Cap K22 vs. Deflection Y 4-89
4.5.46 Graphs - Pile Cap K33 vs. Rotation Z 4-90
4.5.47 Graphs - Pile Cap K33 vs. Deflection Z 4-91
4.5.48 Graphs - Pile Cap K44 vs. Rotation X 4-92
4.5.49 Graphs - Pile Cap K55 vs. Rotation Y 4-93
4.5.50 Graphs - Pile Cap K66 vs. Rotation Z 4-94
4.5.51 Graphs – Export Graphs to Excel 4-95
4.5.52 Graphs – Moment Curvature 4-96
4.6 3D View Results Menu 4-97
4.6.1 Show 3D View -Deformed Shape 4-98
4.6.2 Show 3D View -Contour Plot 4-99
4.6.3 Show 3D View -Pile Diagrams 4-100
4.6.4 Show 3D View -Pile Heads Response 4-101
4.6.5 Show 3D View -Pile Top Effects 4-102

4.6.6 Show 3D View -Lateral Effects in Piles 4-104
4.6.7 Show 3D View -Flexible Cap Contour Plot - Flexible Cap Version 4-105
4.6.8 Show 3D View -Flexible Cap Section Cut - Flexible Cap Version 4-106
4.6.9 Show 3D View – Flexible Cap Beam Diagrams - Flexible Cap Version 4-107
4.6.10 Show 3D View -Flexible Cap Node Results - Flexible Cap Version 4-108
4.6.11 Show 3D View -Flexible Cap Plate Results - Flexible Cap Version 4-109
4.6.12 Show 3D View -Flexible Cap Beam Results - Flexible Cap Version 4-110

List of Figures

Figure 1.1 Example of modeling a bridge on piles 1-4
Figure 2.1 Selection of Type of License during installation of the program 2-2
Figure 2.2 Selection of Type of Network Installation 2-3
Figure 2.3 Sample organizational and operational flow chart 2-6
Figure 2.4 Window to personalize the program 2-7
Figure 2.5 Check for Updates Window 2-7
Figure 2.6 Starting window 2-8
Figure 2.7 Options contained in the File Menu (includes 3D View options) 2-9
Figure 2.8 Options contained in the Data Menu 2-11
Figure 2.9 Options contained in the Data Menu – Flexible Cap Version 2-12
Figure 2.10 Options contained in the Edit Menu (3D View Active) 2-15
• Figure 2.11 Options contained in the Edit Menu (3D View Active) – Flexible Cap Version 2-17
Figure 2.12 Options contained in the Options Menu 2-19
Figure 2.13 Options contained in the Options Menu – Flexible Cap Version 2-20
Figure 2.14 Options contained in the View Menu (3D View Active) 2-24
Figure 2.15 Options contained in the Computation Menu 2-25
Figure 2.16 Options contained in the Computation Menu – Flexible Cap Version 2-26
Figure 2.17 Options contained in the Graphs Menu. 2D Model 2-28
Figure 2.18 Options contained in the Graphs Menu. 3D Model 2-29
Figure 2.19 Options contained in the Plot Menu 2-33
Figure 2.20 Options contained in the Show 3D View Menu 2-35
Figure 2.21 Options contained in the Show 3D View Menu – Flexible Cap Version 2-37
Figure 2.22 Options contained in the 3D View Results Menu 2-39
Figure 2.23 Options contained in the 3D View Results Menu – Flexible Cap Version 2-39
Figure 2.24 Options contained in the Windows Menu 2-41
Figure 2.25 Options contained in the Help Menu 2-41
Figure 3.1 Options contained in the File Menu 3-3
Figure 3.2 Window shown after File - New is selected 3-4
Figure 3.3 New from Template Window, 2D rectangular option 3-5
Figure 3.4 New from Template Window, 3D rectangular option 3-6
Figure 3.5 New from Template Window, 3D circular option 3-7
Figure 3.6 Input parameters for pile group generation, rectangular option 3-8
Figure 3.7 Input parameters for pile group generation, circular option 3-9
Figure 3.8 Pile Group Layout using New from Template, 3D rectangular option 3-10

Figure 3.9 Pile Group Layout using New from Template, 3D circular option 3-11
Figure 3.10 New from Template Window, 3D rectangular option – Flexible Cap Version....3-12
Figure 3.11 New from Template Window, 3D circular option – Flexible Cap Version 3-12
Figure 3.12 New from Template Window, 3D rectangular option, Add Plate Elements – Flexible Cap Version 3-13
Figure 3.13 New from Template Window, 3D rectangular option, Add Beam elements – Flexible Cap Version 3-13
Figure 3.14 Input parameters for mesh generation, rectangular option, plate elements – Flexible Cap Version 3-15
Figure 3.15 Input parameters for mesh generation, circular option, plate elements – Flexible Cap Version 3-16
Figure 3.16 Pile Group Layout using New from Template, 3D rectangular option, plate elements – Flexible Cap Version 3-16
Figure 3.17 Pile Group Layout using New from Template, 3D circular option, plate elements – Flexible Cap Version 3-17
Figure 3.18 Input parameters for mesh generation, rectangular option, beam elements – Flexible Cap Version 3-18
Figure 3.19 Input parameters for mesh generation, circular option, beam elements – Flexible Cap Version 3-19
Figure 3.20 Pile Group Layout using New from Template, 3D rectangular option, beam elements – Flexible Cap Version 3-20
Figure 3.21 Pile Group Layout using New from Template, 3D circular option, beam elements – Flexible Cap Version 3-21
Figure 3.22 Open File Dialog 3-22
Figure 3.23 Warning message for possible data loss. 3-23
Figure 3.24 Save As Dialog 3-24
Figure 3.25 Message window advising that changes were not saved to disk 3-25
Figure 3.26 Save Bitmap Dialog 3-26
Figure 3.27 Page Setup Dialog 3-27
Figure 3.28 Print Dialog....3-28
Figure 3.29 Options contained in the Data Menu 3-29
Figure 3.30 Options contained in the Data Menu – Flexible Cap Version 3-30
Figure 3.31 Project Title Window....3-31
Figure 3.32 Cross Sections Definition Window 3-32
Figure 3.33 Edit Cross Section Window 3-34
Figure 3.34 Edit Cross Section Window. Circular Pipe Section 3-35
Figure 3.35 Edit Cross Section Window. Rectangular Pipe Section 3-36
Figure 3.36 Edit Cross Section Window. I Section....3-36
Figure 3.37 Edit Cross Section Window. H Section 3-37
Figure 3.38 Edit Cross Section Window. AISC Section 3-37
Figure 3.39 Edit Cross Section Window for a Tapered Section....3-39
Figure 3.40 Toggle between Section Specifications and Section Draw. H section 3-40

Figure 3.41 Edit Cross Section Window (Nonlinear Section) 3-46
Figure 3.42 Edit Cross Section Window. Rectangular Concrete Pile 3-48
Figure 3.43 Edit Cross Section Window. Round Concrete Shaft 3-48
Figure 3.44 Edit Cross Section Window. Round Concrete Shaft with Casing (and Steel Section) 3-49
Figure 3.45 Edit Cross Section Window. Round Prestressed Pile 3-49
Figure 3.46 Edit Cross Section Window. Square Prestressed Pile 3-50
Figure 3.47 Edit Cross Section Window. Octagonal Prestressed Pile 3-50
Figure 3.48 Edit Cross Section Window. Steel Pipe Pile 3-51
Figure 3.49 Edit Cross Section Window. Steel I, H Section Pile 3-51
Figure 3.50 Edit Cross Section Window. Steel AISC Section Pile 3-52
Figure 3.51 Edit Cross Section Window. User Defined EI 3-52
Figure 3.52 Edit Cross Section Window. Thrust Forces 3-53
Figure 3.53 Show Stress Curve Window. 3-60
Figure 3.54 Show Diagrams Window. 3-61
Figure 3.55 Adjust Softening Parameter. 3-62
Figure 3.56 Compute Elastoplastic Moment Curvature 3-63
Figure 3.57 Pile Properties Window 3-64
Figure 3.58 Pile Sections Window 3-64
Figure 3.59 Pile Group Properties Window. 2D Model 3-65
Figure 3.60 Pile Head Connections 3-68
Figure 3.61 Pile Group Properties Window. 3D Model 3-69
Figure 3.62 Pile Head Coordinates Window. 2D Model 3-71
Figure 3.63 Example of a 2D model 3-73
Figure 3.64 Distance from pile top to ground line to simulate slopping ground 3-74
Figure 3.65 Pile Head Coordinates Window. 3D Model 3-75
Figure 3.66 Local Axes Definition 3-76
Figure 3.67 Local Axes for 2D models 3-77
Figure 3.68 Equations to estimate orientation for local axes. 2D Model 3-77
Figure 3.69 Local Axes for Horizontal Piles. 3D model 3-78
Figure 3.70 Local Axes for Vertical Piles. 3D model 3-78
Figure 3.71 Local Axes for Batter Piles in the XY plane. 3D model 3-79
Figure 3.72 Local Axes for Batter Piles in the XZ plane. 3D model 3-79
Figure 3.73 Local Axes for Batter Piles. General Orientation. 3D model 3-80
Figure 3.74 Equations to estimate orientation for local axes, no section rotation. 3D Model 3-80
Figure 3.75 Equations to estimate orientation for local axes, including section rotation. 3D Model 3-81
Figure 3.76 Pile-Cap Options Window 3-82
Figure 3.77 Pile-Cap Options. General Shape 3-84
Figure 3.78 Pile-Cap Options. General Shape 3-85
Figure 3.79 Node Coordinates Window – Flexible Cap Version 3-86
Figure 3.80 Plate Connectivity Window – Flexible Cap Version 3-88
Figure 3.81 Plate Connectivity Numbering 3-89

Figure 3.82 Plate Offsets Window – Flexible Cap Version 3-90
Figure 3.83 Example Plate Offsets – Flexible Cap Version 3-92
Figure 3.84 Plate Section Properties Window – Flexible Cap Version 3-92
Figure 3.85 Beam Connectivity Window – Flexible Cap Version 3-94
Figure 3.86 Beam Connectivity Numbering 3-95
Figure 3.87 Beam Offsets Window – Flexible Cap Version 3-96
Figure 3.88 Example Beam Offsets – Flexible Cap Version 3-97
Figure 3.89 Beam End Zones Window – Flexible Cap Version 3-98
Figure 3.90 Beam End Zones – Flexible Cap Version 3-99
Figure 3.91 Plate Offsets Window – Flexible Cap Version 3-100
Figure 3.92 Beam Section Rotation – Flexible Cap Version 3-101
Figure 3.93 Concentrated Springs at Nodes Window -Flexible Cap Version 3-102
Figure 3.94 Distributed Springs on Plates Window - Flexible Cap Version 3-104
Figure 3.95 Distributed Springs on Beams Window - Flexible Cap Version 3-106
Figure 3.96 Loading Definition Window 3-108
Figure 3.97 Edit Load Case Window 3-111
Figure 3.98 Edit Load Case Window for a model with a flexible pile cap -Flexible Cap Version .. 3-112
Figure 3.99 Concentrated Loads at Pile Cap Window 3-116
Figure 3.100 Concentrated Load Window. Loading for 3D Analysis 3-118
Figure 3.101 Concentrated Load Window. Loading for 2D Analysis 3-119
Figure 3.102 Specified Displacements/Loads Window. 3D Analysis Case 3-120
Figure 3.103 Specified Displacements/Loads Window. 2D Analysis Case 3-121
Figure 3.104 Displacements at Nodes Window - Flexible Cap Version 3-122
Figure 3.105 Concentrated Loads at Nodes Window -Flexible Cap Version 3-124
Figure 3.106 Distributed Loads on Plates Window - Flexible Cap Version 3-126
Figure 3.107 Distributed Loads on Beams Window - Flexible Cap Version 3-128
Figure 3.108 Concentrated Loads on Beams Window - Flexible Cap Version 3-130
Figure 3.109 Distributed Lateral Loads Window 3-132
Figure 3.110 Equivalent distributed load in global and local axes 3-134
Figure 3.111 Soil Movement Window 3-135
Figure 3.112 Foundation Stiffness Options Window based on Nonlinear Soil Response 3-138
Figure 3.113 Foundation Stiffness Options Window based on Nonlinear Soil Response 3-140
Figure 3.114 Flexibility Matrix Components due to a force in X direction (2D case) 3-142
Figure 3.115 Flexibility Matrix Components due to a force in Y direction (2D case) 3-142
Figure 3.116 Flexibility Matrix Components due to a moment in Z direction (2D case) 3-143
Figure 3.117 Flexibility Matrix for a two-dimensional model 3-143
Figure 3.118 Flexibility Matrix for a three-dimensional model 3-144
Figure 3.119 Estimation of Stiffness Matrix from a Flexibility Matrix 3-145
Figure 3.120 Stiffness Matrix for a two-dimensional model 3-145
Figure 3.121 Stiffness Matrix for a three-dimensional model 3-146
Figure 3.122 Stiffness Matrix considering only diagonal terms for a three-dimensional model ... 3-147
Figure 3.123 Foundation Stiffness Options Window based on Equivalent Elastic Springs 3-150

Figure 3.124 Analysis due to a load or target displacement....3-152
Figure 3.125 Estimation of equivalent elastic springs....3-152
Figure 3.126 Static Condensation to find Lateral Stiffness at Pile Head....3-153
Figure 3.127 Equivalent Elastic Stiffness for a Pile Group....3-153
Figure 3.128 Analysis Option Window....3-154
Figure 3.129 Output Variables Window....3-157
Figure 3.130 Load Combinations Window....3-159
Figure 3.131 Sample of a LRFD File....3-161
Figure 3.132 Edit Load Combination Window....3-162
Figure 3.133 Soil Layers Window....3-164
Figure 3.134 Set of Soil-Resistance Window....3-4
Figure 3.135 Soil-Resistance Set Window....3-5
Figure 3.136 Soil-Resistance Curve Window....3-6
Figure 3.137 Set of Axial Load vs. Displacement Window....3-8
Figure 3.138 Axial Load-Displacement Curve Window....3-8
Figure 3.139 Set of Torque vs. Rotation Window....3-10
Figure 3.140 Torque-Rotation Curve Window....3-11
Figure 3.141 Set of Nonlinear EI Window....3-12
Figure 3.142 Moment Curvature Set Window....3-13
Figure 3.143 Moment Curvature Data Window....3-14
Figure 3.144 Set of Reduction Factors along Depth Window....3-16
Figure 3.145 Reduction Factors Window....3-17
Figure 3.146 Set of Soil Movement Window....3-18
Figure 3.147 Soil Movement Window....3-19
Figure 3.148 Set of Distributed Lateral Loads Window....3-20
Figure 3.149 Distributed Load Window....3-21
Figure 3.150 Printing Depths for p-y Curves Window....3-23
Figure 3.151 Displacements for p-y Curves Window....3-24
Figure 3.152 Import Soil Layers from EnCPT Window....3-25
Figure 3.153 Import p-y curves from LPILE Window....3-27
Figure 3.154 Import p-y curves from LPILE Window....3-29
Figure 3.155 Import Nonlinear EI Data from LPILE Window....3-31
Figure 3.156 Edit Menu with 3D View Active....3-35
Figure 3.157 Edit Menu with 3D View Active - Flexible Cap Version....3-36
Figure 3.158 Selection by clicking....3-37
Figure 3.159 Selection using a box....3-38
Figure 3.160 Selection of plates - Flexible Cap Version....3-40
Figure 3.161 Selection of beams - Flexible Cap Version....3-41
Figure 3.162 Selection of nodes - Flexible Cap Version....3-43
Figure 3.163 Selection of piles, plates and nodes - Flexible Cap Version....3-44
Figure 3.164 New Pile Window. 2D Model....3-46
Figure 3.165 New Pile Window. 3D Model....3-47

Figure 3.166 Pile Layout Window. No piles Selected....3-48
Figure 3.167 Pile Layout Window. Piles Selected....3-49
Figure 3.168 Window for confirmation of deletion....3-50
Figure 3.169 Move Options Window. Absolute Coordinates....3-51
Figure 3.170 Move Options Window. Relative Coordinates....3-51
Figure 3.171 Paste Options Window. Absolute Coordinates....3-52
Figure 3.172 Paste Options Window. Relative Coordinates....3-53
Figure 3.173 Paste Array Options Window. Rectangular array....3-54
Figure 3.174 Pile layout after paste using rectangular array....3-54
Figure 3.175 Paste Array Options Window. Polar array....3-55
Figure 3.176 Pile layout after paste using polar array....3-55
Figure 3.177 New Plate Window - Flexible Cap Version....3-56
Figure 3.178 Edit Plate Window. No plates Selected - Flexible Cap Version....3-57
Figure 3.179 Edit Plate Window. Plates Selected - Flexible Cap Version....3-58
Figure 3.180 Window for confirmation of plates deletion - Flexible Cap Version....3-59
Figure 3.181 Divide Plates Window - Flexible Cap Version....3-59
Figure 3.182 Divide Plates option - Flexible Cap Version....3-60
Figure 3.183 New Beam Window - Flexible Cap Version....3-61
Figure 3.184 Edit Beam Window. No beams Selected - Flexible Cap Version....3-62
Figure 3.185 Edit Beam Window. Beams Selected - Flexible Cap Version....3-63
Figure 3.186 Window for confirmation of beams deletion - Flexible Cap Version....3-63
Figure 3.187 Divide Beams Window - Flexible Cap Version....3-64
Figure 3.188 Divide Beams option - Flexible Cap Version....3-65
Figure 3.189 New Node Window - Flexible Cap Version....3-65
Figure 3.190 Edit Node Window. No nodes Selected - Flexible Cap Version....3-66
Figure 3.191 Edit Node Window. Nodes Selected - Flexible Cap Version....3-67
Figure 3.192 Window for confirmation of nodes deletion - Flexible Cap Version....3-68
Figure 3.193 Window for confirmation of unconnected nodes deletion - Flexible Cap Version....3-68
Figure 3.194 Move All Selected Window – Flexible Cap Version....3-69
Figure 3.195 Copy All Objects Window – Flexible Cap Version....3-70
Figure 3.196 Window for confirmation of all selected elements deletion - Flexible Cap Version....3-71
Figure 3.197 Pile Sets Definition Window....3-72
Figure 3.198 Edit Pile Set Window....3-73
Figure 3.199 Plate Sets Definition Window - Flexible Cap Version....3-75
Figure 3.200 Edit Plate Set Window - Flexible Cap Version....3-76
Figure 3.201 Beam Sets Definition Window - Flexible Cap Version....3-78
Figure 3.202 Edit beam Set Window - Flexible Cap Version....3-79
Figure 3.203 Node Sets Definition Window - Flexible Cap Version....3-81
Figure 3.204 Edit Node Set Window - Flexible Cap Version....3-82
Figure 3.205 Options contained in the Options Menu....3-84
Figure 3.206 Options contained in the Options Menu - Flexible Cap Version....3-85
Figure 3.207 Units Option....3-86

Figure 3.208 Analysis Mode Option 3-87
Figure 3.209 Message window to confirm conversion from 3D layout to a 2D layout 3-88
Figure 3.210 Pile-Cap Type Options - Flexible Cap Version 3-88
Figure 3.211 Modify Group Layout using Template Window. Defining new pile properties. 3-90
Figure 3.212 Modify Group Layout using Template Window. Using existing pile properties. 3-91
Figure 3.213 Modify Group Layout using Template Window. Defining new properties – Flexible Cap Version 3-92
Figure 3.214 Modify Group Layout using Template Window. Using existing properties – Flexible Cap Version. 3-92
Figure 3.215 Modify Flexible Cap Layout using Template Window - Flexible Cap Version 3-93
Figure 3.216 Shifting Options 3-95
Figure 3.217 Shifting Options - Flexible Cap Version 3-95
Figure 3.218 Reduction Factors for Lateral Loads Window 3-97
Figure 3.219 Default Control Options Window 3-100
Figure 3.220 Output Variables Window 3-103
Figure 3.221 Text Editor Options Window 3-104
Figure 3.222 Message window to confirm checking for updates on ENSOFT site 3-105
Figure 3.223 Effect of the Include Load Case/Comb Number on Graphs option. 3-106
Figure 3.224 Options contained in the View Menu when 3D View is not active 3-107
Figure 3.225 Options contained in the View Menu when 3D View is active 3-108
Figure 3.226 Sample of Pile Top View 3-109
Figure 3.227 Sample of 3D View 3-110
Figure 3.228 XY View (Front) 3-111
Figure 3.229 XY View (Back) 3-111
Figure 3.230 XZ View (Right) 3-112
Figure 3.231 XZ View (Left) 3-112
Figure 3.232 YZ View (Top) 3-113
Figure 3.233 YZ View (Bottom) 3-113
Figure 3.234 XYZ View (Iso) 3-114
Figure 3.235 XYZ View (Reversed) 3-114
Figure 3.236 Panning in the 3D-View display 3-115
Figure 3.237 Zooming in the 3D-View display 3-115
Figure 3.238 Zoom by defining a window in the 3D-View display 3-116
Figure 3.239 Rotating in the 3D View display 3-116
Figure 3.240 Rotating in the 3D View by defining the rotation angles 3-117
Figure 3.241 Effect of the Zoom-in option 3-117
Figure 3.242 Effect of the Zoom-out option 3-118
Figure 3.243 Effect of the Restart View option 3-119
Figure 3.244 Effect of the Refresh View option, after reducing pile length 3-119
Figure 3.245 General Toolbar 3-120
Figure 3.246 Data Input Toolbar 3-120
Figure 3.247 Data Input Toolbar. Additional Buttons for Flexible Cap Version 3-121

Figure 3.248 Edit Toolbar 3-121
Figure 3.249 Edit Toolbar. Additional Buttons for Flexible Cap Version 3-122
Figure 3.250 Data Input Toolbar 3-122
Figure 3.251 View Toolbar 3-123
Figure 3.252 View Toolbar 3-123
Figure 3.253 Computations Toolbar 3-124
Figure 3.254 Output Toolbar 3-124
Figure 3.255 Output Toolbar. Additional Buttons for Flexible Cap Version 3-125
Figure 3.256 Plots Toolbar 3-126
Figure 3.257 Plots Toolbar (Continuation) 3-127
Figure 3.258 Help Toolbar 3-127
Figure 3.259 Options contained in the Show 3D View Menu 3-128
Figure 3.260 Options contained in the Show 3D View Menu – Flexible Cap Version 3-129
Figure 3.261 Effects on 3D model display using Piles option 3-130
Figure 3.262 Effects on 3D model display using Pile Numbers option 3-131
Figure 3.263 Effects on 3D model display using Pile Properties option 3-132
Figure 3.264 Effects on 3D model display using Pile Wire/Pile Shade options 3-133
Figure 3.265 Effects on 3D model display using Local Axes option 3-134
Figure 3.266 A 3D model display using Pile Head Connections option 3-135
Figure 3.267 Effects on 3D model display using Pile Cap option 3-136
Figure 3.268 Effects on 3D model display using FEM Nodes option. Flexible Cap Version 3-137
Figure 3.269 Node Numbers option. Flexible Cap Version 3-138
Figure 3.270 Effects on 3D model display using FEM Edges option - Flexible Cap Version 3-139
Figure 3.271 Effects on 3D model display using FEM Plates option - Flexible Cap Version 3-140
Figure 3.272 Effects on 3D model display using Plate Thickness option. Flexible Cap Version 3-141
Figure 3.273 Plate Numbers option - Flexible Cap Version 3-142
Figure 3.274 Effects on 3D model display using FEM Beams option - Flexible Cap Version 3-143
Figure 3.275 Effects on 3D model display using Shade Beams option. Flexible Cap Version 3-144
Figure 3.276 Beam Numbers option - Flexible Cap Version 3-145
Figure 3.277 Beam Numbers option - Flexible Cap Version 3-146
Figure 3.278 Plate/Beam Offsets option - Flexible Cap Version 3-147
Figure 3.279 Beam Numbers option - Flexible Cap Version 3-148
Figure 3.280 Concentrated Springs at Nodes option - Flexible Cap Version 3-149
Figure 3.281 Distributed Springs on Plates option - Flexible Cap Version 3-150
Figure 3.282 Distributed Springs on Plates option - Flexible Cap Version 3-151
Figure 3.283 Effects on 3D model display using Soil Layers option 3-152
Figure 3.284 Effects on 3D model display using Layer Labels option 3-153
Figure 3.285 Effects on 3D model display using Layer Depth Labels option 3-154
Figure 3.286 Effects on 3D model display using Concent. Loads at Pile Cap option 3-155
Figure 3.287 Effects on 3D model display using Distributed Loads at Piles option 3-156
Figure 3.288 Specified Displacements at Nodes option - Flexible Cap Version 3-157
Figure 3.289 Concentrated Loads at Nodes option - Flexible Cap Version 3-158

Figure 3.290 Distributed Loads on Plates option - Flexible Cap Version 3-159
Figure 3.291 Distributed Loads on Beams option - Flexible Cap Version 3-160
Figure 3.292 Distributed Loads on Beams option - Flexible Cap Version 3-161
Figure 3.293 Effects on 3D model display using Global Axes option 3-162
Figure 3.294 Effects on 3D model display using Vertical Lines option 3-163
Figure 3.295 Show Loading Options Window 3-164
Figure 3.296 Show Loading Options Window - Flexible Cap Version 3-165
Figure 3.297 Show Sprimgs Options Window – Flexible Cap Version 3-166
Figure 3.298 Display Values Options Window 3-167
Figure 3.299 Effects on 3D model display using Show Grid option 3-168
Figure 3.300 Grid Options Window 3-169
Figure 3.301 Effects on 3D model display using Pile Color option 3-170
Figure 3.302 Effects on 3D model display using Highlight Piles option 3-171
Figure 3.303 Effects on 3D model display using Highlight Plates option – Flexible Cap Version ... 3-172
Figure 3.304 Effects on 3D model display using Highlight Beams option – Flexible Cap Version .. 3-173
Figure 3.305 Effects on 3D model display using Highlight Nodes option – Flexible Cap Version... 3-174
Figure 3.306 Effects on 3D model display using Select Piles option 3-175
Figure 3.307 Effects on 3D model display using Select Plates option- Flexible Cap Version 3-176
Figure 3.308 Effects on 3D model display using Select Beams option- Flexible Cap Version 3-177
Figure 3.309 Effects on 3D model display using Select Nodes option- Flexible Cap Version 3-178
Figure 3.310 Effects on 3D model display using Show/Hide Piles option 3-179
Figure 3.311 Effects on 3D model display using Show/Hide Piles, Plates, Beams option – Flexible Cap Version 3-180
Figure 3.312 Options contained in the Windows Menu 3-181
Figure 3.313 Options contained in the Help Menu 3-182
Figure 3.314 Sample of User's Manual 3-183
Figure 3.315 Sample of Example Manual 3-184
Figure 3.316 Sample of Technical Manual 3-185
Figure 3.317 About Window 3-186
Figure 3.318 Sample of Check for Updates 3-187
Figure 3.319 Sample of Ensoft Key Inquirer 3-188
Figure 3.320 Sample of Check Network Dongle 3-189
Figure 4.1 Options contained in the Computation Menu 4-2
Figure 4.2 Options contained in the Computation Menu – Flexible Cap Version 4-3
Figure 4.3 Sample message if an input data is missing 4-4
Figure 4.4 Sample message while running the analysis 4-4
Figure 4.5 Sample message to stop the analysis 4-5
Figure 4.6 Sample message when analysis has finished 4-5
Figure 4.7 Sample message when analysis is unsuccessful 4-6
Figure 4.8 Sample message when analysis has finished 4-7
Figure 4.9 Select Load Case or Combination window 4-8
Figure 4.10 Sample of Input Text File 4-9

Figure 4.11 Sample of Echo Print File 4-10
Figure 4.12 Sample of Processor Run Notes Text File 4-11
Figure 4.13 Sample of Output Results File 4-13
Figure 4.14 Sample of Summary Results File 4-14
Figure 4.15 Sample of Moment Curvature File 4-15
Figure 4.16 Sample of Echo File (Flexible Cap) – Flexible Cap Version 4-16
Figure 4.17 Sample of Output Results File (Flexible Cap) – Flexible Cap Version 4-18
Figure 4.18 Sample of Summary Results File 4-19
Figure 4.19 Special Plots Window 4-20
Figure 4.20 Incremental/PushOver Plots Window 4-21
Figure 4.21 Effects on graph display using mouse commands. Defining box for zooming in (Left click) 4-24
Figure 4.22 Effects on graph display using mouse commands. After zooming in 4-25
Figure 4.23 Effects on graph display using mouse commands. Zooming out (Right Click) 4-26
Figure 4.24 Effects on graph display using mouse commands. Hiding a curve (Double Click on Legend) 4-27
Figure 4.25 Options contained in the Plot Menu 4-28
Figure 4.26 Effects on graph display using Plot Legend option. Plot Legend check-marked 4-29
Figure 4.27 Effects on graph display using Plot Legend option. Plot Legend not check-marked 4-30
Figure 4.28 Effects on graph display using Plot Markers option. Using Markers every 5 increments 4-31
Figure 4.29 Effects on graph display using Plot Markers option. Using no markers (Never) 4-32
Figure 4.30 Effects on graph display using Font Size option. Using twice the Default Size 4-33
Figure 4.31 Effects on graph display using Line Width option. Using four times the Default Size 4-34
Figure 4.32 Using Graph Title option 4-35
Figure 4.33 Using Edit Legend option 4-36
Figure 4.34 Using Active Graphs option 4-37
Figure 4.35 Using Print option 4-38
Figure 4.36 Using Page Setup option 4-39
Figure 4.37 Using Save To Disk option 4-40
Figure 4.38 Options contained in the Graphs Menu. 2D Model 4-41
Figure 4.39 Options contained in the Graphs Menu. 3D Model 4-42
Figure 4.40 Sign Convention for pile under lateral loading. (xy-plane loading) 4-43
Figure 4.41 Sign Convention for pile under lateral loading. (xz-plane loading) 4-44
Figure 4.42 Sample graphics screen of p-y Curves command under the Graphs menu 4-46
Figure 4.43 Sample graphics screen of Axial Load vs. Displacement command under the Graphs menu 4-47
Figure 4.44 Sample graphics screen of Torque vs. Rotation command under the Graphs menu . 4-48
Figure 4.45 Sample graphics screen of p-y Curves command under the Graphs menu 4-49
Figure 4.46 Sample graphics screen of Axial Load vs. Displacement command under the Graphs menu 4-50
Figure 4.47 Sample graphics screen of Torque vs. Rotation command under the Graphs menu . 4-51

Figure 4.48 Sample graphics screen of Deflection vs. Distance command under the Graphs menu .4-52
Figure 4.49 Sample graphics screen of Moment vs. Distance command under the Graphs menu4-53
Figure 4.50 Sample graphics screen of Shear vs. Distance command under the Graphs menu 4-54
Figure 4.51 Sample graphics screen of Soil Reaction vs. Distance command under the Graphs menu 4-55
Figure 4.52 Sample graphics screen of Bending Stiffness vs. Distance command under the Graphs menu 4-56
Figure 4.53 Sample graphics screen of Def., Mom., Shear vs. Distance command under the Graphs menu 4-57
Figure 4.54 Sample graphics screen of Deflection y dir vs. Distance command under the Graphs menu 4-58
Figure 4.55 Sample graphics screen of Deflection z dir vs. Distance command under the Graphs menu 4-59
Figure 4.56 Sample graphics screen of Moment z dir vs. Distance command under the Graphs menu 4-60
Figure 4.57 Sample graphics screen of Moment y dir vs. Distance command under the Graphs menu 4-61
Figure 4.58 Sample graphics screen of Shear y dir vs. Distance command under the Graphs menu 4-62
Figure 4.59 Sample graphics screen of Shear z dir vs. Distance command under the Graphs menu.4-63
Figure 4.60 Sample graphics screen of Soil Reaction y dir vs. Distance command under the Graphs menu 4-64
Figure 4.61 Sample graphics screen of Soil Reaction z dir vs. Distance command under the Graphs menu 4-65
Figure 4.62 Sample graphics screen of Bending Stiffness z dir vs. Distance command under the Graphs menu.....4-66
Figure 4.63 Sample graphics screen of Bending Stiffness y dir vs. Distance command under the Graphs menu.....4-67
Figure 4.64 Sample graphics screen of Def. y, Mom. z, Shear y vs. Distance command under the Graphs menu.....4-68
Figure 4.65 Sample graphics screen of Def. z, Mom. y, Shear z vs. Distance command under the Graphs menu.....4-69
Figure 4.66 Sample graphics screen of Deflection x Dir vs. Distance command under the Graphs menu 4-70
Figure 4.67 Sample graphics screen of Def. z, Mom. y, Shear z vs. Distance command under the Graphs menu.....4-71
Figure 4.68 Sample graphics screen of Resultant Deflection vs. Distance command under the Graphs menu.....4-72
Figure 4.69 Sample graphics screen of Resultant Moment vs. Distance command under the Graphs menu 4-73

Figure 4.70 Sample graphics screen of Resultant Shear vs. Distance command under the Graphs menu 4-74
Figure 4.71 Sample graphics screen of Resultant Soil Reaction vs. Distance command under the Graphs menu 4-75
Figure 4.72 Sample graphics screen of Def. y, Mom. z, Shear y vs. Distance command under the Graphs menu 4-76
Figure 4.73 Sample graphics screen of Pile Cap Vertical Load vs. Deflection command under the Graphs menu 4-77
Figure 4.74 Sample graphics screen of Pile Cap Load vs. Lateral Deflection command under the Graphs menu 4-78
Figure 4.75 Sample graphics screen of Pile Cap Load vs. Rotation command under the Graphs menu 4-79
Figure 4.76 Sample graphics screen of Pile Cap Moment vs. Deflection command under the Graphs menu 4-80
Figure 4.77 Sample graphics screen of Pile Cap Moment vs. Rotation command under the Graphs menu 4-81
Figure 4.78 Sample graphics screen of Pile Cap Vertical Load vs. Deflection X command under the Graphs menu 4-82
Figure 4.79 Sample graphics screen of Pile Cap Lateral Load vs. Deflection Y command under the Graphs menu 4-83
Figure 4.80 Sample graphics screen of Pile Cap Lateral Load vs. Deflection Z command under the Graphs menu 4-84
Figure 4.81 Sample graphics screen of Pile Cap Moment vs. Rotation X command under the Graphs menu 4-85
Figure 4.82 Sample graphics screen of Pile Cap Moment vs. Rotation Y command under the Graphs menu 4-86
Figure 4.83 Sample graphics screen of Pile Cap Moment vs. Rotation Z command under the Graphs menu 4-87
Figure 4.84 Sample graphics screen of Pile Cap K11 vs. Deflection X command under the Graphs menu 4-88
Figure 4.85 Sample graphics screen of Pile Cap K22 vs. Deflection Y command under the Graphs menu 4-89
Figure 4.86 Sample graphics screen of Pile Cap K33 vs. Rotation Z command under the Graphs menu 4-90
Figure 4.87 Sample graphics screen of Pile Cap K33 vs. Deflection Z command under the Graphs menu 4-91
Figure 4.88 Sample graphics screen of Pile Cap K44 vs. Rotation X command under the Graphs menu 4-92
Figure 4.89 Sample graphics screen of Pile Cap K55 vs. Rotation Y command under the Graphs menu 4-93
Figure 4.90 Sample graphics screen of Pile Cap K66 vs. Rotation Z command under the Graphs menu 4-94

Figure 4.91 Plot to Excel Window 4-95
Figure 4.92 Plot EI Diagrams Window....4-96
Figure 4.93 Options contained in the 3D View Results Menu 4-97
Figure 4.94 Options contained in the 3D View Results Menu – Flexible Cap Version....4-97
Figure 4.95 Sample graphics screen for Deformed Shape command....4-98
Figure 4.96 Sample graphics screen for Contour Plot command 4-99
Figure 4.97 Sample graphics screen for Pile Diagrams command 4-100
Figure 4.98 Sample graphics screen for Pile Cap Response command....4-101
Figure 4.99 Sample graphics screen for Pile Top Effects command 4-102
Figure 4.100 Sample screen for Pile Top Results Window including Axial Demand Plot 4-103
Figure 4.101 Sample graphics screen for Pile Top Effects command 4-104
Figure 4.102 Sample graphics screen for Flexible Cap Contour Plot command – Flexible Cap Version 4-105
Figure 4.103 Sample graphics screen for Flexible Cap Section Cut command – Flexible Cap Version 4-106
Figure 4.104 Sample graphics screen for Beam Diagrams command – Flexible Cap Version 4-107
Figure 4.105 Sample graphics screen for Flexible Cap Node Results – Flexible Cap Version 4-108
Figure 4.106 Sample graphics screen for Flexible Cap Plate Results – Flexible Cap Version....4-109
Figure 4.107 Sample graphics screen for Flexible Cap Beam Results – Flexible Cap Version....4-110

List of Tables

Table 2.1 Files created in GROUP 2022 runs 2-10
Table 3.1 Supported mathematical operations 3-2
Table 3.2 Supported mathematical constants 3-2
Table 3.3 Commonly used batter angles 3-72
Table 3.4 Signs for coordinates and batter angles for an example of a 2D model 3-73
Table 3.5 Soil Types and Parameters Needed per Soil Type 3-1
Table 3.6 Soil Types and Parameters Needed per Soil Type (continuation) 3-2
Table 3.7 Values of ε_{50} for Clays 3-1
Table 3.8 Values of ε_{50} for Stiff Clays 3-1
Table 3.9 Soil-Modulus Parameter k for Sands 3-2
Table 3.10 Soil Modulus Parameter k for Clays 3-2
Table 3.11 Unit systems 3-86
Table 4.1 Files created in GROUP 2022 runs 4-6
Table 4.2 Files created in GROUP 2022 runs – Flexible Cap Version 4-6
Table 4.3 File created using Moment Curvature Calculations 4-7
Table 4.4 Mouse commands in the graphics mode 4-23

CHAPTER 1. Introduction

1.1 General Description

GROUP has been well-received as a useful tool for analyzing the behavior of piles in a group subjected to both axial and lateral loadings. The program was developed to compute the distribution of loads (vertical, lateral, and overturning moment) from the pile cap to individual piles in the group. GROUP v8.0 allows the user to analyze the behavior of pile groups using either a 2-dimensional model for piles in a symmetric layout or a 3-dimensional model for a group of piles that is not symmetrically arranged.

The piles may be installed vertically or on a batter, and their heads may be fixed, pinned, or elastically restrained by the pile cap. The cap is assumed to act as a rigid body, and may settle, translate, and rotate.

The program will internally generate the nonlinear response of the soil, in the form of *t-z* and *q-w* curves for axial loading, *t-r* curves for torsional loading, and *p-y* curves for lateral loading. An iterative solution accommodates the nonlinear response of each of the piles, and will satisfy the equations of equilibrium. This achieves compatibility between pile movement and soil response, and between cap and pile-head movement.

For the lateral analysis of piles, second order effects due to the axial load are included in the formulation, so the P-delta effects are considered automatically by the program.

1.2 Program Features and Version History

1.2.1 GROUP 1.0 (1987)

The first version of GROUP internally generated the nonlinear response of the soil in the form of *p-y* curves for soft clay, stiff clay with or without free water, and sand. The user was also allowed to input externally generated *p-y* curves. The relationship between vertical load and pile settlement (or uplift) had to be externally specified by the user.

1.2.2 GROUP 2.0 (1990)

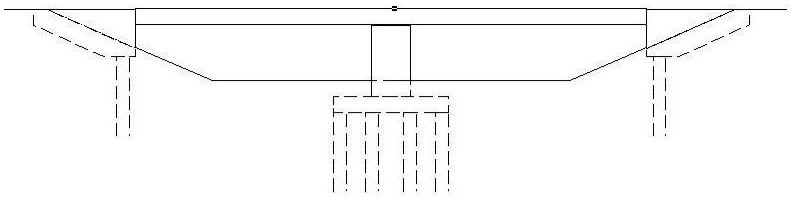
The second version of GROUP included a number of additional, user-friendly features. The main improvements were on the input and graphics modules. The input module introduced the use of pull-down menus and grid cells for data entry. The graphics module provided enhanced plots for pile deflection, bending moment, shear, and soil resistance.

A subroutine was added to the main program to allow the implementation of the *t-z* method to solve for the settlement (or uplift) of a pile, based on soil properties and pile geometry. The user was able to specify that the program internally generate the settlement of a pile as a function of axial load based on nonlinear load transfer in skin friction and end bearing, which in turn may vary as a function of distance below the ground surface.

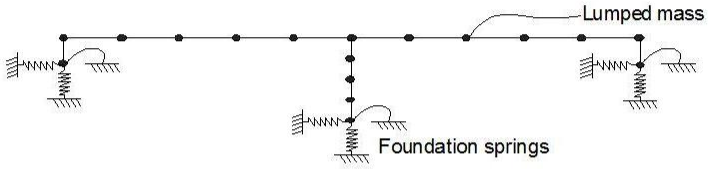
1.2.3 GROUP 3.0 (1994)

The third version of GROUP featured more computational options.

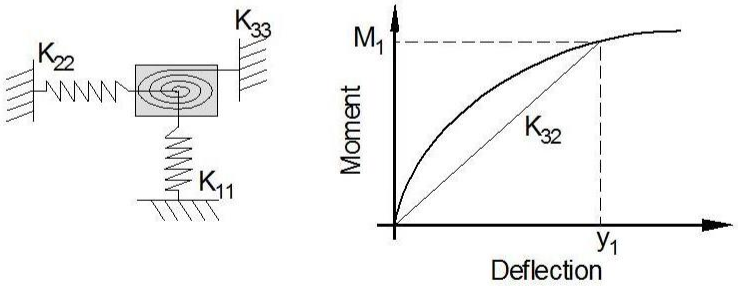
- The user was provided with updated technical information on group-reduction factors for piles under lateral loads. If the user wished to account for pile-soil-pile interaction, the program could optionally generate *p*-multipliers to automatically reduce soil resistance.
- The new version provided data for evaluating pile-cap stiffness levels, which are often used in structural engineering models. Under this new option, the program created curves of incremental loading versus pile-cap movements and rotations. A conceptual modeling application for the analysis of the behavior of a bridge is shown in Figure 1.1. With curves similar to the one shown in Figure 1.1c, the user was able to specify pile-cap stiffness levels corresponding to a given range of loading for their models of structural analyses.
- GROUP 3.0 could accommodate up to 50 rows of piles in each group analysis, compared to only 20 rows in previous versions of the program.
- Also, the input-editing module was improved to handle more data points and soil layers.
- New example problems were presented, including one for the special case where the pile cap is installed below the ground surface.



a. Elevation



b. Analytical Model



$$\begin{bmatrix} K_{11} & 0 & 0 \\ 0 & K_{22} & K_{23} \\ 0 & K_{32} & K_{33} \end{bmatrix}$$

c. Stiffness Matrix

Figure 1.1 Example of modeling a bridge on piles

1.2.4 GROUP 4.0 (1996)

The fourth version of GROUP featured a major upgrade for compatibility with new operating-systems, enhancements in ease of use, and new technical and computational features.

- Standard Windows operations, such as dialog boxes, grid cells, clickable buttons, drop-down list of options, and pull-down submenus were all incorporated to ease data entry.
- Improved file-management features were included to help the user during program execution. A standard single-program module was introduced for data input, program execution, graphical observation of output-data curves, and optional direct viewing of input or output text files.
- The online help file was also enhanced and could be loaded during preparation of input data for quick consultation of main operational features and basic technical recommendations.
- The graphical observation of output curves featured a new interface that allowed the user to zoom into areas of particular interest. The user may thus observe the detailed behavioral measurements of any portion of the depth of each pile group.
- Three additional *p-y* criteria were added to the previously existing four soil types. New soil criteria are available for *c-ϕ* soils, the sand model recommended by the American Petroleum Institute (API), and rock.
- A new option that takes into account the lateral soil resistance against embedded pile caps was introduced. The theory and specific assumptions utilized for the computation of lateral soil resistance on embedded pile caps were included in the new *Technical Manual*.
- The user was also provided with the option to specify a reduction factor on the axial capacity of closely-spaced piles.

1.2.5 GROUP 5.0 (2000)

The fifth version of GROUP was introduced in October, 2000, and included a new computation module for 3-dimensional pile-group analyses.

- GROUP v5.0 allowed the user to analyze the behavior of pile groups using either a 2 or 3-dimensional model. The 2-dimensional model is sufficient and convenient if the piles are arranged symmetrically to the applied load. Otherwise, a 3-dimensional model is recommended for analyzing the nonlinear behavior of pile groups.
- GROUP v5.0 solves the nonlinear response of each pile under six components of loadings and assures compatibility of geometry and equilibrium of forces between the applied external loads and the reactions of each pile head.
- Torsional resistance curves could be generated by the program.

1.2.6 GROUP 6.0 (2003)

The sixth version of GROUP was introduced in January, 2003. The new features for the sixth version are summarized below:

- The program allowed for up to 500 piles in a group, and each pile could have different EI values in orthogonal directions.
- GROUP v6.0 considered the nonlinear bending stiffness (EI) of each pile during computation. The user can enter nonlinear moment-curvature curves under different axial loads for each pile.
- For closely-spaced piles, the pile-soil-pile interaction can be taken into account by introducing reduction factors for the *p-y* curves used for each single pile. These factors were modified based on a study by Reese and Van Impe (2001).
  GROUP v6.0 included advanced 3-dimensional graphics to represent the response of each pile in the group. The user has options that allow for the display of the deformed/undeformed shape of pile groups, stress contours, soil layers, and many others.

1.2.7 GROUP 7.0 (2006)

The seventh version of GROUP was introduced in February, 2006. The new features for the seventh version are summarized below:

- The *p-y* criteria for liquefiable sand developed by Rollins, et al. (2003), and p-y criteria for stiff clay with user-specified initial *k* values, recommended by Brown (2002), were added in Version 7.0.
- The program requires data for unit side friction at the top and bottom of each layer as well as the unit tip resistance for generating the axial stiffness (axial load-displacement relationship) at the pile head. Version 7.0 internally provided the estimated unit side friction and tip resistance based on the strength data of soil layers and commonly accepted empirical equations. However, the program still allowed those data specified externally for the user to control the axial capacity of piles based on the local recommendations.
- Version 7.0 generated more types of graphs than previous versions. More importantly, the program plots the force and displacement on the pile cap, such as the axial force (tension and compression) of each pile on the pile cap, the lateral forces (shear and moment) of each pile on the pile cap, and the displacement of the pile cap in different directions.
- The program provided a summary table of output data, which is convenient when the size of the output file becomes very large due to a large number of piles in the application.
- The program implemented a new algorithm that significantly improved the nonlinear iterations and accuracy of the solution.

1.2.8 GROUP 8.0 (2010)

This is the eight version of GROUP. The new features are summarized below:

- The program allows for a maximum of 2000 piles in a group.
- A New from Template option has been added, so the program can easily generate basic rectangular or circular pile group layouts.
- The program includes various additional options for loading.
    - The program allows the user to perform analyses under various load cases. Each load case may be due to concentrated loads at the pile cap and/or distributed lateral loads at the piles.
    - Concentrated loads at the pile cap may be defined for any position; the program will compute the equivalent concentrated load at the origin for analyses.
    - Distributed lateral loads at the piles can be defined relative to the local axes as well as the global axes. For loads oriented at the global axes in batter piles, the axial component may be included as a concentrated load at the pile head.
    - The program allows the user to perform analyses under various load combinations. Load combinations are set by load factors applied at the load cases previously defined by the user.
    - Maximum and minimum envelopes may be computed for both load cases and load combinations.
    - The program includes the option to compute both the Flexibility and Stiffness Matrix for different levels of loading.
- A Cross Section dialog was included to allow the user the option to define circular, rectangular and I sections. However, for noncircular sections the equivalent diameter must be defined by the user.
- The program will be able to transform a 2D pile layout into a 3D layout. The program can also convert a 3D pile layout into a 2D layout; however, this conversion may not be accurate for non-symmetrical configurations.
- Inputs for soil properties have been rearranged to allow the user enter the information by soil type.
- The program allows a mix of input defined curves (*p-y* curves, axial load-displacement curves and torque-rotation curves) and automatically generated curves in the same model.
- The program provides a summary table of output data, which includes maximum and minimum effects for pile heads over the pile length.
- User can select the information to be printed, which is convenient when the size of the output file becomes very large due to a large number of piles and load cases.

1.2.9 GROUP 2013 (2013)

Group is now released under maintenance contract. All users with active maintenance contracts can download all program updates and revisions from the Ensoft website (www.ensoftinc.com) without additional charges. Thus, users are no longer purchasing version updates to the program. Instead, they are purchasing extensions to their maintenance contracts. This make using traditional version numbers no longer necessary.

The version numbers now used for GROUP has three parts that indicate the year of the release, the data file format version number, and the revision number. The meaning of the versions numbers is described below.

- The first number refers to the year of the program that indicates the features of the program.
- The second number is the data file version number that is changed whenever there is a change in the data file format.
- The third number indicates the revision number. The revision number is incremented for every update of the program or documentation.

The new features for release GROUP are summarized below:

- Estimation of an equivalent elastic stiffness matrix.
- Include analysis due to lateral soil movement.
- Include option to define reduction factors for lateral loading along depth.
- Estimate the axial displacement and axial forces along piles. This option is not available when user defined axial load displacement curve is used.
- Compute the displacements in global directions.
- Option to define a tapered section along depth
- Option to change the modulus of elasticity along depth

1.2.10 GROUP 2014 (2014)

The new features for release GROUP are summarized below:

- Import moment curvature generated for LPILE program.
- Import p-y curves generated for LPILE program.
- Add two soil models, API Soft Clay with variable J and Elastic Subgrade.
- Revise input units for various parameters, i.e., depth, specific weight, section dimensions.
- Help the user to estimate some soil parameters, i.e., e50, k.

- Add an option to analyze a case specifying displacements at the pile cap level. The user may combine specifying displacements and applied forces in different degree of freedoms.
- Add tool buttons to navigate through load cases and combinations.
- Use of parallel processing, so the program can perform the analysis using more than one processor that may reduce the analysis time.

1.2.11 GROUP 2016 (2016)

The new features for release GROUP are summarized below:

- Compute Nonlinear Bending Stiffness for various sections.
- Enhance program to use negative vertical coordinates for pile heads.
- Enhance program so the ground line can start at any soil layer.
- Add an option to use vertical projection while using batter piles.
- Add an option to calculate reduction factors for lateral loading based on average diameter.
- Allow the rotation of the section respect to the x axis.
- Add option to draw pile caps of irregular shape.
- Add option to shift piles and soil layers in vertical direction.
- Add option to color piles based on pile properties and cross sections
- Add option to export plots to Excel
- Add option to import/export pile group layout and loading to Excel

1.2.12 GROUP 2019 (2019)

The new features for release GROUP are summarized below:

- Add option to perform incremental/pushover analysis for a load case.
- Add option to compute or not layering correction.
- Add option for the analysis of embedded pile caps using nonlinear springs.
- Add option to consider non-symmetric sections for the estimation of nonlinear bending stiffness.
- Add option to import soil layers generated by EnCPT.
- Enhance estimation of load displacement curves.
- Enhance program so the pile head can start at any soil layer.

1.2.13 GROUP 2022 (2022)

The new features for release GROUP are summarized below:

- Add soil model: Liquified Sand Hybrid, Piedmont Residual, Massive Rock, Loess.
- Add option to consider a flexible pile cap. The flexible pile cap is modeled using plate elements and/or beam elements.

1.3 Technical Support

Although GROUP was designed to be distinguished by its ease of use, some users may still have questions. The ENSOFT technical staff strongly encourages users to utilize electronic means of support via email or fax:

- electronic mail to: support@ensoftinc.com
- fax to: (512) 244-6067

In any support request, please include the following information:

- Software version (obtained from the Help/About menu)
- Description of the problem
- Copy, file attachment, or printout of the input-data file
- Name, fax, and telephone number of the contact person and of the registered user (or name and office location of the registered company)

Although immediate answers are offered on most technical support requests, please allow up to two business days for a response in case of difficulties or scheduling conflicts.

1.3.1 Support by Telephone

Technical help by means of direct calls to our local telephone number (512-244-6464) is available, but limited. The current policy of ENSOFT is that all telephone calls for software support will be answered free of charge for the first 10 minutes of each call if the user has a current maintenance for

the latest version of GROUP. A charge of $1.50 per minute comes into effect for every additional minute after the initial 10 minutes. Applicable charges will be billed to the user.

Telephone support will be provided free of charge for a period of three months after the first software-support call has been made. The charge of $1.50 per minute comes into effect after the free support period.

Software support by any of the preferred methods (email or fax) is free for all users that have licenses with current maintenance for the latest version of GROUP.

1.3.2 Upgrade Notifications and Internet Site

All registered users who keep their address current at ENSOFT will receive notifications by regular mail when new versions become available. Users may also consult our internet site (www.ensoftinc.com) for additional information on software updates, demos, new applications, technical news, and company information.

Many times, software updates are placed at our internet site and available for free to all registered users. New software updates are notified by electronic mail to users that voluntarily subscribe to our users email list. Please note that users must have a current maintenance in order to run the latest update available.

1.3.3 Software Customization Services

Customized versions of GROUP may also be designed with a contractual agreement for the costs of additional programming. ENSOFT uses modern computational techniques and broad experience to obtain fast and reliable results to engineering problems.

1.3.4 Changes to Support Policy

The software support policy and associated expenses are subject to change at ENSOFT's discretion and without specific mailed notices to the users. However, any change of rules will be verbally provided during telephone calls for software support.

CHAPTER 2. Installation and Getting Started

2.1 Installation Procedures

GROUP 2022 is distributed with a hardware lock. The hardware lock is a device (USB dongle) that is attached to the USB port of the computer in use. This method of distribution is compatible with existing operating systems, has better stability than other alternatives, and allows users to obtain software updates or replacements via internet download.

Before installing, your computer should be equipped with the following:

- An open USB port
- At least 200 MB of free space
- At least 4GB of memory
- Windows 7 or later

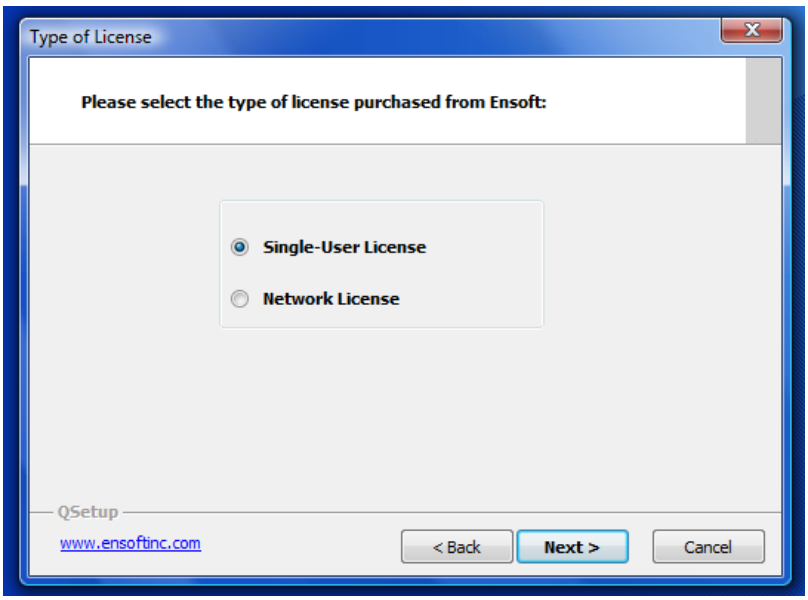


Figure 2.1 Selection of Type of License during installation of the program

To install GROUP 2022:

- Insert the installation CD. If Autoplay is enabled, Windows will ask if the user wants to run setup.exe. If the user does not have that feature enabled, then from *My Computer* double click the drive into which the installation CD is inserted.
- Select to install GROUP 2022, and then click on the "Install Standard" button at the bottom of the window.
- Follow the on screen instructions until the window shown above is reached.

2.1.1 Single User License

If your license is for a single-user, select the "Single-User License" option and click Next. Follow the dialog boxes until the installation finishes. After the installation is completed, insert the black USB dongle into any USB port and follow USB driver installation if it shows up. In some cases, the user may be requested to restart Windows before executing the program.

2.1.2 Network License

If a network license is being installed, select the "Network License" option and click Next. At the following window, select the type of network installation that is being installed.

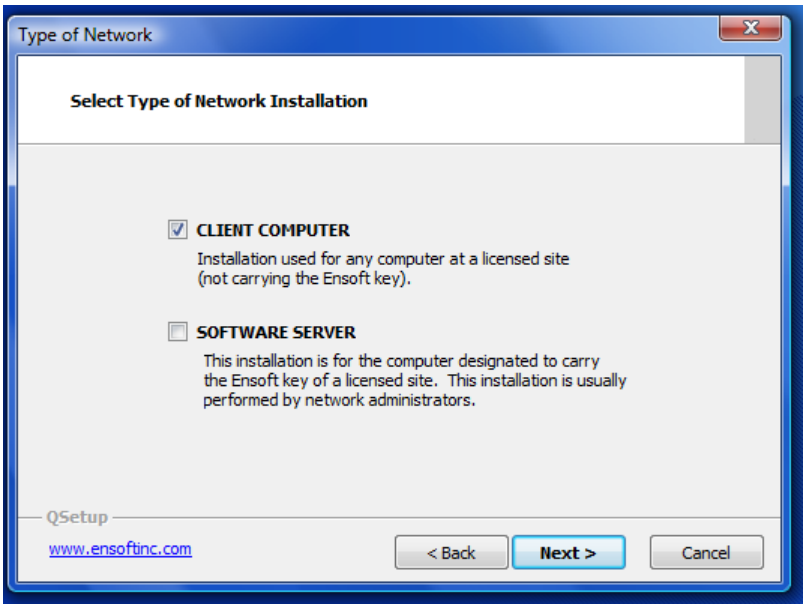


Figure 2.2 Selection of Type of Network Installation

2.1.2.1 Installation to a Server

The "Software Server" is known as the computer that will be carrying the network lock provided by ENSOFT, INC. The software server is not necessarily the same as the existing network server. Any computer in the existing Windows network may be designated "Software Server" for GROUP, as long as the network lock is attached to the USB port and the “server” version is installed in its hard drive.

To install, select the "Software Server" option and click Next. Follow the dialog boxes until the installation finishes. Be sure to point to the Unique Company File provided by ENSOFT, INC when requested.

After the installation is completed, insert the black USB dongle into any USB port and follow the Windows automatic USB driver installation if it appears. In general, this installation should be performed by the network administrator.

2.1.2.2 Installation to Clients

"Client Computers" may be all other computers on the network that have the program installed as clients. Client Computers do not need a USB dongle. "Client" installations will be allowed to run as long as the IP address of the client is within the IP range defined for the company network and the network service is running on the "server".

To install, select the "Client Computer" option and click Next. Follow the dialog boxes until the installation finishes. Be sure to set the IP address of the "server" when requested. In some cases, the user may be requested to restart Windows after installation is complete.

2.2 Software Updates

Occasionally, ENSOFT will produce software improvements and/or fixes. Software users may freely download the latest updates from the following site (Please note that users must have a current maintenance in order to run the latest update available):

http://www.ensoftinc.com

Download the update files to an appropriate local directory, then run the executable. Follow the on-screen instructions until installation is complete.

2.3 Versions Available

GROUP 2022 has two versions the Standard Version and the Flexible Cap Version. The Flexible Cap Version includes all the options available in the Standard Version plus the option to model the Pile Cap as a Flexible Cap using Plate Elements and/or Beam Elements.

2.4 Getting Started

A general diagram showing the menu choices and operational flow chart of program GROUP 2022 is presented in Figure 2.3. The following paragraphs provide a short description on the operational features of GROUP 2022 and should quickly enable the user to get started with the program.

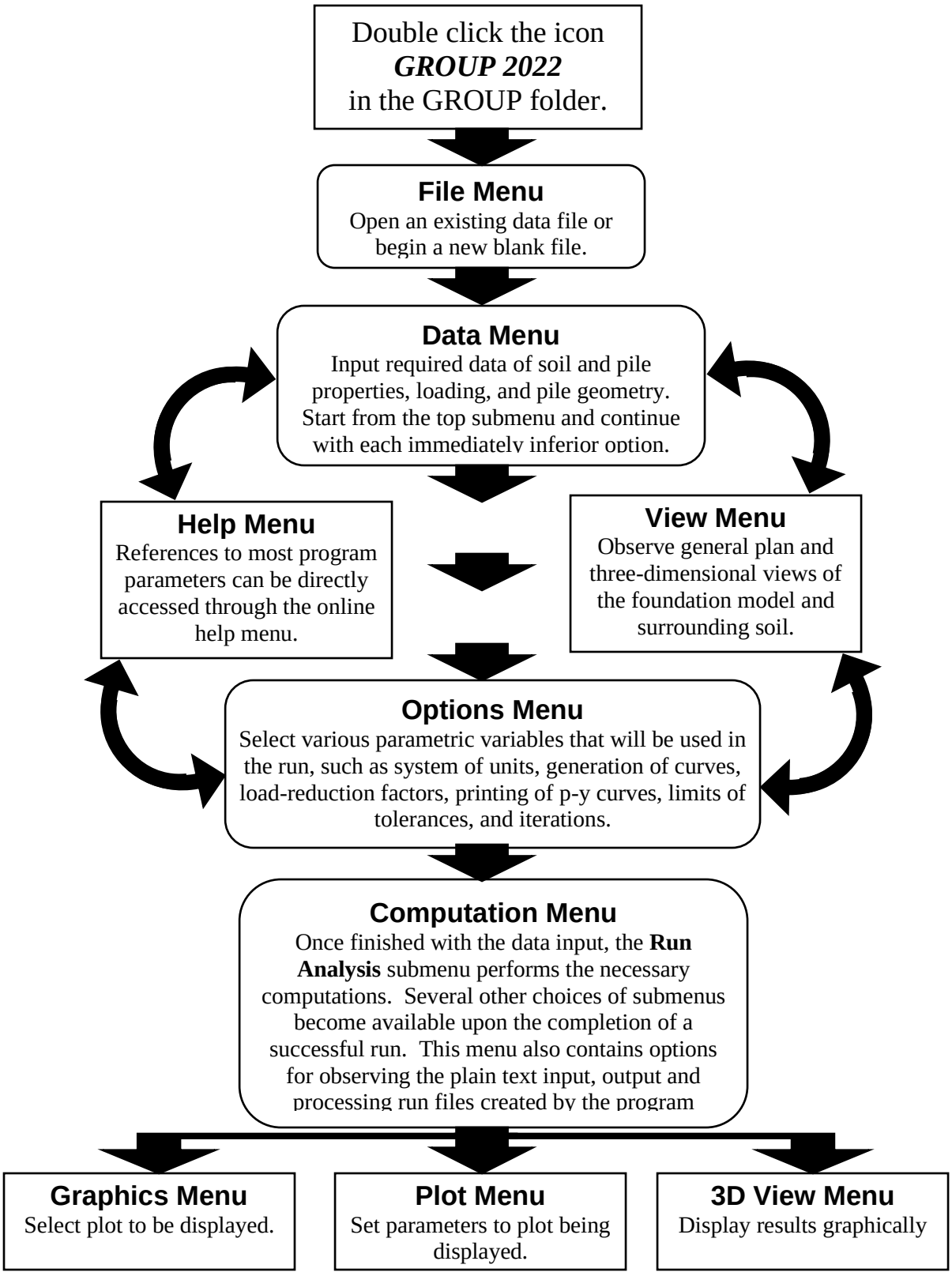


Figure 2.3 Sample organizational and operational flow chart

2.4.1 Program Start

Double click the GROUP 2022 icon to start: . When the program runs for the first time, users have the opportunity to personalize the program. This information is printed in all the output files generated by the program.

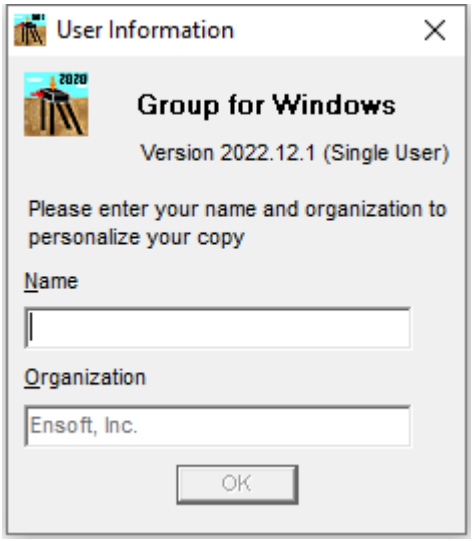


Figure 2.4 Window to personalize the program

When the program runs for the first time, the program shows the Check for Updates Window

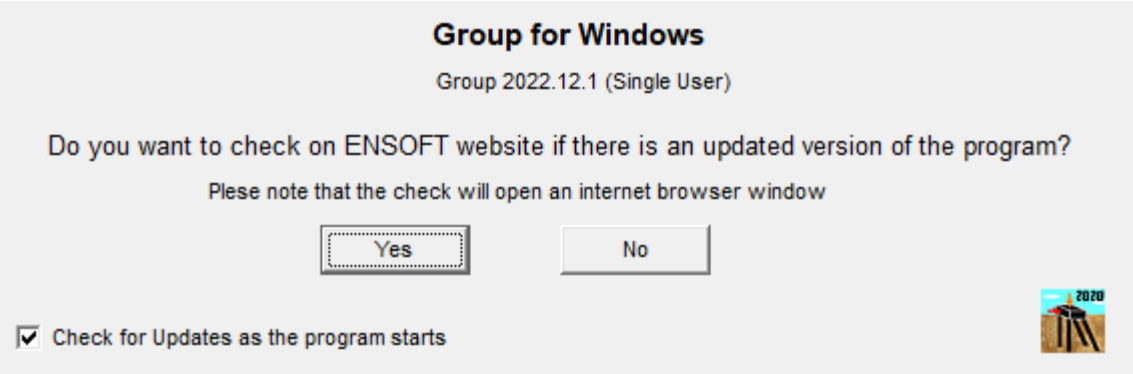


Figure 2.5 Check for Updates Window

This window also shows up when the user checks the item: **Checks for Updates as the Program Starts** under the **Options** Menu, or the **Option Check for Updates as the program starts** is still checked in the Check for Updates Window. By clicking on yes, the program will open an internet website window to check in the ENSOFT website if there is a new update available. After closing this window, the GROUP 2022 Window will show up

A blank workspace with a toolbar at the top will appear. As a standard Windows feature, pressing Alt displays the menu operations with underlined letters. Pressing the underlined letter after pressing Alt is the same as clicking the operation. For example, to open a **New File**, the user could press Alt, F, N, in sequence, Ctrl+N, or click **File** then **New**. Additionally, holding the mouse cursor over an icon displays the function of the icon.

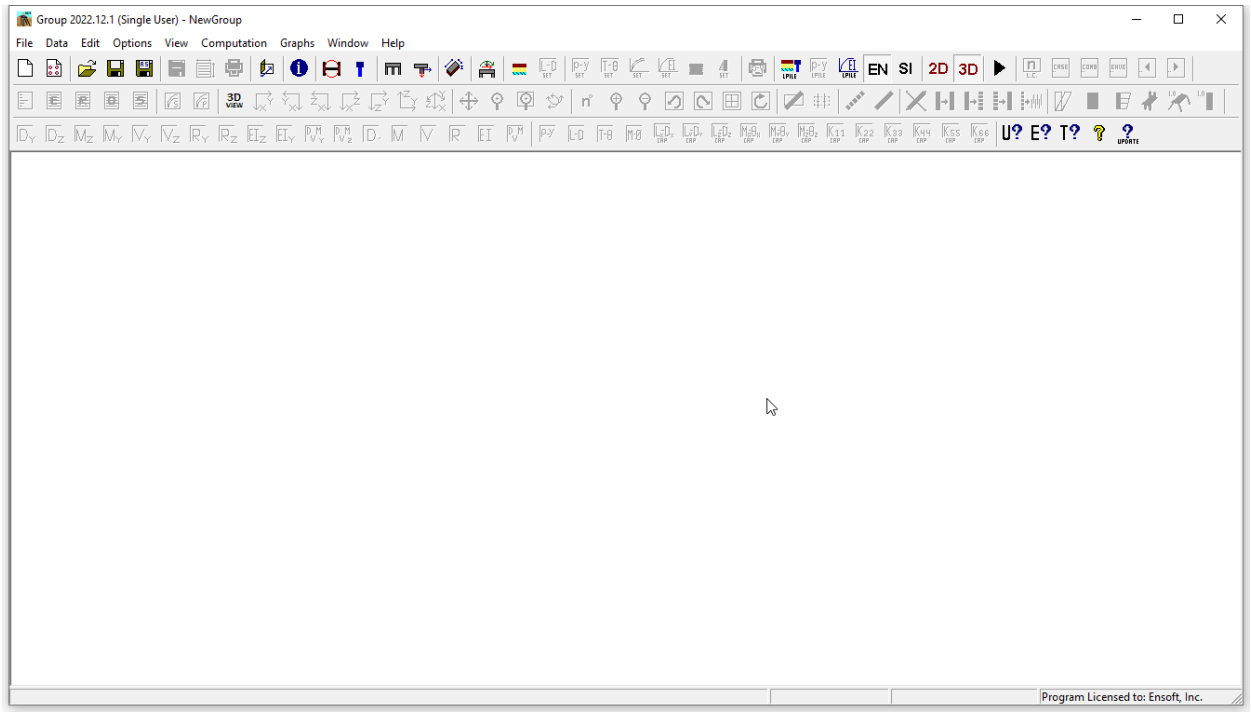


Figure 2.6 Starting window

2.4.2 File

This menu contains the following submenus. The choices are listed below, along with a general description of their use.

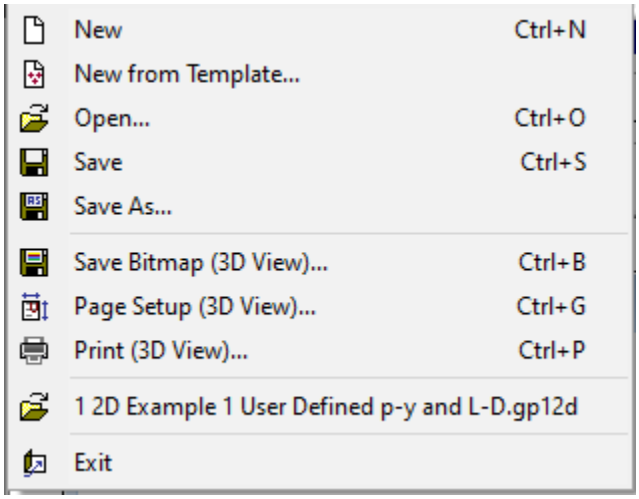


Figure 2.7 Options contained in the File Menu (includes 3D View options)

- **New** : Create a new data file. This opens up a blank workspace and closes the current project.
- **New from Template...** : Create a new data file for a rectangular or circular pile group configuration.
- **Open…** : Open an existing data file. Closes the current project.
- **Save** : Save input data under the current file name.
- **Save As…** : Save input data under a different file name. This option allows the user to save any opened or new input data file under a different file name or in a different directory. Any input data file saved under an existing file name will replace the contents of the existing file.
- **Exit** : Exit the program. This is selected to exit GROUP 2022. Any input data file that was modified and not yet saved to disk will produce a confirmation window before exiting the program.
- A list of recently opened files is displayed in between **Save As…** and **Exit**. By clicking on the menu item, the program will open the file.

If the **3D View** window is active three additional options appear.

  - **Save Bitmap (3D View)...** : Save the current view as a .bmp file.
  - **Page Setup (3D View)...** : Printing options.
  - **Print (3D View)...** :Print the current view.

Several additional files are created in every new GROUP 2022 project. A general description of these files is presented in Table 2.1. These files will be created in the same folder as the input file. Any of these files may be opened with standard text editors or word-processing programs.

File Name Extension	File Description	Example File
.gp12d	Input Data file	example.*gp12d*
.gp12e	Echo Print Data file	example.*gp12e*
.gp12o	Output Results file	example.*gp12o*
.gp12p	Plot Results file	example.*gp12p*
.gp12r	Processor Notes file	example.*gp12r*
.gp12t	Summary Results file	example.*gp12t*

Table 2.1 Files created in GROUP 2022 runs

2.4.3 Data

This menu contains several submenu choices as shown in Figure 2.8. The data menu for the Flexible Cap Version is shown in Figure 2.9. The choices are listed below, along with a general description of their use.

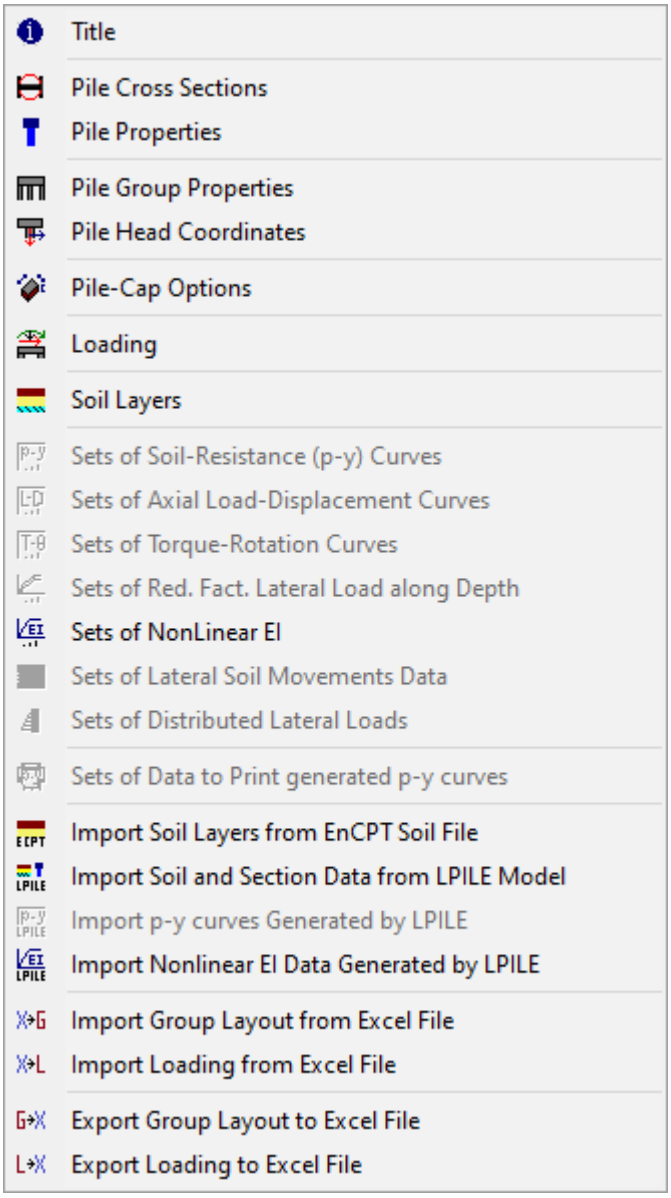


Figure 2.8 Options contained in the Data Menu

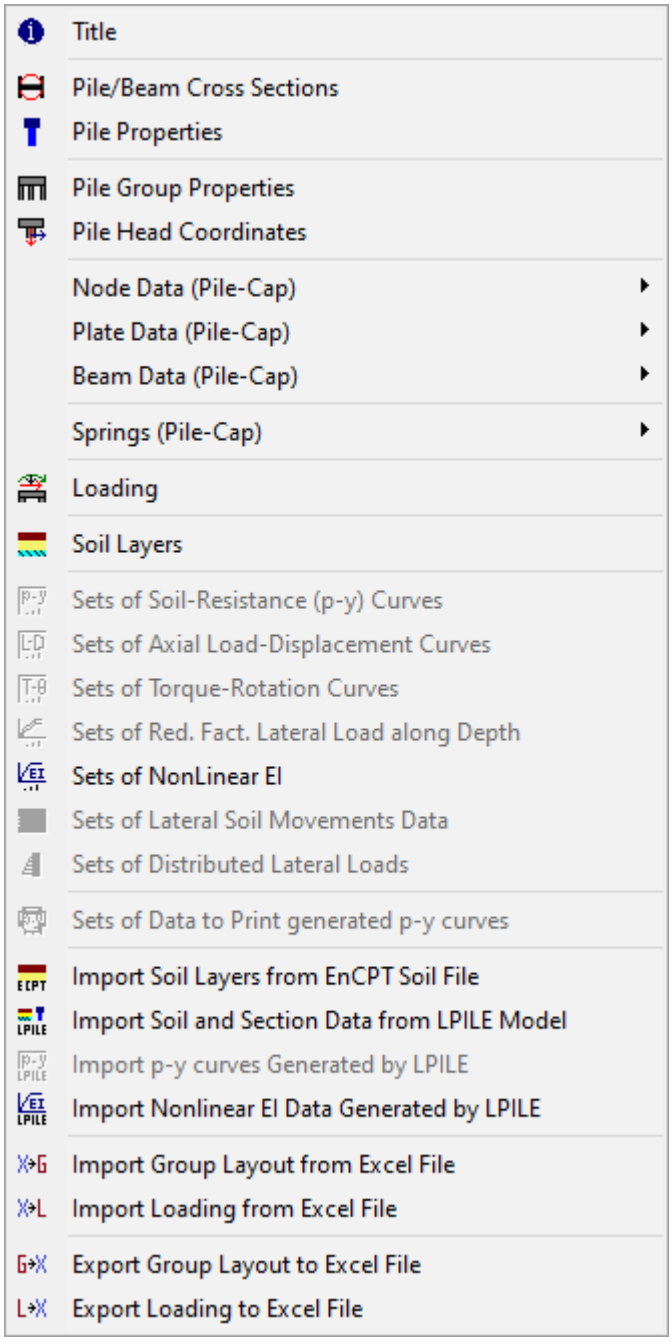


Figure 2.9 Options contained in the Data Menu – Flexible Cap Version

- **Title** : Input a single line of text with a general description of the project.
- **Pile Cross Sections :** Define the cross sections of the piles.
- **Pile Properties :** Specify material and dimensions for each pile type included in the different subgroups.
- **Pile Group Properties :** Specify attributes for pile subgroups.

- **Pile Head Coordinates :** Specify coordinates for the pile heads, as well as batter angles for the pile subgroups.
- **Loading** : Define load cases.
- **Pile-Cap Options :** Input the dimensions of the pile cap (Not available for the Flexible Cap Version).
- **Soil Layers :** Input the soil layers and associated soil properties.
- **Sets of Soil-Resistance (p-y) Curves :** An optional input that is disabled, as a default, for all new runs. It becomes available if the user chooses to input p-y curves to describe lateral behavior of soils (to do this, the user must check the item ***Enter User-Defined Soil Resistance (p-y) curves*** under the ***Options*** menu). Then the user can input data for specific p-y curves that represent the lateral behavior of the soil.
- **Sets of Axial Load-Displacement Curves :** An optional input that is disabled by default. Available if the user chooses to input specific curves that describe the axial behavior of all pile subgroups (to do this the user must check the item ***Enter User-Defined Load Displacement Curves*** under the ***Options*** menu).
- **Sets of Torque-Rotation Curves :** An optional input that is disabled by default for all new runs. Available if the user chooses to input curves to describe the torsional behavior of all pile subgroups (to do this the user must check the item ***Enter User-Defined Torque Rotation curves*** under the ***Options*** menu).
- **Sets of Red. Fact. Lateral Load along Depth :** An optional input that is disabled by default. Available if the user chooses to input reduction factors for lateral load along depth (to do this the user must check the item ***Reduction Factors for Lateral Load Along Depth*** under the ***Options*** menu).
- **Sets of Nonlinear EI :** This submenu is used to input specific curves that describe the nonlinear bending behavior of pile sections, i.e. moment curvature. This submenu is enabled if user defined moment curvature is assigned to any load case.
- **Sets Lateral Soil Movements Data :** This submenu is used to input the data for soil movement as a function of depth. This submenu is enabled if soil movement data is assigned to any load case.
- **Sets of Distributed Lateral Loads :** This submenu is used to input the data for distributed lateral loads in piles as a function of depth. The unit of the distributed load is force per unit length. The effect of the width of each pile should be taken into account in the data. This submenu is enabled if distributed lateral load is assigned to any load case.
- **Sets of Data to Print generated p-y Curves :** An optional input that is disabled by default. Available if the user chooses to print *p-y* curves for different pile subgroups and soil depths (to do this the user must check the item ***Print Generated p-y Curves*** under the ***Options*** menu).
- **Import Soil Layers from EnCPT Soil File :** This submenu is used to import soil layering data generated by the EnCPT program.
- **Import Soil and Section Data from LPILE Model :** This submenu is used to import soil layering data and/or cross sections data from a LPILE model (v2015/2016/2018/2019).
- **Import p-y Curves Generated by LPILE :** An optional input that is disabled, as a default, for all new runs. It becomes available if the user chooses to input p-y curves to describe lateral behavior of soils (to do this, the user must check the item ***Enter User-Defined Soil Resistance (p-y) curves*** under the

Options menu). Then the user can import specific p-y curves that represent the lateral behavior of the soil that were generated by the program LPILE (v2012/2013/2015/2016/2018/2019).

- **Import Nonlinear EI Data Generated by LPILE :** This submenu is used to import specific curves that describe the nonlinear bending behavior of pile sections, i.e. moment curvature, that were generated by the program LPILE (v2012/2013/2015/2016/2018/2019).
- **Import Group Layout from Excel File :** This submenu is used to import a group layout from an Excel File. User should prepare the Excel File following the template to avoid problems while importing the file.
- **Import Loading from Excel File :** This submenu is used to import loading from an Excel File. User should prepare the Excel File following the template to avoid problems while importing the file.
- **Export Group Layout to Excel File :** This submenu is used to export the group layout to an Excel File. User may use this option to create a basic template in order to define more complicated group layouts.
- **Export Loading to Excel File :** This submenu is used to export loading to an Excel File. User may use this option to create a basic template in order to define more complicated loading.

For the Flexible Cap Version the following options are included:

- **Node Data (Pile-Cap) :** This submenu is used to define node data for the nodes that belong to the flexible cap. It includes the Node Coordinates option.
- **Node Coordinates :** This submenu is used to define and or edit the node coordinates that belong to the flexible cap.
- **Plate Data (Pile-Cap) :** This submenu is used to define plate data for the plates that belong to the flexible cap. It includes the Plate Connectivity, Plate Offsets and Plate Sections options.
- **Plate Connectivity :** This submenu is used to define and or edit the element connectivity for the plate elements that belong to the flexible cap.
- **Plate Offsets:** This submenu is used to define and or edit plate offsets of plate elements that belong to the flexible cap.
- **Plate Sections:** This submenu is used to define and or edit the geometric and material properties of the plates that belong to the flexible cap.
- **Beam Data (Pile-Cap) :** This submenu is used to define beam data for the beams that belong to the flexible cap. It includes the Beam Connectivity, Beam Offsets, Beam End Rigid Zones and Beam Section Rotation options.
- **Beam Connectivity :** This submenu is used to define and or edit the element connectivity for the beam elements that belong to the flexible cap.
- **Beam Offsets :** This submenu is used to define and or edit plate offsets of beam elements that belong to the flexible cap.
- **Beam End Rigid Zones :** This submenu is used to define and or end rigid zones of beam elements that belong to the flexible cap.
- **Beam Section Rotation:** This submenu is used to define and or edit section rotation angles of beam elements that belong to the flexible cap.

- **Springs (Pile-Cap) :** This submenu is used to define springs for the elememts that belong to the flexible cap. It includes the Concentrated Spring at Nodes, Distributed Springs at Plates and Distributed Springs at Beams options.
- **Concentrated Springs at Nodes:** This submenu is used to define and or edit concentrated springs applied at the nodes.
- **Distributed Springs at Plates :** This submenu is used to define and or edituniform distributed springs applied at plates.
- **Distributed Springs at Beams :** This submenu is used to define and or edit uniform distributed springs applied at beams.

2.4.4 Edit

This menu is used to graphically edit the pile group layout.

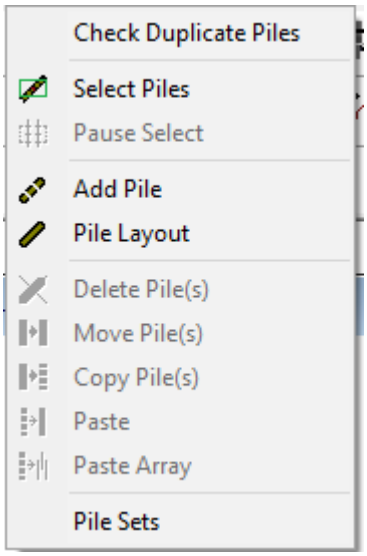
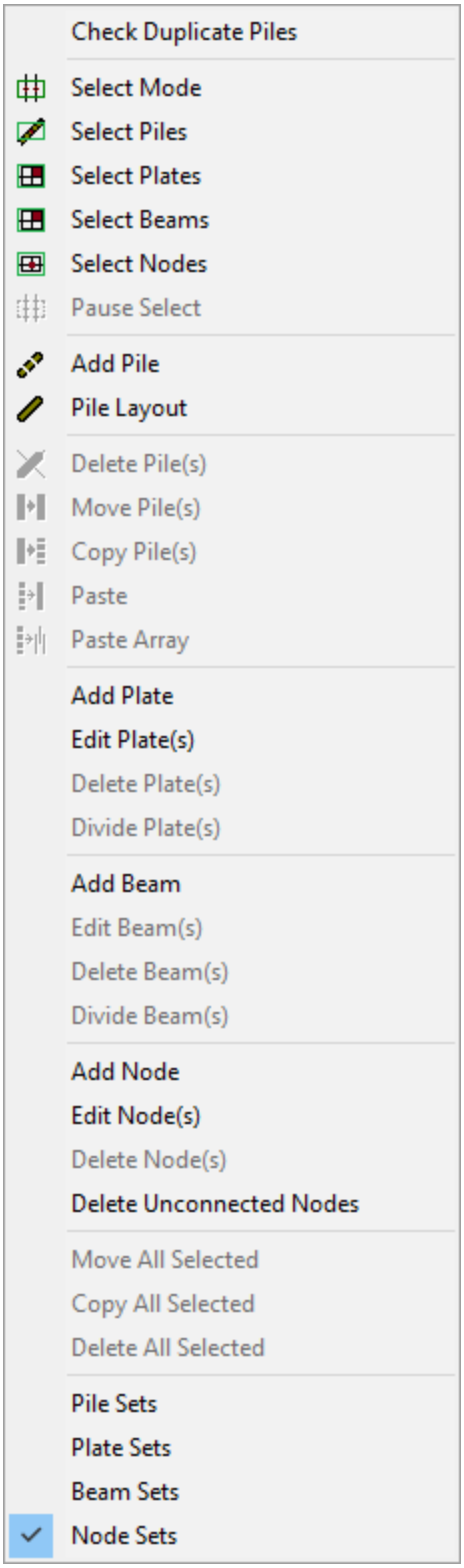


Figure 2.10 Options contained in the Edit Menu (3D View Active)


Check Duplicate Piles
Select Mode
Select Piles
Select Plates
Select Beams
Select Nodes
Pause Select
Add Pile
Pile Layout
Delete Pile(s)
Move Pile(s)
Copy Pile(s)
Paste
Paste Array
Add Plate
Edit Plate(s)
Delete Plate(s)
Divide Plate(s)
Add Beam
Edit Beam(s)
Delete Beam(s)
Divide Beam(s)
Add Node
Edit Node(s)
Delete Node(s)
Delete Unconnected Nodes
Move All Selected
Copy All Selected
Delete All Selected
Pile Sets
Plate Sets
Beam Sets
Node Sets

- **Figure 2.11** Options contained in the Edit Menu (3D View Active) – Flexible Cap Version

- **Check Duplicate Piles :** By clicking on this option the program will check to see if two or more piles have the same coordinates and properties. If a duplicate pile(s) is found, the program will ask if the user wants to delete the duplicate pile(s). This is the only option active in 2D mode.
- **Select Piles :** In this mode piles can be selected graphically.
- **Pause Select :** This option is to pause the selection, in order to change the view by translating, zooming and/or rotating.
- **Add Pile :** Add a new pile into the model.
- **Pile Layout :** Review or edit existing pile information.
- **Delete Pile(s) :** Delete the selected pile.
- **Move Pile(s) :**Translate the selected piles.
- **Copy Pile(s) :** Copy the selected pile's information to the clipboard.
- **Paste:** Paste the piles already in the clipboard.
- **Paste Array :** Paste selected piles in the clipboard using a rectangular or circular pattern.
- **Pile Sets :** This option is useful to define pile sets than can be used later to apply loads and/or boundary conditions.

For the Flexible Cap Version the following options are included:

- **Select mode :** In this option all the objects (piles, plates, beams and/or nodes) that belong to the model can be selected graphically.
- **Select Plates :** In this mode the plates that belong to the flexible cap can be selected graphically.
- **Select Beams :** In this mode the beams that belong to the flexible cap can be selected graphically.
- **Select Nodes :** In this mode the nodes that belong to the flexible cap can be selected graphically.
- **Add Plate :** Add a new plate into the model.
- **Edit Plate(s) :** Edit the connectivity, offsets and material properties index of a plate or plates.
- **Delete Plate(s) :** Delete selected plate(s).
- **Divide Plate(s) :** Divide selected plate(s).
- **Add Beam :** Add a new beam into the model.
- **Edit Beam(s) :** Edit the connectivity, offsets, rigid end zones and section index of a beam or beams.
- **Delete Beam(s) :** Delete selected beam(s).
- **Divide Beam(s) :** Divide selected beam(s).
- **Add Node :** Add a new node into the model.
- **Edit Node(s) :** Edit the coordinates of a node or nodes.
- **Delete Node(s) :** Delete selected node(s).
- **Delete Unconnected Node(s) :** Delete the nodes that are not connected to a plate and/or beam.
- **Move All Selected :** Move all objects selected: piles, plates, beams and/or nodes.
- **Copy All Selected :** Copy all objects selected: piles, plates, beams and/or nodes.
- **Delete All Selected :** Delete all objects selected: piles, plates, beams and/or nodes.

- **Plate Sets :** This option is useful to define plate sets than can be used later to apply loads and/or boundary conditions.
- **Beam Sets :** This option is useful to define beam sets than can be used later to apply loads and/or boundary conditions.
- **Node Sets :** This option is useful to define node sets than can be used later to apply loads and/or boundary conditions.

2.4.5 Options

This menu controls several program variables and preferences and is shown below.

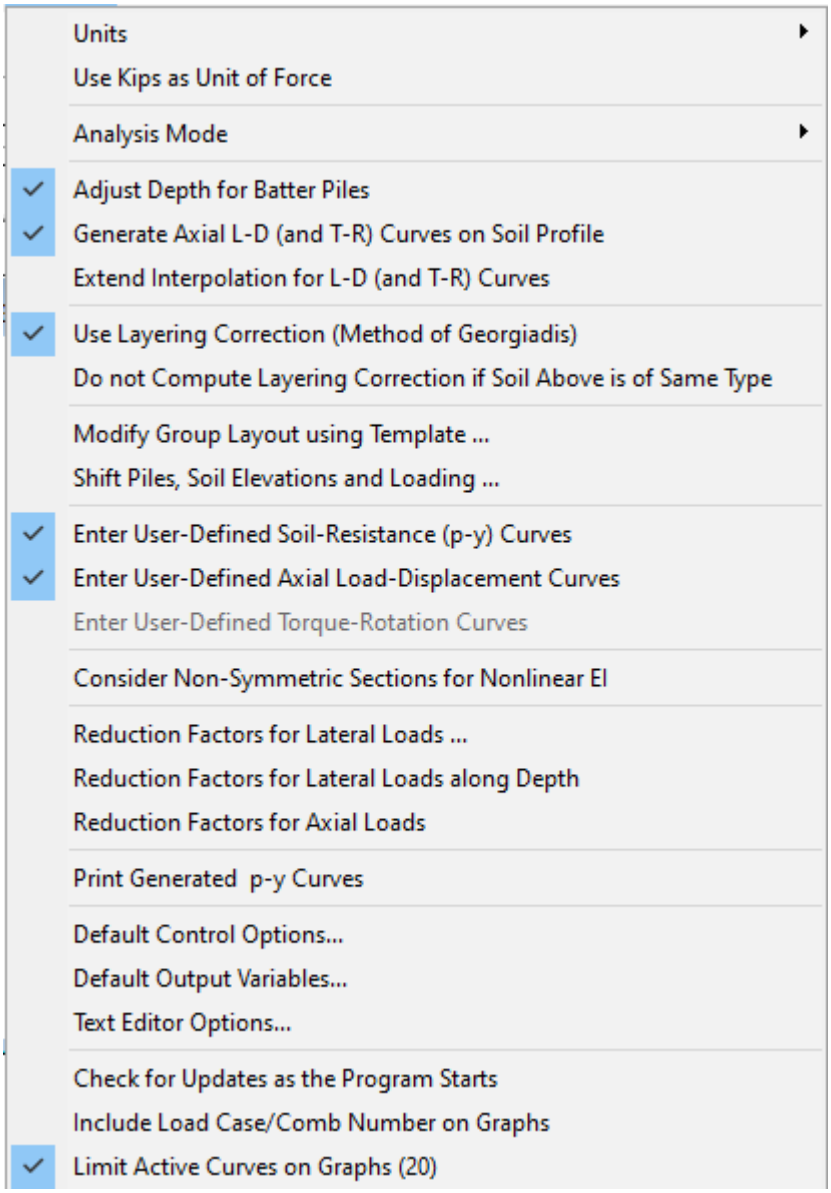


Figure 2.12 Options contained in the Options Menu

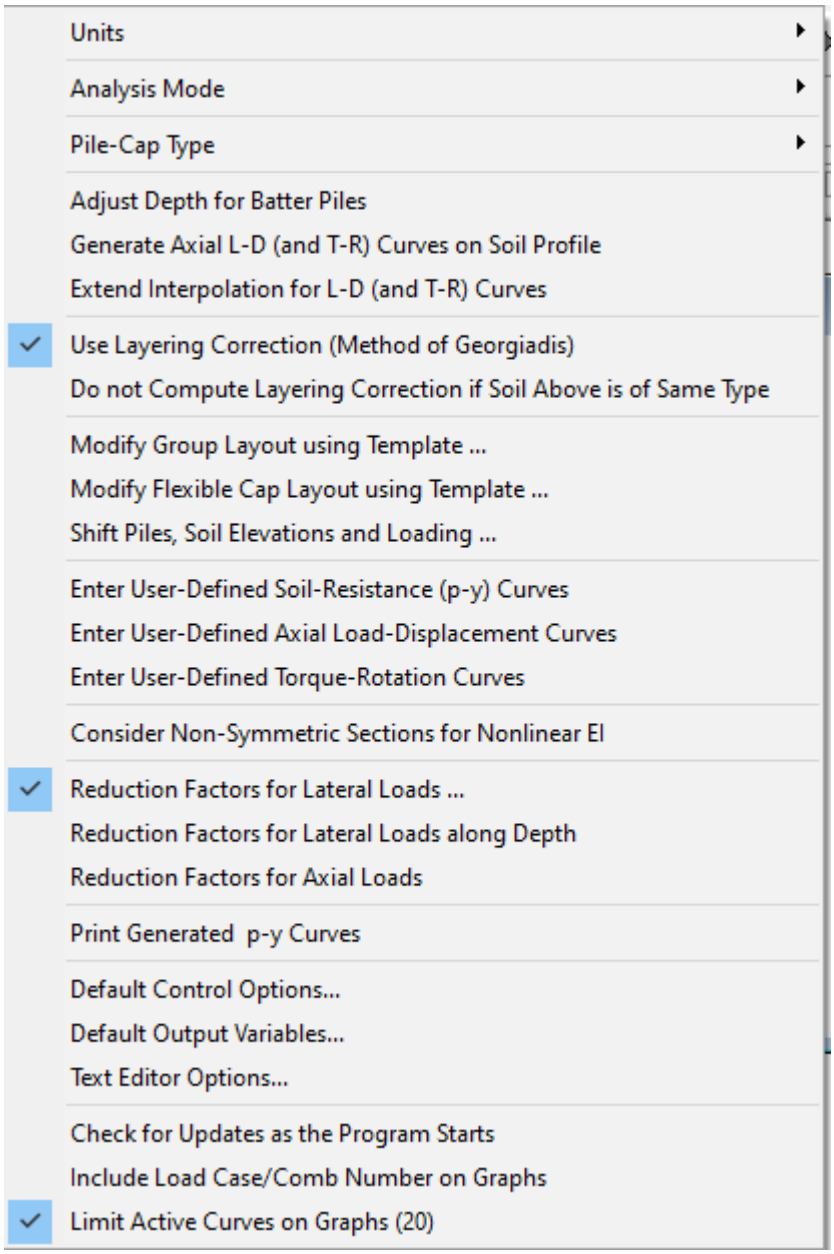


Figure 2.13 Options contained in the Options Menu – Flexible Cap Version

- **Units** : This option allows the user to select either **English Units,** using pounds(lbs), and inches(in) or **SI Units,** using kilonewtons (kN), and meters (m). English units are enabled by default. Whenever the user changes the units of measurement, previously inputted values will be automatically converted to the new units.
- **Use Kips as Unit of Force** : This option allows the user to define Kips instead of pounds for units of loading while using **English Units**. Whenever the user changes the units of measurement, previously inputted values will be automatically converted to the new units.
- **Analysis Mode** : This option is used to specify analysis based on either a two-dimensional (**2D**) or three-dimensional model (**3D**).
- **Adjust Depth for Batter Piles** : By selecting this option the program will adjust the depth for estimation of p-y curves, t-z and Q-w curves in batter piles. If the option if not selected, the program will use the length (as previous versions of GROUP).
- **Generate Axial L-D (and T-R) curves on Soil Profile** : By selecting this option the program will generate the axial load versus displacement curves (as well as the torque versus rotation curves) considering the soil profile and the batter angle. If the option if not selected, the program will generate one curve per pile property (as previous versions of GROUP).
- **Extend Interpolation for L-D (and T-R) curves** : By selecting this option the program will extend the interpolation of the L-D (and T-R curves) towards the maximum defined. The program will use the value of load (torque) corresponding the to the maximum displacement (rotation) if the displacement (rotation) is larger than the maximum defined. The program will use the value of load (torque) corresponding the to the minimum displacement (rotation) if the displacement (rotation) is smaller than the minimum defined. If this option is not checked the program will use a value of zero in those events (as previous versions of GROUP).
- **Use Layering Correction (Method of Georgiadis)** : By selecting this option the program will use layering correction. By unchecking this option, the program will not use a layering correction, so the equivalent depth of each layer will be equal to actual depth of the layer.
- **Do not Compute Layering Correction if Soil Above is of Same Type** : By selecting this option the program will not estimate layering correction if soil above of current layer is of same material.
- **Modify Group Layout using Template :** Create a new pile configuration for rectangular or circular pile groups.
- **Shift Piles and Soil Elevations :** This option allows the user to move vertically the piles and/or the soil layers.
- **Enter User-Defined Soil-Resistance (p-y) Curves** : By selecting this option, the user is able to enter curves that define the lateral behavior of soils in the item **Sets of Soil-Resistance (p-y) Curves** under the **Data** menu.
- **Enter User-Defined Load-Displacement Curves :** By selecting this option, the user is able to enter curves that define the axial behavior of piles in the item **Sets of Axial Load-Displacement Curves** under the **Data** menu.
- **Enter User-Defined Torque-Rotation Curves :** By selecting this option, the user is able to enter curves that define the torsional behavior of piles in the item **Sets of Torque-Rotation Curves** under the **Data** menu.

- **Consider Non-Symmetric Sections for Nonlinear EI :** By selecting this option, the program will be able to estimate nonlinear bending stiffness for non-symmetric sections. By unchecking this option, the program will be able to consider only symmetric sections for the estimation of nonlinear bending stiffness.
- **Reduction Factors for Lateral Loads :** This option controls the method to be used for computing lateral-load reduction factors to account for closely-spaced piles. The submenu options are:
    - **Automatically Generate :** Reduction factors will be internally generated by the program based on data input.
    - **Specify :** Reduction factors will be specified by the user in the **Pile Properties Window**.
    - **Ignore Effects :** The effect due to closely-spaced piles will be ignored.
- **Reduction Factors for Lateral Load Along Depth** : By selecting this option, the user is able to enter the curves that define the reduction factors for lateral load along depth in the item **Sets of Red. Fact. Lateral Load along Depth** under the **Data** menu.
- **Reduction Factors for Axial Loads :** This option allows the user to input a reduction factor for pile axial capacity in order to account for closely-spaced piles.
- **Print Generated p-y Curves** : Activates **Sets of Data to Print p-y Curves** under the **Data** menu.
- **Default Control Options…:** This option allows the user to set the default values for analysis parameters.
- **Default Output Variables…:** This option allows the user to set default variables along the pile to be printed in the output file.
- **Text Editor Options…:** This option allows the user to set the preferred text editor or word processor. As a default, the "c:\windows\notepad.exe" is used to set Microsoft Notepad as the default text editor.
- **Check for Updates as the Program Starts** : Instructs GROUP to check the Ensoft website for software updates when the program starts. An active internet connection is required.
- **Include Load Case/Comb. Number on Graphs** : This option allows the user to include the load case or combination number in the result graphs.
- **Limit Active Curves on Graphs (20)** : This option allows the user to limit the number of curves displayed in the graphs to 20.

For the Flexible Cap Version, the following options are included:

- **Pile-Cap Type :** This option allows the user to select the model for the pile cap either rigid or flexible.
- **Modify Flexible Cap Layout using Template :** Create a new flexible cap configuration for rectangular pile-cap geometry.

2.4.6 View

This menu is used to graphically review the geometric configuration of the modeled pile group. It also allows the user to activate or deactivate the different toolbars. The **View** menu is shown in Figure 2.14 and the different submenu choices are described below.

- **Model Summary:** Provides a list of the parameters for the current model.
- **Pile Top:** Activates a view of the group model from above.
- **3D View:** Provides a 3D view of the group model and activates several different preset viewing angles, as well as transformation options that allow the user to customize the viewing angle.
- **View Options:** These options allow the user to switch the viewing angle to one of the presets.
- **Transform Options:** These options allow the user to pan, zoom in or out, and rotate the viewing angle.
- **Toolbar Options:** These options activate or deactivate the various toolbars at the top of the screen.

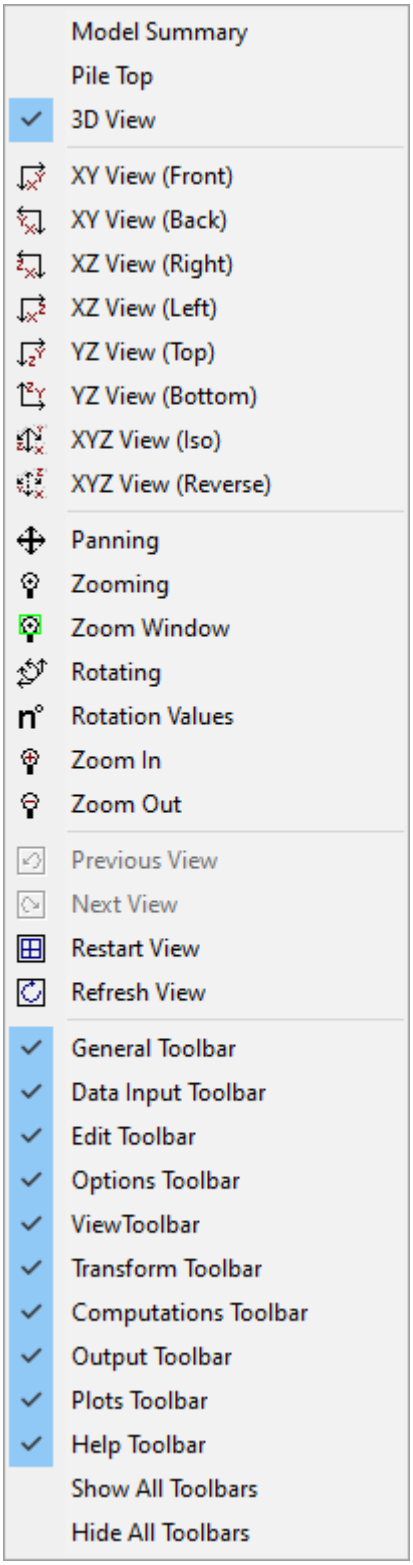


Figure 2.14 Options contained in the View Menu (3D View Active)

2.4.7 Computation Options

The **Computation** menu is provided to run the analytical computations after all data is entered and saved. After the computations are executed, this menu also provides options for the review of plain-text input and output data. The listing of the **Computation** menu is shown in Figure 2.15 and the different submenu choices are described below.

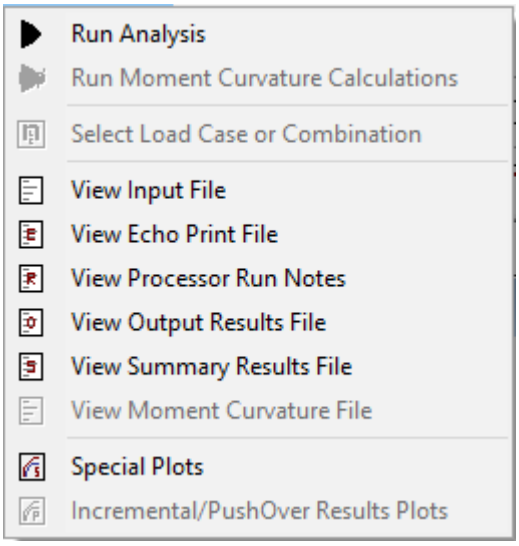


Figure 2.15 Options contained in the Computation Menu

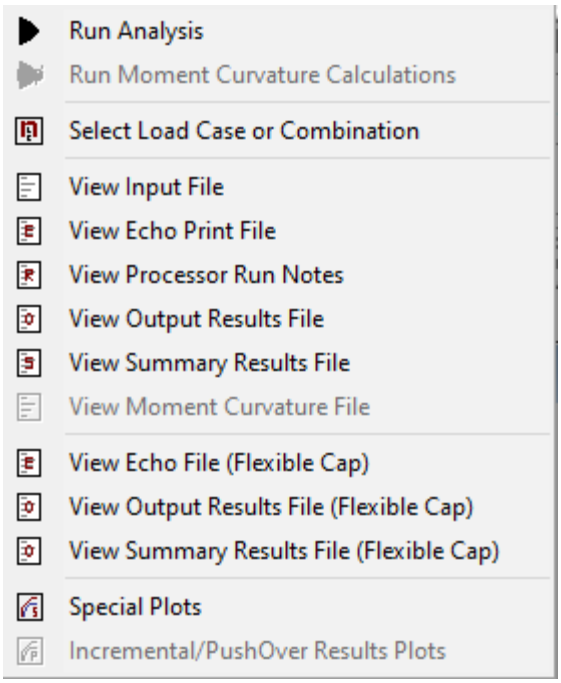


Figure 2.16 Options contained in the Computation Menu – Flexible Cap Version

- **Run Analysis:** This option runs the analytical computations. Only use this after all data has been entered and *saved*.
- **Run Analysis:** This option runs the moment curvature calculations only if a nonlinear section if at least one nonlinear section is defined.
- **Select Load Case or Combination:** This option opens the **Select Load Case or Combination** window. There the user can select the load case or load combination that will be active for displaying the graphical results.
- **View Input File:** This submenu is used to view the input data file in text mode. This command becomes active after new data files have been saved to disk or when opening existing data files.
- **View Echo Print File:** This submenu is used to view the echo print data file in text mode that is automatically produced during each analytical run. This command becomes active after an analytical run, or when opening previously executed data files
- **View Processor Run Notes:** This submenu is used to view the notes provided during processing. The command becomes active after an analytical run, or when opening previously executed data files. Reviewing the notes is essential for debugging unsuccessful runs.
- **View Output Results File:** This submenu is used to view the output file that is automatically produced during each analytical run. This command becomes active after an analytical run, or when opening previously executed data files.

- **View Summary Results File:** This submenu is used to view the summary file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files.
- **View Moment Curvature File:** This submenu is used to view the moment curvature file that is produced after using the run moment curvature calculations option. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files.
- **Special Plots:** This submenu is used to open the Special Plots Windows that allows the user to produce various plots. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files.
- **Incremental/PushOver Results Plots:** This submenu is used to open the Incremental/PushOver Plots Windows that allows the user to produce various plots. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files if at least one load case includes Incremental/PushOver analysis.

For the Flexible Cap Version the following options are included:

- **View Echo File (Flexible Cap):** This submenu is used to view the echo print data file of the flexible cap in text mode that is automatically produced during each analytical run. This command becomes active after an analytical run, or when opening previously executed data files when a flexible cap is defined.
- **View Output Results File (Flexible Cap):** This submenu is used to view the output file of the flexible cap that is automatically produced during each analytical run. This command becomes active after an analytical run, or when opening previously executed data files when a flexible cap is defined.
- **View Summary Results File (Flexible Cap):** This submenu is used to view the summary file of the flexible cap that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files when a flexible cap is defined.

2.4.8 Graphs Menu

The **Graphs** menu is provided to display the plots available after the analysis has been completed. Most of the graphs are based on the current load case or load combination. The current Load Case is displayed in the second box of the status line. The listing of the **Graphs** menu is shown in Figure 2.17 and Figure 2.18 and the different submenu choices are described below.

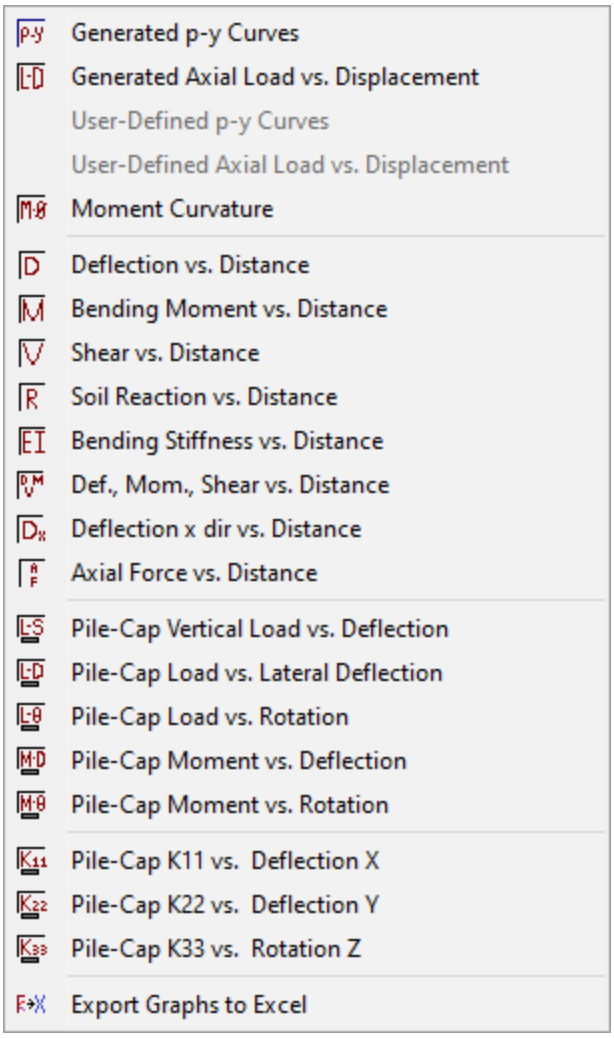


Figure 2.17 Options contained in the Graphs Menu. 2D Model

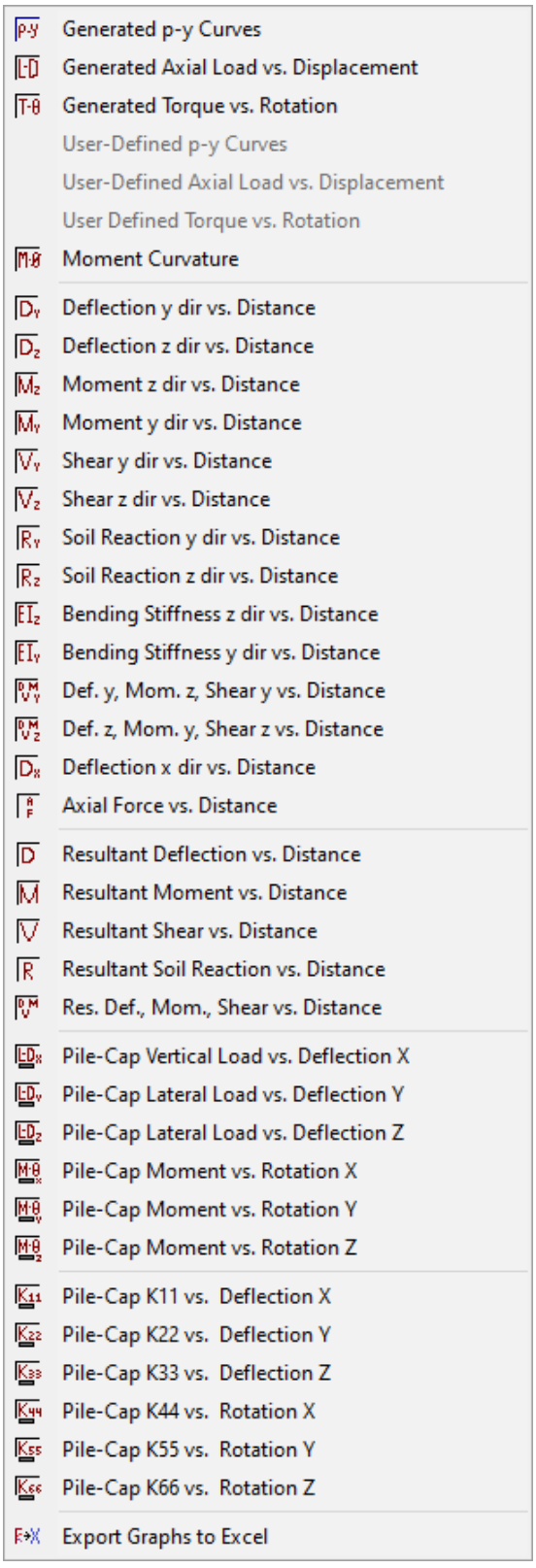


Figure 2.18 Options contained in the Graphs Menu. 3D Model

- **Generated p-y Curves:** Plots internally generated p-y curves at user defined depths.
- **Generated Axial Load vs. Displacement:** Plots internally generated axial load versus pile-head displacements.
- **Generated Torque vs. Rotation:** Plots internally generated torque versus pile-head rotations. (3D only)
- **User Defined p-y Curves:** Plots user defined p-y curves.
- **User Defined Axial Load vs. Displacement:** Plots user-defined axial load versus pile-head displacements.
- **User Defined Torque vs. Rotation:** Provides a plot of user-defined torque versus pile-head rotations. (3D Only)
- **Moment Curvature:** Plots moment curvature for all sections, for both internally generated and user-defined.
- **Deflection vs. Distance:** Plots lateral pile deflection versus distance from pile top on each individual pile. (2D Only)
- **Bending Moment vs. Distance:** Plots the bending moment versus distance from pile top on each individual pile.
- **Shear vs. Distance:** Plots the shear versus distance from pile top on each individual pile. (2D Only)
- **Soil Reaction vs. Distance:** Plots of soil reaction versus distance from pile top on each individual pile. (2D Only)
- **Bending Stiffness vs. Distance:** Plots of bending stiffness versus distance from pile top on each individual pile. (2D Only)
- **Def., Mom., Shear vs. Distance:** Plots in one graph the lateral pile deflection, bending moment and shear versus distance from pile top on each individual pile. (2D Only)
- **Deflection y dir vs. Distance:** Plots lateral pile deflection in the local y-direction versus distance from pile top on each individual pile. (3D Only)
- **Deflection z dir vs. Distance:** Plots lateral pile deflection in the local z-direction versus distance from pile top on each individual pile. (3D Only)
- **Moment z dir vs. Distance:** Plots bending moment about the local z-axis versus distance from pile top on each individual pile. (3D Only)
- **Moment y dir vs. Distance:** Plots bending moment about the local y-axis versus distance from pile top on each individual pile. (3D Only)
- **Shear y dir vs. Distance:** Plots lateral shear in the local y-direction versus distance from pile top on each individual pile. (3D Only)
- **Shear z dir vs. Distance:** Plots lateral shear in the local z- direction versus distance from pile top on each individual pile. (3D Only)
- **Soil Reaction y dir vs. Distance:** Plots lateral soil reaction the local y- direction versus distance from pile top on each individual pile. (3D Only).
- **Soil Reaction z dir vs. Distance:** Plots lateral soil reaction the local z- direction versus distance from pile top on each individual pile. (3D Only)

- **Bending Stiffness z dir vs. Distance:** Plots of bending stiffness in the local z-direction versus distance from pile top on each individual pile. (3D Only)
- **Bending Stiffness y dir vs. Distance:** Plots of bending stiffness in the local y direction versus distance from pile top on each individual pile. (3D Only)
- **Resultant Deflection vs. Distance:** Plots lateral pile resultant deflection versus distance from pile top on each individual pile. (3D Only)
- **Def. y, Mom. z, Shear y vs. Distance:** Plots in one graph the lateral pile deflection y, bending moment z and shear y versus distance from pile top on each individual pile. (3D Only).
- **Def. z, Mom. y, Shear z vs. Distance:** Plots in one graph the lateral pile deflection z, bending moment y and shear z versus distance from pile top on each individual pile. (3D Only).
- **Resultant Moment vs. Distance:** Plots the resultant moment versus distance from pile top on each individual pile. (3D Only)
- **Resultant Shear vs. Distance:** Plots the resultant shear versus distance from pile top on each individual pile. (3D Only)
- **Resultant Soil Reaction vs. Distance:** Plots resultant soil reaction versus distance from pile top on each individual pile. (3D Only)
- **Res. Def., Mom., Shear vs. Distance:** Plots in one graph the resultant lateral pile deflection, resultant bending moment and resultant shear versus distance from pile top on each individual pile. (3D Only)

The following are only available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case.

- **Pile Cap Vertical Load vs. Deflection:** Plots incremental vertical load versus vertical deflection at the pile cap. (2D Only)
- **Pile Cap Load vs. Lateral Deflection:** Plots incremental lateral load versus lateral deflection at the pile cap. (2D Only)
- **Pile Cap Load vs. Rotation** Plots incremental lateral load versus rotation at the pile cap. (2D Only)
- **Pile Cap Moment vs. Deflection:** Plots incremental bending moment versus lateral deflection at the pile cap. (2D Only)
- **Pile Cap Moment vs. Rotation:** Plots incremental bending moment versus rotation at the pile cap. (2D Only)
- **Pile Cap Vertical Load vs. Deflection X/Y/Z:** Plots incremental vertical load versus vertical deflection at the pile cap. (3D Only)
- **Pile Cap Moment vs. Rotation X/Y/Z:** Plots incremental moment versus rotation about the X/Y/Z-axis at the pile cap. (3D Only)
- **Pile Cap K11 vs. Deflection X:** Plots the stiffness coefficient K11 (Kxx) versus vertical deflection at the pile cap.
- **Pile Cap K22 vs. Deflection Y:** Plots the stiffness coefficient K22 (Kyy) versus lateral deflection in the Y-direction at the pile cap.

- **Pile Cap K33 vs. Rotation Z:** Plots the stiffness coefficient K33 (Krzrz) versus rotation along the Z-axis at the pile cap.
- **Pile Cap K33 vs. Deflection Z:** Plots the stiffness coefficient K33 (Kzz) versus lateral deflection in the Z-direction at the p**ile cap.** (3D Only)
- **Pile Cap K44 vs. Rotation X:** Plots the stiffness coefficient K44 (Krxrx) versus rotation about the X-axis at the pile cap. (3D Only)
- **Pile Cap K55 vs. Rotation Y:** Plots the stiffness coefficient K55 (Kryry) versus rotation about the Y-axis at the pile cap. (3D Only)
- **Pile Cap K66 vs. Rotation Z:** Plots the stiffness coefficient K66 (Krzrz) versus rotation about the Z-axis at the pile cap. (3D Only)

The following are only available if **Perform Incremental PushOver/Analysis** is specified in the input for the current load case.

- **Pile Cap Vertical Load vs. Deflection:** Plots incremental vertical load versus vertical deflection at the pile cap. (2D Only)
- **Pile Cap Load vs. Lateral Deflection:** Plots incremental lateral load versus lateral deflection at the pile cap. (2D Only)
- **Pile Cap Load vs. Rotation** Plots incremental lateral load versus rotation at the pile cap. (2D Only)
- **Pile Cap Moment vs. Deflection:** Plots incremental bending moment versus lateral deflection at the pile cap. (2D Only)
- **Pile Cap Moment vs. Rotation:** Plots incremental bending moment versus rotation at the pile cap. (2D Only)
- **Pile Cap Vertical Load vs. Deflection X/Y/Z:** Plots incremental vertical load versus vertical deflection at the pile cap. (3D Only)
- **Pile Cap Moment vs. Rotation X/Y/Z:** Plots incremental moment versus rotation about the X/Y/Z-axis at the pile cap. (3D Only)

2.4.9 Plot Menu

The **Plot** menu is available when a graph is displayed. The listing of the **Plot** menu is shown in Figure 2.19 and the different submenu choices are briefly described below

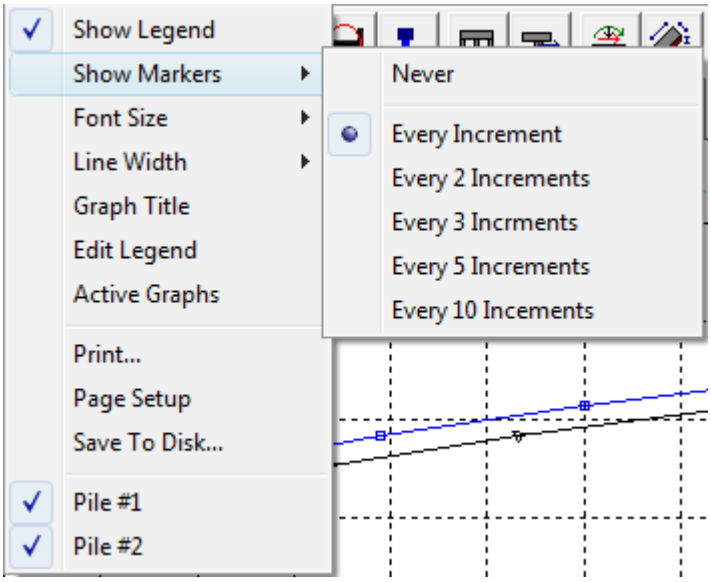


Figure 2.19 Options contained in the Plot Menu

- **Show Legend:** Display the legend in the current plot.
- **Show Markers:** Change the increment for displaying the curve markers. The submenu options are:
  - **Never:** Use this option to not include markers at each curve.
  - **Every Increment:** Show markers every increment (default).
  - **Every 2/3/5/10 Increments:** Show markers every 2/3/5/10 increments.
- **Font Size:** Change the font size. The submenu options are:
  - **Default:** Use this option to use the default size estimated by the program.
  - **1.5*Default:** Uses a font size equal to 1.5 times the default size.
  - **2*Default:** Uses a font size equal to 2 times the default size.
  - **2.5*Default:** Uses a font size equal to 2.5 times the default size.
  - **3*Default:** Uses a font size equal to 3 times the default size.
  - **4*Default:** Uses a font size equal to 4 times the default size.

- **Line Width:** Change the line width. The submenu options are:
    - **Default:** Use this option to use the default width estimated by the program.
    - **2*Default:** Uses a line width equal to 2 times the default width.
    - **4*Default:** Uses a line width equal to 4 times the default width.
    - **6*Default:** Uses a line width equal to 6 times the default width.
    - **10*Default:** Uses a line width equal to 10 times the default width.
- **Print... :** Print the current graph.
- **Page Setup:** Set parameters (margins, orientation) for printing the current graph.
- **Save To Disk.. :** Save the current graph as a bitmap file.
- **Graph Title:** Include a title for the current graph.
- **Edit Legend:** Edit the default legend.
- **Active Graphs:** Hide or show selected curves. Very useful if the graph contains a lot of curves.
- **Curve # n:** If this option is selected (default), the curve #n will be shown. The number of curves listed will be equal to the number of the active curves.

2.4.10 Show 3D View Menu

The **Show 3D View** menu is available when the **3D View** window is displayed.

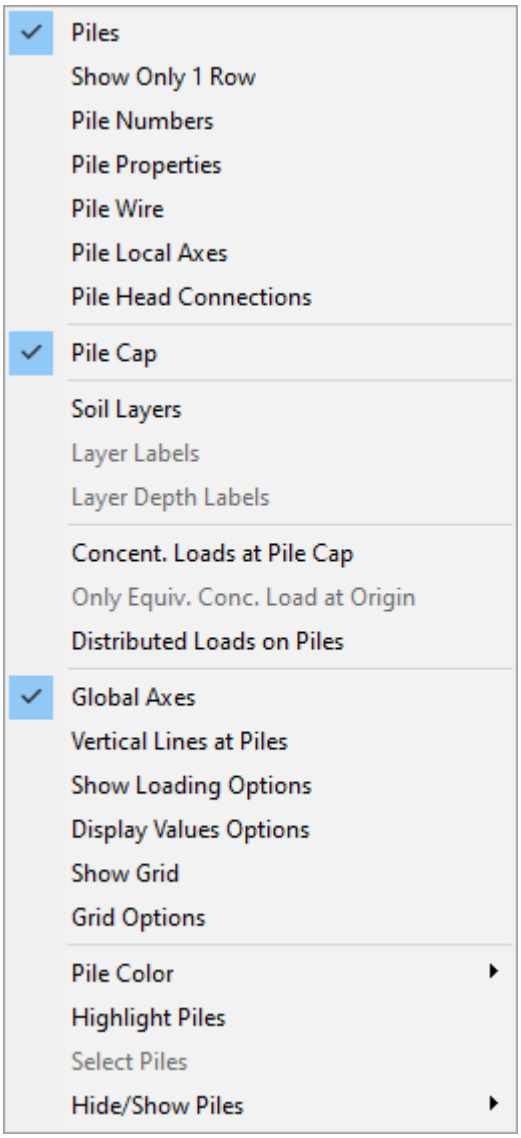


Figure 2.20 Options contained in the Show 3D View Menu

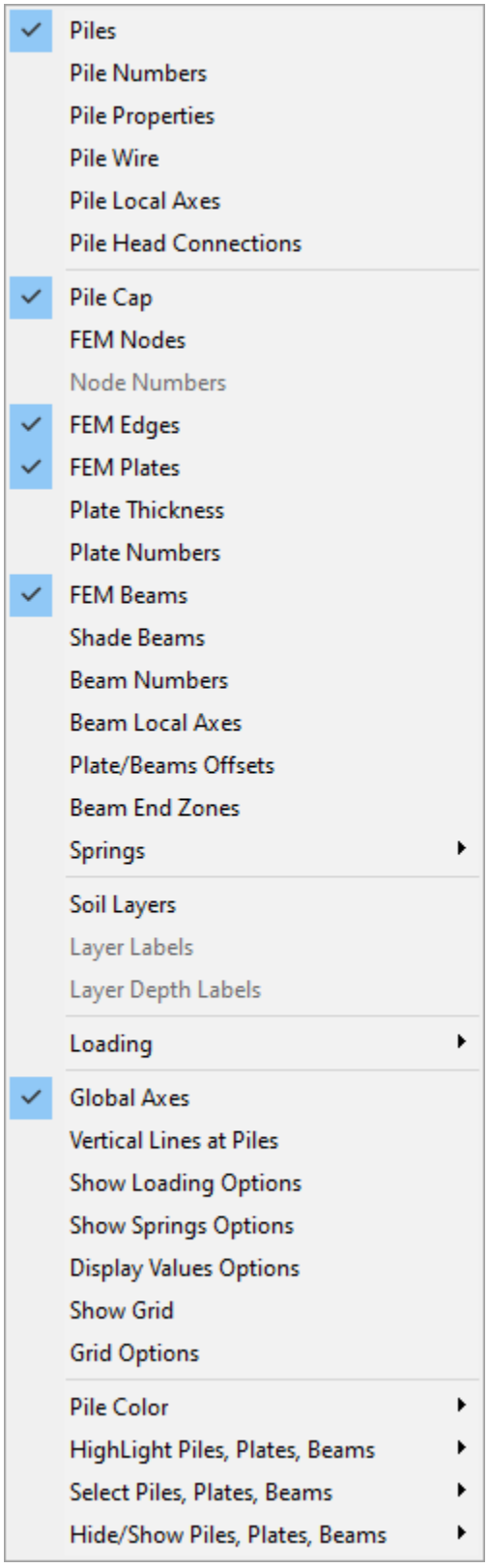

Piles
Pile Numbers
Pile Properties
Pile Wire
Pile Local Axes
Pile Head Connections
Pile Cap
FEM Nodes
Node Numbers
FEM Edges
FEM Plates
Plate Thickness
Plate Numbers
FEM Beams
Shade Beams
Beam Numbers
Beam Local Axes
Plate/Beams Offsets
Beam End Zones
Springs
Soil Layers
Layer Labels
Layer Depth Labels
Loading
Global Axes
Vertical Lines at Piles
Show Loading Options
Show Springs Options
Display Values Options
Show Grid
Grid Options
Pile Color
HighLight Piles, Plates, Beams
Select Piles, Plates, Beams
Hide/Show Piles, Plates, Beams

Figure 2.21 Options contained in the Show 3D View Menu – Flexible Cap Version

- **Piles:** Show/Hide the piles.
- **Show Only One Row:** Show only one row of piles (only for 2D models)
- **Pile Numbers**: Toggles the pile numbers at each pile head.
- **Pile Wire/Shade:** Toggle between showing the piles as lines, or as based on cross section dimensions, respectively.
- **Pile Local Axes:** Show/ Hide the local axes at the top of each pile head. This option is very useful for checking the direction of local axes in the piles.
- **Pile Head Connections:** Show/ Hide the pile head connections at the top of each pile head. This option is very useful for checking the pile head connection defined for the piles.
- **Pile Properties**: Toggles the pile properties at each pile head.
- **Pile Cap:** Show/ Hide the pile cap.
- **Soil Layers:** Show/ Hide the soil layers.
- **Layer Labels:** Toggle the soil labels. Only available if the **Soil Layers** option is selected.
- **Layer Depths Labels:** Toggle the depth labels. Only available if the **Soil Layers** option is selected.
- **Concent. Loads at Pile Cap:** Show the concentrated loads at the pile cap for the current load case.
- **Onlt Equiv. Conc. Load at Origin:** Show onlt the equivalent concentrated load applied at the origin of coordinates for the current load case.
- **Distributed Loads at Piles:** Show/Hide the distributed loads at piles for the current load case.
- **Global Axes:** Show/ Hide the global axes.
- **Vertical Lines at Piles:** Show/ Hide vertical lines that start from the top of each pile head. This option is very useful in the case of batter piles.
- **Show Loading Options:** Open the Loading Options Window. There the user can change the default drawing for concentrated and distributed loads.
- **Display Values Options:** Open the Display Values Window. There the user can change the default format for values displayed in the 3D View.
- **Show Grid:** Show/Hide a three-dimensional grid. This option is very useful for checking the extent of the model.
- **Grid Options:** Change the increment values for the 3D grid.
- **Pile Color:** This option is useful to color the piles for a specific property. The options available are:
    - **Default:** All the piles have the same color.
    - **By Pile Properties:** Piles with same pile property have the same color.
    - **By Cross Sections:** Pile increments with same pile cross section have the same color.
    - **By Pile Numbers:** Each pile has a different color. It may be repeated every 15 piles.
- **Highlight Piles:** Navigate through the pile numbers, showing the current pile with a different color.
- **Select Piles:** Open a dialog box for various ways to select piles.
- **Hide/Show Piles:** This option is useful when a user wants to review a specific section of the pile group. Here the user can:
    - **Hide Selected Piles:** Hide the selected piles. (Use **Edit→ Select Mode)**

- **Hide Except Selected Piles:** Hide the piles that are not currently selected.
- **Reverse Hiding:** Hide piles that are shown and show piles that are hidden.
- **Show All Piles:** Show all the piles.

For the Flexible Cap Version the following options are included:

- **FEM Nodes:** Show/Hide the nodes of the finite element nodes that dbelong to the flexible pile cap.
- **Node Numbers**: Toggles the node numbers.
- **FEM Edges:** Show/Hide the edges of the plates that belong to the flexible pile cap.
- **FEM Plates:** Show/Hide the plates that belong to the flexible pile cap.
- **Plate Thickenss:** Toggle between showing the plates as 2D planes, or showing them as 3D solids.
- **Plate Numbers**: Toggles the plate numbers.
- **FEM Plates:** Show/Hide the plates that belong to the flexible pile cap.
- **Plate Thickenss:** Toggle between showing the plates as 2D planes, or showing them as 3D solids.
- **Plate Numbers**: Toggles the plate numbers.
- **Select Plates:** Open a dialog box for various ways to highlight plates.
- **Select Plates:** Open a dialog box for various ways to select plates.
- **Select Nodes:** Open a dialog box for various ways to highlight nodes.
- **Select Nodes:** Open a dialog box for various ways to select nodes.
- **Hide/Show Piles, Plates:** This option is useful when a user want to review a specific section of the pile group. Here the user can:
  - **Hide Selected Piles, Plates:** Hide the selected piles and or plates. (Use **Edit→ Select Mode)**
  - **Hide Except Selected Piles, Plates:** Hide the piles and or plates that are not currently selected.
  - **Reverse Hiding:** Hide piles and or plates that are shown and show piles that are hidden.
  - **Show All Piles, Plates:** Show all the piles and or plates.

2.4.11 3D View Results

The **3D View Results** menu is available when the **3D View** window is displayed and after running analysis under the **Computation** menu.

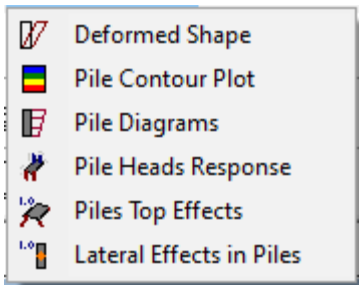


Figure 2.22 Options contained in the 3D View Results Menu

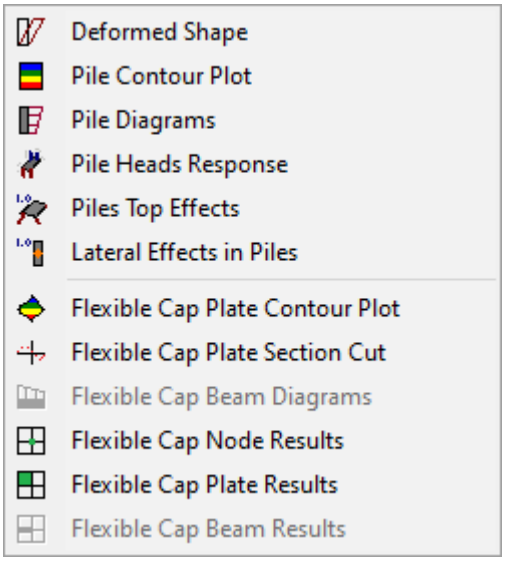


Figure 2.23 Options contained in the 3D View Results Menu – Flexible Cap Version

- **Deformed Shape:** Display the deflected shape for the current load case or combination.
- **Pile Contour Plot:** Display a contour plot on piles for the current load case or combination.
- **Pile Diagrams:** Display the diagrams along the piles for the current load case or combination.
- **Pile Heads Response:** Display the results of the pile heads for the current load case or combination.
- **Pile Top Effects:** Display the results of the pile heads for the current load case or combination.
- **Lateral Effects in Piles:** Display the lateral results along the piles for the current load case or combination.

For the Flexible Cap Version the following options are included:

- **Flexible Cap Contour Plot:** Display a contour plot on the flexible pile-cap for the current load case or combination.
- **Flexible Cap Section Cut:** Open a window to display a distribution of the results available at the flexible pile-cap at a section cut.
- **Flexible Cap Beam Diagrams:** Open a window to display the diagrams along the beams for the current load case or combination.
- **Flexible Cap Node Results:** Open a window to display the values of the results available calculated or interpolated at the nodes.
- **Flexible Cap Plate Results:** Open a window to display the values of the results available calculated at the beams.
- **Flexible Cap Node Results:** Open a window to display the values of the results available calculated the beams.

2.4.12 Window Menu

The **Window** menu provides standard functions for organizing open-screen windows.

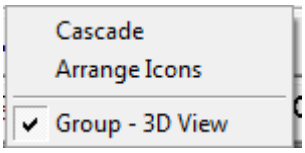


Figure 2.24 Options contained in the Windows Menu

- **Cascade:** This option organizes all open windows so that they are all visible. See above for an example.
- **Arrange Icons**: Any minimized windows will be aligned at the bottom left of the screen.
- After the options **Cascade** and **Arrange Icons**, the windows opened are listed in order of depth. Clicking on a window name through the menu brings it to the top
- **Group – 3D View**: This option switches the view to the 3D View window.

2.4.13 Help Menu

The **Help** menu provides an online help reference.

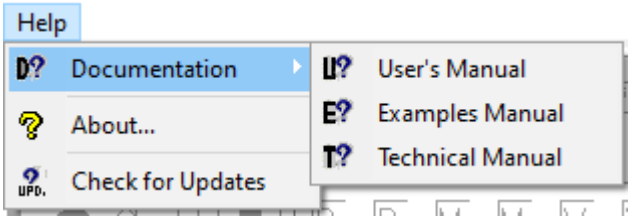


Figure 2.25 Options contained in the Help Menu

- **Documentation**: Access to the digital versions of the Manuals for the Program.

  - **User's Manual**: Open the User's Manual of the Program (pdf version).
  - **Examples Manual**: Open the Examples Manual of the Program (pdf version).
  - **Technical Manual**: Open the Technical Version of the Program (pdf version).
- **About**: This option provides a screen describing the program version, date, license user and methods for accessing technical support.
- **Check for Updates**: This opens the user's default internet browser to an Ensoft webpage with specific update information for GROUP 2022.

CHAPTER 3. References for Data Input

3.1 Numeric Data Entries

Cells that require numeric data may accept mathematical expressions as entries. Entering a mathematical expression works similarly to entering normal numeric data—the user simply types the expression then presses the "Enter" key. The following table shows a list of supported operations and constants, with the order of operations following the order listed. Implicit multiplications using constants is not supported (use 2*e instead of 2e). Negation of the constants PI or e is not allowed. For instance, instead of entering -PI the user must enter -(PI). Scientific notation (i.e. 1.65e8 or 1.65e-8) may be used to input very large or very small numbers. After an expression is calculated, very large or very small numbers will be displayed using scientific notation.

Symbol	Description
()	Parenthesis (may be nested)
^	Exponentiation
*****	Multiplication
/	Division
+	Addition
-	Subtraction/Negation

Table 3.1 Supported mathematical operations

Symbol	Value
PI (or pi)	3.1415927
e (or E)	2.7182818

Table 3.2 Supported mathematical constants

3.2 File Menu

This menu contains options related to the management of input-data files and also for exiting the program. Input-data files created for GROUP 2022 are provided with a standard file-name extension in the form of **.gp12d* (where * represents any allowable file name). Input files created from previous versions have an extension **.gp11d* ,**.gp10d, *.gp9d*, **.gp8d* or **.gpd* . All input data files are standard text files and may be edited with any text editor or word-processing program.

If the **3D View** window is active 3 additional options appear.

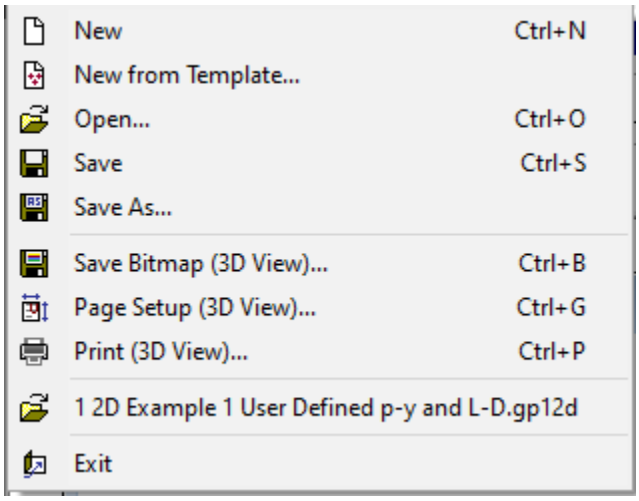


Figure 3.1 Options contained in the File Menu

3.2.1 File - New

Once the program is started, default values are used for certain operating parameters, and a blank input-data file is created. Selecting this menu item resets all GROUP 2022 variables to either default or blank values, as appropriate. This option should be selected when a new data file is desired to be created. This menu option may also be accessed using the [Ctrl]+[N] keyboard combination.

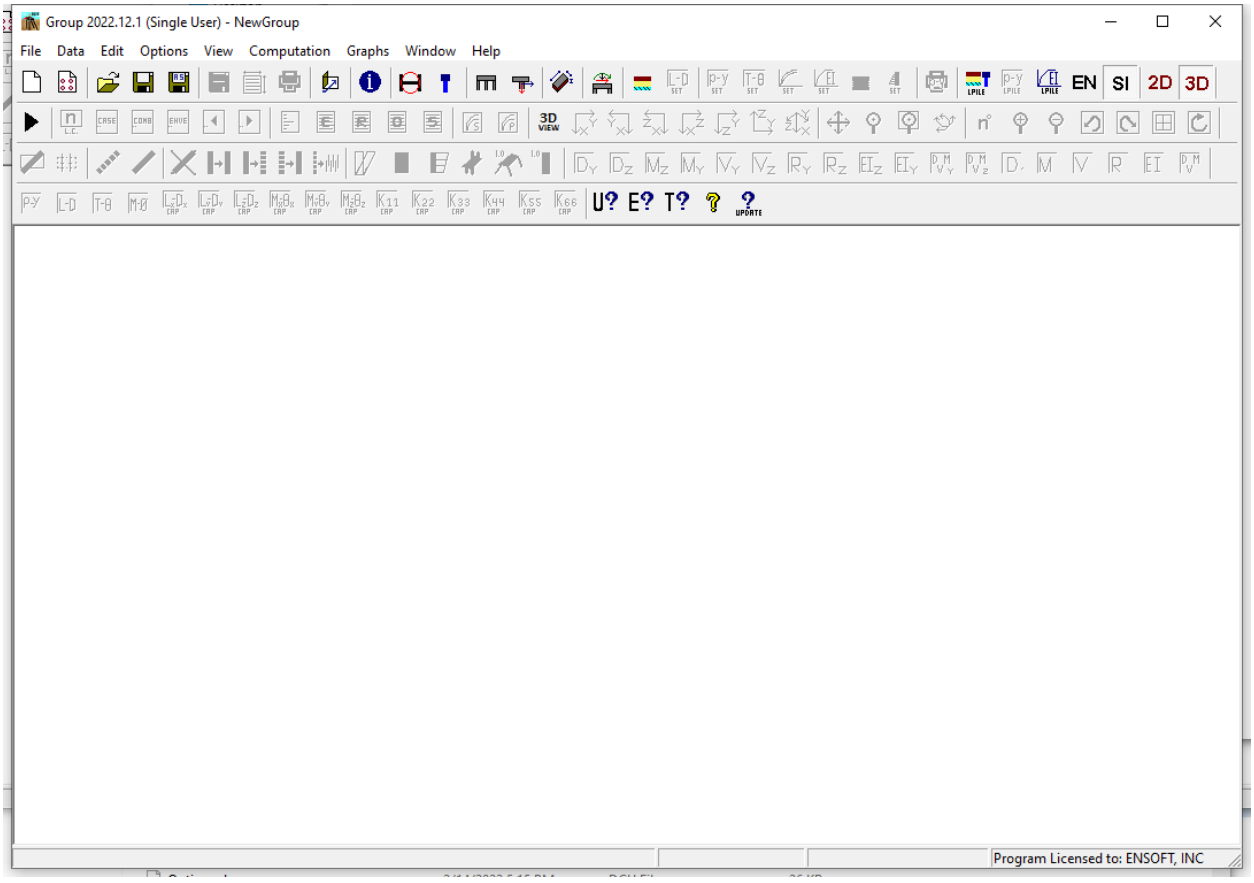


Figure 3.2 Window shown after File - New is selected

3.2.2 File - New from Template

Create a new data file for a regular rectangular or circular pile group configuration. After selecting this option, the New from Template Window will appear.

Three pile group templates are included:

- 2D Rectangular: Generates a rectangular 2D pile group layout (Figure 3.3)
- 3D Rectangular: Generates a rectangular 3D pile group layout. (Figure 3.4)
- 3D Circular: Generates a circular 3D pile group layout (Figure 3.5)

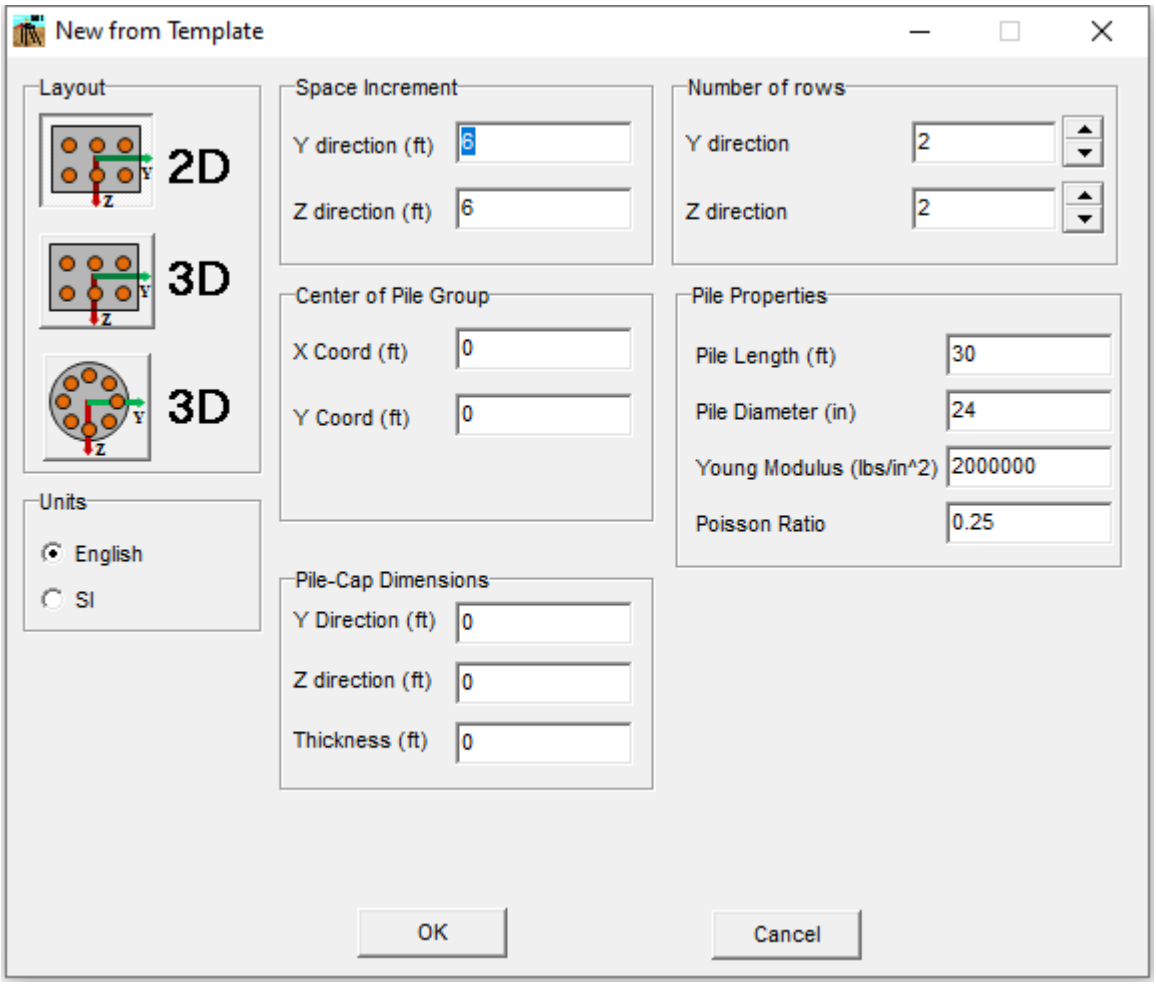


Figure 3.3 New from Template Window, 2D rectangular option

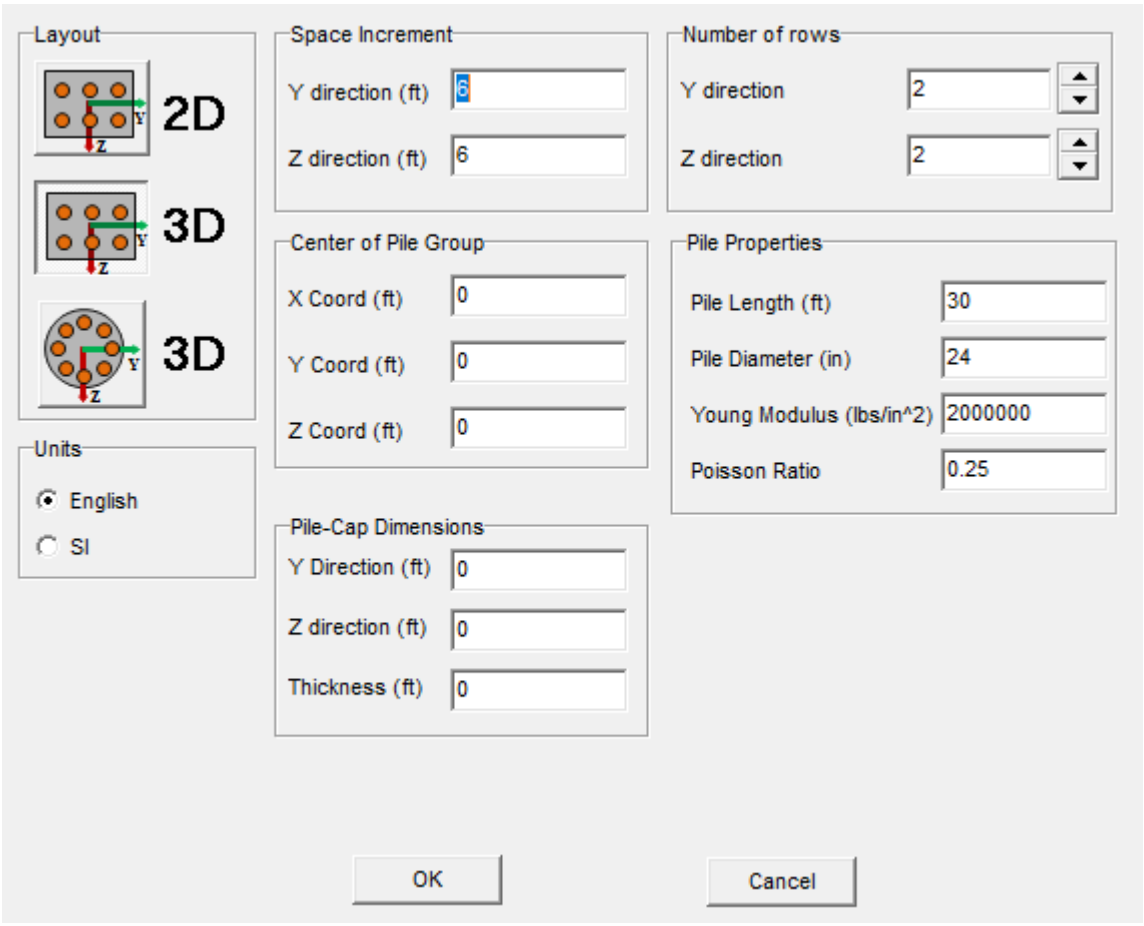


Figure 3.4 New from Template Window, 3D rectangular option

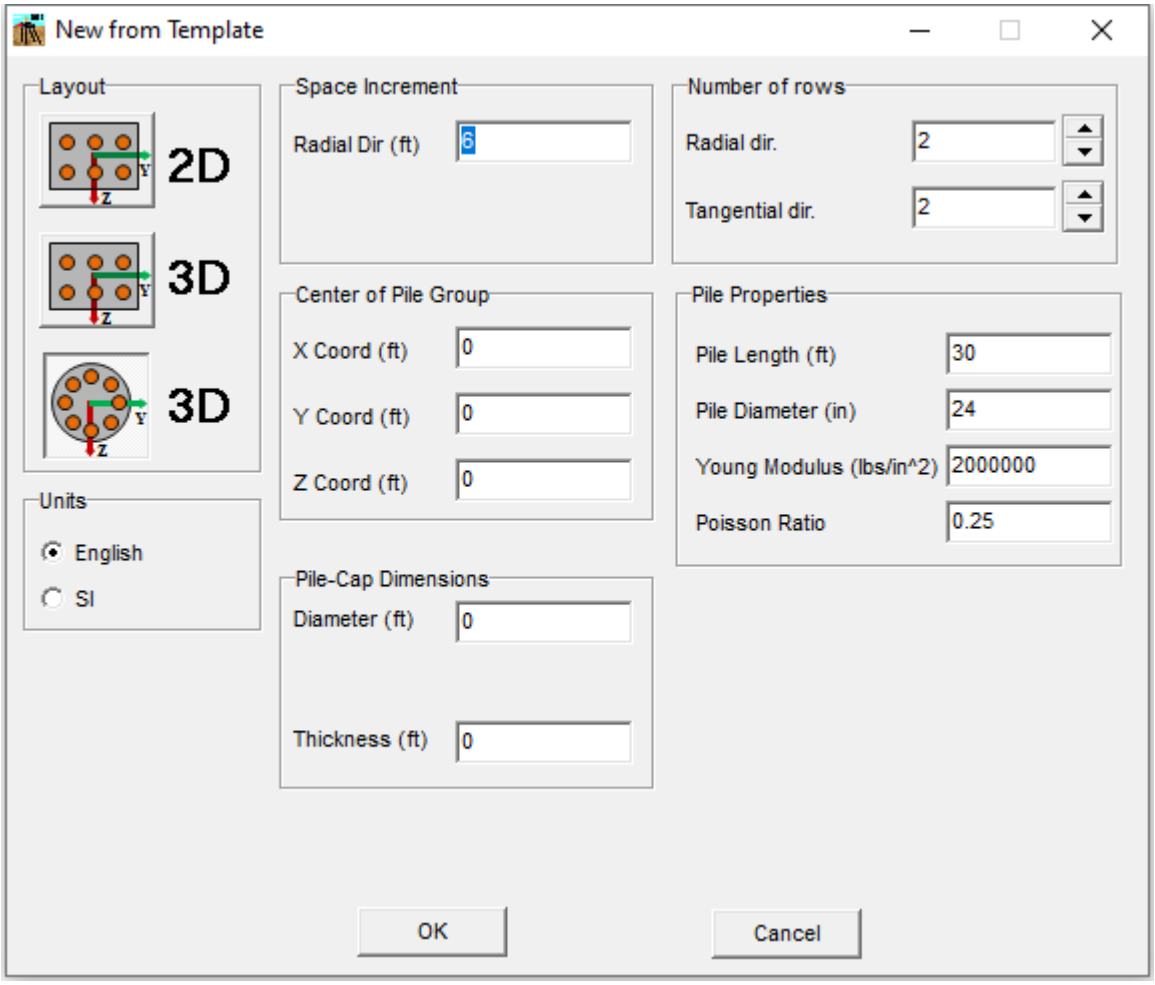


Figure 3.5 New from Template Window, 3D circular option

The following parameters can be defined in the New from Template window:

- **Layout**: Select the pile group configuration to generate. Various entries may show up or hide based on the layout selected.
- **Units**: Select either English or SI units.
- **Space increment**: Enter the space increment for model generation. For rectangular layouts, enter Space Increment for Y and Z directions. For Circular layouts enter Space Increment in the radial direction. Refer to Figure 3.6 and Figure 3.7.
- **Number of Rows**: Enter the Number of Rows for model generation. For rectangular layouts, enter number of rows for Y and Z directions. For Circular layouts enter Number of Rows in the radial direction. Refer to Figure 3.6 and Figure 3.7.
- **Center of Pile Group**: Enter the coordinates for the center of the pile group (Y and Z coordinates) and the coordinate of the pile heads (X coordinate). Refer to Figure 3.6 and Figure 3.7.

- **Pile Cap Dimensions**: Enter the dimensions of the pile cap for drawing purposes in the 3D window. If these values ar left as zero, the dimensions for 3D drawing will be estimated. For rectangular layouts, enter dimensions in Y and Z directions. For Circular layouts enter the diameter. Refer to Figure 3.6 and Figure 3.7.
- **Pile Properties**: Enter the Pile Length, Pile Diameter, Modulus of Elasticity and Poisson Ratio for all the piles to be generated. Refer to Figure 3.6 and Figure 3.7.

An example of a rectangular layout generated is displayed in Figure 3.8, while a circular layout is shown in Figure 3.9.

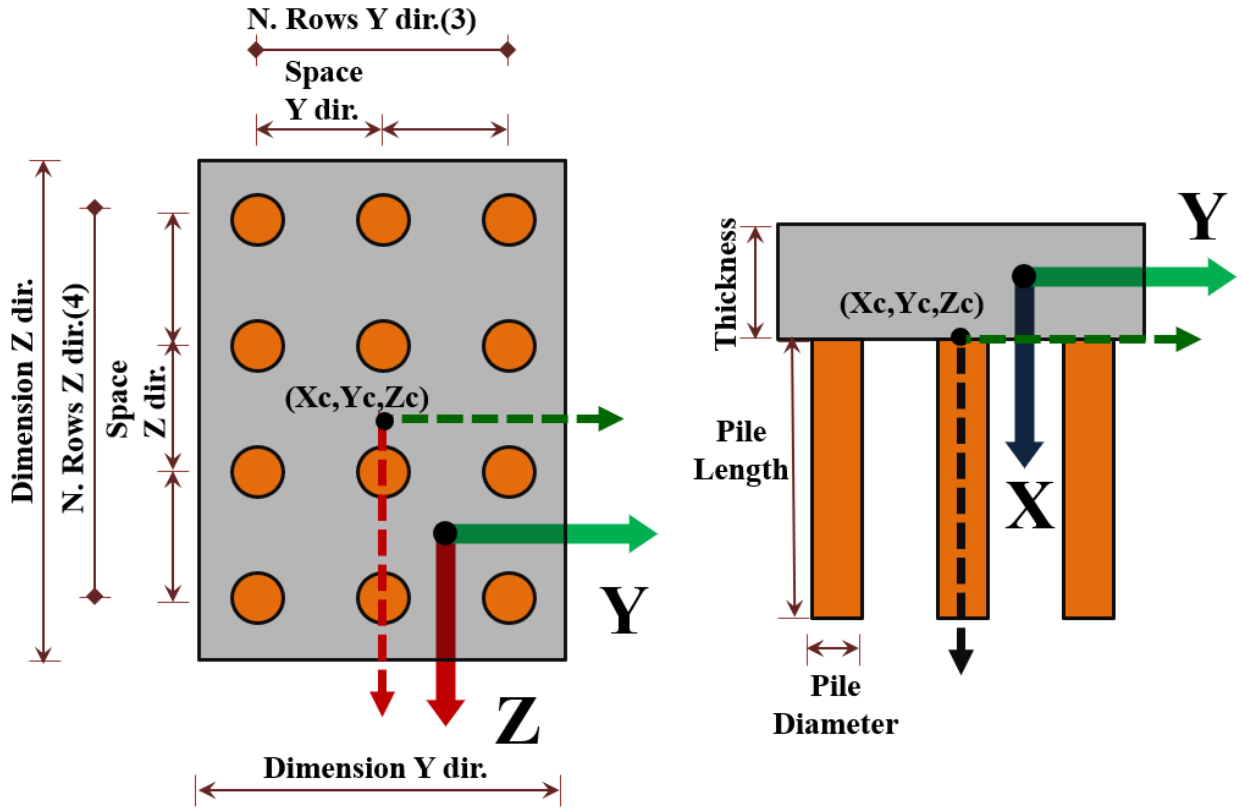


Figure 3.6 Input parameters for pile group generation, rectangular option

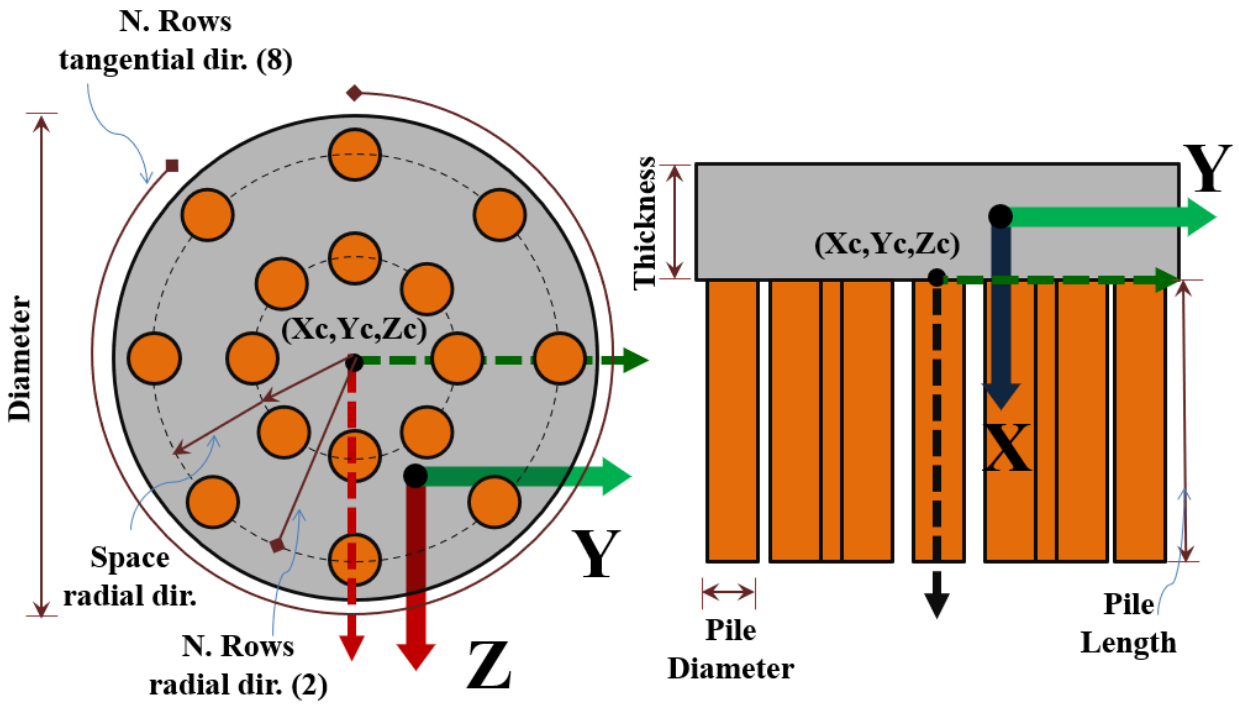


Figure 3.7 Input parameters for pile group generation, circular option

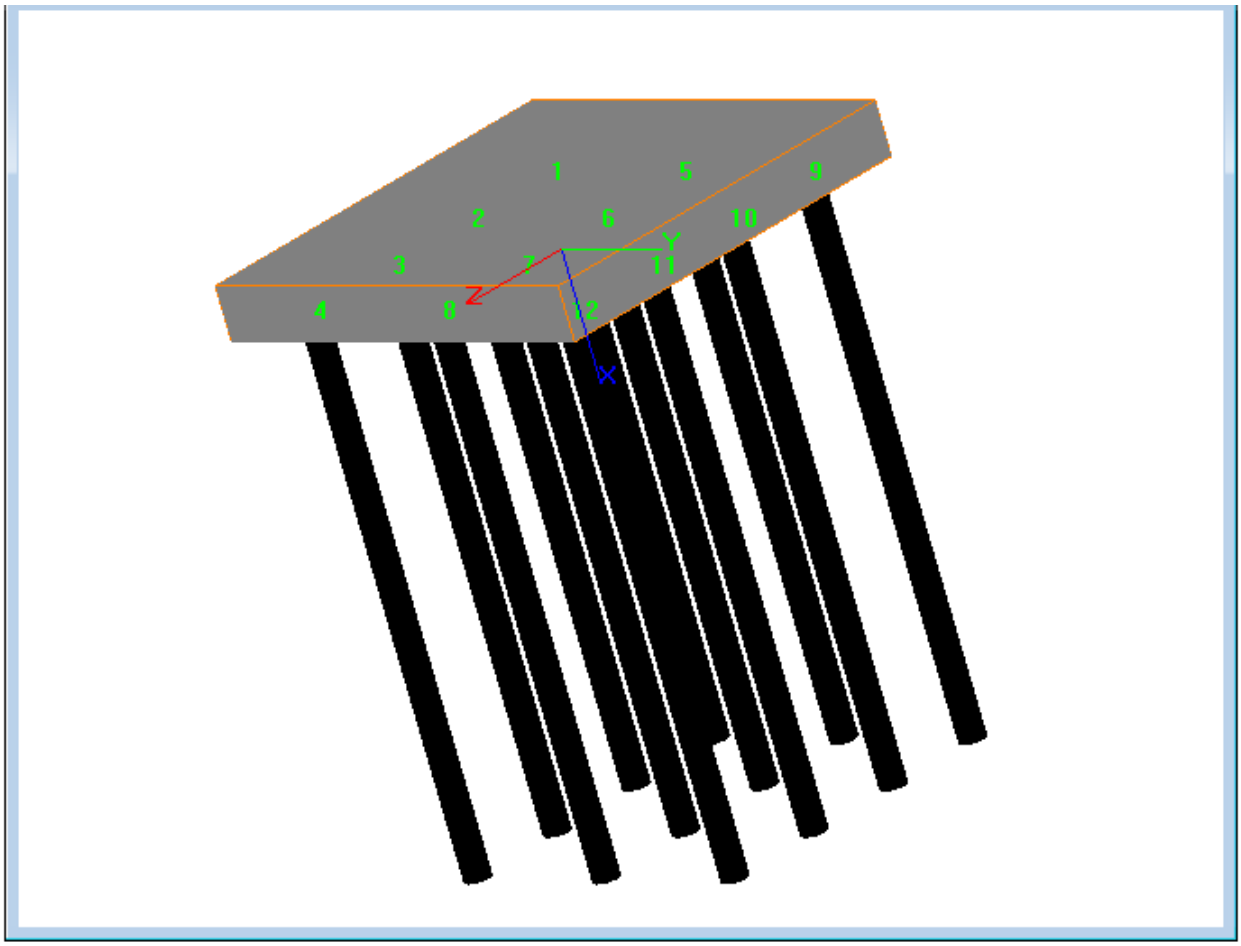


Figure 3.8 Pile Group Layout using New from Template, 3D rectangular option

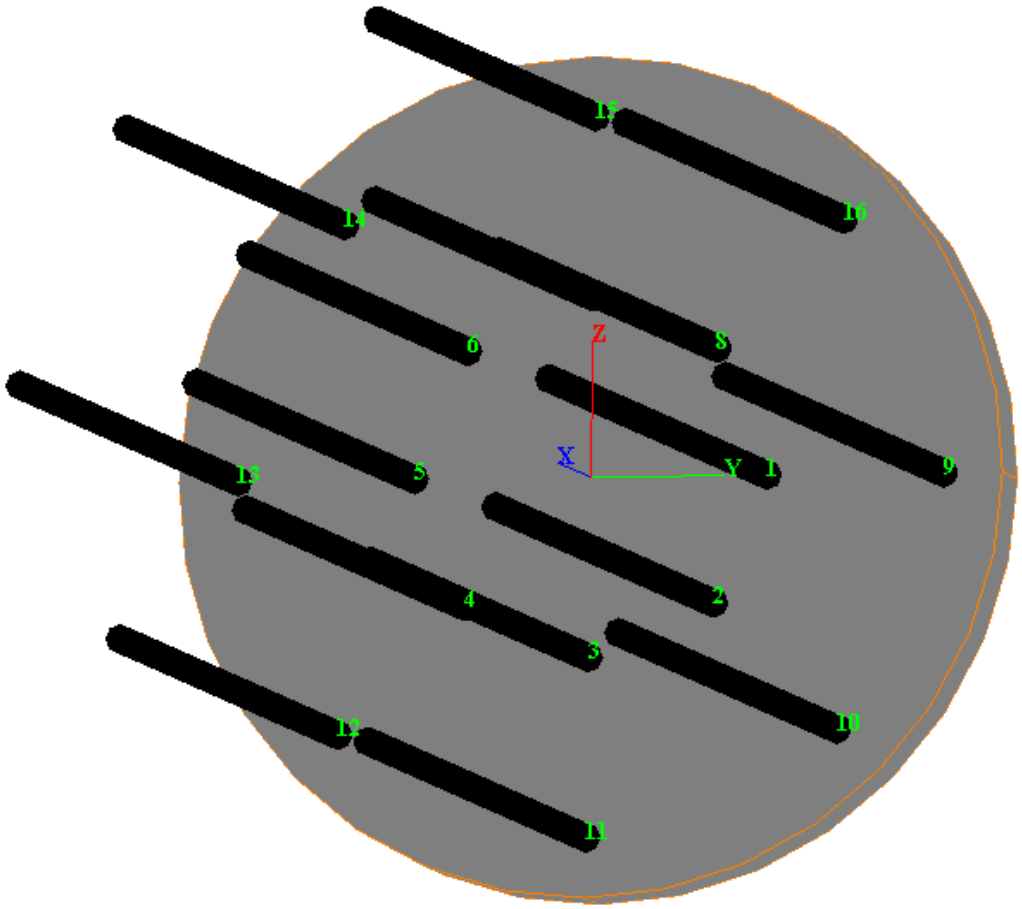


Figure 3.9 Pile Group Layout using New from Template, 3D circular option

3.2.3 File - New from Template – Flexible Cap Version

The New from Template Window for the Flexible Cap Version has two options for pile cap type: Rigid and Flexible. The rigid option is similar to the Standard Version, and it is described in detail in the previous item (3.2.2). The options related to the flexible cap type will be described below.

Two pile group templates that also included flexible cap are included:

- 3D Rectangular: Generates a rectangular 3D pile group layout and flexible cap. (Figure 3.10)
- 3D Circular: Generates a circular 3D pile group layout and flexible cap (Figure 3.11)

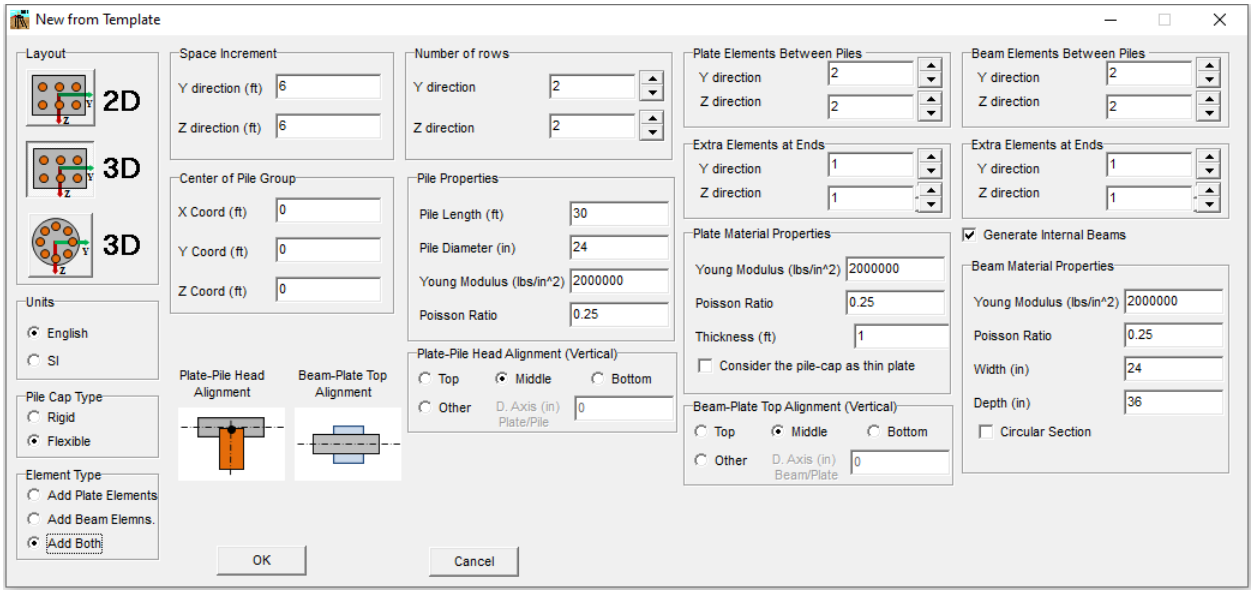


Figure 3.10 New from Template Window, 3D rectangular option – Flexible Cap Version

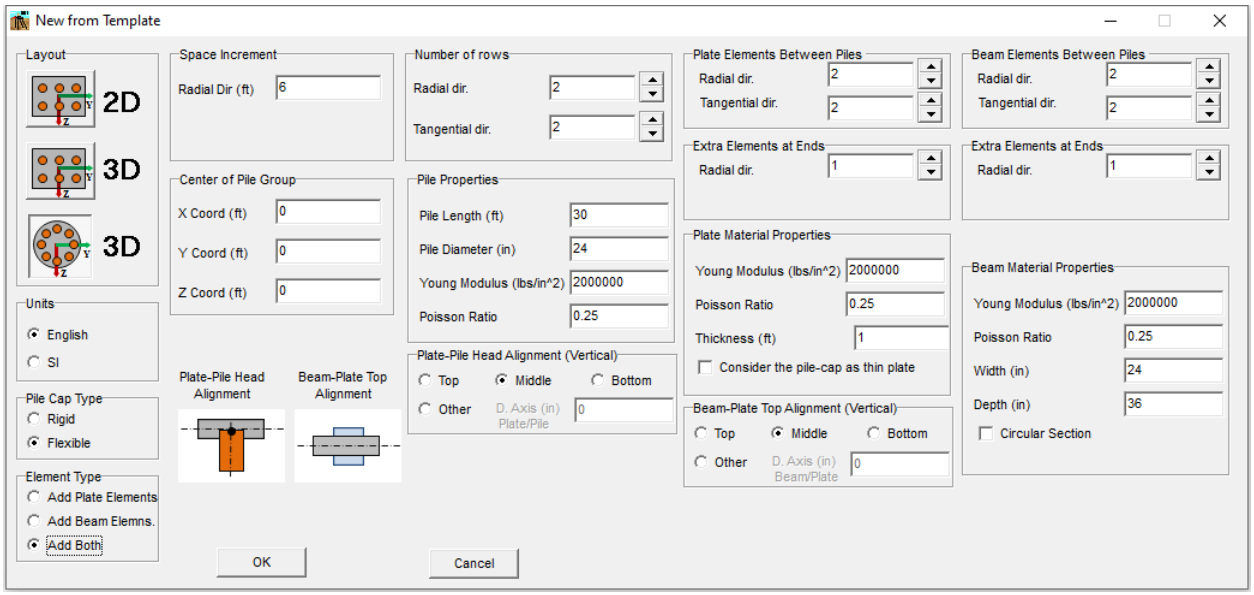


Figure 3.11 New from Template Window, 3D circular option – Flexible Cap Version

The flexible cap can be defined as a combination of plate and beam elements (Figure 3.10) , or only plate elements (Figure 3.12), or only beam elements (Figure 3.13).

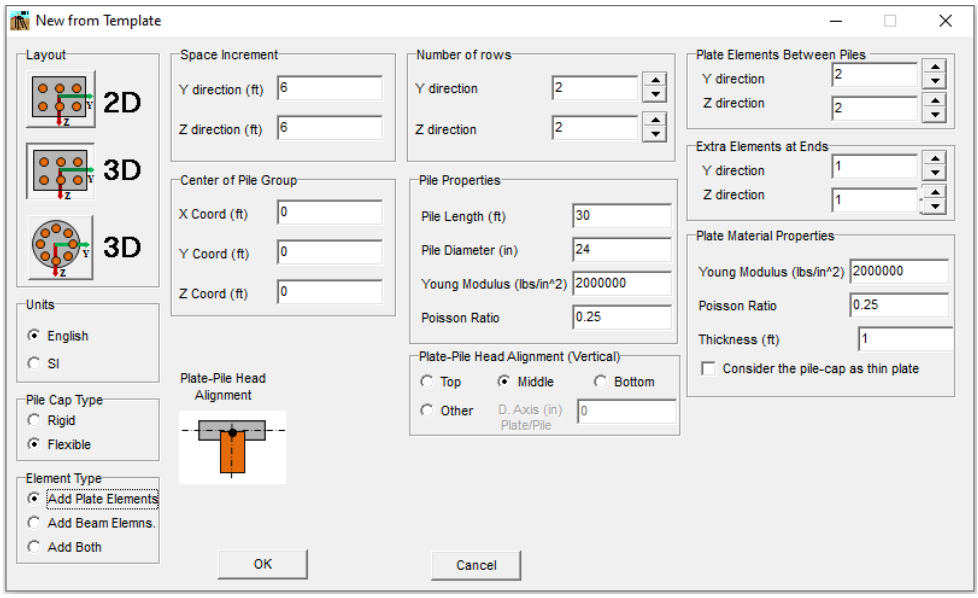


Figure 3.12 New from Template Window, 3D rectangular option, Add Plate Elements – Flexible Cap Version

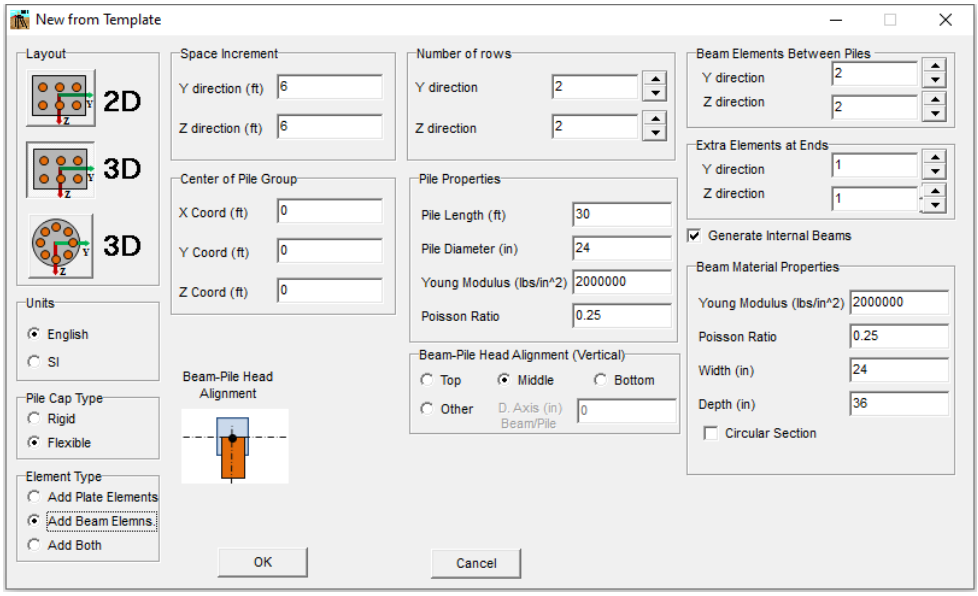


Figure 3.13 New from Template Window, 3D rectangular option, Add Beam elements – Flexible Cap Version

In addition to the parameters in the previous item (3.2.2), the following parameters can be defined in the New from Template window for the Flexible Cap Version:

- **Plate Elements Between Piles** : Enter the number of plates to be defined between piles. For rectangular layouts, enter number of plates for Y and Z directions. For Circular layouts enter number of plates in the radial and tangential direction. Refer to Figure 3.14 and Figure 3.15.
- **Beam Elements Between Piles** : Enter the number of beams to be defined between piles. For rectangular layouts, enter number of plates for Y and Z directions. For Circular layouts enter number of plates in the radial and tangential direction. Refer to Figure 3.18 and Figure 3.19.
- **Extra Elements at Ends** : Enter the number of plates or beams to be defined from the outer pile to the boundary of plate pile cap. For rectangular layouts, enter number of plates or beams for Y and Z directions. For Circular layouts enter number of plates or beams in the radial direction. Refer to Figure 3.14 and Figure 3.15 for plate elements and Figure 3.18 and Figure 3.19 for beam elements.
- **Generate Internal Beams** : This option is available while defining only beams for a rectangular sahape pile cap. By clicking this option the program will add the internal beams, otherwise the program will include only the principal beams. Refer to Figure 3.18.
- **Plate Material Properties**: Enter the Modulus of Elasticity, Poisson Ration and Thickness of the flexible pile cap. Check the option is the pile cap is to be considered as thin.
- **Beam Material Properties**: Enter the properties to define the cross section and material properties of the beam. Enter Modulus of Elasticity, Poisson Ratio and Width and Depth or Diameter for circular sections..
- **Plate-Pile Head Alignment (Vertical)**: Select the alignment between the pile head and the plates. If selected other than middle or the D.Axis value is different from zero, a rigid element will be used to connect each pile head with a vertical projected node that belongs to the finite element mesh defining the flexible pile-cap (vertical offset).
- **Beam-Pile Head Alignment (Vertical)**: Select the alignment between the pile head and the beams. If selected other than middle or the D.Axis value is different from zero, a rigid element will be used to connect each pile head with a vertical projected node that belongs to the finite element mesh defining the flexible pile-cap (vertical offset).
- **Plate-Beam Head Alignment (Vertical)**: Select the alignment between the beams and plates. If selected other than middle or the D.Axis value is different from zero, a rigid element will be used to connect each pile head with a vertical projected node that belongs to the finite element mesh defining the flexible pile-cap (vertical offset).

An example of a rectangular layout generated is displayed in Figure 3.16 and Figure 3.20, while a circular layout is shown in Figure 3.17 and Figure 3.21.

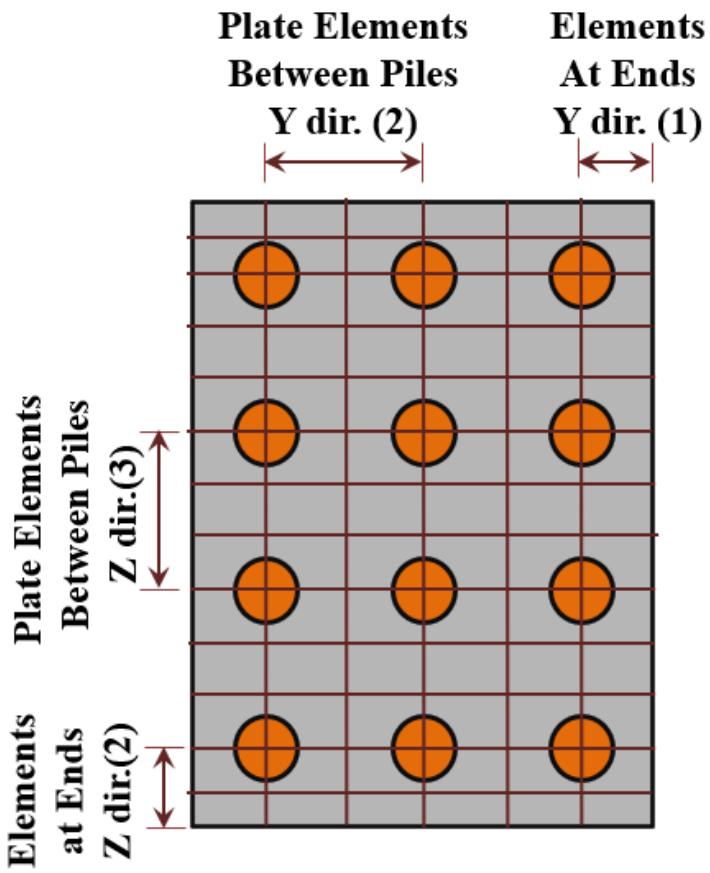


Figure 3.14 Input parameters for mesh generation, rectangular option, plate elements – Flexible Cap Version

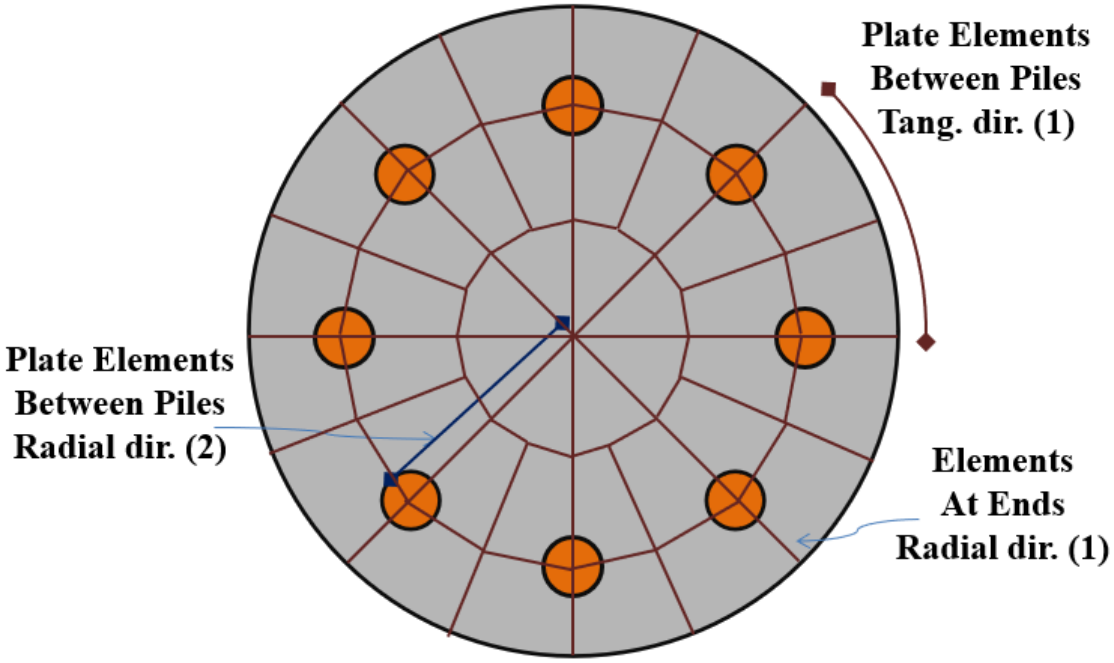


Figure 3.15 Input parameters for mesh generation, circular option, plate elements – Flexible Cap Version

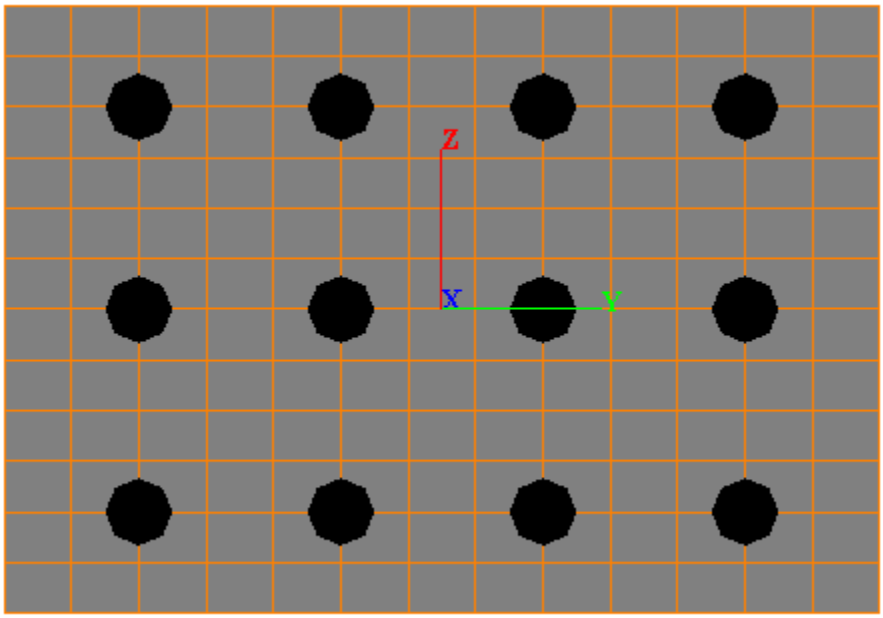


Figure 3.16 Pile Group Layout using New from Template, 3D rectangular option, plate elements – Flexible Cap Version

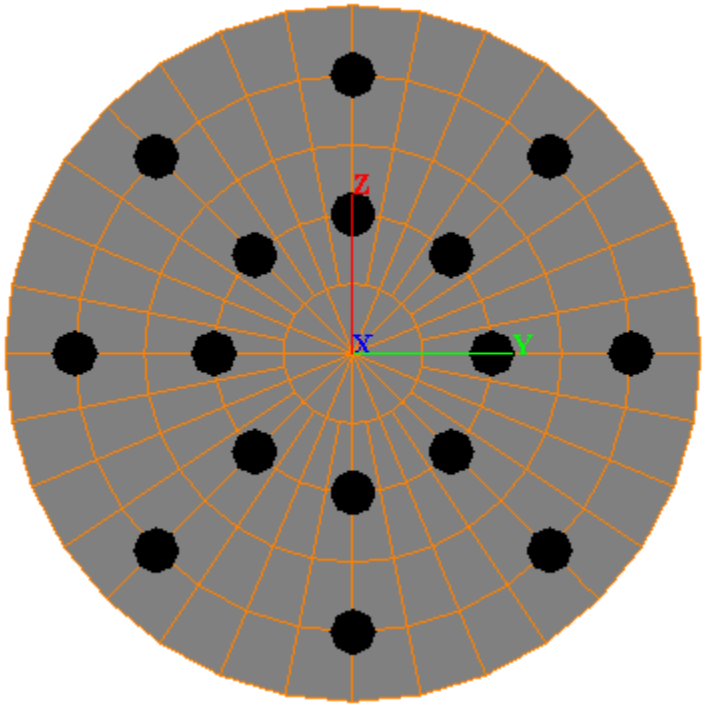


Figure 3.17 Pile Group Layout using New from Template, 3D circular option, plate elements – Flexible Cap Version

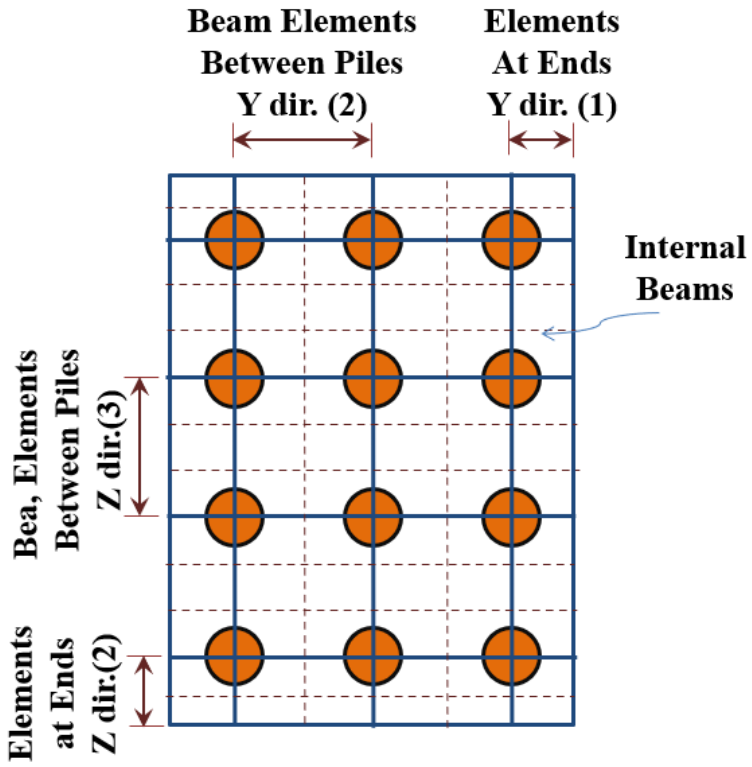


Figure 3.18 Input parameters for mesh generation, rectangular option, beam elements – Flexible Cap Version

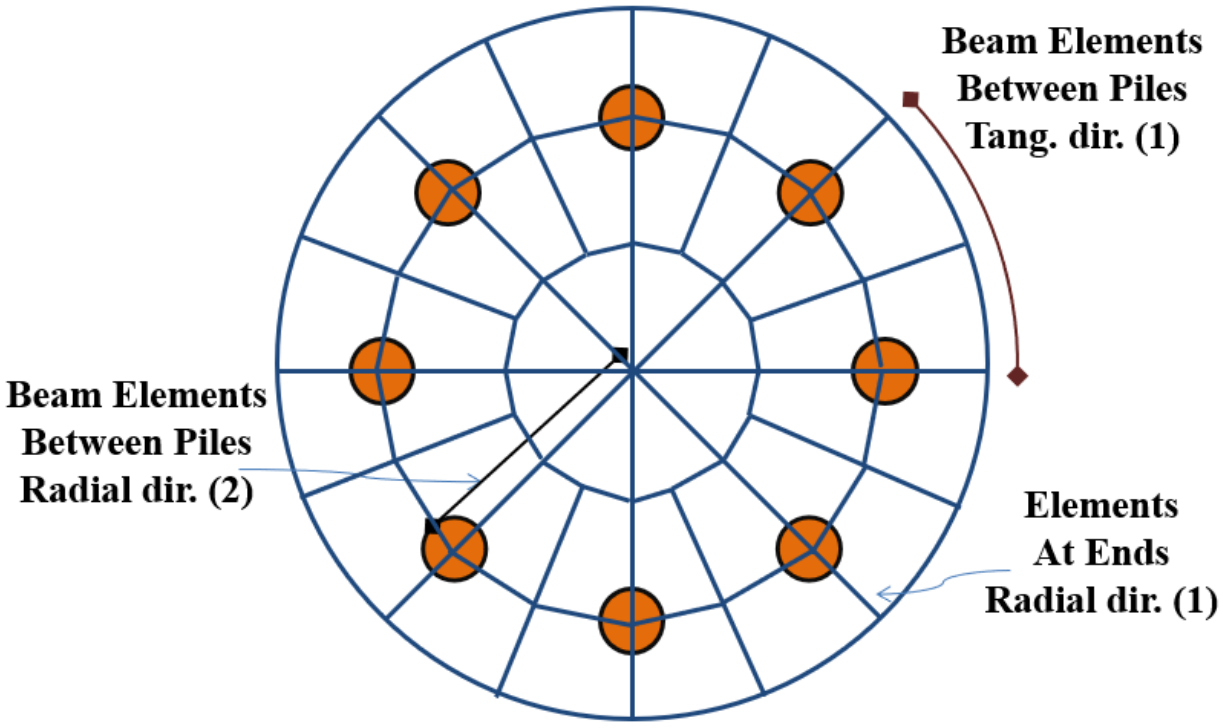


Figure 3.19 Input parameters for mesh generation, circular option, beam elements – Flexible Cap Version

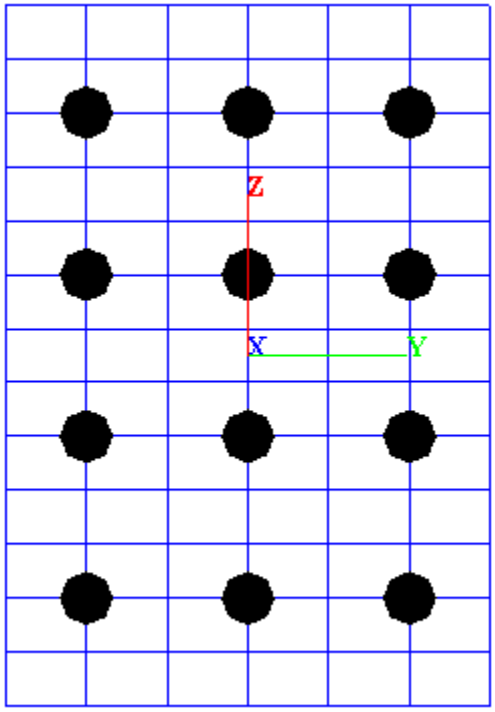


Figure 3.20 Pile Group Layout using New from Template, 3D rectangular option, beam elements – Flexible Cap Version

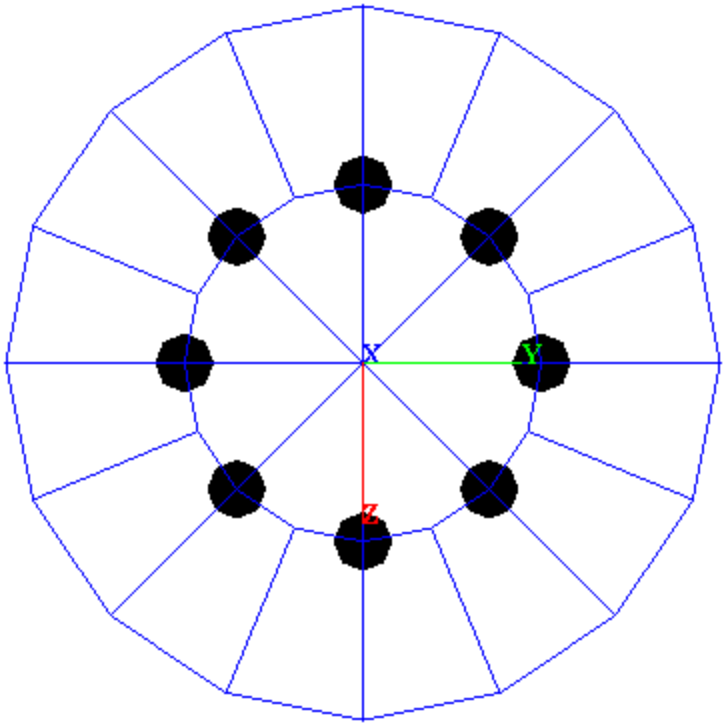


Figure 3.21 Pile Group Layout using New from Template, 3D circular option, beam elements – Flexible Cap Version

3.2.4 File - Open

This is used to open a file that has been previously prepared and saved to disk. The Open File window dialog, shown in Figure 3.22, is used to search for an existing input data file. The default directory is the GROUP 2022 install directory. Standard Windows navigation procedures may be used to locate the name and directory of the desired project file. This menu option may also be accessed with the [Ctrl]+[O] keyboard combination.

Every analytical run of GROUP 2022 produces several additional files (listed on Table 4.1 of this manual). The name of the input data file indicates the names of all related files (output, graphics and processor-text files) produced by a successful program run. All the additional files will be created in the same directory as the input file.

Opening some partially-completed GROUP 2022 input files or invalid data files may produce an information window reporting that an "invalid or incomplete" file is being opened. The user should click the OK button and all partial-input data that was previously prepared should become available.

The program allows users to read input data files created from previous versions of GROUP by selecting the drop-down arrow under the Files of Type option at the bottom of the Open window screen. The program will automatically convert the open GROUP file to GROUP 12.0 Data when the user saves the opened file.

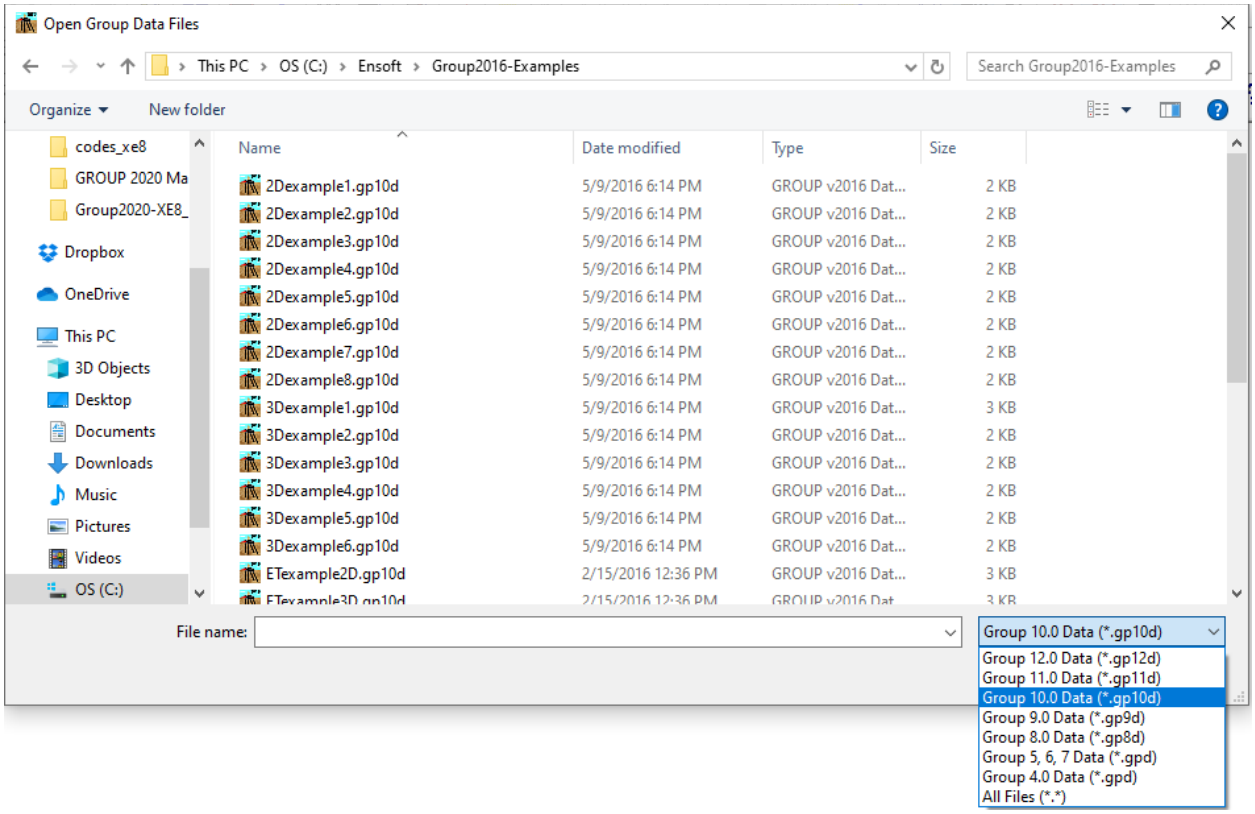


Figure 3.22 Open File Dialog

Please note that the Flexible Cap Version will be able open input files generated by the Standard Version. The Standard version will be able to open files generated by the Flexible Cap Version; however, if the model in the Flexible Cap Version was defined using a flexible pile cap, the program will display a warning message indicating that all the flexible cap definitions will be lost if the file is saved (Figure 3.23).

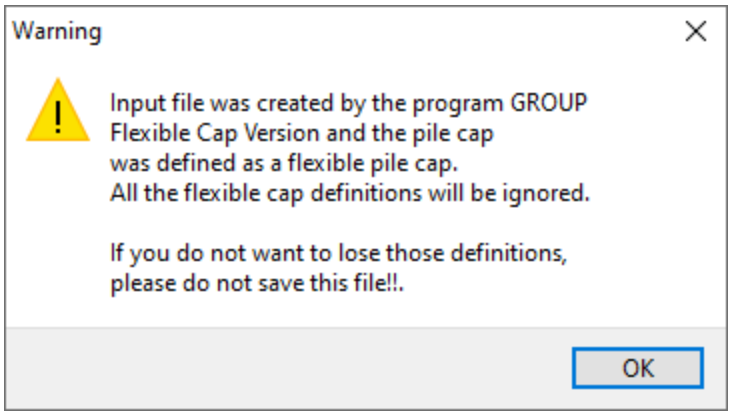


Figure 3.23 Warning message for possible data loss.

3.2.5 File - Save

This option is used to save input data under the current file name. Any input data that was previously saved with the same file name is replaced by the current one. Input data files should be saved every time before proceeding with analytical runs. This menu option is accessed using the [Ctrl]+[S] keyboard combination.

3.2.6 File - Save As

This option allows the user to save any opened or new input data file under a different file name or different directory. The Save As window dialog, shown in Figure 3.24, is used to navigate between folders and specify a file name. If a file with the same name already exists, a confirmation message indicating so will appear. By clicking OK in the confirmation window, the input data file under the existing file name will be replaced with the current one.

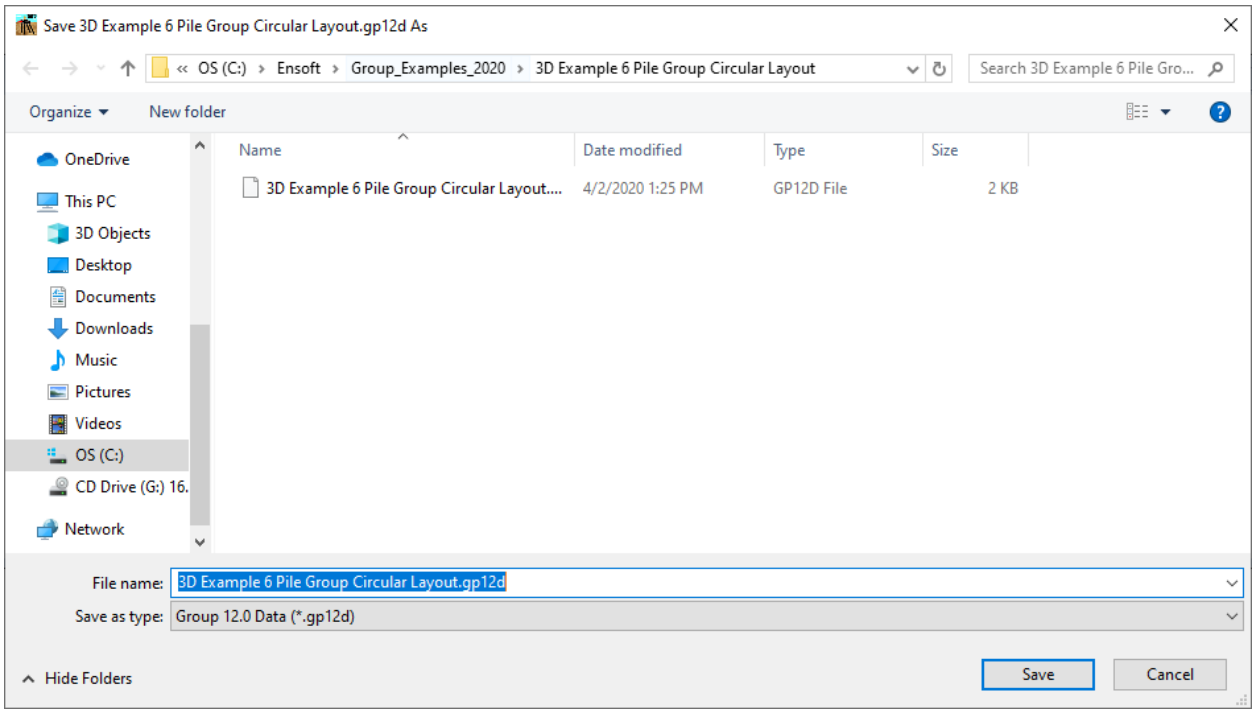


Figure 3.24 Save As Dialog

3.2.7 File - Exit

This is selected to exit GROUP 2022. Any input data file that was modified but not yet saved to disk will produce a confirmation window before exiting the program.

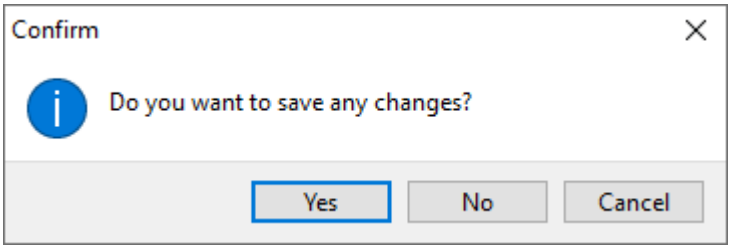


Figure 3.25 Message window advising that changes were not saved to disk

3.2.8 File - Recent Files

A list of recently opened files is displayed in between **Save As…** and **Exit**. By clicking on the menu item, the program will open the file.

3.2.9 File - Save Bitmap (3D View)

This option is available when the **3D View** window is active. This option allows the user to save the model displayed in the **3D View** window as bitmap picture file. The Save As window dialog, shown in Figure 3.26, is used to navigate between folders and provide a name for saving the file. If a file with the same name already exists, a confirmation message indicating so will appear. By clicking OK in the confirmation window, the graphics file under the existing file name will be replaced with the current model view.

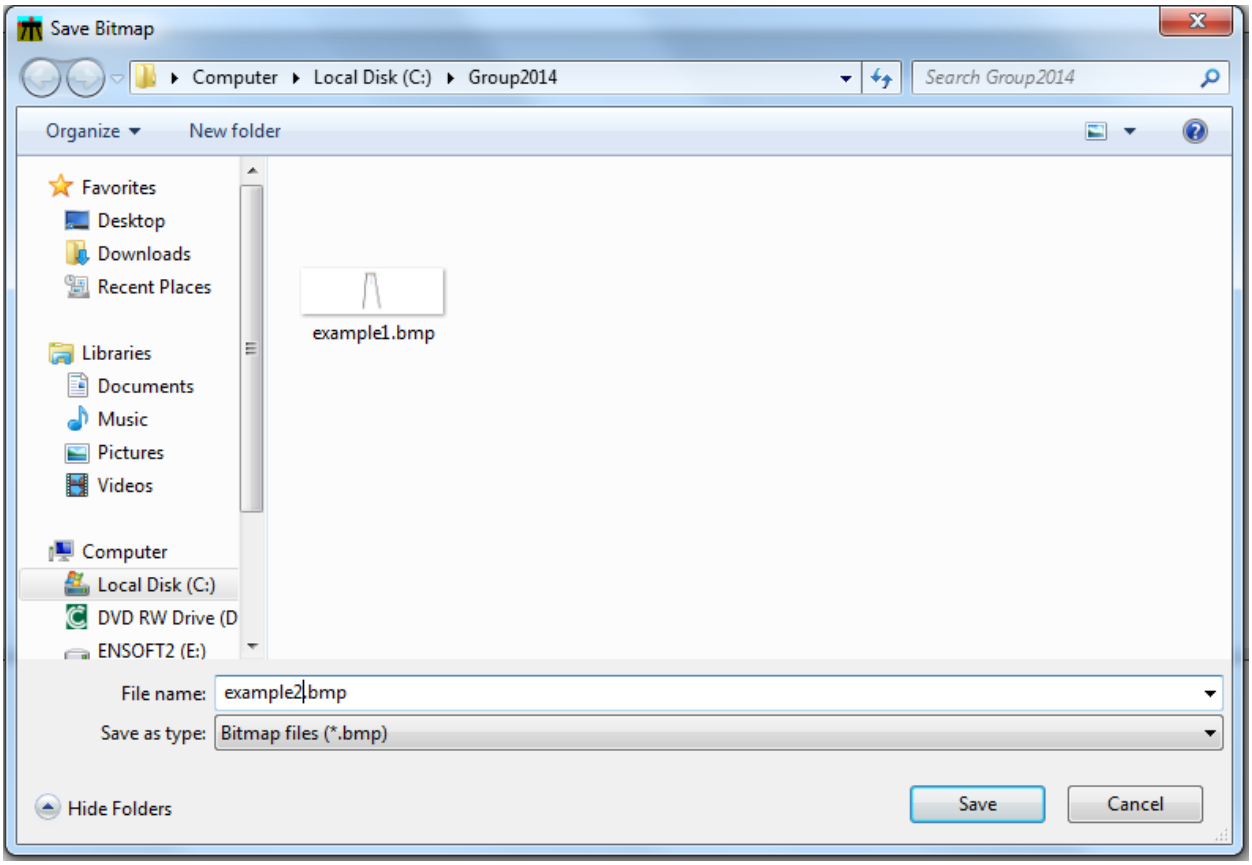


Figure 3.26 Save Bitmap Dialog

3.2.10 File - Page Setup (3D View)

This option is available when the **3D View** window is active. This option allows the user to set the margins for printing. The Page Setup window dialog, shown in Figure 3.27, is used to define the margins for printing.

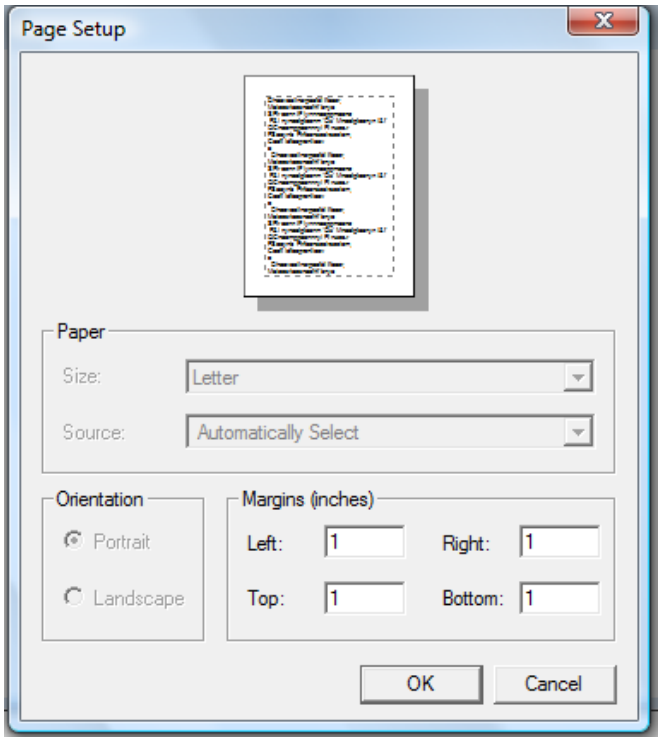


Figure 3.27 Page Setup Dialog

3.2.11 File - Print (3D View)

This option is available when the **3D View** window is active. This option allows the user to print the model displayed in the **3D View** window. The Print window dialog, shown in Figure 3.28, is used to select the printer and other parameters.

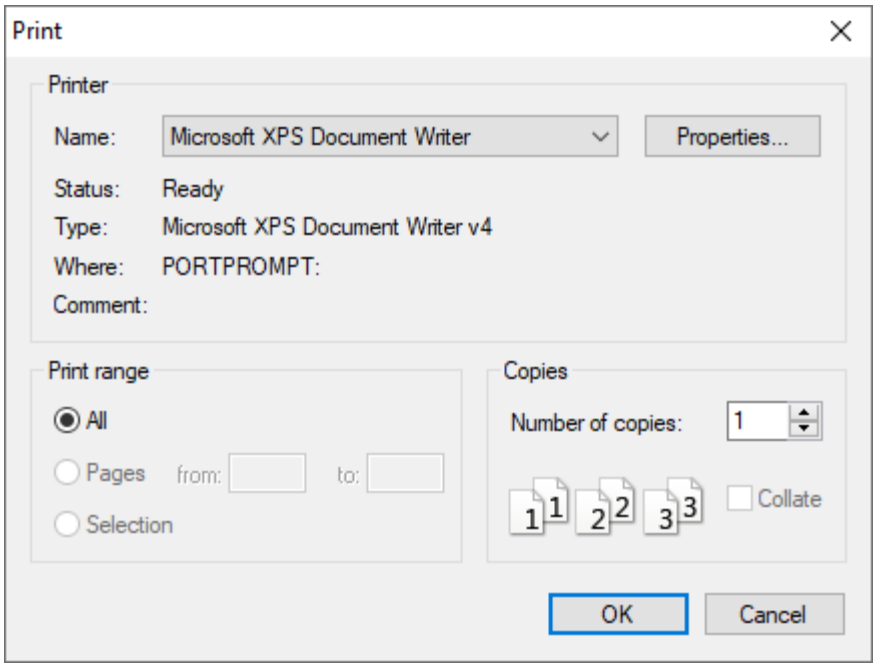


Figure 3.28 Print Dialog

3.3 Data Menu

The input of specific parameters for an application is controlled by options in the **Data** menu. It is recommended that the user choose each submenu and enter parameters in a consecutive manner, starting with the top option. Some input windows may optionally be left open on the screen. The selection of other menu options will then produce new windows on top of those that were left open. Many of the submenus will have **Add Row, Insert Row**, and **Delete Row** buttons. **Add Row** always adds a new row to the end of all existing rows, **Insert Row** inserts a new row above the currently selected row, and **Delete Row** deletes the row where the mouse cursor is currently at.

Selecting or clicking any of the submenu choices contained in the Data menu produces various types of windows. As a reminder of standard commands of Microsoft Windows®, open windows may be closed by all or some of the following methods:

- Clicking the "OK" button (if available).
- Clicking the X box on the upper-right corner of the window.
- Double-clicking the GROUP 2022 icon on the upper left corner of the window.
- Using the [Ctrl]+[F4] keyboard combination.
- Clicking once the GROUP 2022 icon on the upper left corner of the window and then choosing Close.

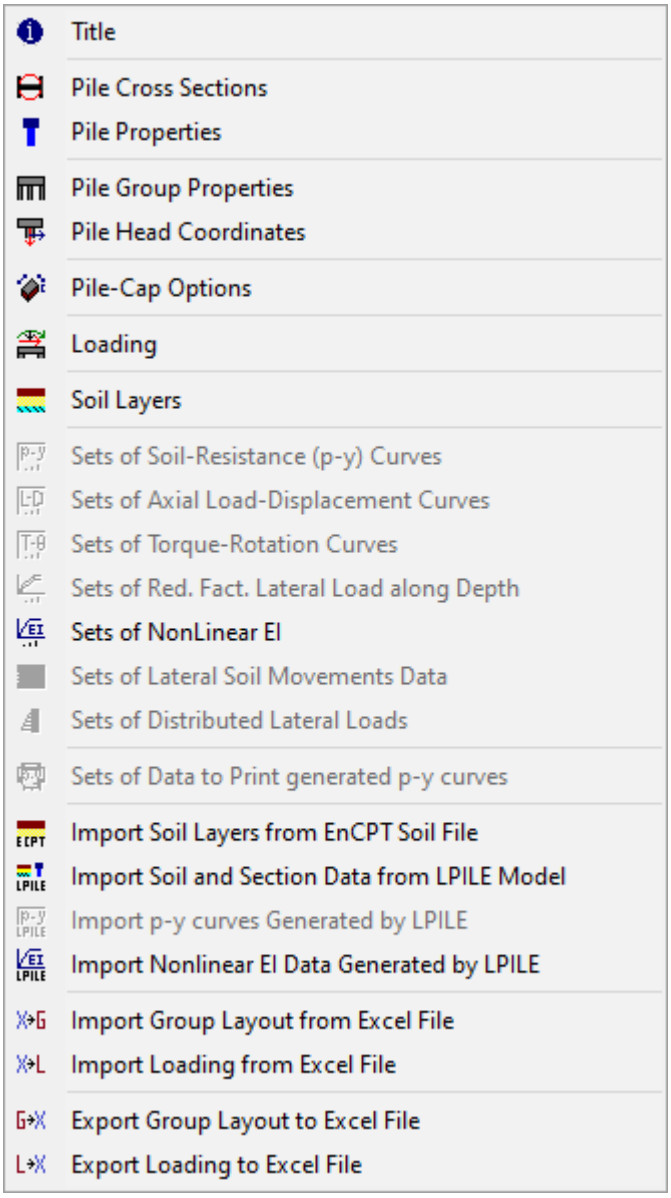


Figure 3.29 Options contained in the Data Menu

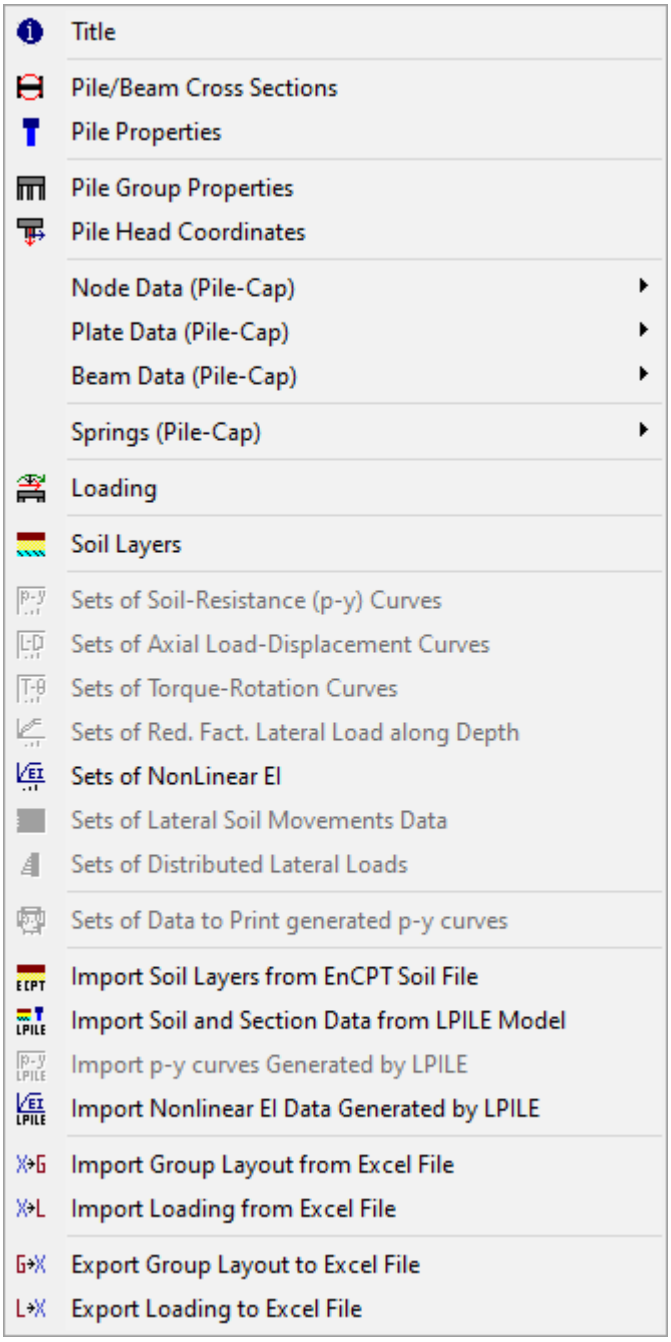


Figure 3.30 Options contained in the Data Menu – Flexible Cap Version

3.3.1 Data - Title

This option activates the **Project Title** window, shown in Figure 3.31, where the user can enter a line of text containing a general description of the application problem. Any combination of characters may be entered in the text box in order to describe a particular application. The user input will be restrained automatically once the maximum length of text is reached.

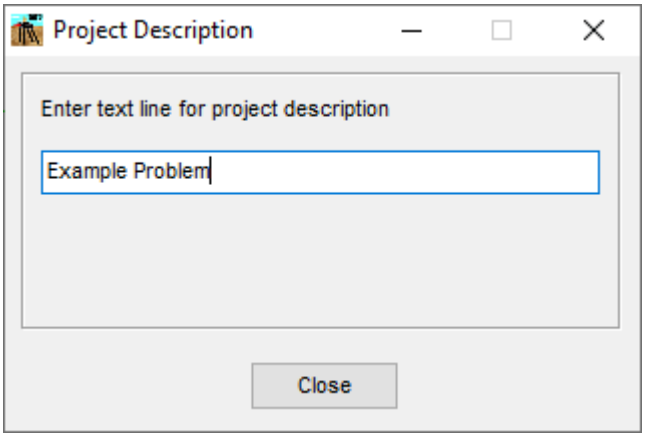


Figure 3.31 Project Title Window

3.3.2 Data - Pile Cross Sections

This submenu option is needed to define the cross sections on the piles and to enter the cross-section dimensions.

By selecting this option, the **Cross Sections Definition** window will appear

3.3.2.1 Cross Sections Definition Window

In this window the cross sections definition is carried out. A sample of this window is shown in Figure 3.32

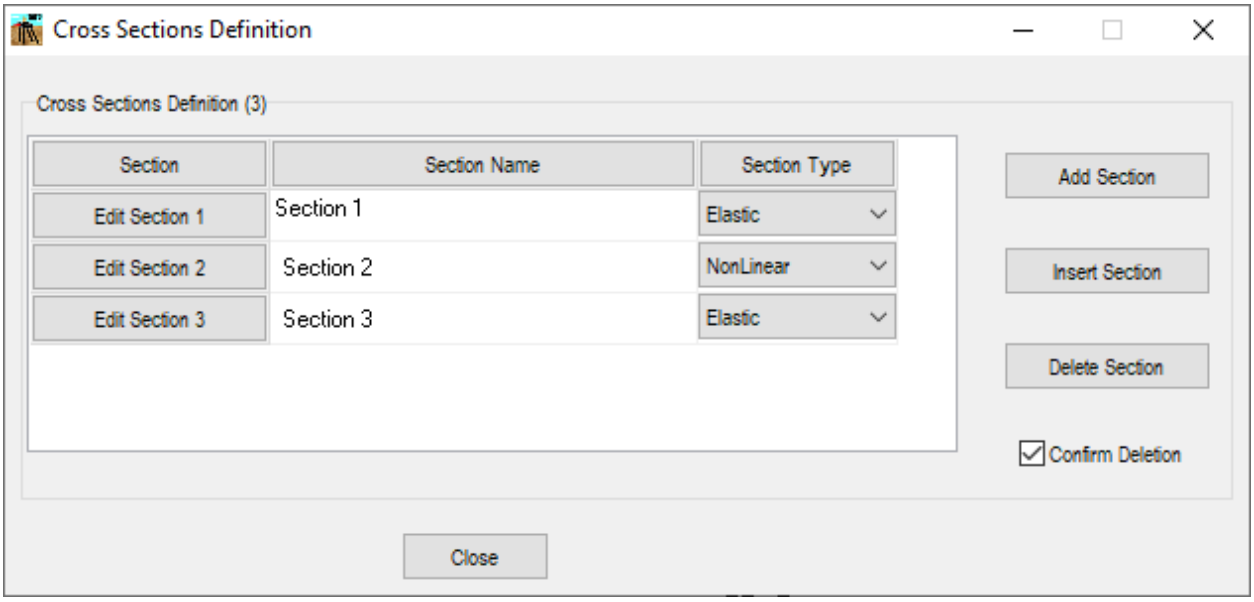


Figure 3.32 Cross Sections Definition Window

Cross Sections Definition

- **Add Section.** Add a new row at the end of the list.
- **Insert Section.** Add a new row above the current row.
- **Delete Section.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a section.

For each cross section (row) the following columns are available:

- **Edit Section**. Click on the **Edit Section** button to open the **Edit Cross Section** window for defining the cross-section dimensions.
- **Section Name**. In the second column enter the name associated with the cross section being defined.
- **Section Type**. In the third column select the type of section. It could elastic or nonlinear.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.3.2.2 Edit Cross Section Window (Elastic Section)

In this window the user can define each cross section (elastic). Bending stiffness, axial stiffness and torsional stiffness of the pile section are considered elastic while selecting this option. The section number and section name are displayed at the top of the window. A sample of this window is shown in Figure 3.33

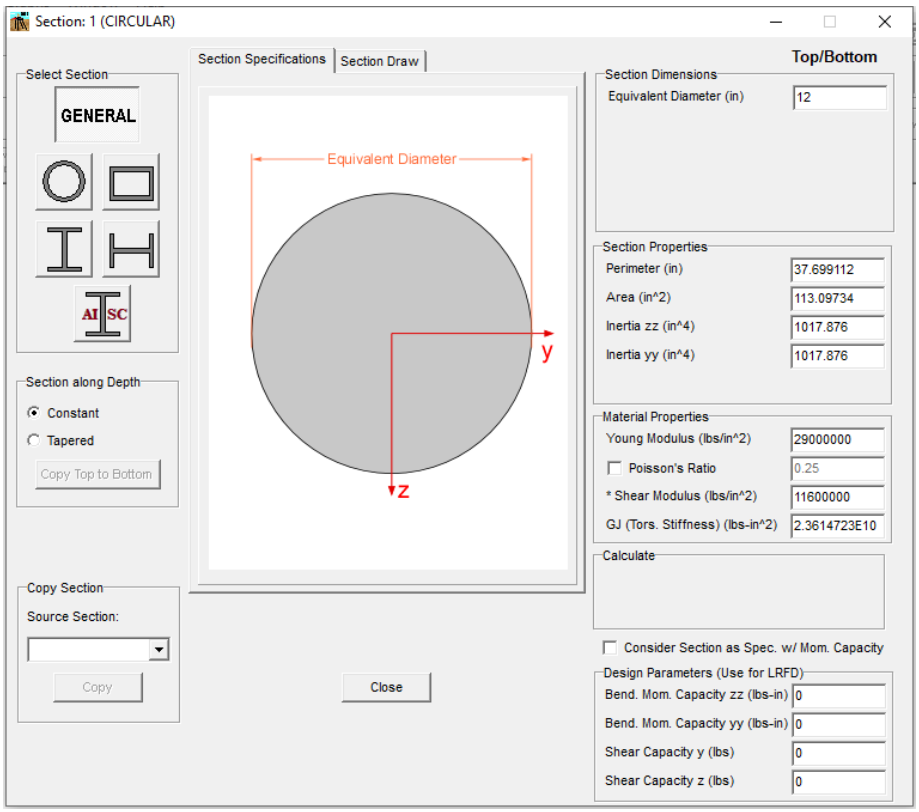


Figure 3.33 Edit Cross Section Window

Select Section

While using the **Select Section** feature, the user can choose from 5 different pile section types which will be graphically displayed in the middle of the window. On the right, the **Section Dimensions** will adjust depending on the section type selected. The five sections available are:

- **General Section.** Use for a general shape section. This section will be assigned automatically to models created from previous versions of GROUP. See Figure 3.34.

- **Circular Pipe Section.** Also used to define a solid circular section. See Figure 3.34.
- **Rectangular Pipe Section.** Also used to define a solid rectangular section. See Figure 3.35.
- **I Section.** See Figure 3.36.
- **H Section.** See Figure 3.37.
- **AISC Section.** See Figure 3.38.

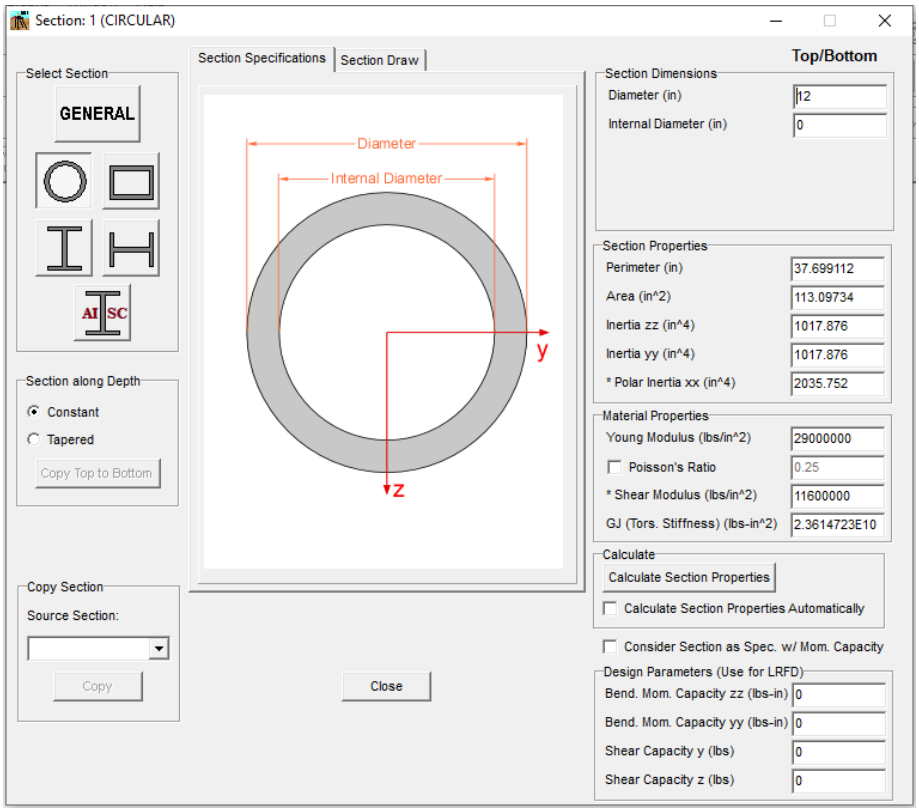


Figure 3.34 Edit Cross Section Window. Circular Pipe Section

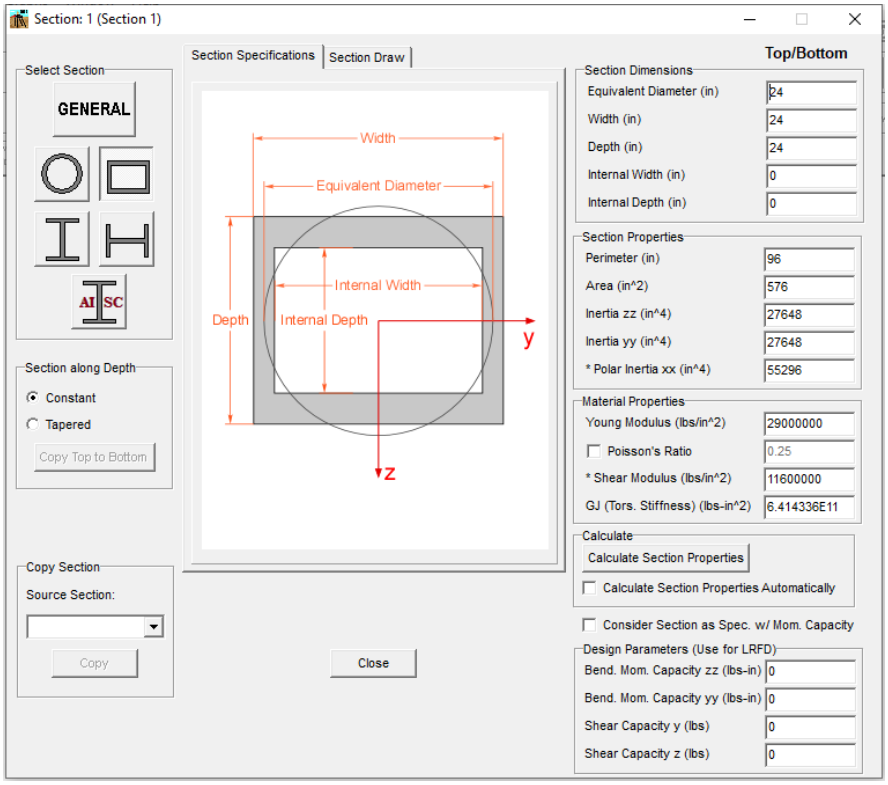


Figure 3.35 Edit Cross Section Window. Rectangular Pipe Section

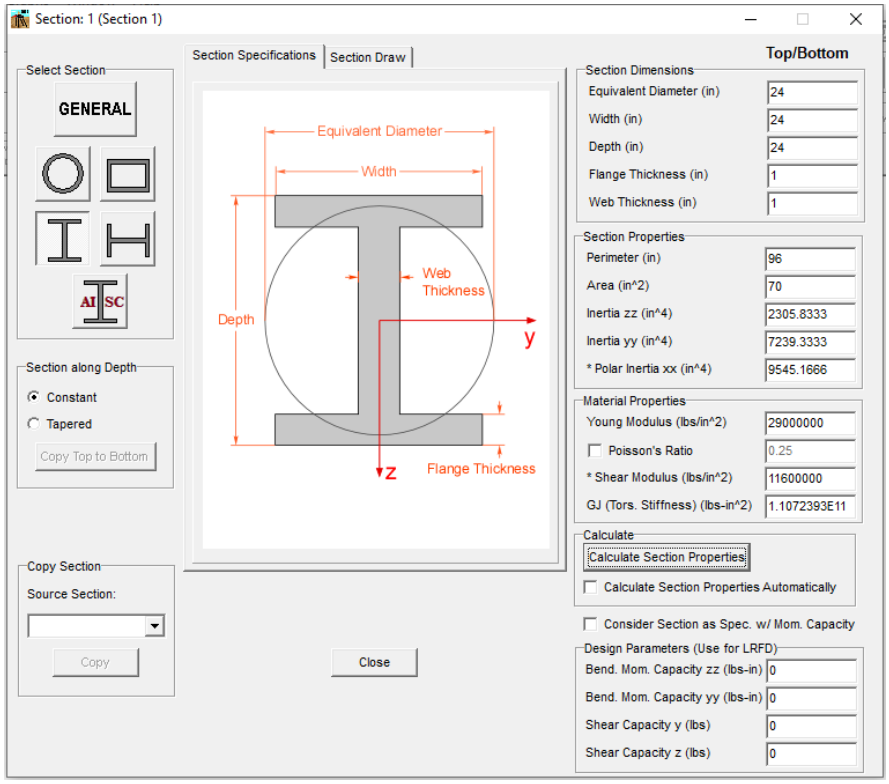


Figure 3.36 Edit Cross Section Window. I Section

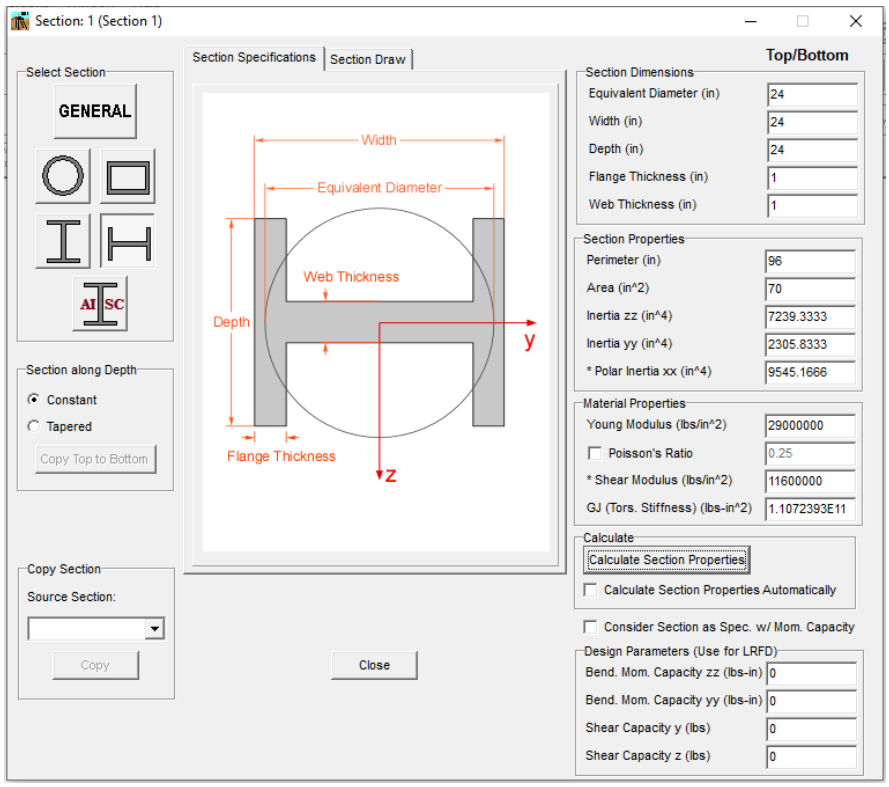


Figure 3.37 Edit Cross Section Window. H Section

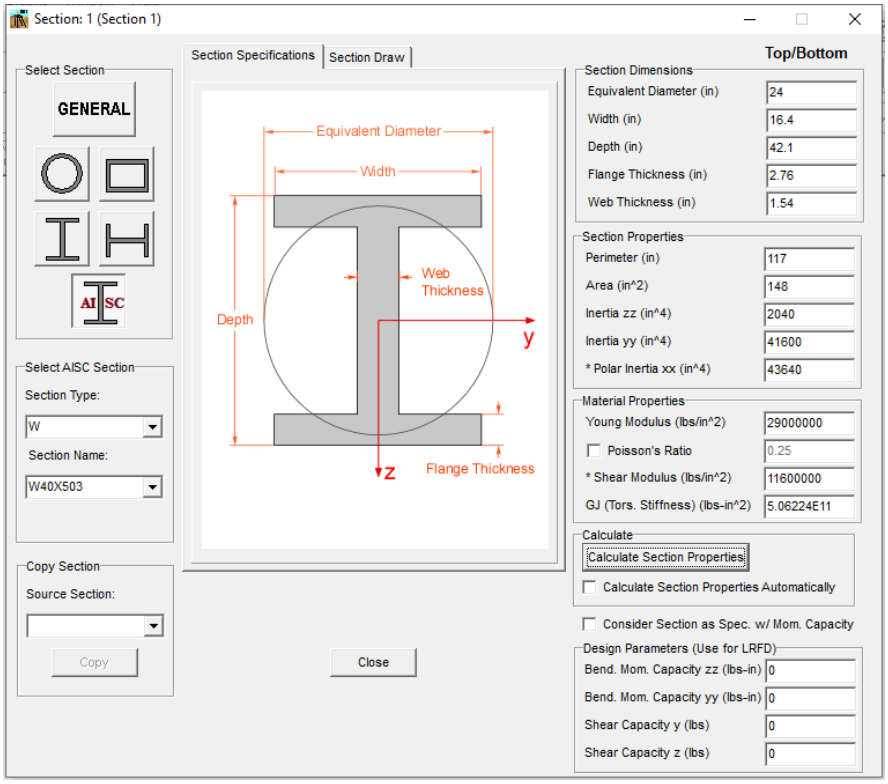


Figure 3.38 Edit Cross Section Window. AISC Section

Copy Section

The option is helpful for when there are cross sections that only have a few differences. In this case, the user can simply copy the section and make the desired changes. Select the cross section that is going to be the source (already defined), and then click on the **Copy** button to duplicate the data from the source. This option is available if at least two elastic cross sections have been defined.

Section along Depth

While using the **Section along Depth** feature, the user can define a constant section or a tapered section along the depth of the pile. For a constant section only one set of input parameters are required; however, for a tapered section property both top and bottom are required.

If a tapered section is defined the Button **Copy Top to Bottom** is available. While clicking on this button, all the parameters defined for the top are copied to the bottom.

Please note that if a tapered section is defined the program will use a linear interpolation for all the variables to estimate values between top and bottom of the section.

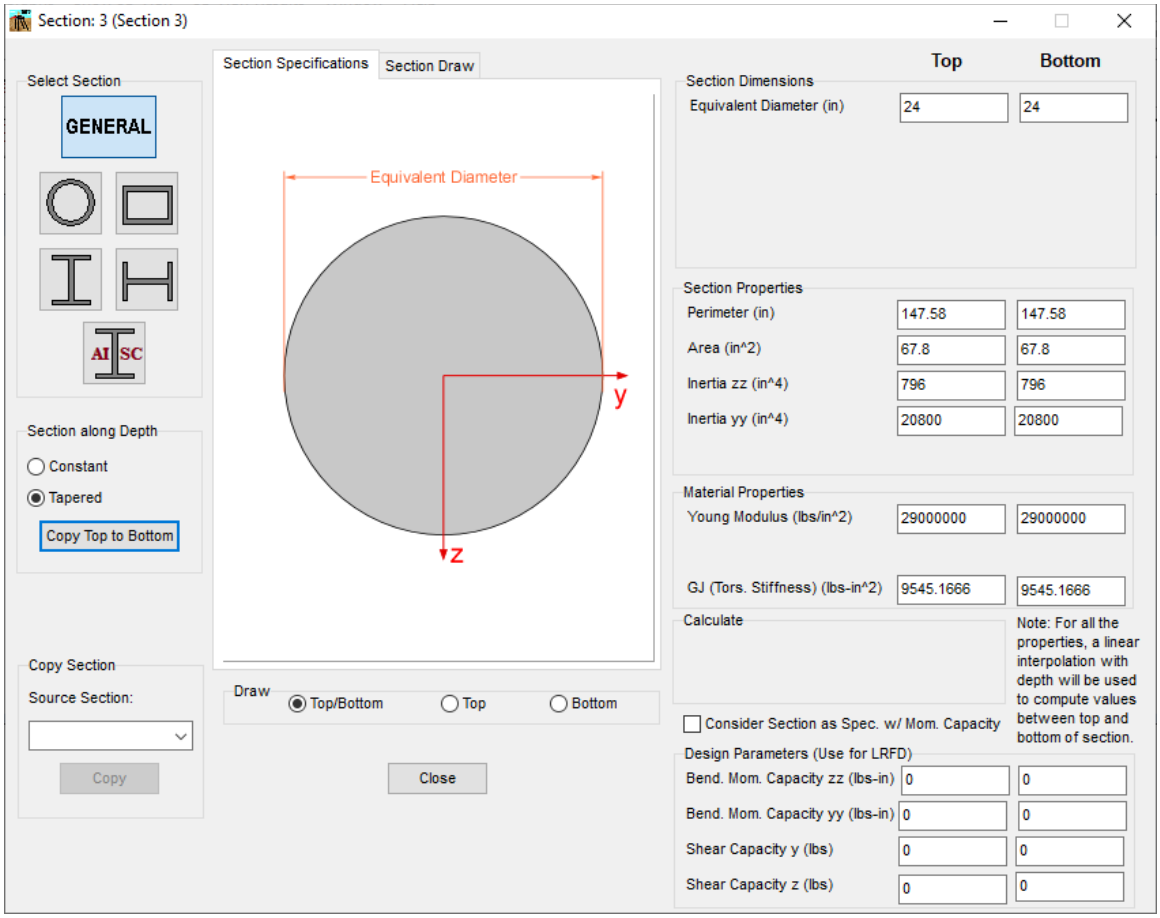


Figure 3.39 Edit Cross Section Window for a Tapered Section

Section Specifications/Section Draw

In the Section Specifications tab, a picture showing the cross section with labels will appear. When the user clicks on a label, the corresponding field will be selected. This is an alternate method for entering section dimensions.

By selecting the Section Draw tab, a scale drawing of the current section will be shown.

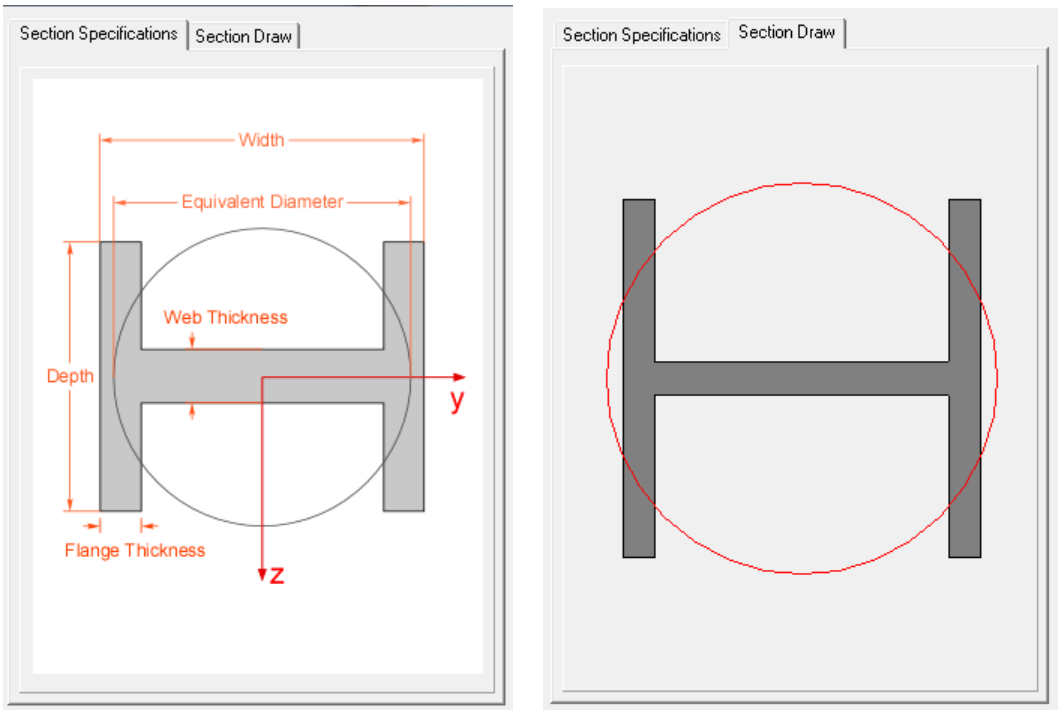


Figure 3.40 Toggle between Section Specifications and Section Draw. H section

If a tapered section is defined the user can choose to draw both sections (top and bottom), only top or only bottom one.

Section Dimensions

Input dimensions parameters will depend on the cross-section type selected. The following parameters are needed

- **General Section.**
  - *Equivalent Diameter*. Diameter for circular sections or an equivalent diameter for noncircular sections. Program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis. Program does not use this value for sections of beam elements (flexible cap version).

- **Circular Pipe Section.**
  - *Diameter*. External diameter of the pile. Program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis.
  - *Internal Diameter*. Internal diameter of the pile. Leave as zero for a solid pile.

- **Rectangular Pipe Section.**
  - *Equivalent Diameter*. For non-circular sections user must enter the value for equivalent diameter. Program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis. Program does not use this value for sections of beam elements (flexible cap version).
  - *Width*. Width of the cross section.
  - *Depth*. Depth of the cross section.
  - *Internal Width*. Internal width of the cross section. Leave as zero for a solid pile.
  - *Internal Depth*. Internal depth of the cross section. Leave as zero for a solid pile.

- **I or H Section.**
  - *Equivalent Diameter*. For non-circular sections user must enter the value for equivalent diameter. Program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis. Program does not use this value for sections of beam elements (flexible cap version).
  - *Width*. Width of the cross section.
  - *Depth*. Depth of the cross section.
  - *Flange Thickness*. Flange thickness of the cross section.
  - *Web Thickness*. Web thickness of the cross section.

- **AISC Section.**
    - *Equivalent Diameter*. For non-circular sections user must enter the value for equivalent diameter. Program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis. Program does not use this value for sections of beam elements (flexible cap version).
    - Other parameters will be populated based on the section selected.

Select AISC Section

These options are available if the AISC Section has been selected. While selecting a section, section parameters will be populated. For non-circular sections, user must enter the value for equivalent diameter.

- **Section Type.**
  Select the section type required. The following sections are available:
    - *W*.
    - *M*
    - S
    - HP
    - HSS
    - PIPE

- **Section Name.**
  Select the name of the section required:

- **Rotate Section.**
  Check this option to rotate the section 90 degrees. Useful for 2D models for loading along the weak axes of the section.

Section Properties

The user can input section properties, or let the computer estimate those properties. For AISC sections the properties are populated after selecting the section

- **Perimeter.** Perimeter of the section being defined. Program uses this value to estimate the side resistance contribution for axial analysis.
- **Area.** Cross sectional area (A) of the section being defined. Program uses this value to estimate the tip resistance contribution for axial analysis, to estimate the contribution to the axial stiffness of the pile for axial analysis and to estimate the maximum stress on the section.
- **Inertia zz.** Moment of inertia referred to the local axis z (Izz). Program uses this value to estimate the bending stiffness in the zz direction.
- **Inertia yy.** Moment of inertia referred to the local axis y (Iyy). This value is needed for three-dimensional models. Program uses this value to estimate the bending stiffness in the zz direction.
- **Polar Inertia xx.** Polar moment of inertia referred to the local axis x (Ixx=J). This value is used only to estimate the value of torsional stiffness (GJ) for three-dimensional models.

Material Properties

User should input material properties.

- **Young Modulus.** Young Modulus (Modulus of elasticity) of element material. Program uses this value to estimate the contribution to the axial stiffness of the element for axial analysis and to estimate the bending stiffness for lateral analysis.
- **Poisson Patio.** Poisson (Modulus of elasticity) of the pile material. Program uses this value to estimate the contribution to the axial stiffness of the pile for axial analysis and to estimate the bending stiffness for lateral analysis.
- **Shear Modulus.** Shear modulus of the pile material. This value is used only to estimate the value of torsional stiffness (GJ) for three-dimensional models.
- **GJ (Torsional Stiffness).** This number correspond to the torsional stiffness of the section (GJ). This value is needed for three-dimensional models.

Calculate

These options are available for sections other than the general section.

- **Calculate Section Properties.** Click on this button to calculate the section properties based on the section dimensions. For three-dimensional models it also estimates the torsional stiffness based on the polar moment of inertia and the shear modulus.
- **Calculate Section Properties Automatically.** By selecting this option, the program will automatically compute the section properties. Any change in the section dimensions will be reflected automatically in the section properties.

Note: For the Torsional Constant (J) the program will assume the value of Polar Moment of Inertia (Ixx=Iyy+Izz).

Design Parameters

By entering these parameters, GROUP will compute the demand/capacity ratio for bending moment and shear capacity for the load combinations. This ratio is useful for checking the level of stresses in the pile. This ratio is not intended to replace a design procedure.

- **Bending Moment Capacity.** Capacity in bending moment of the cross section.
- **Shear Capacity.** Capacity in shear of the cross section.

If the capacity values are left as zero, no ratio is computed.

Considered Section as Spec. w/Moment Capacity

By checking this option, the section will be assumed to have an elastoplastic behavior. If the demanded moment is smaller than the moment capacity, the section will behave elastic. If the demanded moment reaches the moment capacity, a plastic hinge will be formed.

Close Button

Click on the **Close** button to close the window and go back to the **Cross Section Definition** window.

Note about estimation of Total Stress:

The program will output the maximum total normal stress along the pile depth. For elastic sections, the total stress is estimated as the stress at the outer fiber per section. The normal stress includes the contribution due to the axial force and the contribution due to the bending moment. Please disregard the total stress values printed by the program for nonlinear sections.

For general and circular sections, the total stress is computed as:

$$\sigma = \left|\frac{P}{A}\right| + \left|\frac{\sqrt{M_z{}^2 + M_y{}^2}\ d}{2\ I_z}\right|$$

For other sections the total stress is computed as:

$$\sigma = \left|\frac{P}{A}\right| + \left|\frac{M_z\ wd}{2\ I_z}\right| + \left|\frac{M_y\ dp}{2\ I_y}\right|$$

where:

σ, is the total stress at specified depth

A, is the area of the cross section

I_z, is the moment of inertia along the z axis

I_y, is the moment of inertia along the y axis (not considered for 2D analysis)

d, is the diameter of the section

wd, is the width of the section

dp, is the depth of the section

P, is the axial force

M_z, is the bending moment along the z axis

M_y, is the bending moment along the y axis (not considered for 2D analysis)

3.3.2.3 Edit Cross Section Window (NonLinear Section)

In this window the user can define each cross section (nonlinear). Program will use parameters to estimate the nonlinear bending stiffness (moment curvature diagram) of the pile section. Axial stiffness and torsional stiffness of the pile section are still considered elastic. The section number and section name are displayed at the top of the window. A sample of this window is shown in Figure 3.41

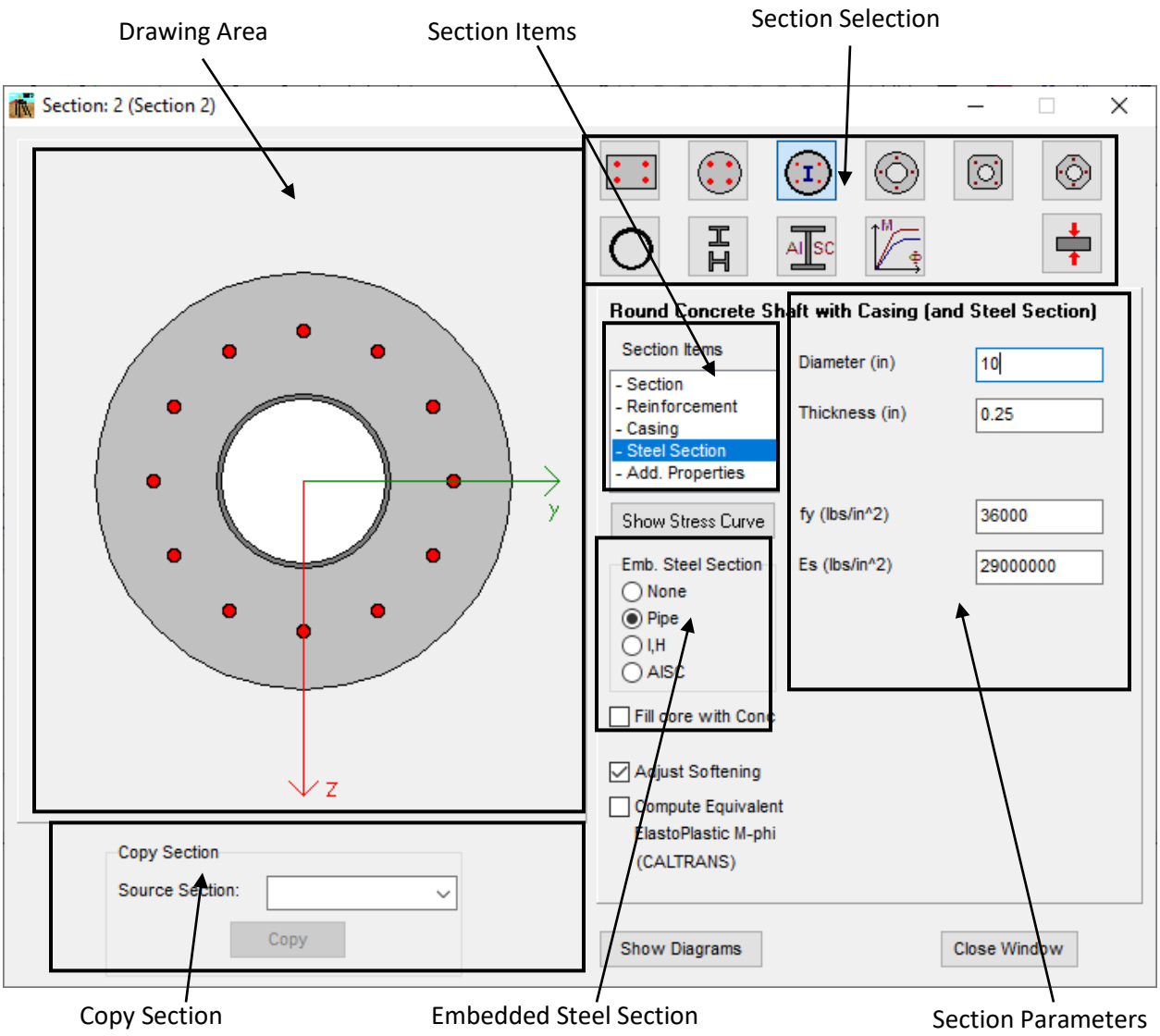


Figure 3.41 Edit Cross Section Window (Nonlinear Section)

Section Selection

While using the **Section Selection** feature, the user can choose from 10 different pile section types which will be graphically displayed in drawing area of the window. On the right, the **Section Items** will adjust depending on the section type selected. The ten sections available are:

- **Rectangular Concrete Pile.** See Figure 3.42.
- **Round Concrete Shaft.** See Figure 3.43.
- **Round Concrete Shaft with Casing (and Steel Section).** See Figure 3.44.
- **Round Prestressed Pile.** See Figure 3.45.
- **Square Prestressed Pile.** See Figure 3.46.
- **Octagonal Prestressed Pile.** See Figure 3.47.
- **Steel Pipe Pile.** See Figure 3.48.
- **Steel I, H Section Pile.** See Figure 3.49.
- **Steel AISC Section Pile.** See Figure 3.50.
- **User Defined EI.** See Figure 3.51.

  Also, the user can select the thrust forces option to define the forces at which the moment curvature curves will be computed.
- **Thrust Forces.** See Figure 3.52.

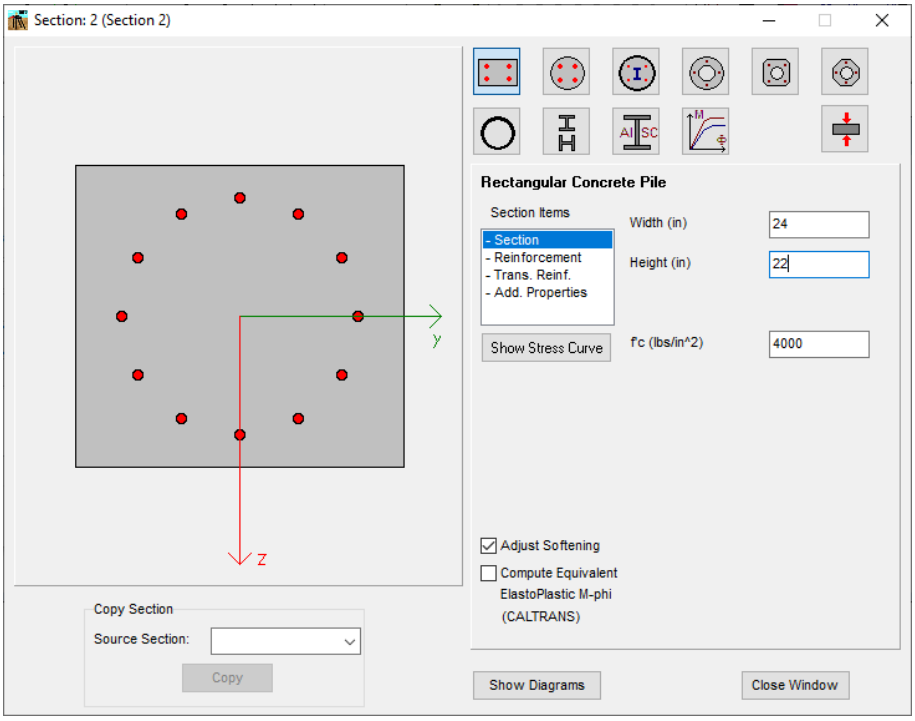


Figure 3.42 Edit Cross Section Window. Rectangular Concrete Pile

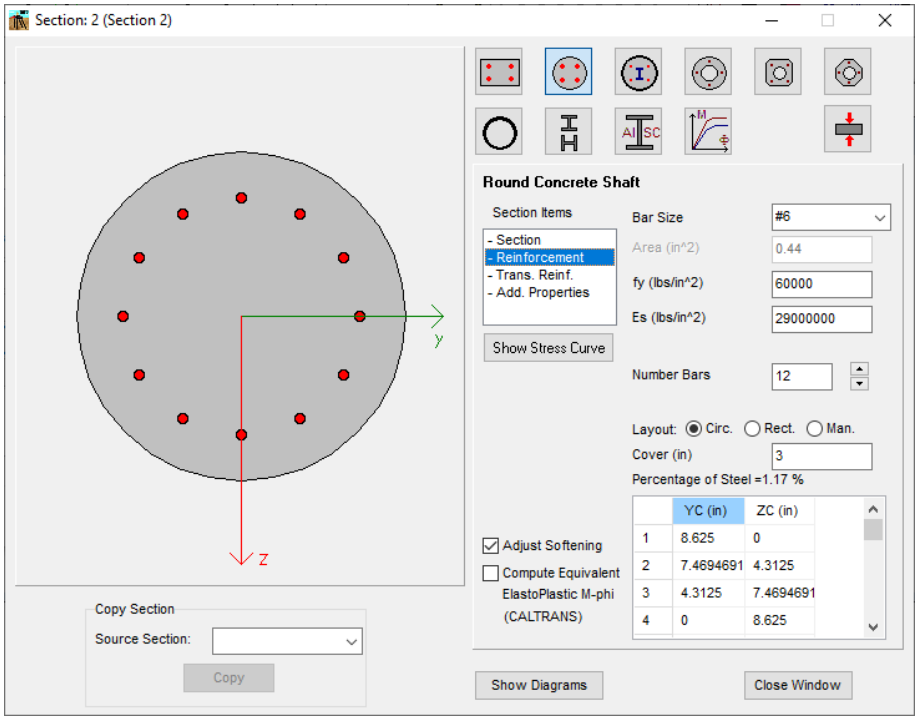


Figure 3.43 Edit Cross Section Window. Round Concrete Shaft

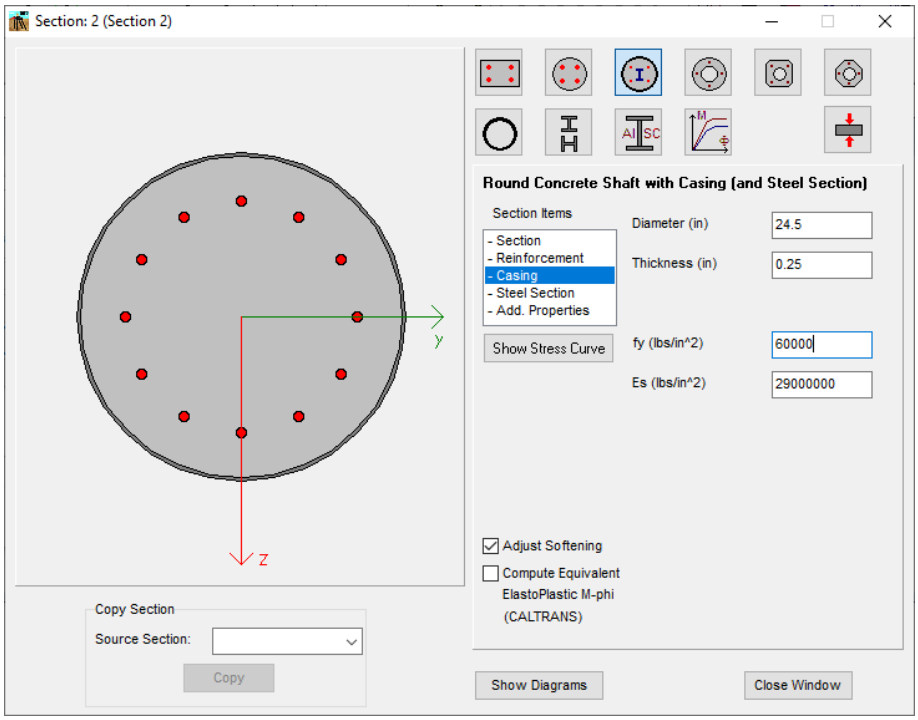


Figure 3.44 Edit Cross Section Window. Round Concrete Shaft with Casing (and Steel Section)

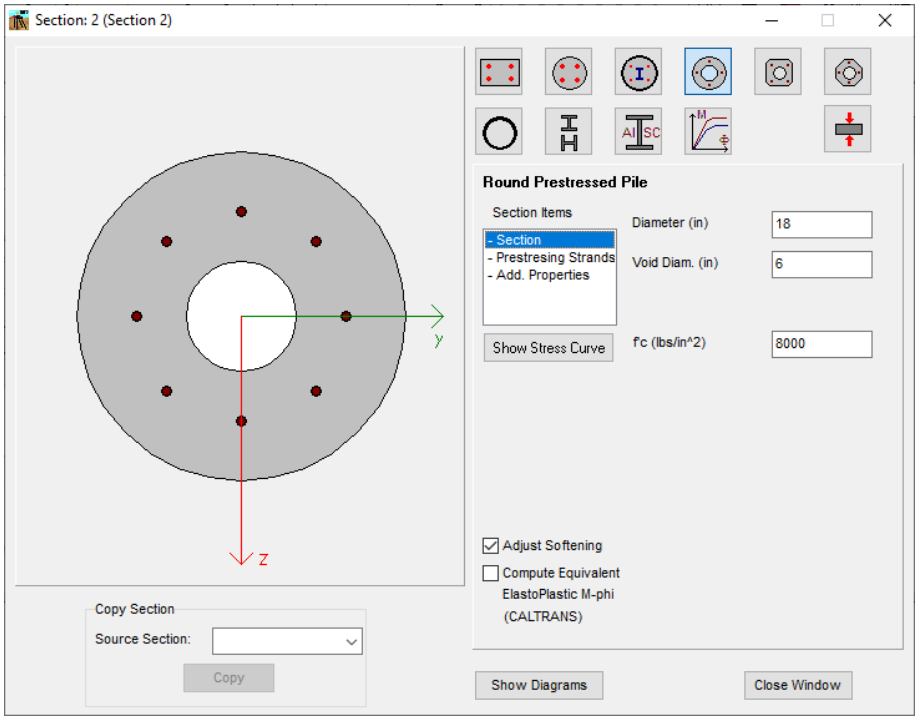


Figure 3.45 Edit Cross Section Window. Round Prestressed Pile

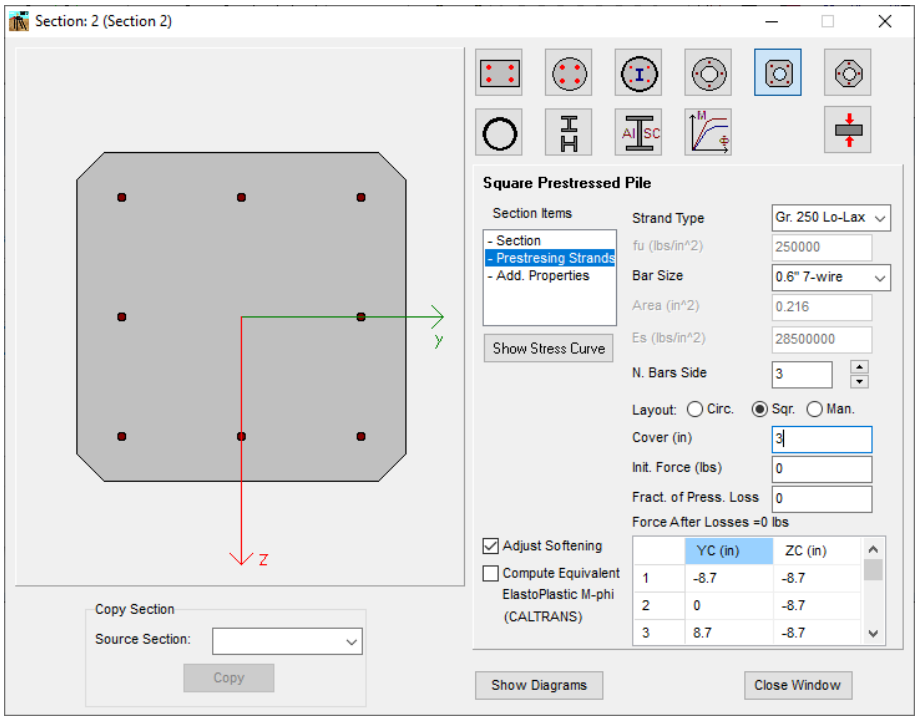


Figure 3.46 Edit Cross Section Window. Square Prestressed Pile

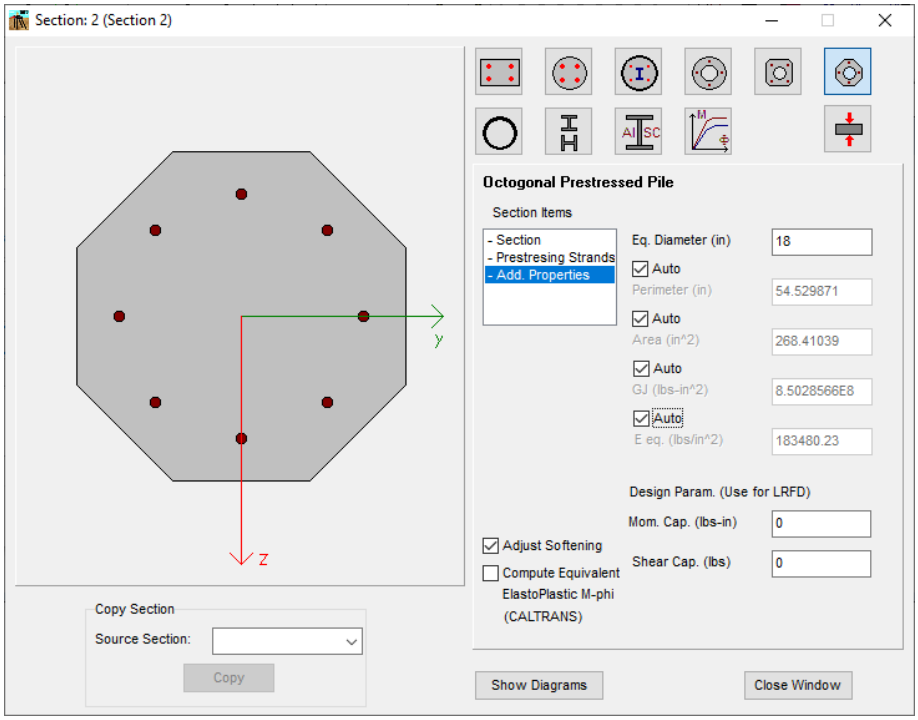


Figure 3.47 Edit Cross Section Window. Octagonal Prestressed Pile

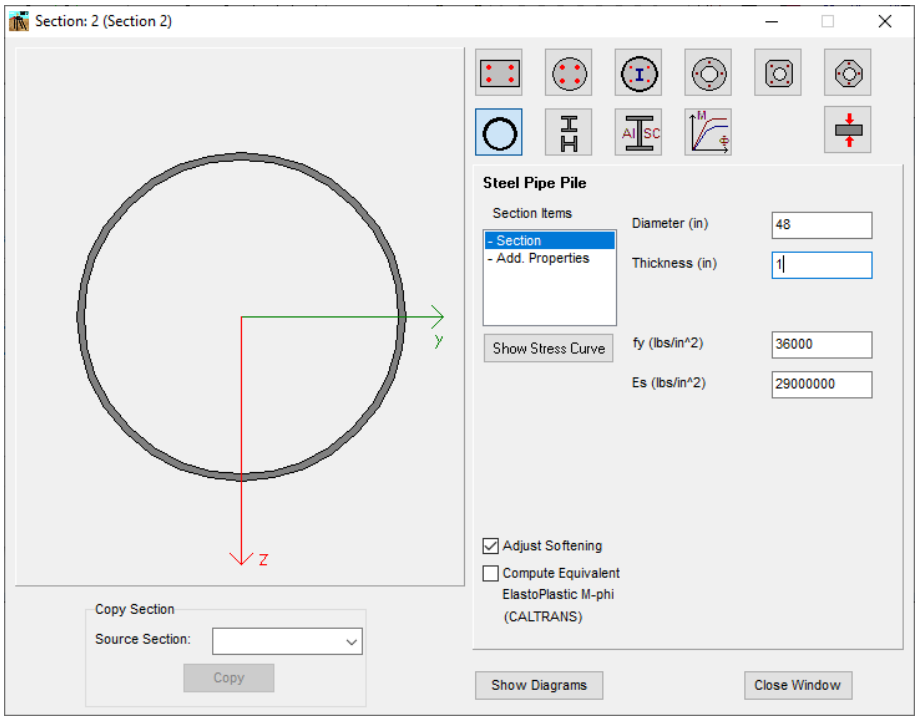


Figure 3.48 Edit Cross Section Window. Steel Pipe Pile

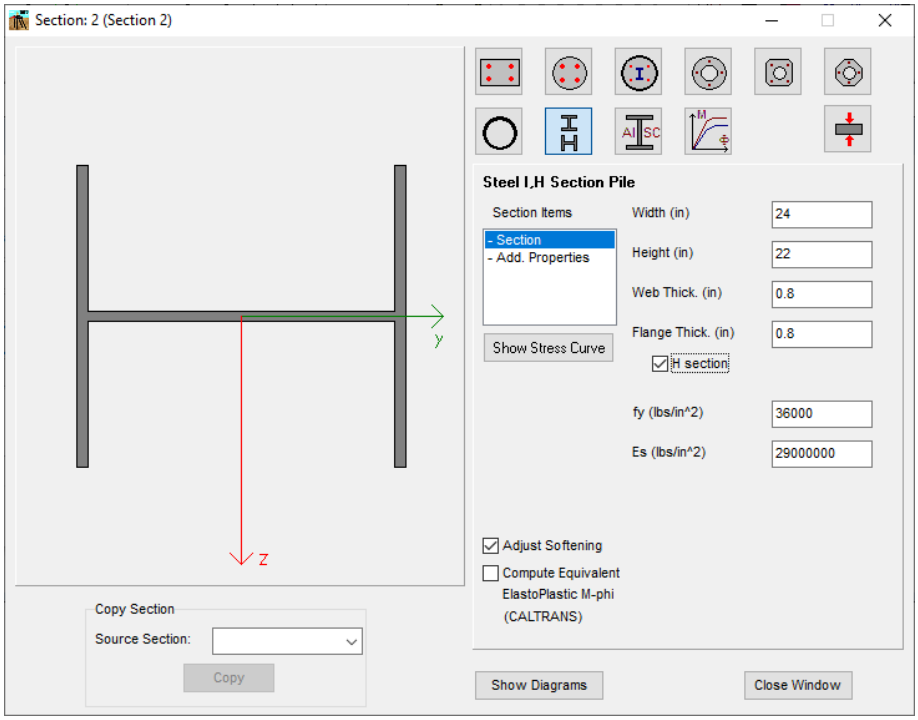


Figure 3.49 Edit Cross Section Window. Steel I, H Section Pile

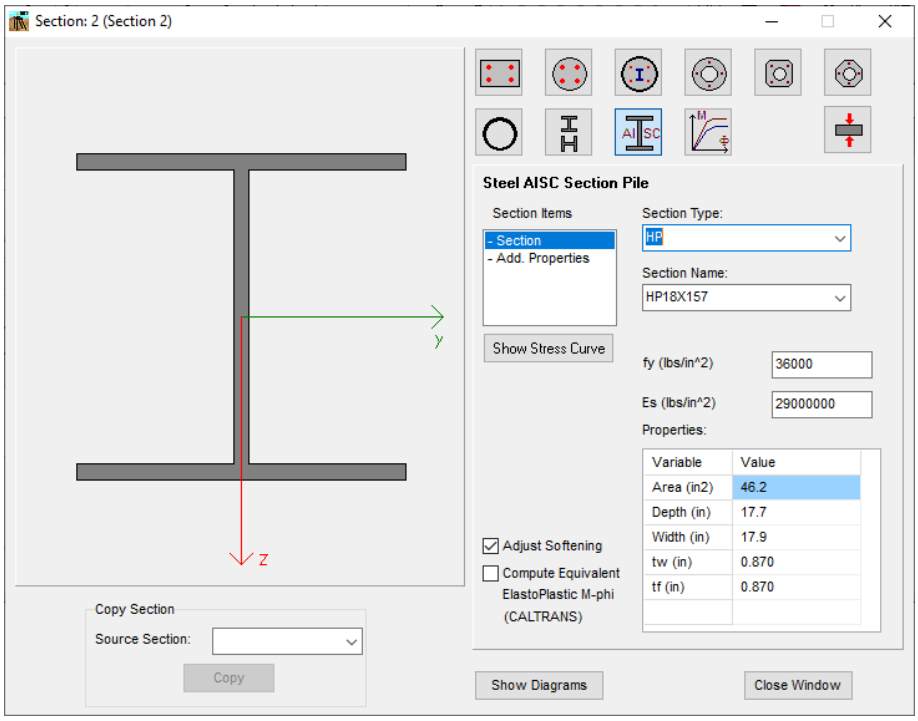


Figure 3.50 Edit Cross Section Window. Steel AISC Section Pile

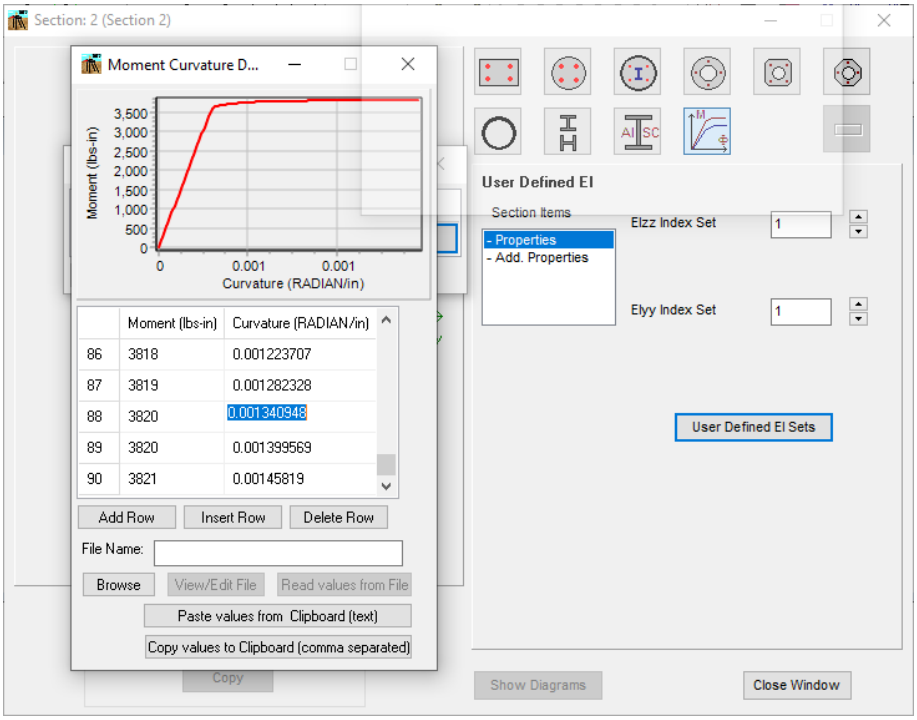


Figure 3.51 Edit Cross Section Window. User Defined EI

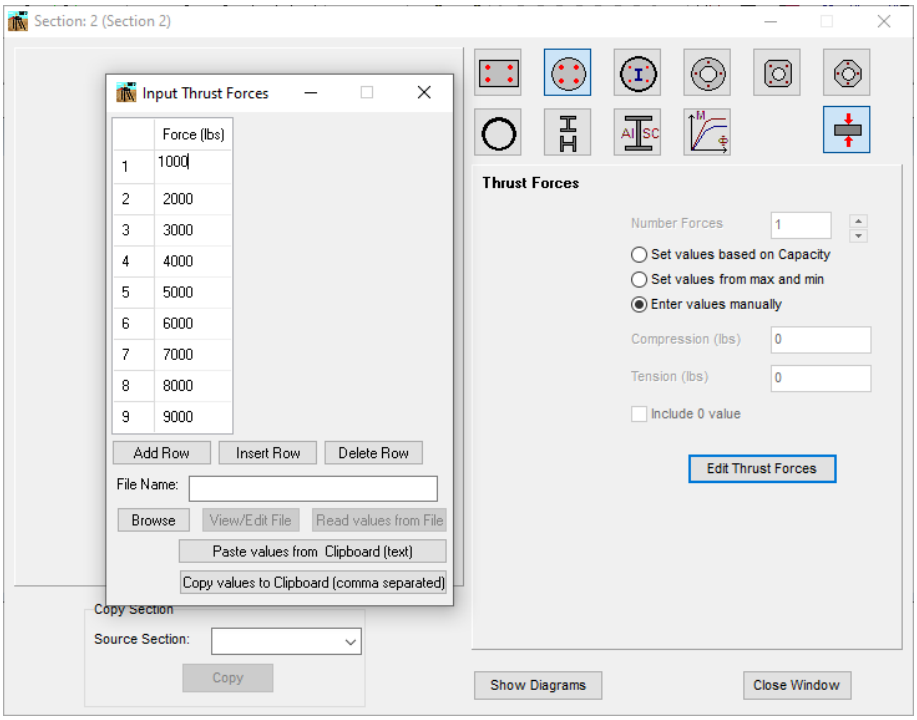


Figure 3.52 Edit Cross Section Window. Thrust Forces

Section Items

Each section has different section items available. On the right, the **Section Parameters** will adjust depending on the section item selected. The following items are available:

- **Section.** Available to all sections except User Define EI. Input parameters are the section dimensions and concrete compressive strength for concrete sections or yield strength and Young modulus for steel sections.
- **Reinforcement.** Available for all reinforced concrete sections. Input parameters are the reinforcement properties and reinforcement layout.
- **Trans. Reinf.** Available for both rectangular and round reinforced concrete sections. Input parameters are the transverse reinforcement properties needed to estimate the concrete confinement following Mander equations.
- **Prestressing Strands.** Available for all prestressed concrete sections. Input parameters are the strands properties and strands layout.
- **Casing.** Available only for the round concrete shaft with casing section. Input parameters are the casing dimensions and yield strength and Young modulus for the casing.
- **Steel Section.** Available only for the round concrete shaft with casing section. While selecting this option the Embedded Steel Section will appear. The input required will depend on the embedded section selected.

- **Add. Properties.** Available to all sections. Input parameters are the additional parameters needed to complete the analyses and will depend on the section selected.
- **Properties.** Available only to the User Define EI. Input parameters are the indices of the user defined EI set associated with the section.

Embedded Steel Section

This option is available only for the round concrete shaft with casing section if the Steel Section was selected under Section Items On the right, the **Section Parameters** will adjust depending on the section selected. The following items are available:

- **None.**
- **Steel Pipe Pile.**
- **Steel I, H Section Pile.**
- **Steel AISC Section Pile.**

Section Parameters

Input dimensions parameters will depend on the cross-section type and section item selected. The input parameters, per section item, are:

- **Section.**
    - *Width*. Width of the cross section.
    - *Height*. Height of the cross section.
    - *Diameter*. External diameter of the section. In addition to estimate the bending stiffness of the section, the program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis.
    - *Void Diam.* Internal diameter of the section for prestressed sections. Leave as zero for a solid section.
    - *Corner Chamf.*. Length of the corner chamfer. Used for square prestressed section.
    - *Thickness*. Thickness of the steel pipe section.
    - *Web Thick*. Thickness of the web for I or H steel section.

- *Flange Thick*. Thickness of the flange for I or H steel section.
- *Section Type*. Select the AISC section type. The following sections are available:
  - *W*.
  - *M*
  - S
  - HP
  - HSS
  - PIPE
- *Section Name*. Select the AISC section name. The properties table will be populated.
- *f'c*. Concrete compressive strength.
- *fy*. Steel yield strength.
- *Es*. Steel Young's modulus.
- *H Section*. Check this option to assign an H section.
- *Rotate Section*. Available for AISC Sections. By check this option the section is rotated 90 degrees. Useful for 2D models for loading along the weak axes of the section.
- *EIzz set*. Select the nonlinear EI set in the zz direction. Select 0 por an elastic section in the zz direction.
- *EIyy set*. Select the nonlinear EI set in the yy direction. Select 0 por an elastic section in the yy direction.
- *Izz*. Moment of Inertia in the zz direction. Available only if 0 was selected for the EI zz set.
- *Iyy*. Moment of Inertia in the yy direction. Available only if 0 was selected for the EI yy set.

- **Reinforcement.**
  - *Bar Size*. Select rebar from table or User Defined to enter the area.
  - *Area*. Enter area of rebar for user defined rebar, otherwise the program estimates it based on selected bar size.
  - *fy*. Reinforcement yield strength.
  - *Es*. Reinforcement Young's modulus.
  - *Number of Bars*. Number of reinforcement rebars in the cross section. Used for circular layout or manual layout.
  - *Number Bars Y*. Number of reinforcement rebars in the Y direction. Used for rectangular layout.
  - *Number Bars Z*. Number of reinforcement rebars in the Z direction. Used for rectangular layout.

- *Circ*. By checking this option, the program will automatically compute the rebar coordinates based on a circular layout.
- *Rect*. By checking this option, the program will automatically compute the rebar coordinates based on a rectangular layout.
- *Man*. By checking this option, the user should enter the rebar coordinates.
- *Cover*. Distance from edge of section to edge of rebar. Used for circular or rectangular layout.
- *Y, Z*. Rebar coordinates. The program will automatically compute these values if circular or rectangular layout are selected. Otherwise, user should enter these values.

- **Trans. Reinf.**
  - *Confined Section*. Check on this option to let the program estimate the concrete confinement due to the transverse reinforcement.
  - *Spiral*. Available for circular sections. Select this option for spiral transverse reinforcement.
  - *Hoop*. Available for circular sections. Select this option if the transverse reinforcement is made up of hoops.
  - *Bar Size*. Select transverse rebar from table or User Defined to enter the area.
  - *As*. Enter area of transverse rebar for user defined rebar, otherwise the program estimates it based on selected bar size.
  - *Number Bars*. Number of transverse reinforcement rebars along the spacing.
  - *Set Legs from Long. Reinforcement*. Available for rectangular sections. Select this option to assign automatically the transverse rebars based on the longitudinal reinforcement.
  - *Number Legs Y*. Available for rectangular sections. Number of legs along the Y direction.
  - *Number Legs Z*. Available for rectangular sections. Number of legs along the Z direction.
  - *Spacing*. Distance between transverse reinforcement along the pile, from center to center.
  - *fys*. Transverse reinforcement yield strength.
  - *Use Strain Hardening Model*. Check on this option to let the program use the strain hardening model for longitudinal reinforcement.
  - *fu/fy*. Enter the ratio of ultimate stress (fu) over yield stress (fy). To be used for a hardening model.
  - *esh*. Enter the strain at which the hardening start. To be used for a hardening model.
  - *esu*. Enter the ultimate strain. To be used for a hardening model.

- **Prestressing Strands.**
  - *Strand Type*. Select strand type from table or User Defined to enter the ultimate strength.
  - *fu*. Enter the ultimate strength of the strand for user defined strand, otherwise the program estimates it based on selected strand type.
  - *Bar Size*. Select rebar from table or User Defined to enter the area.
  - *Area*. Enter area of strand for user defined strand, otherwise the program estimates it based on selected bar size.
  - *Number of Bars*. Number of prestressing strands in the cross section. Used for circular layout or manual layout.
  - *Number Bars Side*. Number of prestressing strands in each side. Used for square layout.
  - *Circ*. By checking this option, the program will automatically compute the prestressing strand coordinates based on a circular layout.
  - *Sqr*. By checking this option, the program will automatically compute the prestressing strand coordinates based on a square layout.
  - *Man*. By checking this option, the user should enter the prestressing strand coordinates.
  - *Cover*. Distance from edge of section to edge of rebar. Used for circular or rectangular layout.
  - *Init. Force*. Enter the initial force in the section after losses.
  - *Y, Z*. Strand coordinates. The program will automatically compute these values if circular or rectangular layout are selected. Otherwise, user should enter these values.

- **Casing.**
  - *Diameter*. External diameter of the steel casing. In addition to estimate the bending stiffness of the section, the program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis and to estimate the side friction contribution for axial analysis.
  - *Thickness*. Thickness of the steel casing.
  - *fy*. Steel casing yield strength.
  - *Es*. Steel casing Young's modulus.

- **Steel Section.**
  The input parameters will depend on the embedded steel section selected
  - *Diameter*. External diameter of the section. In addition to estimate the bending stiffness of the section, the program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis and to estimate the side friction contribution for axial analysis.
  - *Thickness*. Thickness of the steel pipe section.
  - *Web Thick*. Thickness of the web for I or H steel section.
  - *Flange Thick*. Thickness of the flange for I or H steel section.
  - *Section Type*. Select the AISC section type. The following sections are available:
    - *W*.
    - *M*
    - S
    - HP
    - HSS
    - PIPE
  - *Section Name*. Select the AISC section name. The properties table will be populated.
  - *fy*. Steel yield strength.
  - *Es*. Steel Young's modulus.
  - *H Section*. Check this option to assign an H section.
  - *Rotate Shape*. Check this option to rotate AISC sections 90 degrees. Useful to assign loading into weak axis for 2D models.

- **Add. Properties.**
  - *Eq. Diameter*. Enter the equivalent diameter for non-circular sections. The program uses this value to estimate the contribution of soil resistance (p-y curve) for lateral analysis.
  - *Perimeter*. Enter the perimeter of the section. The program uses this value to estimate the side friction contribution for axial analysis.
  - *Area*. Enter the area of the section. Click on Auto (if available) to let the program estimate de area based on the input parameters. Program uses this value to estimate the tip resistance contribution for axial analysis and to estimate the contribution to the axial stiffness of the pile for axial analysis.

- *GJ*. Enter the product of Shear Modulus and Polar Moment of Inertia of the section (GJ). Click on Auto (if available) to let the program estimate the value of GJ based on the input parameters.
- *E eq*. Enter the equivalent Young's of the section. Click on Auto (if available) to let the program estimate the value of equivalent Young's Modulus based on the input parameters. Program uses this value only to estimate the contribution to the axial stiffness of the pile for axial analysis.
- *Mom. Cap.* Capacity in bending moment of the cross section. Used for LRFD Analysis.
- *Shear Cap.* Capacity in shear of the cross section. Used for LRFD Analysis.

- **Properties.**
  - *EIzz set*. Select the nonlinear EI set in the zz direction. Select 0 por an elastic section in the zz direction.
  - *EIyy set*. Select the nonlinear EI set in the yy direction. Select 0 por an elastic section in the yy direction.
  - *Izz*. Moment of Inertia in the zz direction. Available only if 0 was selected for the EI zz set.
  - *Iyy*. Moment of Inertia in the yy direction. Available only if 0 was selected for the EI yy set.

- **Thrust Forces**
  - *Number Forces*. Number of thrust forces used to compute the moment curvature curves.
  - *Set values based on Capacity*. Check this option to let the program generate the values of axial loads based on the capacity on tension and compression.
  - *Set values from max and min*. Check this option to let the program generate the values of axial loads based on the maximum and minimum values entered.
  - *Enter values manually*. Check this option if the user will enter the values of axial loads.
  - *Compression*. Value estimated by the program for the compression capacity of the section. To avoid numerical errors the program uses a smaller value (about 5% smaller) to generate the axial loads.
  - *Tension*. Value estimated by the program for the tensile capacity of the section. To avoid numerical errors the program uses a larger value (about 5% larger) to generate the axial loads

- *Maximum*. Maximum value to be used to generate the axial loads if Set values from max and min is selected.
- *Minimum*. Minimum value to be used to generate the axial loads if Set values from max and min is selected.
- *Include 0 value*. Check this option to include the zero value in the list of axial loads generated by the program.
- *Axial Thrust*. The program will automatically compute these values if one of the Set values option was selected. Otherwise, user should enter these values.

Show Stress Curve

Click on the **Show Stress Curve** button to open a window displaying the stress curve used for the material which properties are being defined. It could show the stress curve for concrete, steel reinforcement or prestressing strand. See Figure 3.53 for an example displaying a concrete curve

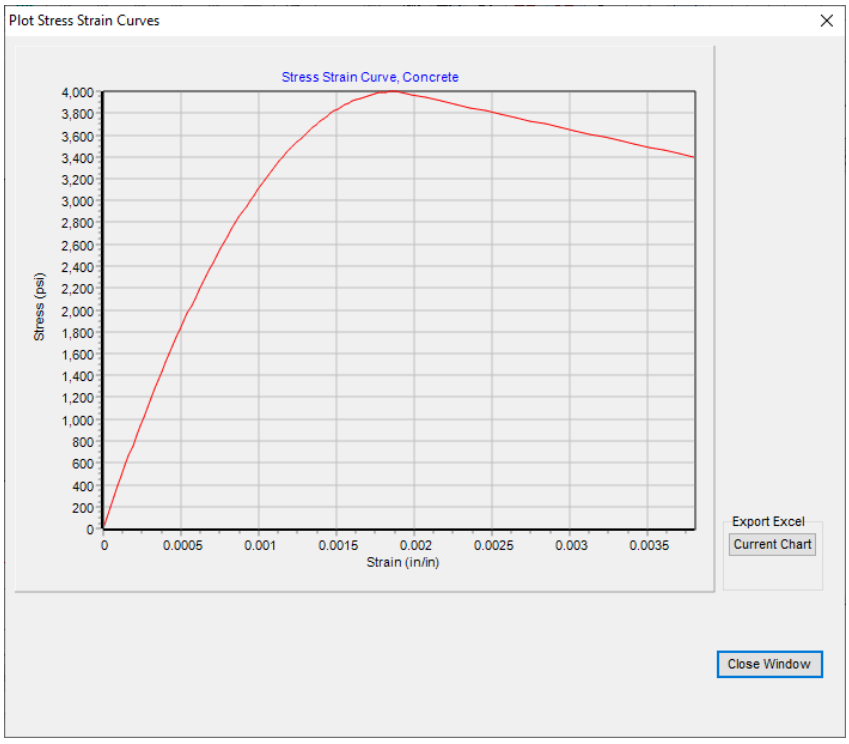


Figure 3.53 Show Stress Curve Window.

Copy Section

The option is helpful for when there are cross sections that only have a few differences. In this case, the user can simply copy the section and make the desired changes. Select the cross section that is going to be the source (already defined), and then click on the **Copy** button to duplicate the data from the source. This option is available if at least two non-linear cross sections have been defined.

Show Diagrams

Click on the **Show Diagrams** button to display the nonlinear EI curves for the current section. See Figure 3.54 for an example displaying a concrete curve.

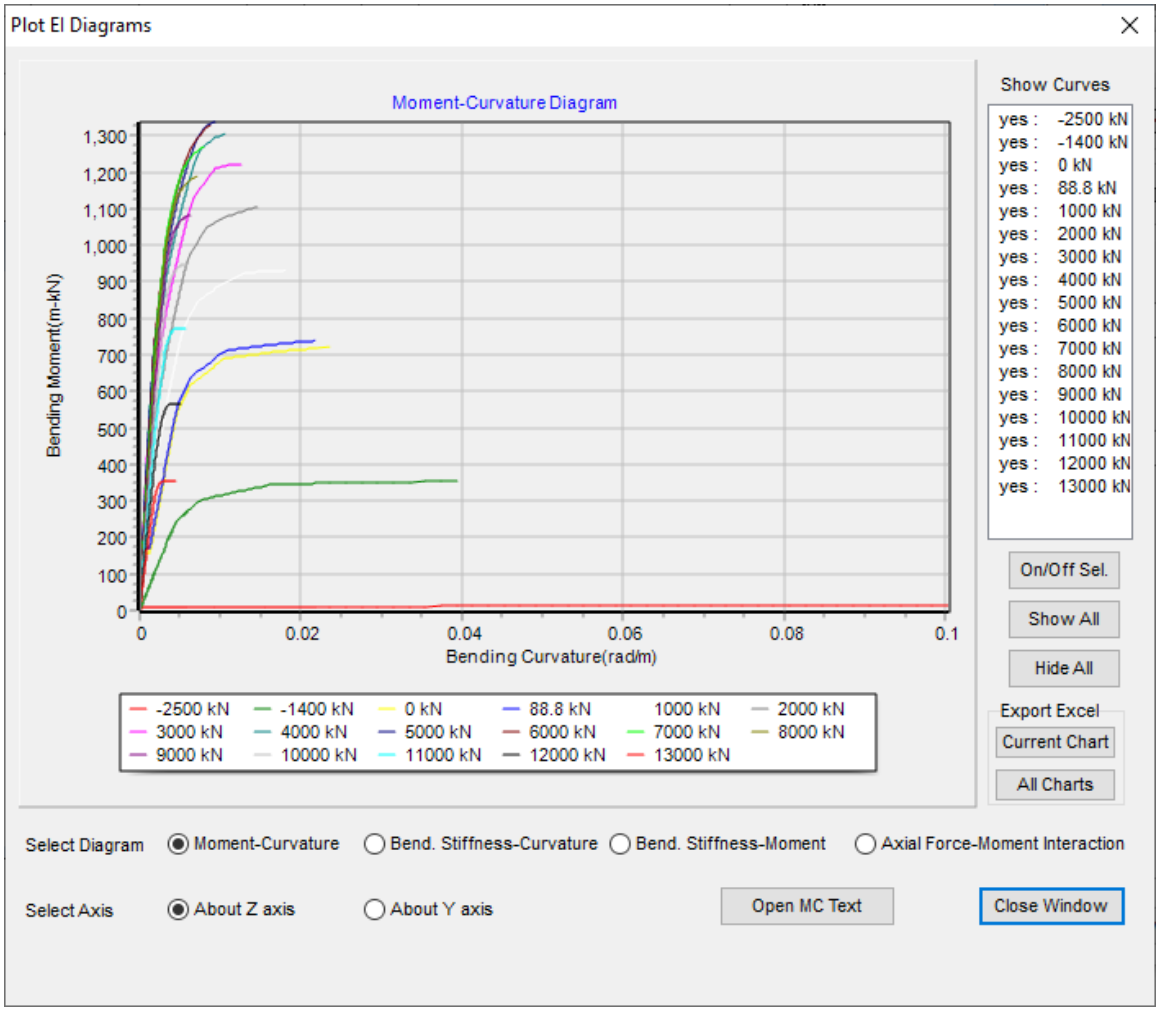
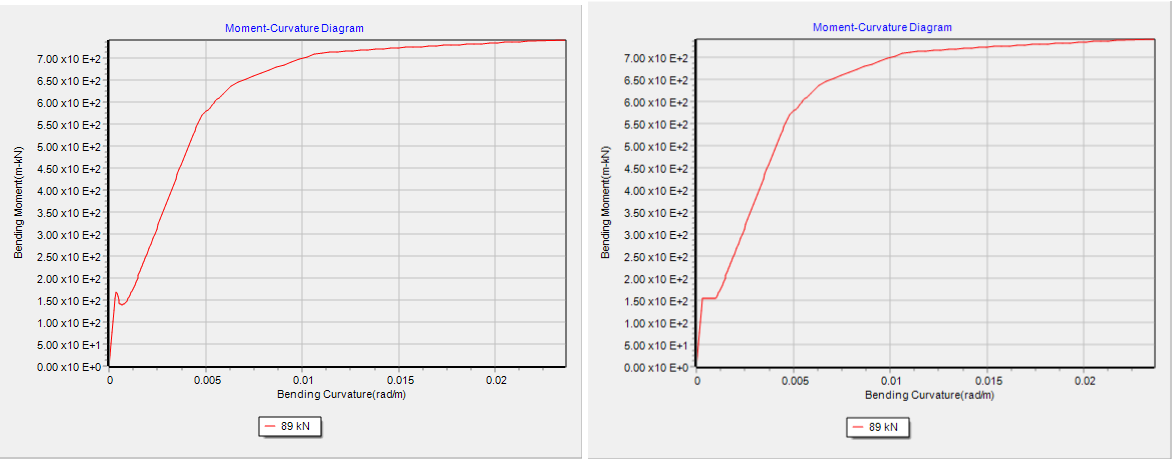


Figure 3.54 Show Diagrams Window.

Adjust Softening

Click on the **Adjust Softening** button to adjust the moment curvature when the moment decreases mainly due to initial cracking of the section. See Figure 3.55 for a comparison of the effect of this parameter in the moment curvature computed.



a) Without Adjust softening b) With Adjust Softening

Figure 3.55 Adjust Softening Parameter.

Compute Elastoplastic M-phi (CALTRANS)

Click on the **Compute Elastoplastic M-phi (CALTRANS)** button to estimate the equivalent elastoplastic moment curvature based on CALTRANS criteria.

By checking this option, GROUP will use the estimated elastoplastic moment curvature for analysis.

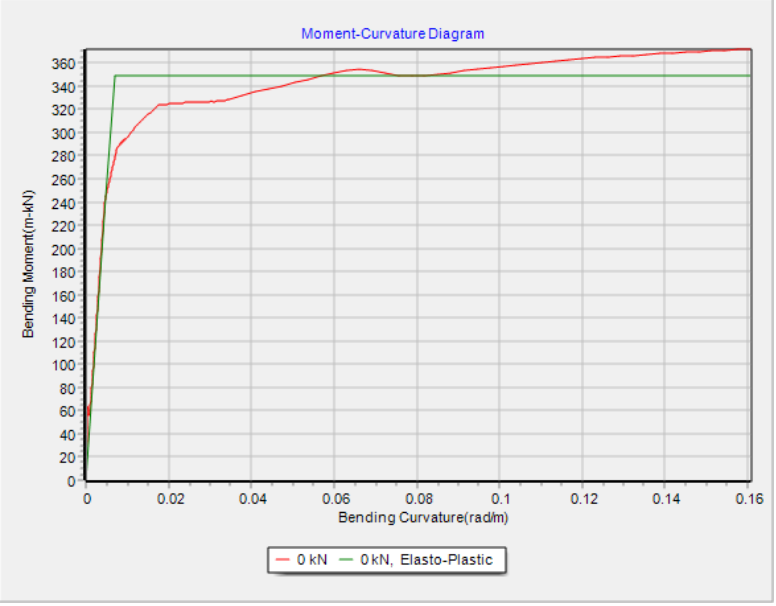


Figure 3.56 Compute Elastoplastic Moment Curvature.

Close Window Button

Click on the **Close Window** button to close the window and go back to the **Cross Section Definition** window.

3.3.3 Data - Pile Properties

This submenu option allows the user to input pile dimensions and section properties corresponding to each pile type used in the group analysis. For each type of pile, the user may enter up to 50 sections with different properties.

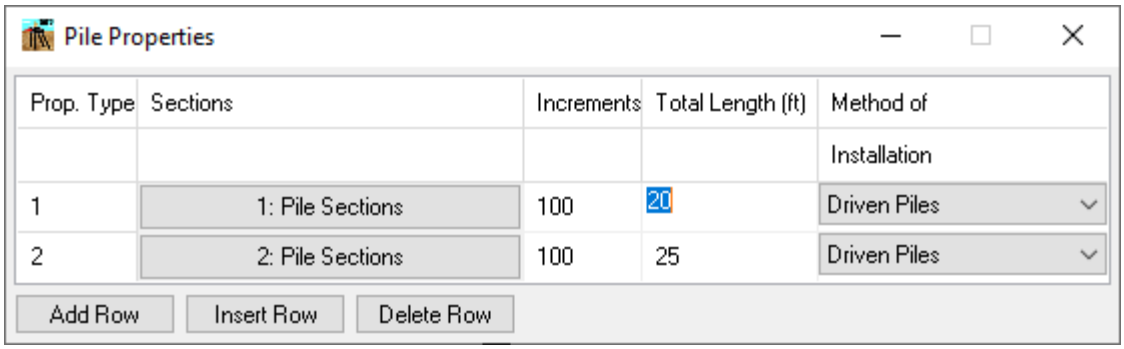


Figure 3.57 Pile Properties Window

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row above the current row.
- **Delete Row.** Delete the current row.

For each pile property (row) the following columns are available:

- **Prop. Type.** This is a sequential number that is provided to each type of pile used in the group analysis. This number is automatically assigned by the program as new types of piles are added. The maximum number of pile properties that may be used in a group analysis is limited to 50.
- **Sections.** Clicking the button under **Sections** in the **Pile Properties** window brings up the **Pile Sections** window.
- **Increments.** The number of increments in which each pile will be discretized. The maximum number of increments that can be defined is 200.
- **Total Length.** The total length specifies the length between the pile head and the pile tip. This number should not include the thickness of the pile cap.
- **Method of Installation.** The user should specify either Driven Piles or Drilled Shafts from the pull-down menu.

Pile Sections Window

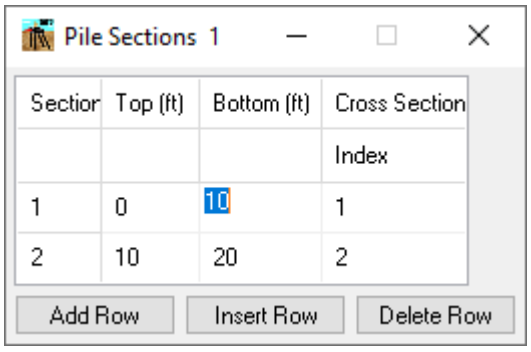


Figure 3.58 Pile Sections Window

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row above the current row.
- **Delete Row.** Delete the current row.

For each pile section (row) the following columns are available:

- **Section:** This is a sequential number that is provided to each pile section. This number is automatically assigned by the program as new rows are added. The maximum number of pile sections that may be used is limited to 20.
- **Top:** This number defines the top coordinate of each pile section being defined. Local x coordinate should be used, so the first pile section of each pile type must always start at level zero.
- **Bottom**: This number defines the bottom coordinate (local x) of each section of pile being defined.
- **Cross Section Index:** This column is used to assign a number corresponding to the type of cross section used for the current section. This number must be an integer. Input is limited to a maximum of 1000 types of cross sections. The types of cross sections are defined in the **Pile Cross Sections** submenu under **Data**.

3.3.4 Data - Pile Group Properties

This submenu option is used to input the pile group properties. The input parameters will be based on the model whether it is a 2D or 3D

3.3.4.1 2D Layout

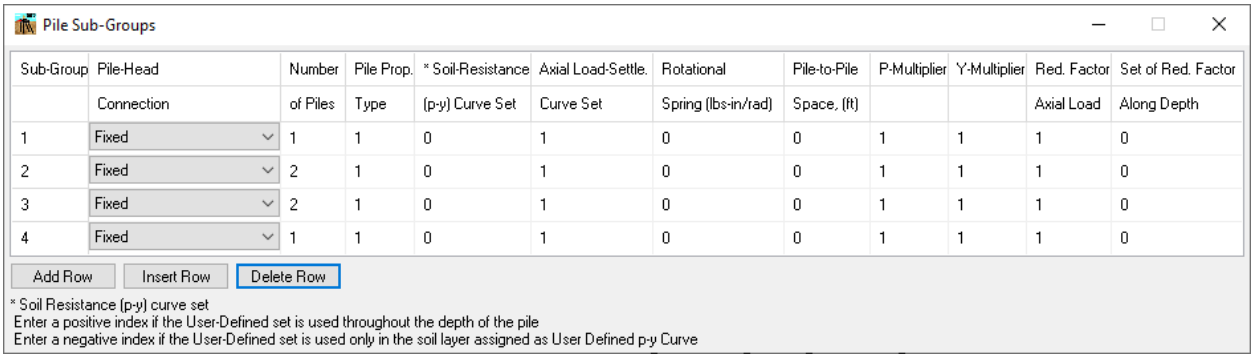


Figure 3.59 Pile Group Properties Window. 2D Model

Depending on the options checked, relevant forms will be available for the user to fill out in this window. Here, the window is shown with all options selected. Note that a minimum of two subgroups of piles is necessary for running analysis; GROUP 2022 will not analyze single piles.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row above the current row.
- **Delete Row.** Delete the current row.

For each pile subgroup (row) the following columns are available:

- **Subgroup.** This is a sequential number that is provided to each pile subgroup. This number is automatically assigned by the program as new pile subgroups are added. The maximum number of pile subgroups that may be used in a group analysis is 2000.
- **Pile-Head Connection.** Clicking on the selection box provides the user with an option for using one of three types of pile-head fixities between the pile and the pile-cap (See Figure 3.60) for all the piles in a particular subgroup (i.e. for each input row):
    - *Pinned*. Provides restraints against translational but not rotational movement.
    - *Fixed*. Provides restraint against all movement.
    - *Elastically-Restrained*. Provides restraint against translational movement but includes a rotational spring. When the user selects this option as a pile-head fixity, the rotational-spring stiffness must be entered in the "Rotational Spring" column. This value must be in units of torque (force times length per unit radian of rotation).
- **Number of Piles.** Used to assign the number of piles that form each pile subgroup. At least one pile must be present in each pile subgroup. The maximum number of piles in each subgroup is limited to 100.
- **Pile Prop. Type.** Used to assign a number corresponding to the type of pile that forms each pile subgroup. The type of pile generally changes with variations in cross-sectional dimensions and/or materials. Only one type of pile may be used in each subgroup. The maximum number of pile types in a group run is limited to 50. The types of pile properties are defined in **Data→Pile Properties**.
- **Soil Resistance (p-y) Curve Set.** In this column, the user types a number that corresponds to the set of curves of lateral-load versus deflection that will be used for the piles in each subgroup row. This number must be an integer. Input is limited to a maximum of 50 sets of soil-resistance curves. The points defining these curves should be specified in the **Data→Sets of Soil-Resistance (p-y) Curves)** menu. This column is available if the user selects **Options→Enter User Defined Soil Resistance (p-y) curves**. Leave the number as 0 if the user wants to use an internally generated p-y curve based on the soil layer properties for the current pile. Enter a negative value if the set is intended to be used only for the layer assigned as a User Defined p-y curve under the Soil Layer Data.

- **Axial Load-Settle. Curve Set.** In this column, the user types a number that corresponds to the set of curves of axial load versus displacement that will be used for the piles of each subgroup row. This number must be an integer and is limited to a maximum of 50 sets of load-displacement curves. The points defining these curves should be specified in **Data→Sets of Axial Load-Displacement Curves**. This column is available if the user selects **Options→Enter User Defined Load-Displacements curves**. Program GROUP 2022 allows internally generated load-displacement curves to be mixed with external (user-input) curves in different pile subgroups. Leave the number as 0 if the user wants to use an internally generated load displacement curve based on the soil layer properties for the current pile.
- **Rotational Spring**. This input depends on the type of restraint selected for the pile-head connections to the pile cap in each subgroup. It will only be taken into account if the user selects an elastically-restrained type of pile-head connection. In all other cases (i.e. fixed and pinned connections), the values on this column are ignored and may be left as the default of zero. The values for spring stiffness must be in units of torque (force times length per unit radian of rotation).
- **Pile-to-Pile Space.** The values in this column should reflect the common center-to-center distance between piles of the same subgroup. The program requires that all piles of the same subgroup are separated by equal distances to their center lines. It is not necessary to specify the spacing of the piles in the same subgroup if users input their own *p-y* modification factors or if the user chooses to ignore group effects.
- **P-Multiplier.** The *p*-multiplier values may be larger or smaller than one. However, in most cases these values are smaller than one to account for the group effect of closely-spaced piles. Chapter 6 in the *Technical Manual*, "Reduction Factor for Soil Resistance in a Closely-Spaced Pile Group," addresses this issue. This column is available if the user selects the **Specify** option in **Options→Reduction Factors for Lateral Loads**.
- **Y-Multiplier.** The *y*-multiplier values may be larger or smaller than one. However, in most cases these values are larger than one to account for the group effect of closely-spaced piles. Chapter 6 in the *Technical Manual*, "Reduction Factor for Soil Resistance in a Closely-Spaced Pile Group," addresses this issue. This column is available if the user selects the **Specify** option in **Options→Reduction Factors for Lateral Loads**.
- **Red. Factor Axial Load.** Under default conditions, group-reduction factors will not be considered for axial loading. This column is available if the user selects **Options→Reduction Factors for Axial Loads**. The user may enter a different reduction factor for each pile subgroup. Information regarding "Reduction Factor for Piles Under Axial Loading in a Closely-Spaced Pile Group" is included in Chapter 7 of the *Technical Manual.*
- **Set of Lateral Red. Factor Along Depth.** Under default conditions, lateral reduction factors along depth will not be considered. This column is available if the user selects **Options→Reduction Factors for Lateral Loads Along Depth**. The user may enter a different set for each pile subgroup.

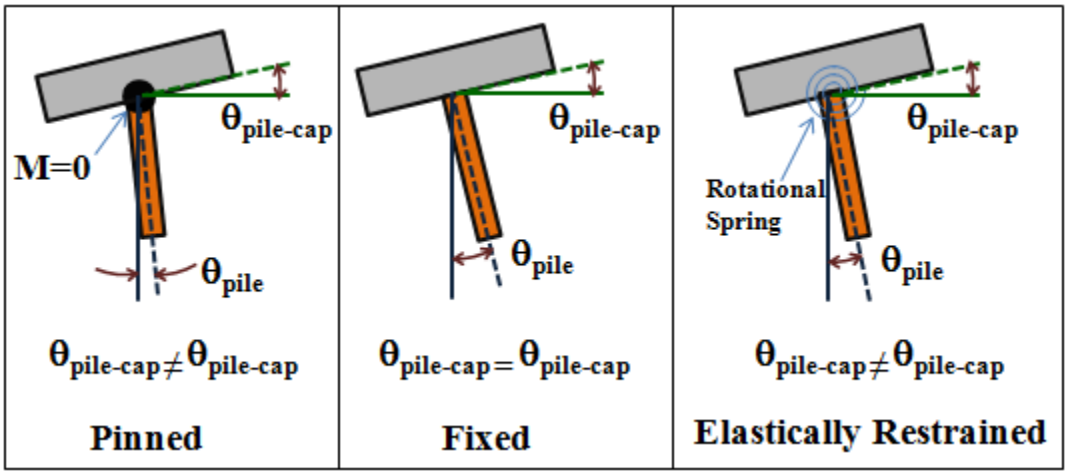


Figure 3.60 Pile Head Connections

3.3.4.2 3D Layout

Pile Group 3D Layout

Pile #	Pile Head Connection (z-z)	Pile Head Connection (y-y)	Pile Prop. Type	* Soil-Resist. (p-y) Curve Set	Axial Load-Settle. Curve Set	Torque-Rotation Curve Set
1	Elast. Restrained	Elast. Restrained	1	0	1	0
2	Elast. Restrained	Elast. Restrained	1	0	1	0
3	Elast. Restrained	Elast. Restrained	1	0	1	0
4	Elast. Restrained	Elast. Restrained	1	0	1	0
5	Elast. Restrained	Elast. Restrained	1	0	1	0
6	Elast. Restrained	Elast. Restrained	1	0	1	0
7	Elast. Restrained	Elast. Restrained	1	0	1	0
8	Elast. Restrained	Elast. Restrained	1	0	1	0

Add Row | Insert Row | Delete Row

* Soil Resistance (p-y) curve set
Enter a positive index if the User-Defined set is used throughout the depth of the pile
Enter a negative index if the User-Defined set is used only in the soil layer assigned as User Defined p-y Curve

Rotational Spring, (z-z) (kN-m/rad)	Rotational Spring, (y-y) (kN-m/rad)	P-Multiplier (Y-dir)	Y-Multiplier (Y-dir)	P-Multiplier (Z-dir)	Y-Multiplier (Z-dir)	Red. Factor Axial Load	Set of Lateral Red. Factor along Depth
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0
400000	0	0	0	0	0	1	0

Figure 3.61 Pile Group Properties Window. 3D Model

The **Pile Group Properties** window is shown above with all options activated and split into two for size reasons. Aside from the inclusion of input parameters for the y and z-axis for the **Pile Head Connection**, **Rotational Spring**, **P-Multiplier**, and **Y-Multiplier**, and the inclusion of the **Torque-Rotation Curve Set**, this window is identical to the one for the 2D layout.

- **Torque-Rotation Curve Set**: In this column, the user types a number that corresponds to the set of curves of torque-rotation that will be used for each individual pile. This number must be an integer and is limited to a maximum of 50 sets of torque-rotation curves. The points defining these curves should be later specified in **Data→Sets of Torque-Rotation Curves**. This column is available if the user selects **Options→Enter User Defined Torque-Rotation**. Program GROUP 2022 allows internally generated torque-rotation curves to be mixed with external (user-input) curves in different pile subgroups. Leave the number as 0 if the user wants to use internally generated torque-rotation curves based on the soil layer properties.

3.3.5 Data - Pile-Head Coordinates

This submenu option is used to input the pile head coordinates. The input parameters will be based on whether the model is 2D or 3D. A review of the theory used by this program is included in Chapter 6 of the *Technical Manual*.

3.3.5.1 2D Layout

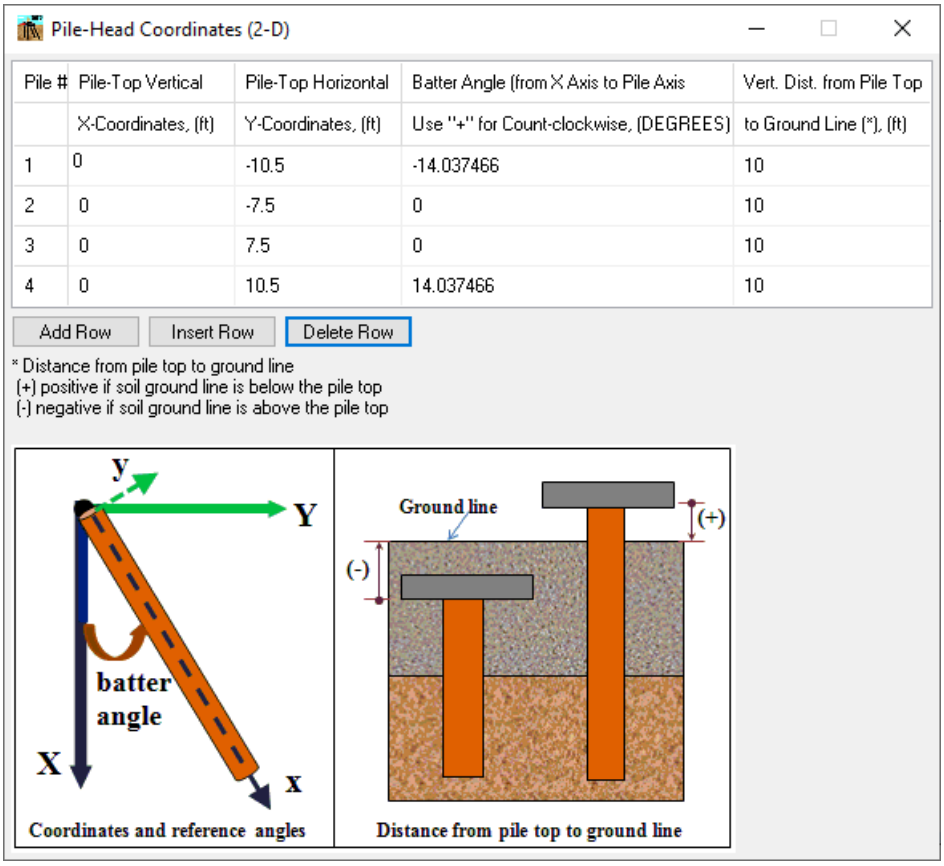


Figure 3.62 Pile Head Coordinates Window. 2D Model

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row above the current row.
  Delete Row. Delete the current row.

For each pile (row) the following columns are available:

- **Pile-Top Vertical X-Coordinate**. Determines the vertical position of a typical pile head for each subgroup. The sign convention is established as positive being downwards from the global origin of coordinates. The user must determine the global origin of coordinates based on the points of load application to the pile group. In most applications, as shown in Figure 3.63, the loads will be applied at the top-middle location of the pile cap. The global origin of coordinates is thus established at that point. Figure 3.63 also describes the local coordinates of pile

subgroups. Please note that if the pile head is embedded in the soil, the pile top vertical X coordinate should be located within the first soil layer. This is necessary in order to properly evaluate the possibility of wedge failure in the top layer.

- **Pile-Top Horizontal Y-Coordinate**. Determines the horizontal position of a typical pile head for each subgroup. The sign convention is established as positive to the right of the global origin of coordinates (X-axis in Figure 3.63). The example shown in Figure 3.63 thus indicates that pile subgroups 1 and 2 would have negative numbers for their **Pile-Top Horizontal Y-Coordinate** values. In the same example, pile subgroups 3 and 4 would have positive values.
- **Batter Angle.** Determines the batter angle of piles in each subgroup. The batter angle is defined as positive if the pile axis is counterclockwise from the downward vertical direction. Thus, the pile tip for a pile with positive batter will be to the right of the pile head. The batter angle must be entered in degrees.

  For reference, some commonly-used batter angles are listed below.

Batter Ratio	Batter Angle (radians)	Batter Angle (degrees)
12:1	0.0831	4.764
10:1	0.0997	5.711
8:1	0.1244	7.125
7:1	0.1419	8.130
6:1	0.1651	9.462
5:1	0.1974	11.310
4:1	0.2450	14.036
3:1	0.3218	18.435

Table 3.3 Commonly used batter angles

- **Vertical Distance from Pile Top to Ground Line**. This number defines the vertical distance (in the global X direction) to the ground line from the top of the typical pile of each subgroup. The sign conventions are established as positive downward. A negative value indicates that the pile head is below ground level. A value of zero indicates that the top of the pile is aligned with the ground surface. All pile subgroups of the example shown in Figure 3.63 would thus need positive entries in this column.
  This distance is also used to simulate a group of piles on sloped ground. An illustration for this case is shown in Figure 3.64. GROUP allows this option only if the distance from pile top to ground surface for all the piles are located within the top soil layer.

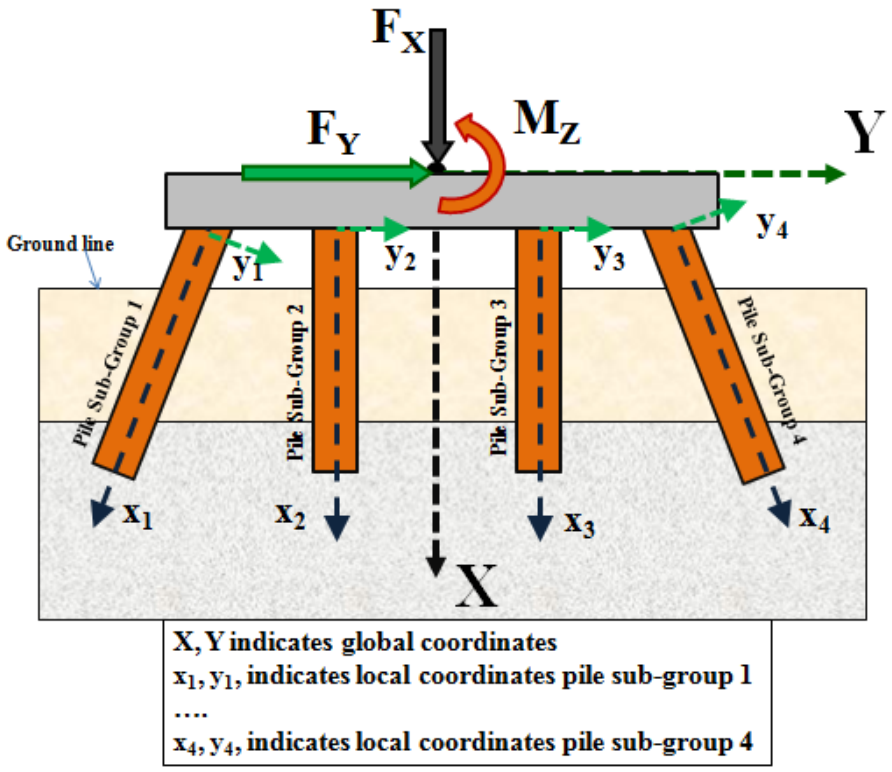


Figure 3.63 Example of a 2D model

For the example in Figure 3.63, the following signs would apply for the general location of their pile heads and batter angle:

Pile Subgroup	Vertical Coordinate	Horizontal Coordinate	Batter Angle	Distance from Pile Top to Ground Line
1	(+)	(-)	(-)	(+)
2	(+)	(-)	0	(+)
3	(+)	(+)	0	(+)
4	(+)	(+)	(+)	(+)

Table 3.4 Signs for coordinates and batter angles for an example of a 2D model

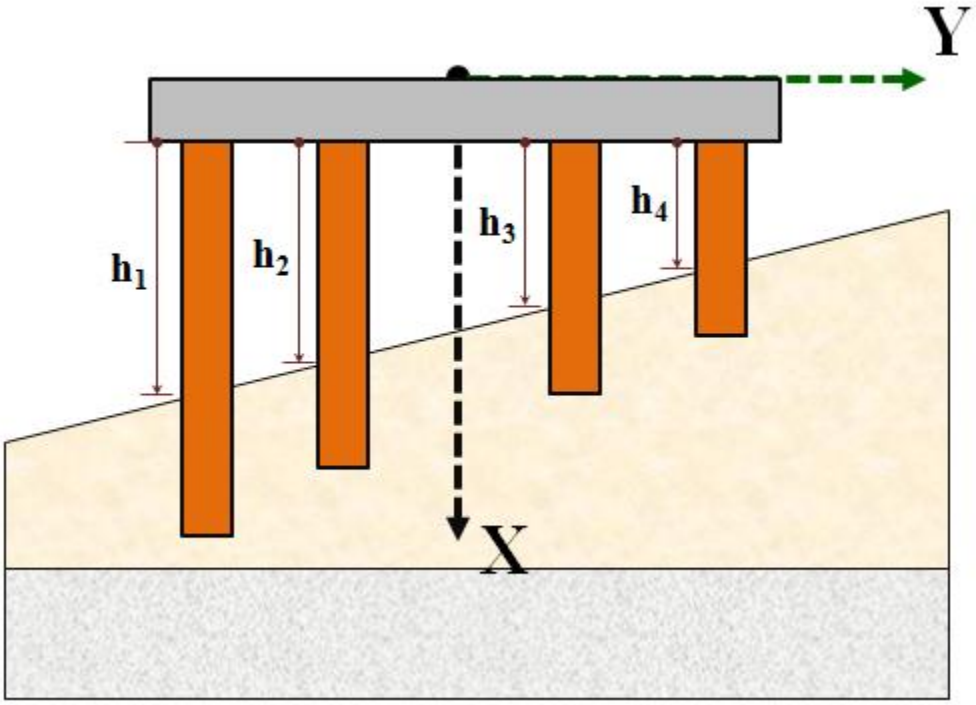
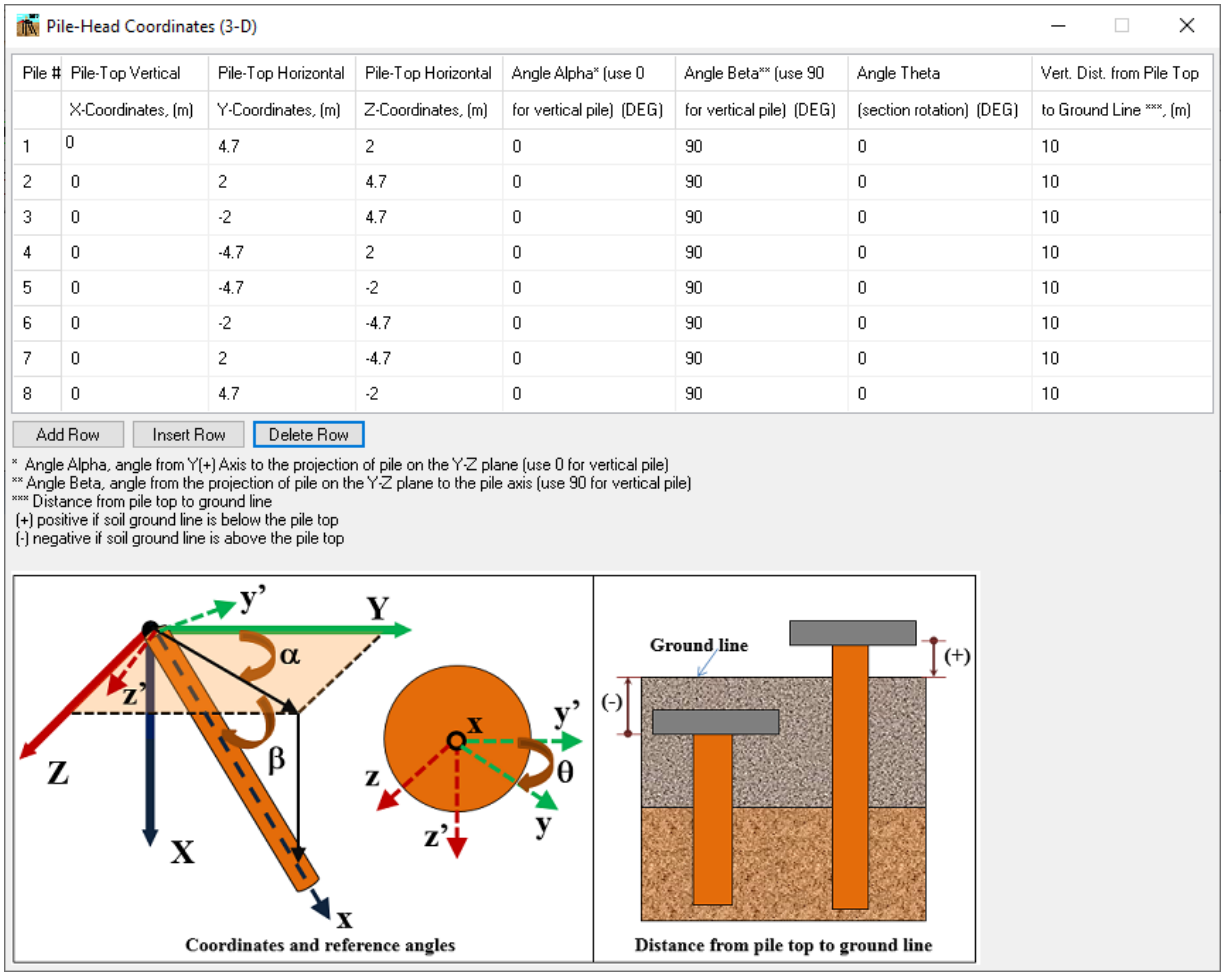


Figure 3.64 Distance from pile top to ground line to simulate slopping ground

3.3.5.2 3D Layout



Pile #	Pile-Top Vertical X-Coordinates, (m)	Pile-Top Horizontal Y-Coordinates, (m)	Pile-Top Horizontal Z-Coordinates, (m)	Angle Alpha* (use 0 for vertical pile) (DEG)	Angle Beta** (use 90 for vertical pile) (DEG)	Angle Theta (section rotation) (DEG)	Vert. Dist. from Pile Top to Ground Line ***, (m)
1	0	4.7	2	0	90	0	10
2	0	2	4.7	0	90	0	10
3	0	-2	4.7	0	90	0	10
4	0	-4.7	2	0	90	0	10
5	0	-4.7	-2	0	90	0	10
6	0	-2	-4.7	0	90	0	10
7	0	2	-4.7	0	90	0	10
8	0	4.7	-2	0	90	0	10

Figure 3.65 Pile Head Coordinates Window. 3D Model

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row above the current row.
- **Delete Row.** Delete the current row.

For each pile (row) the following columns are available:

- **Pile-Top Vertical X- Coordinate**. Determines the vertical position of a typical pile head for each pile. The sign convention is established as positive downwards from the global origin of coordinates. The user must determine the global origin of coordinates.

- **Pile-Top Horizontal Y-Coordinate.** Determines the horizontal position along Y direction of a typical pile head for each pile.
- **Pile-Top Horizontal Z-Coordinate.** Determines the horizontal position along the Z-direction of a typical pile-head for each pile.
- **Angle α**. Determines the angle between the Y-axis and the projection of each pile on the Y-Z plane. The angle must be entered in degrees. Enter zero for vertical piles.
- **Angle β**. Determines the batter angle between the projection of piles on the Y-Z plane and the pile axis of each pile. The batter angle must be entered in degrees. Enter 90 for vertical piles.
- **Angle θ**. Define the rotation angle of the section respect to the local x axis. Enter 0 if no rotation is needed.
- **Vertical Distance from Pile Top to Ground Line**. Defines the vertical distance (in the global X direction) to the ground line from the top of the typical pile of each subgroup. The sign conventions are established as positive downwards. A negative value indicates that the pile head is below ground level. A value of zero indicates that the top of the pile is aligned with the ground surface.

Note on local axes

The local axes for each pile are defined as follow:

- Local x-axis is along the pile. Positive direction is defined from the top of the pile to the pile tip.
- Local y and z-axis are perpendicular to the pile axis.

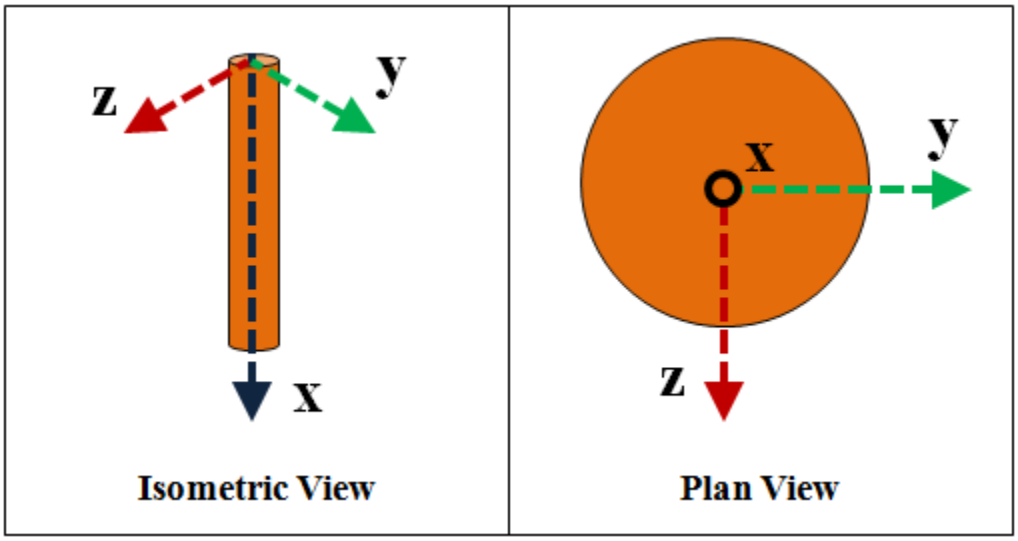


Figure 3.66 Local Axes Definition

The orientation of the local axes depends on the batter angle (2D model) or the α and β angles for 3D modes. For 2D models the local z-axis is always in the direction of the Global Z-axis. So, the local

x and y-axes will be always in the plane XY. Two examples showing the orientation of the local axes (x, y) in terms of the global axes (X, Y) are presented in Figure 3.67. For 2D models, general equations to estimate the orientation of the local axes (x, y, z) in terms of the global axes (X, Y, Z) are presented in Figure 3.68.

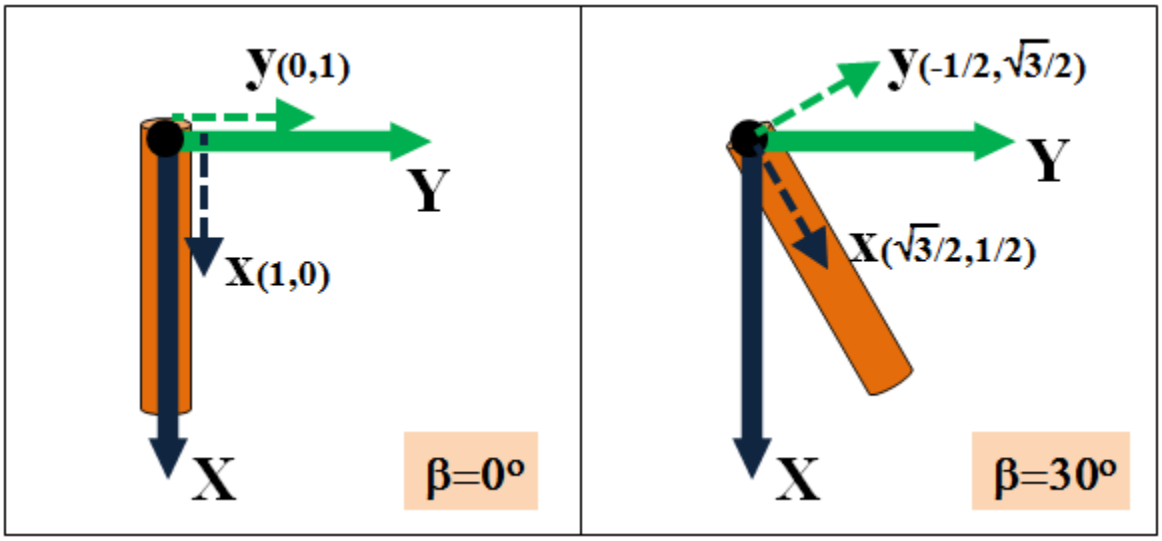


Figure 3.67 Local Axes for 2D models

$$\{x\} = \{\cos(\beta) \quad \sin(\beta) \quad 0\}^T$$

$$\{y\} = \{-\sin(\beta) \quad \cos(\beta) \quad 0\}^T$$

$$\{z\} = \{0 \quad 0 \quad 1\}^T$$

Figure 3.68 Equations to estimate orientation for local axes. 2D Model

Examples showing the orientation of the local axes (x, y, z) in terms of the global axes (X, Y, Z) are presented from Figure 3.69 to Figure 3.73 (section rotation angle is defined as zero). For 3D models, general equations to estimate the orientation of the local axes (x, y, z) in terms of the global axes (X, Y, Z) are presented in Figure 3.74

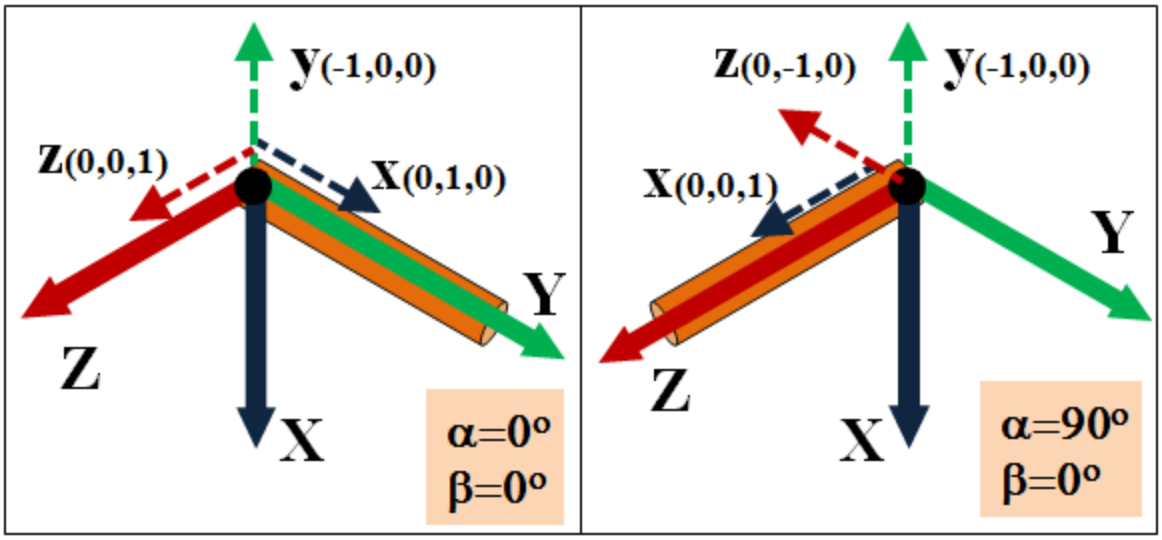


Figure 3.69 Local Axes for Horizontal Piles. 3D model

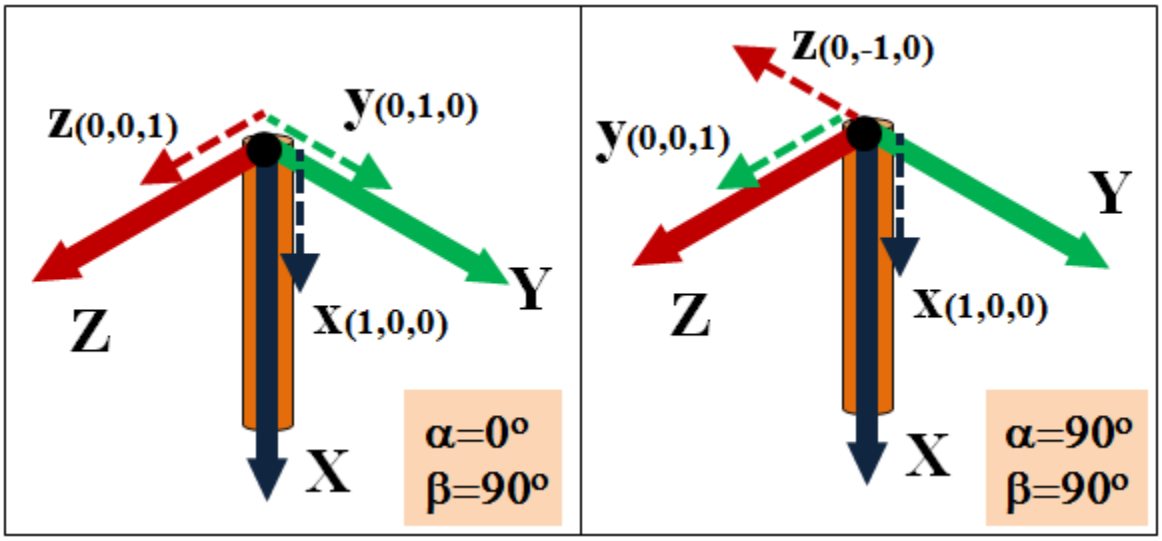


Figure 3.70 Local Axes for Vertical Piles. 3D model

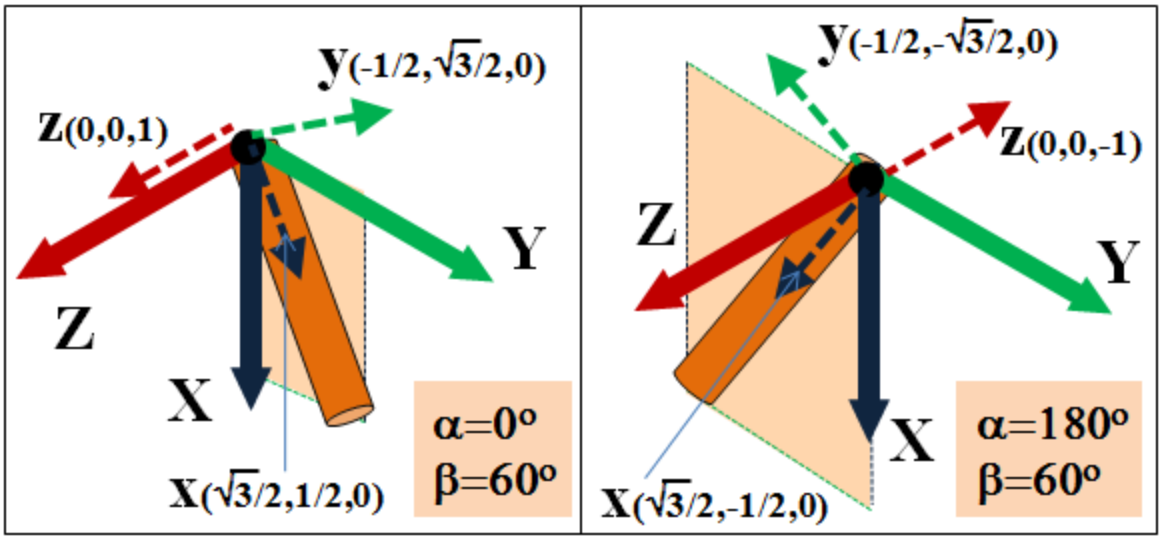


Figure 3.71 Local Axes for Batter Piles in the XY plane. 3D model

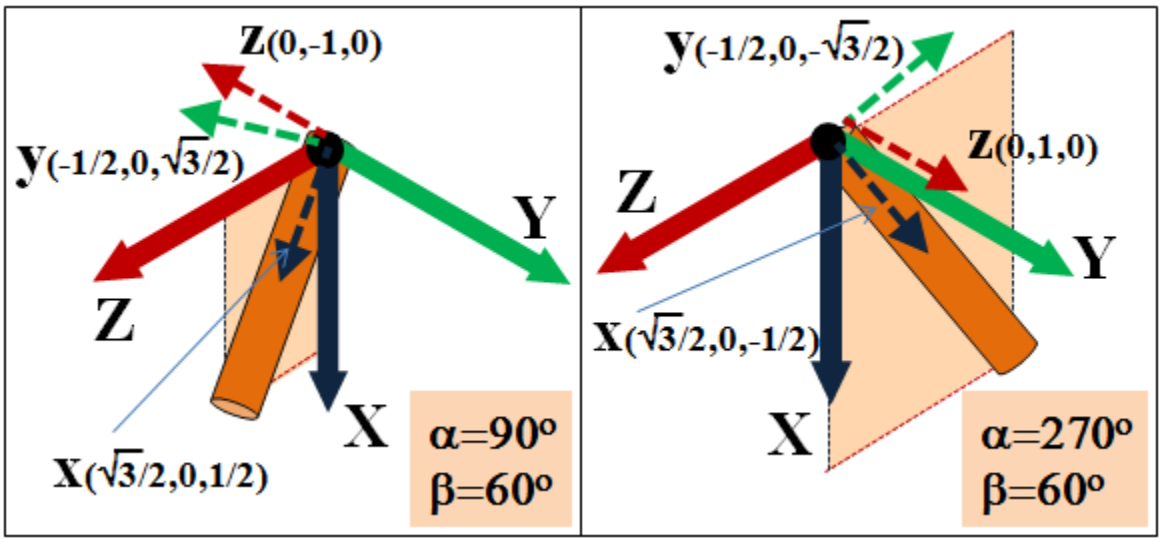


Figure 3.72 Local Axes for Batter Piles in the XZ plane. 3D model

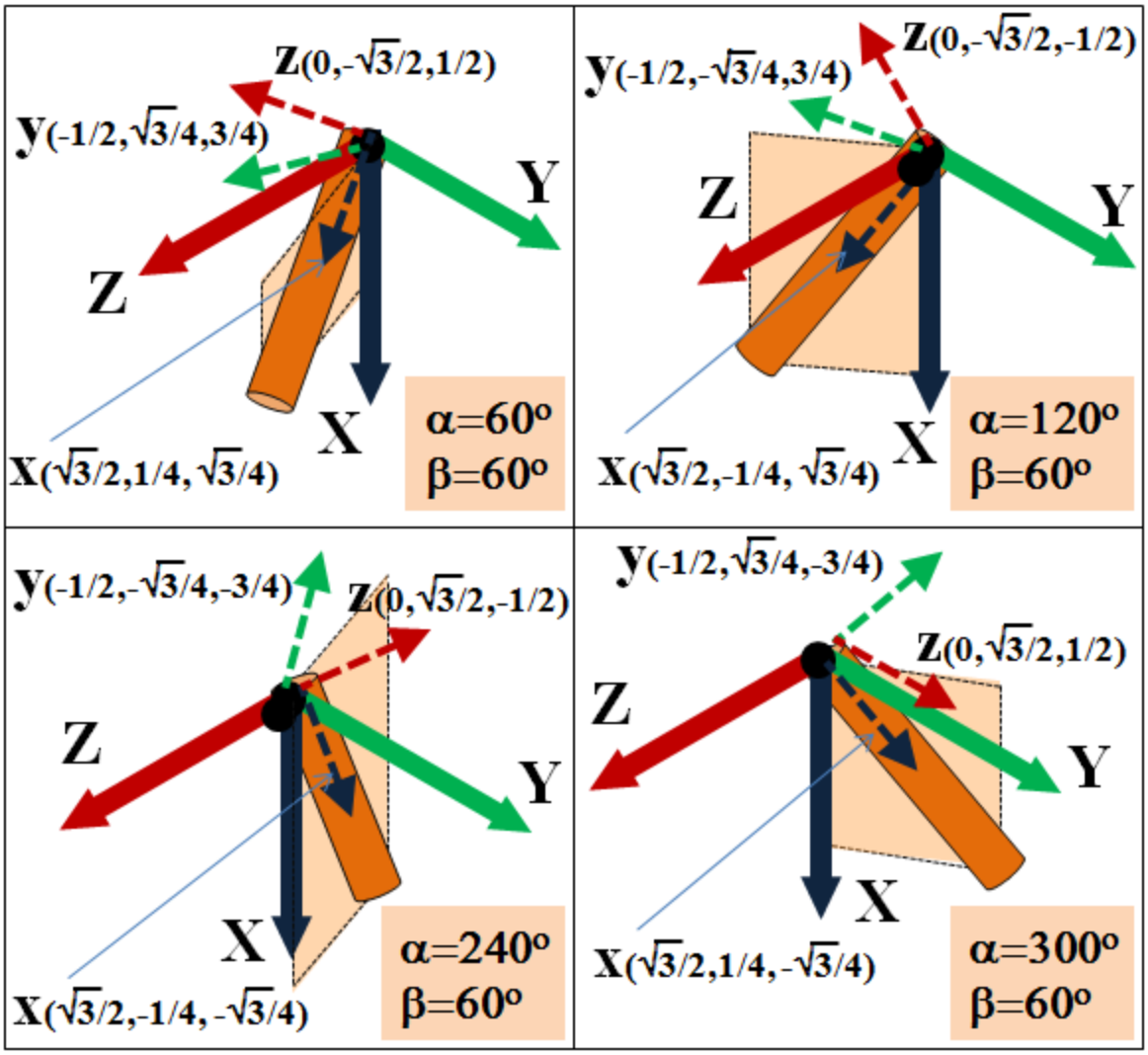


Figure 3.73 Local Axes for Batter Piles. General Orientation. 3D model

$$\{x\} = \{\sin(\beta) \quad \cos(\beta)\cos(\alpha) \quad \cos(\beta)\sin(\alpha)\}^T$$

$$\{y\} = \{-\cos(\beta) \quad \sin(\beta)\cos(\alpha) \quad \sin(\beta)\sin(\alpha)\}^T$$

$$\{z\} = \{0 \quad -\sin(\alpha) \quad \cos(\alpha)\}^T$$

Figure 3.74 Equations to estimate orientation for local axes, no section rotation. 3D Model

$$\{x\} = \{\sin(\beta) \quad \cos(\beta)\cos(\alpha) \quad \cos(\beta)\sin(\alpha)\}^T$$

$$\{y\} = \{-\cos(\beta)\cos(\theta) \quad \sin(\beta)\cos(\alpha)\cos(\theta) - \sin(\alpha)\sin(\theta) \quad \sin(\beta)\sin(\alpha)\cos(\theta) + \cos(\alpha)\sin(\theta)\}^T$$

$$\{z\} = \{\cos(\beta)\sin(\theta) \quad -\sin(\beta)\cos(\alpha)\sin(\theta) - \sin(\alpha)\cos(\theta) \quad -\sin(\beta)\sin(\alpha)\sin(\theta) + \cos(\alpha)\cos(\theta)\}^T$$

Figure 3.75 Equations to estimate orientation for local axes, including section rotation. 3D Model

3.3.6 Data - Pile-Cap Options

This submenu option is used to input the pile-cap dimensions. It is also used to check the option for embedment of the pile-cap.

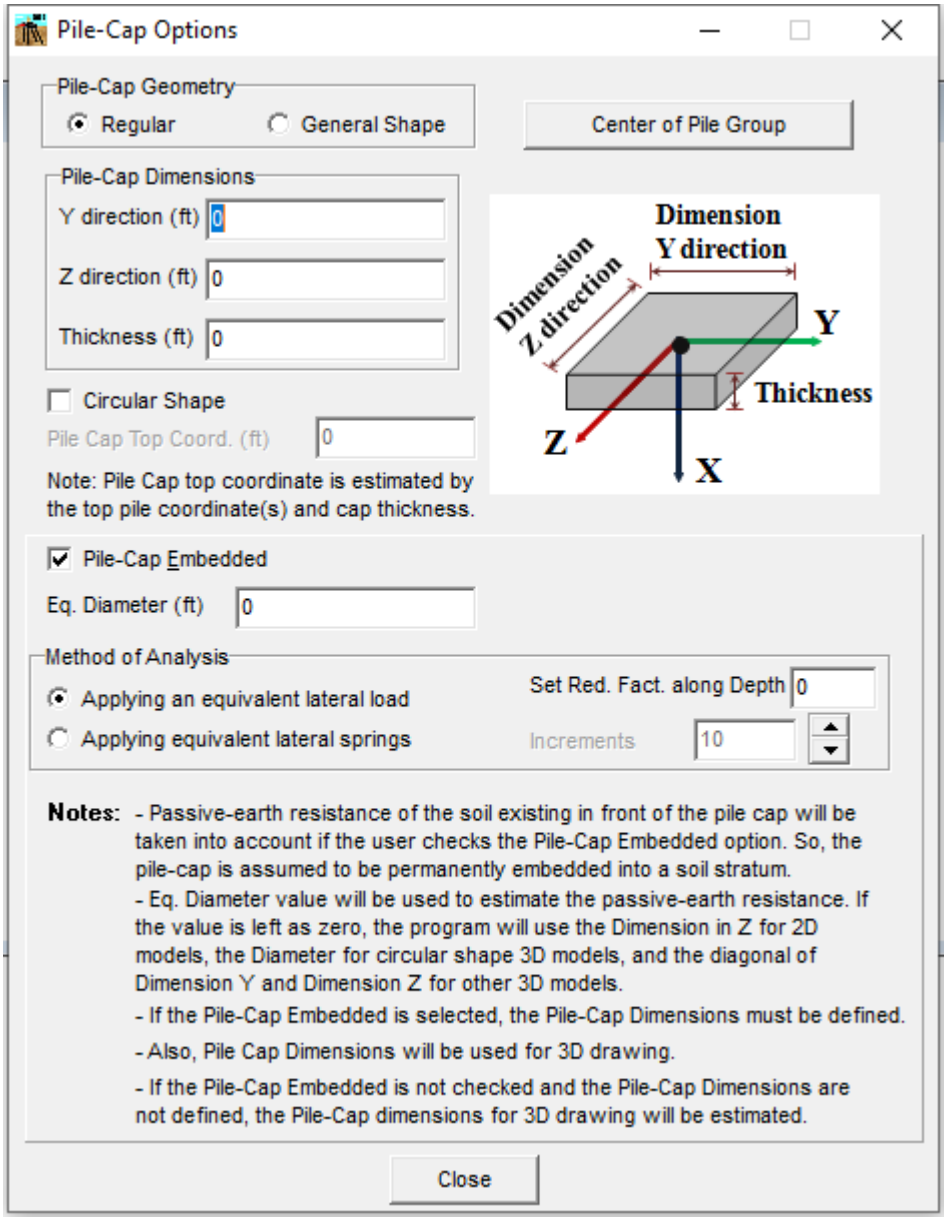


Figure 3.76 Pile-Cap Options Window

Check on the **Regular** option to draw the pile cap as a rectangular or circular shape. If this entry is selected, the user may enter the values for dimensions in the Y direction, Z direction and thickness for a rectangular shape, or the diameter and thickness for a circular shape (Only 3D models).

Check on the **General Shape** option to draw the pile cap as a general shape. The program will draw the pile cap as a combination of concave regions. If this entry the selected the program will show the values in the Y direction, Z direction and thickness based on the coordinates entered.

Check on the **Circular Shape** option if the user wants to draw the pile cap as circular.

Check on the **Pile-Cap Embedded** option only if the user intends to take into account the passive-earth resistance of the soil existing in front of the pile cap. If the pile cap is not embedded into a permanent-soil stratum, or if the user does not want to include the passive-earth pressure because of erosion or other consideration, this option should not be checked.

When using embedded pile caps, GROUP automatically modifies the internally generated *p-y* curves according to the soil overburden pressure. The GROUP program cannot use a user-input *p-y* curve to account for the passive resistance against embedded pile caps. Note that the additional axial loading on top of the pile cap due to the weight of soil above is not computed by the GROUP program. If necessary, users must complete and add this load to their axial compression loads on top of the pile cap.

For embedded pile caps, in 3-D analyses the "length" dimension of an embedded pile cap corresponds to the Y-direction. The "width" dimension thus corresponds to the Z-direction. The user may enter an equivalent diameter for the pile cap under the Eq. Diameter field. If this value is left as zero, the program will use the dimension in Z direction for 2D models, the diagonal of dimensions in Y and Z directions for 3D rectangular models and the diameter for 3D circular models.

If the option **Reduction Factor for Lateral Loads Alond Depth** (under the Options Menu Item) is checked, the user may enter the set under under **Set. Red. Fact. Along Depth** that can be used also for soil resistance in the embedded cap..

Method of Analysis

A description of the method of computation for the passive-earth pressure is presented in Chapter 8 of the *Technical Manual*.

The program can consider the passive pressure either as a concentrated lateral force (equivalent lateral load) or using equivalent lateral spring along the thickness of the cap. Click on **Applying an equivalent lateral load** to use a concentrated lateral force or on **Applying equivalent lateral springs** to use lateral springs along the thickness. For the second option, the user should enter a number on increments to apply the lateral springs, the default value is 10, being the maximum 20. The second option may be used to improve the convergence, particularly in cases of small lateral deformations.

The program will use the dimensions for the tridimensional representation of the model. If the user does not include these values, the program will assume those dimensions based on the pile group configuration

General Shape

After selecting General Shape and Clicking on Geometry Coordinates, the program will show the Cap Surface Window. This window is used to define the number of concave surfaces needed to define the Pile Cap Geometry. Then click under Pile Cap Coordinates to open the Pile Cap Coordinates Window. In the Pile Cap Coordinates Window, the specific surface is defined. For each node, enter the Y and Z coordinates and the vertical coordinates (X) for the top and bottom of the pile cap. The pile cap shown in Figure 3.77 was drawn by defining 5 cap surfaces (presented in Figure 3.78).

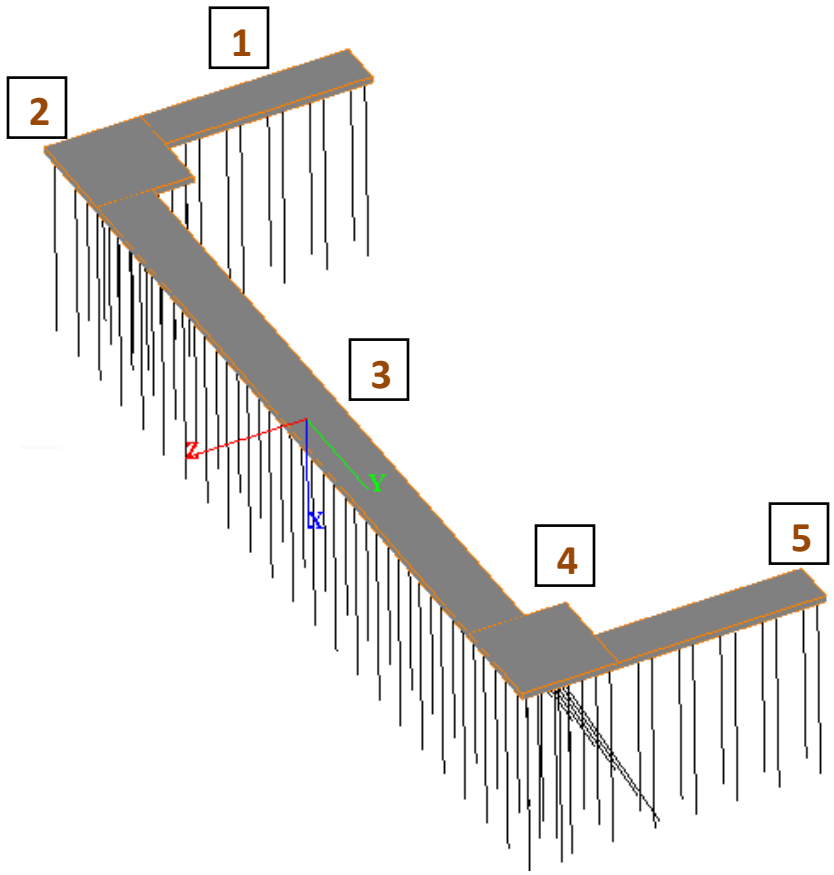


Figure 3.77 Pile-Cap Options. General Shape

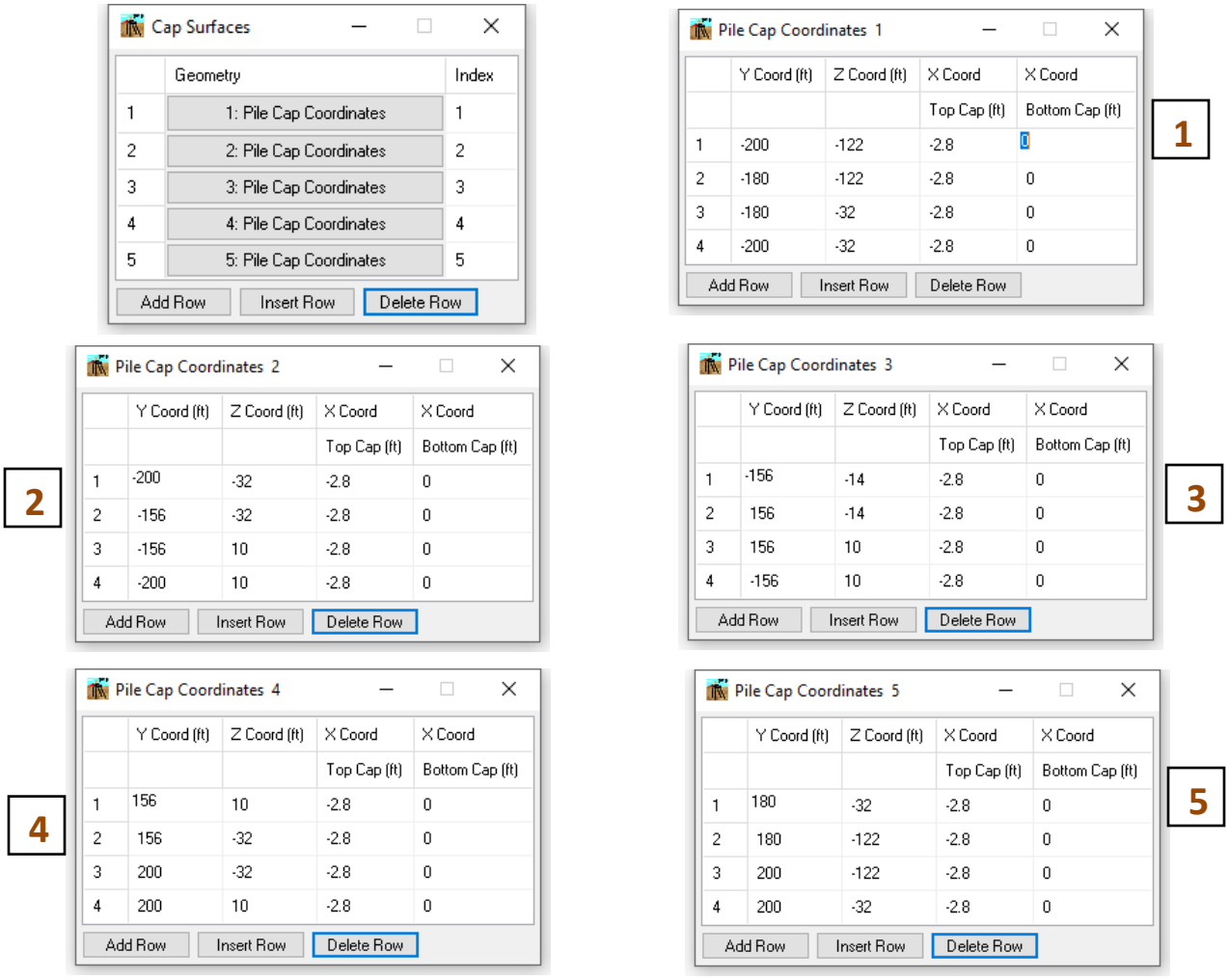


Figure 3.78 Pile-Cap Options. General Shape

3.3.7 Data – Node Data (Pile-Cap)– Flexible Cap Version

This submenu option is used to enter and/or edit the Node Data that belongs to the flexible cap.

3.3.7.1 Data – Node Data (Pile-Cap) - Node Coordinates – Flexible Cap Version

This submenu option is used to open the Node Coordinates Window that allows to input and or edit the node coordinates of the finite element mesh that defines the flexible cap.

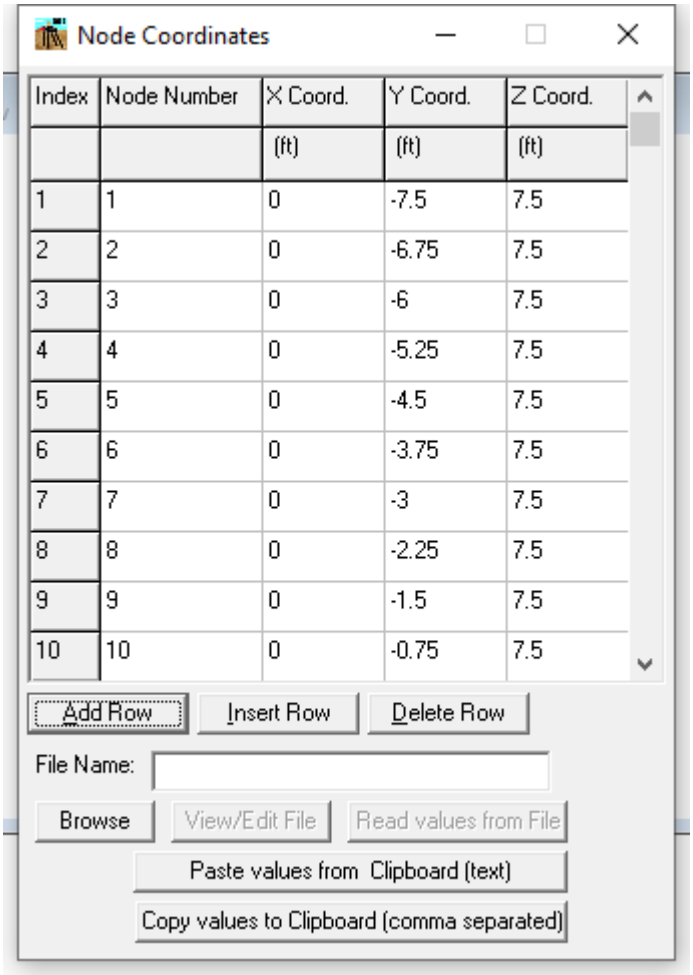


Figure 3.79 Node Coordinates Window – Flexible Cap Version

For each row the following columns are available:

- **Node Number**. Enter the node number assigned to the defined node.
- **X Coord.** Enter the value of the coordinate X for the defined node.
- **Y Coord.** Enter the value of the coordinate Y for the defined node.
- **Z Coord.** Enter the value of the coordinate Y for the defined node.

The maximum number of rows defining nodes is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (4), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the clipboard is not equal to the columns needed (4), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

3.3.8 Data – Plate Data (Pile-Cap) – Flexible Cap Version

This submenu option is used to enter and/or edit the Plate Data that belomgs to the flexible cap.

3.3.8.1 Data – Plate Data (Pile-Cap) – Plate Connectivity – Flexible Cap Version

This submenu option is used to open the Plate Connectivity Window that allows to input and or edit the material index property and the plate connectivity of the finite element mesh that defines the flexible cap.

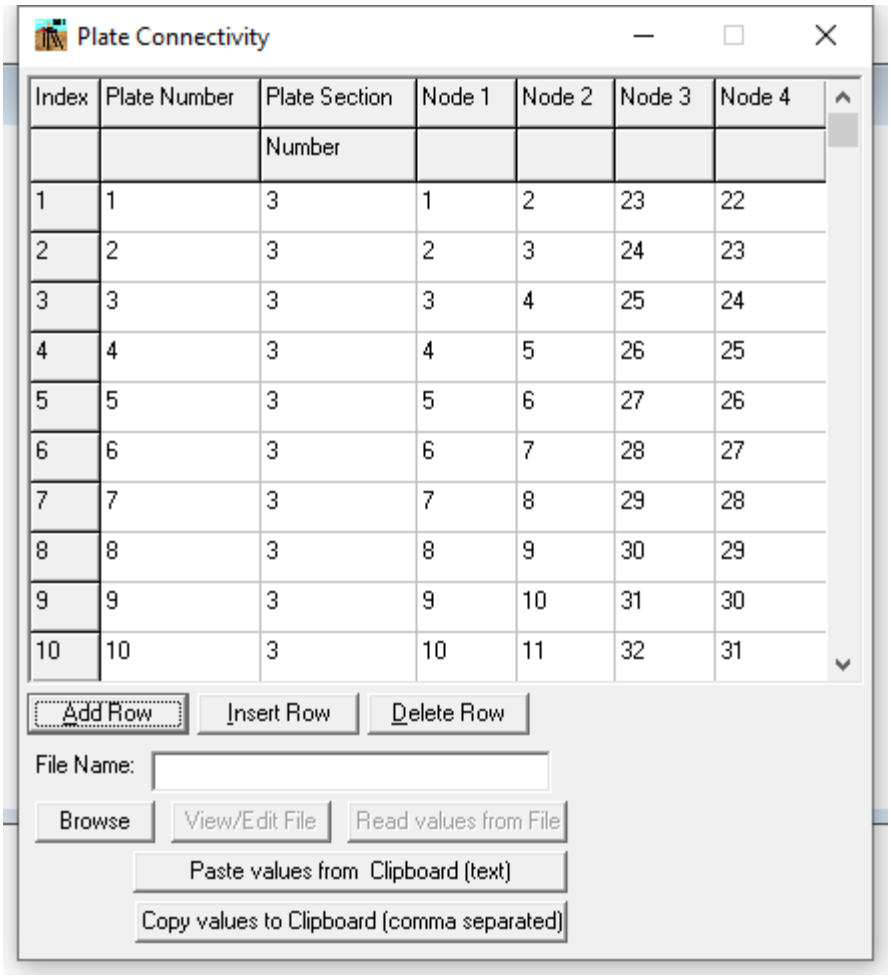


Figure 3.80 Plate Connectivity Window – Flexible Cap Version

For each row the following columns are available:

- **Plate Number**. Enter the plate number assigned to the defined plate.
- **Plate Section Number**. Enter the section number of the plate properties to be used for the defined plate. The section properties are defined under Data->Plate Data->Plate Sections.
- **Node 1.** Enter the node number for Node 1 for the defined plate.
- **Node 2.** Enter the node number for Node 2 for the defined plate.
- **Node 3.** Enter the node number for Node 3 for the defined plate.
- **Node 4.** Enter the node number for Node 4 for the defined plate.

The maximum number of rows defining plates is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (6), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (6), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

The program uses a 4-node element for plates, which is displayed in the figure below.

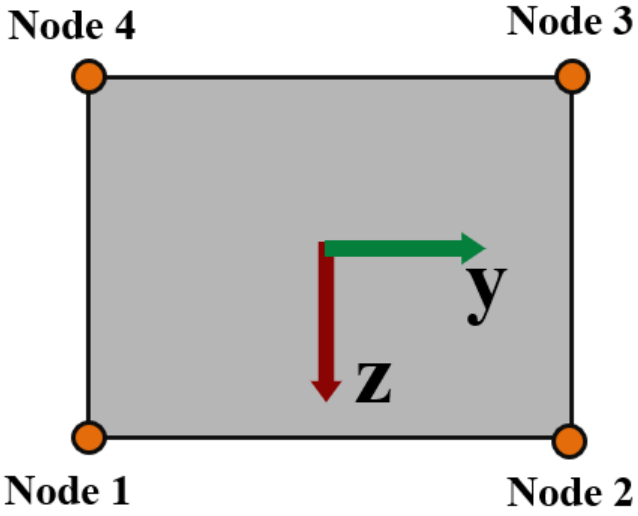


Figure 3.81 Plate Connectivity Numbering

3.3.8.2 Data – Plate Data (Pile-Cap) – Plate Offsets – Flexible Cap Version

This submenu option is used to open the Plate Offsets Window that allows to input and or edit the element offsets in the X, Y and Z directions. Please note that the program will add a rigid element at each node in order to model the offsets.

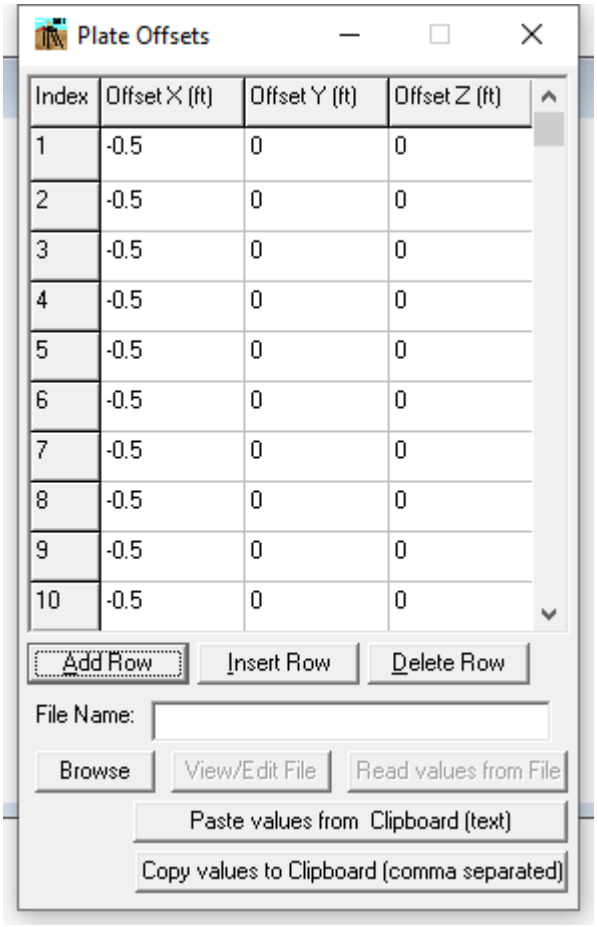


Figure 3.82 Plate Offsets Window – Flexible Cap Version

For each row the following columns are available:

- **Offset X.** Enter the value for offset of the plate in the X direction.
- **Offset Y.** Enter the value for offset of the plate in the Y direction.
- **Offset Z.** Enter the value for offset of the plate in the Y direction.

Similarly to plate connectivity, the maximum number of rows defining plate offsets is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (3), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (3), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

Offsets is an option for the user to define the axis of the element is not in the same position of the node. An example of the use of offsets is presented in the next figure.

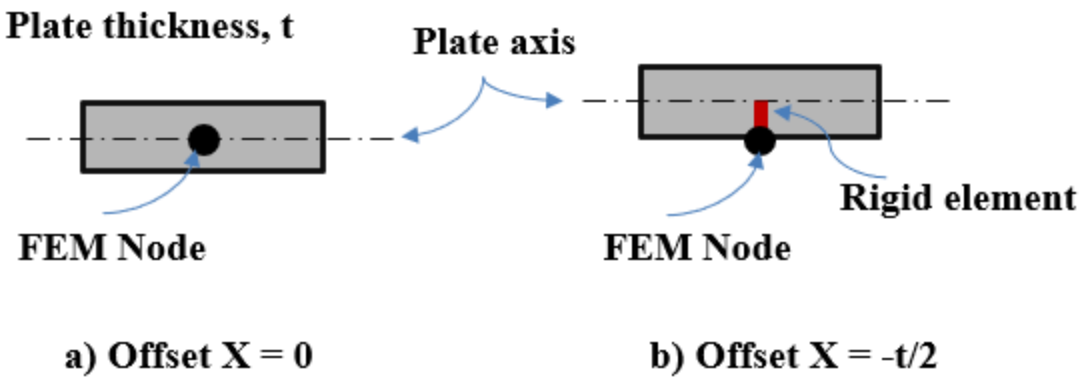


Figure 3.83 Example Plate Offsets – Flexible Cap Version

3.3.8.3 Data – Plate Data (Pile-Cap) – Plate Sections – Flexible Cap Version

This submenu option is used to open the Plate Section Properties Window that allows to input and or edit the material and geometric properties of the plates that defines the flexible cap.

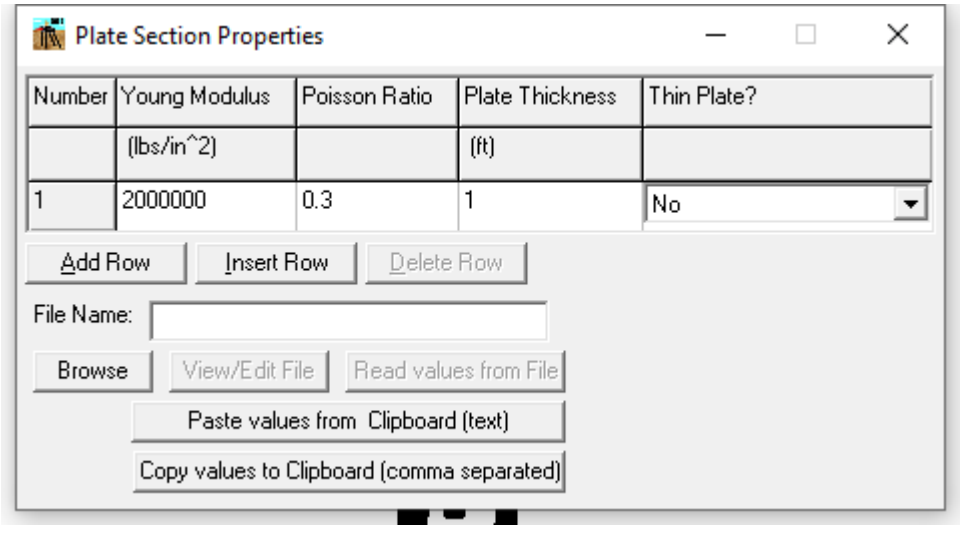


Figure 3.84 Plate Section Properties Window – Flexible Cap Version

For each row the following columns are available:

- **Young Modulus.** Enter the value for the modulus of elasticity of the plate material.
- **Poisson Ratio.** Enter the value for the poisson ratio of the plate material.
- **Plate Thickness.** Enter the value for plate thickness.
- **Thin Plate?.** Select whether is a thick or thin plate. For a thin plate, shear deformation of the plate will not be taken into account.

The maximum number of rows defining plate sections is limited to 20.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (4), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (4), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

3.3.9 Data – Beam Data (Pile-Cap) – Flexible Cap Version

This submenu option is used to enter and/or edit the Plate Data that belomgs to the flexible cap.

3.3.9.1 Data – Beam Data (Pile-Cap) – Beam Connectivity – Flexible Cap Version

This submenu option is used to open the Beam Connectivity Window that allows to input and or edit the beam section number and the beam connectivity of the beam elements that belongs to the flexible cap.

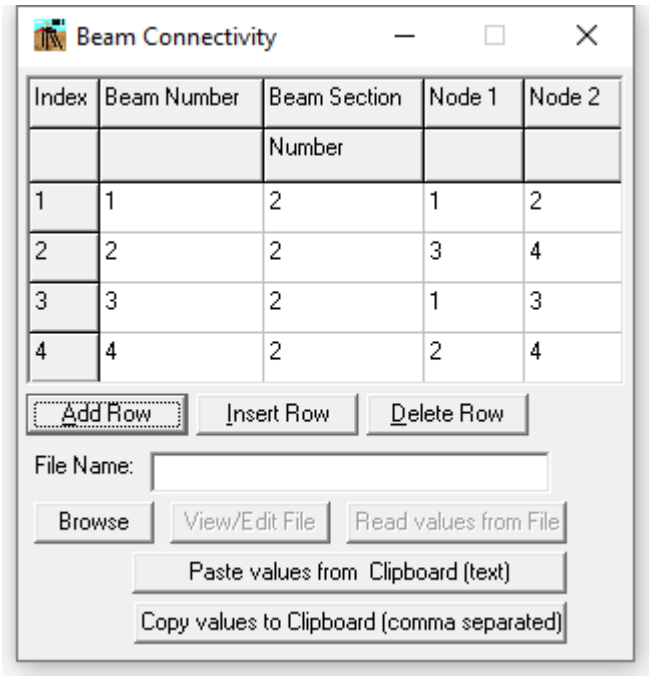


Figure 3.85 Beam Connectivity Window – Flexible Cap Version

For each row the following columns are available:

- **Beam Number**. Enter the beam number assigned to the defined plate.
- **Beam Section Number**. Enter the section number of the cross section properties to be used for the defined beam. The section properties are defined under Data->Pile/Beam Cross Sections. The beam section number must be elastic.
- **Node 1.** Enter the node number for Node 1 for the defined beam.
- **Node 2.** Enter the node number for Node 2 for the defined beam.

The maximum number of rows defining plates is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (4), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (4), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

The program uses the 2-node beam element which is displayed in the figure below.

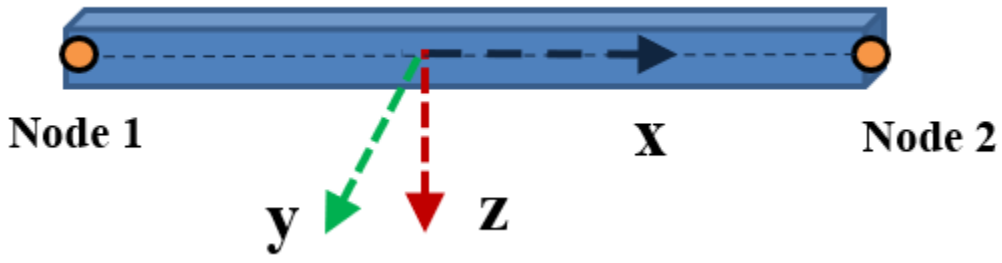


Figure 3.86 Beam Connectivity Numbering

3.3.9.2 Data – Beam Data (Pile-Cap) – Beam Offsets – Flexible Cap Version

This submenu option is used to open the Beam Offsets Window that allows to input and or edit the beam offsets in the X, Y and Z directions. Please note that the program will add a rigid element at each node in order to model the offsets.

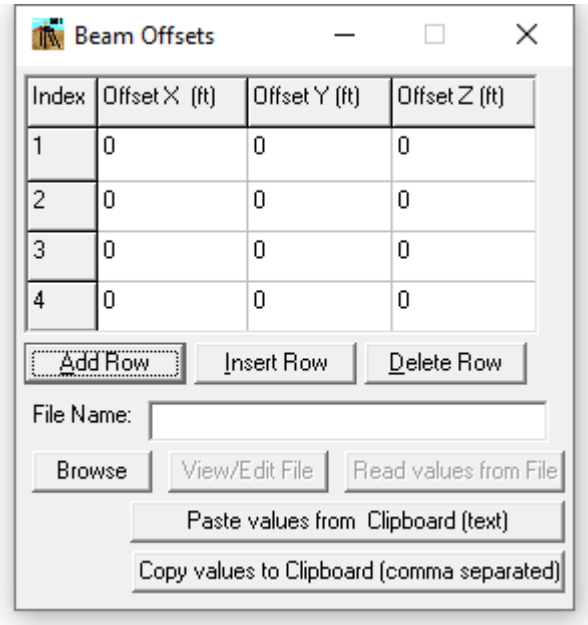


Figure 3.87 Beam Offsets Window – Flexible Cap Version

For each row the following columns are available:

- **Offset X.** Enter the value for offset of the beam in the X direction.
- **Offset Y.** Enter the value for offset of the beam in the Y direction.
- **Offset Z.** Enter the value for offset of the beam in the Y direction.

Similarly to beam connectivity, the maximum number of rows defining beam offsets is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (3), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (3), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

Offsets is an option for the user to define the axis of the element is not in the same position of the node. An example of the use of offsets is presented in the next figure.

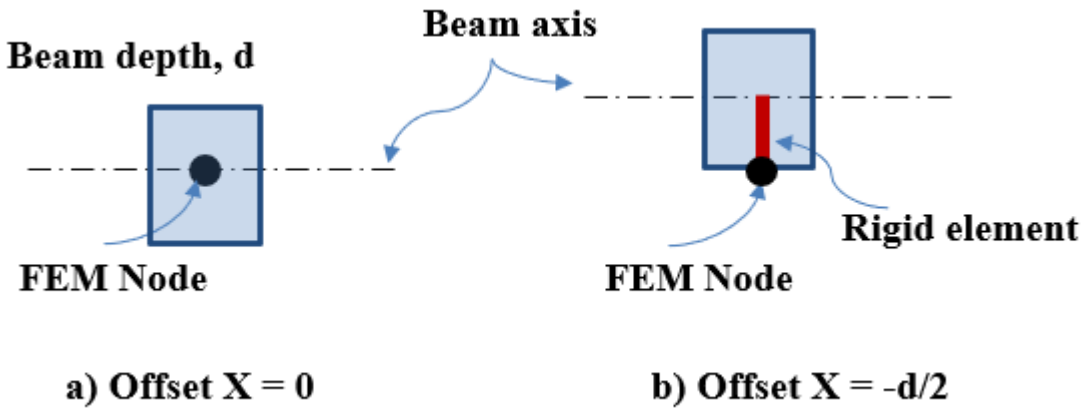


Figure 3.88 Example Beam Offsets – Flexible Cap Version

3.3.9.3 Data – Beam Data (Pile-Cap) – Beam End Zones – Flexible Cap Version

This submenu option is used to open the Beam Offsets Window that allows to input and or edit the beam End Zones at Node1 and Node 2 . Please note that the program will add a flexural rigid element to model the end zones.

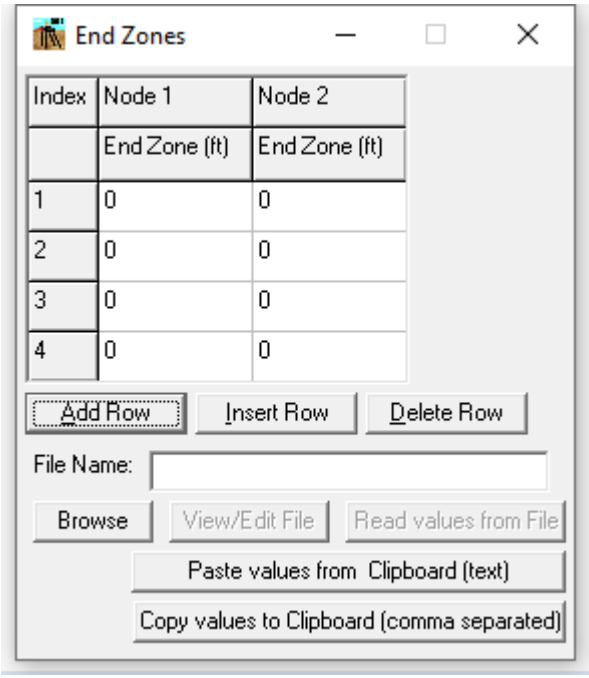


Figure 3.89 Beam End Zones Window – Flexible Cap Version

For each row the following columns are available:

- **Node 1 End Zone.** Enter the value for end zone of the beam at node 1.
- **Node 2 End Zone.** Enter the value for end zone of the beam at node 2.

Similarly to beam connectivity, the maximum number of rows defining beam end zones is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.

- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (2), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (2), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

An example of the behavior of beam end zones is presented in the next figure.

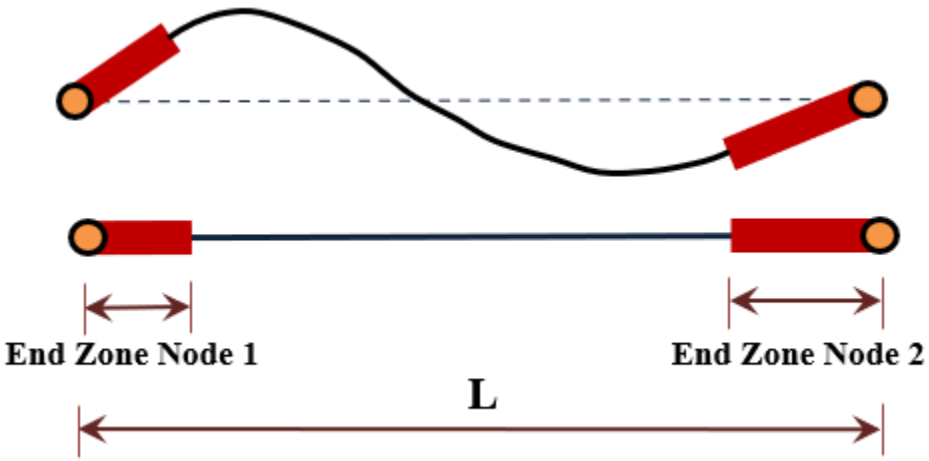


Figure 3.90 Beam End Zones – Flexible Cap Version

3.3.9.4 Data – Beam Data (Pile-Cap) – Beam Section Rotation – Flexible Cap Version

This submenu option is used to open the Beam Section Rotation Window that allows to input and or edit the beam section rotation .

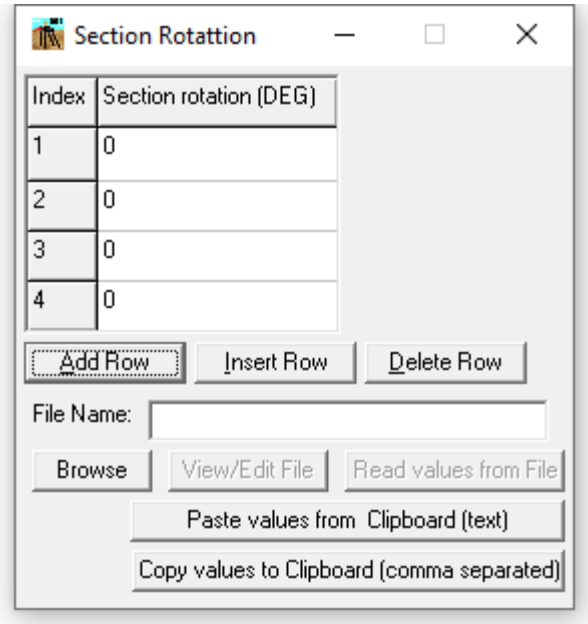


Figure 3.91 Plate Offsets Window – Flexible Cap Version

For each row the following columns are available:

- **Section Rotation.** Enter the value for section rottaion of the beam.

Similarly to beam connectivity, the maximum number of rows defining beam section rotation is limited to 10000.

The following options are available:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

Enter the path of the file name under **File Name:** entry, if user wants to paste the values from a text file. Click on the **Browse** button to browse for the file containing the data. After selecting the file the program will filled the **File Name** entry with the selected file. Click on the **View/Edit** button to view the text file using the default word processor, usually Notepad. Click on the **Read values from File** button to paste the values from the selected file. By using this option, the program will replace all the values already defined. If the number of columns (per row) in the selected file is not equal to the columns needed (1), the program will not paste any values

By clicking on the **Paste values from Clipboard (text)** button, the program will paste the values from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard (per row) is not equal to the columns needed (1), the program will not paste any values.

By clicking on the **Copy values to Clipboard (comma separated)** button, the program will copy the specified data to the clipboard.

An example of the use of section rotation is presented in the next figure.

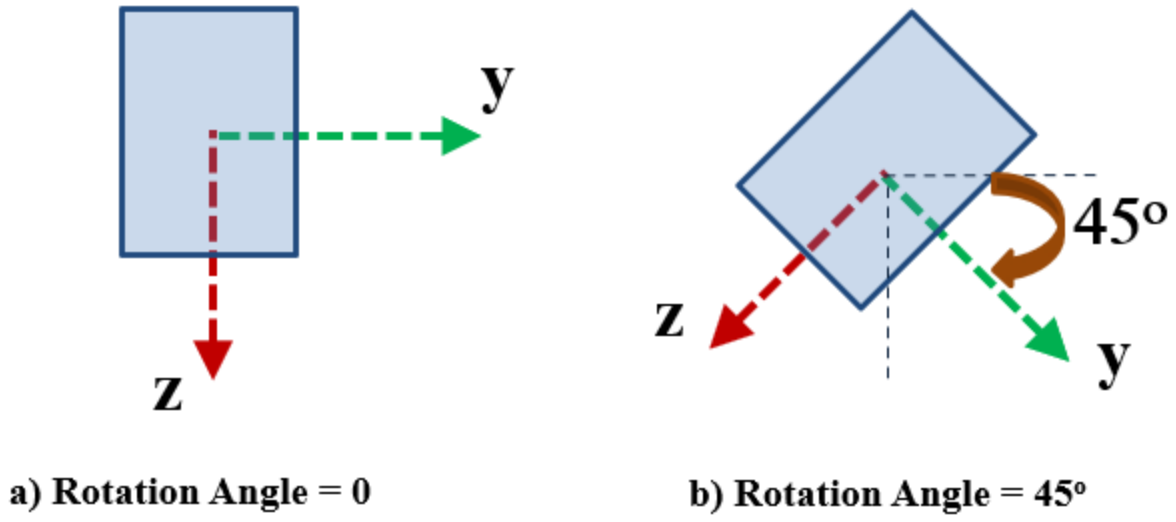


Figure 3.92 Beam Section Rotation – Flexible Cap Version

3.3.10 Data – Springs (Pile-Cap) – Flexible Cap Version

This submenu option is used to enter and/or edit the Plate Data that belomgs to the flexible cap.

3.3.10.1 Data – Springs (Pile-Cap) – Concentrated Springs at Nodes – Flexible Cap Version

This submenu option is used to open the Concentrated Springs at Nodes Window.

In this window the user can define concentrated springs applied at nodes which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.93.

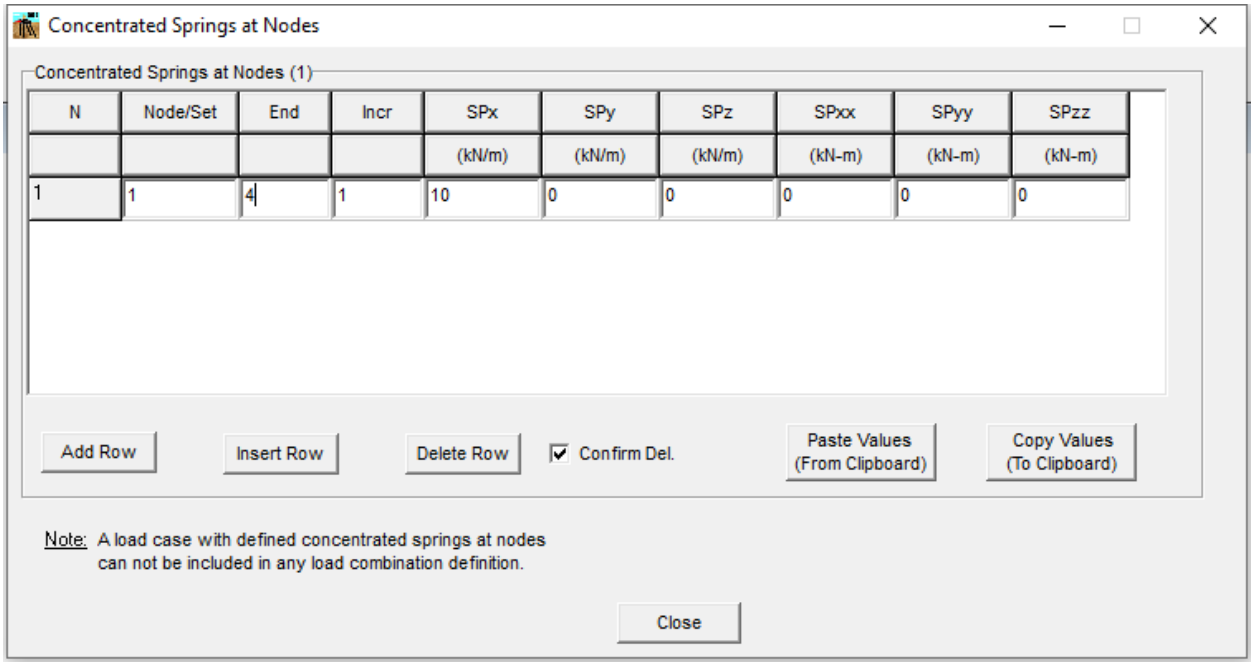


Figure 3.93 Concentrated Springs at Nodes Window -Flexible Cap Version

Concentrated Springs at Nodes

The maximum number of rows defining concentrated load at nodes for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Node/Set**. Enter the node number or the Node Set to which the concentrated springs will be applied. If a Node Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of nodes, enter the highest node number to which the concentrated spring will be applied. For a single node enter zero in this column.
- **Incr**. For a sequence of nodes, enter the increment on node number to get the intermediate node numbers on the sequence to which the concentrated springs will be applied. For a single node enter zero in this column.
- **SPx**. Enter the value for the concentrated spring in the x direction.
- **SPy**. Enter the value for the concentrated spring in the y direction.
- **SPz**. Enter the value for the concentrated spring in the z direction.
- **SPxx**. Enter the value for the concentrated rotational spring in the x direction.
- **SPyy**. Enter the value for the concentrated rotational spring in the y direction.
- **SPzz**. Enter the value for the concentrated rotational spring in the z direction.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated springs at nodes from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (9), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the concentrated springs data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.10.2 Data – Springs (Pile-Cap) – Distributed Springs on Plates – Flexible Cap Version

This submenu option is used to open the Distributed Springs on Plates Window.

In this window the user can define distributed springs applied on plates which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.94.

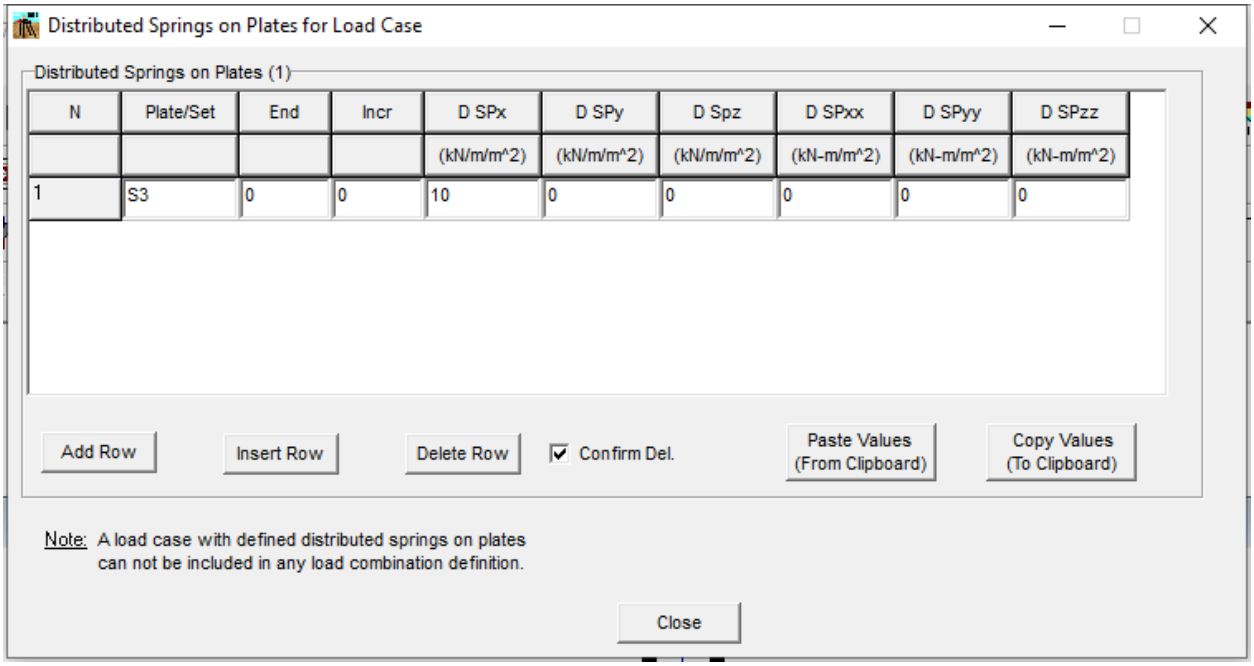


Figure 3.94 Distributed Springs on Plates Window - Flexible Cap Version

Distributed Springs on Plates

The maximum number of rows defining distributed load on plates for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Plate/Set**. Enter the plate number or the Plate Set to which the distributed springs will be applied. If a Plate Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of plates, enter the highest plate number to which the distributed spring will be applied. For a single plate enter zero in this column.
- **Incr**. For a sequence of plates, enter the increment on plate number to get the intermediate plate numbers on the sequence to which the distributed spring will be applied. For a single plate enter zero in this column.
- **D.SPx**. Enter the value for the distributed spring in the x direction.
- **D.SPy**. Enter the value for the distributed spring in the y direction.
- **D.SPz**. Enter the value for the distributed spring in the z direction.
- **D.SPxx**. Enter the value for the distributed rotational spring in the x direction.
- **D.SPyy**. Enter the value for the distributed rotational spring in the y direction.
- **D.SPzz**. Enter the value for the distributed rotational spring in the z direction.

By clicking on the **Paste Values (Clipboard)** button, the program will paste the distributed springs on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (9), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed springs data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.10.3 Data – Springs (Pile-Cap) – Distributed Springs on Beams – Flexible Cap Version

This submenu option is used to open the Distributed Springs on Beams Window.

In this window the user can define distributed loads applied on beams which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.95.

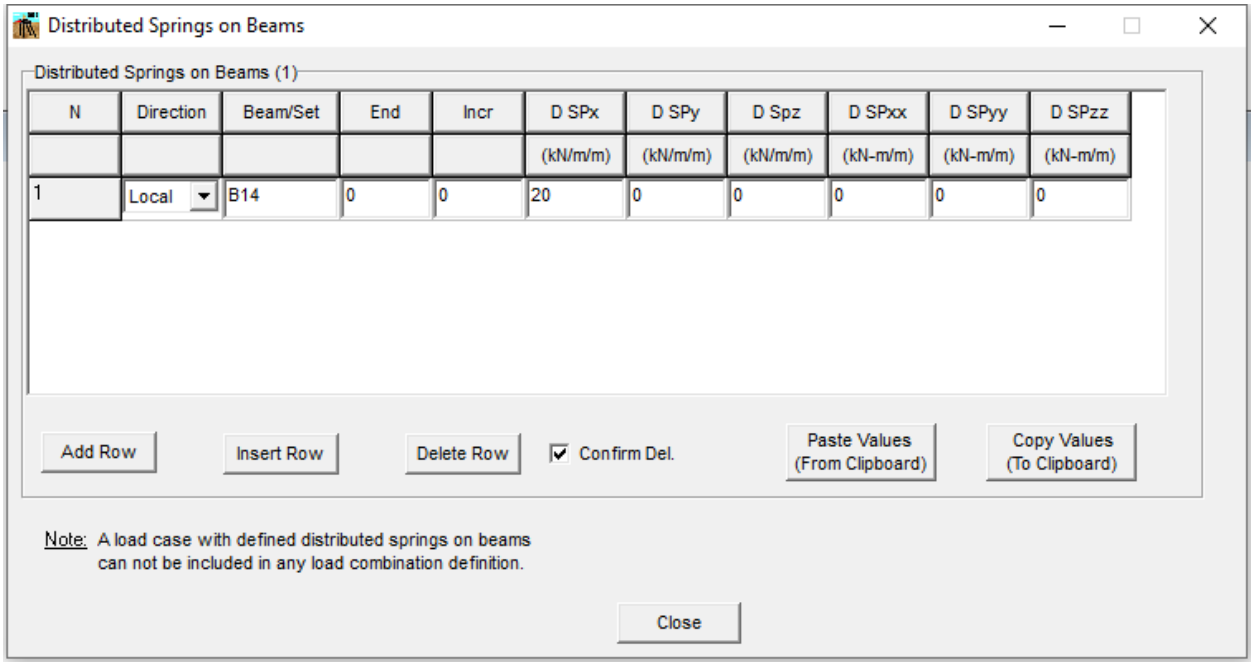


Figure 3.95 Distributed Springs on Beams Window - Flexible Cap Version

Distributed Springs on Beams

The maximum number of rows defining distributed springs on beamss for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Direction**. Select the direction in which the distributed spring will be applied. The user can select along the local or global axes.

- **Beam/Set**. Enter the plate number or the Beam Set to which the distributed spings will be applied. If a Beam Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of beams, enter the highest beam number to which the distributed sping will be applied. For a single beam enter zero in this column.
- **Incr**. For a sequence of beams, enter the increment on beam number to get the intermediate beam numbers on the sequence to which the distributed spring will be applied. For a single beam enter zero in this column.
- **D.SPFx**. Enter the value for the distributed spring in the x direction.
- **D.SPy**. Enter the value for the distributed spring in the y direction.
- **D.SPz**. Enter the value for the distributed spring in the z direction.
- **D.SPxx**. Enter the value for the distributed rotational spring in the x direction.
- **D.SPyy**. Enter the value for the distributed rotational spring in the y direction.
- **D.SPzz**. Enter the value for the distributed rotational spring in the z direction.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the distributed springs on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (10), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed springs data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11 Data - Loading

This submenu option is needed to input concentrated loading in the pile cap and distributed loading along the piles. Also, this menu option allows the user to define different load cases and load combinations for analysis. By selecting this option, the **Loading Definition** window will appear.

3.3.11.1 Loading Definition Window

In this window the user can define the load cases. A sample is shown in Figure 3.96.

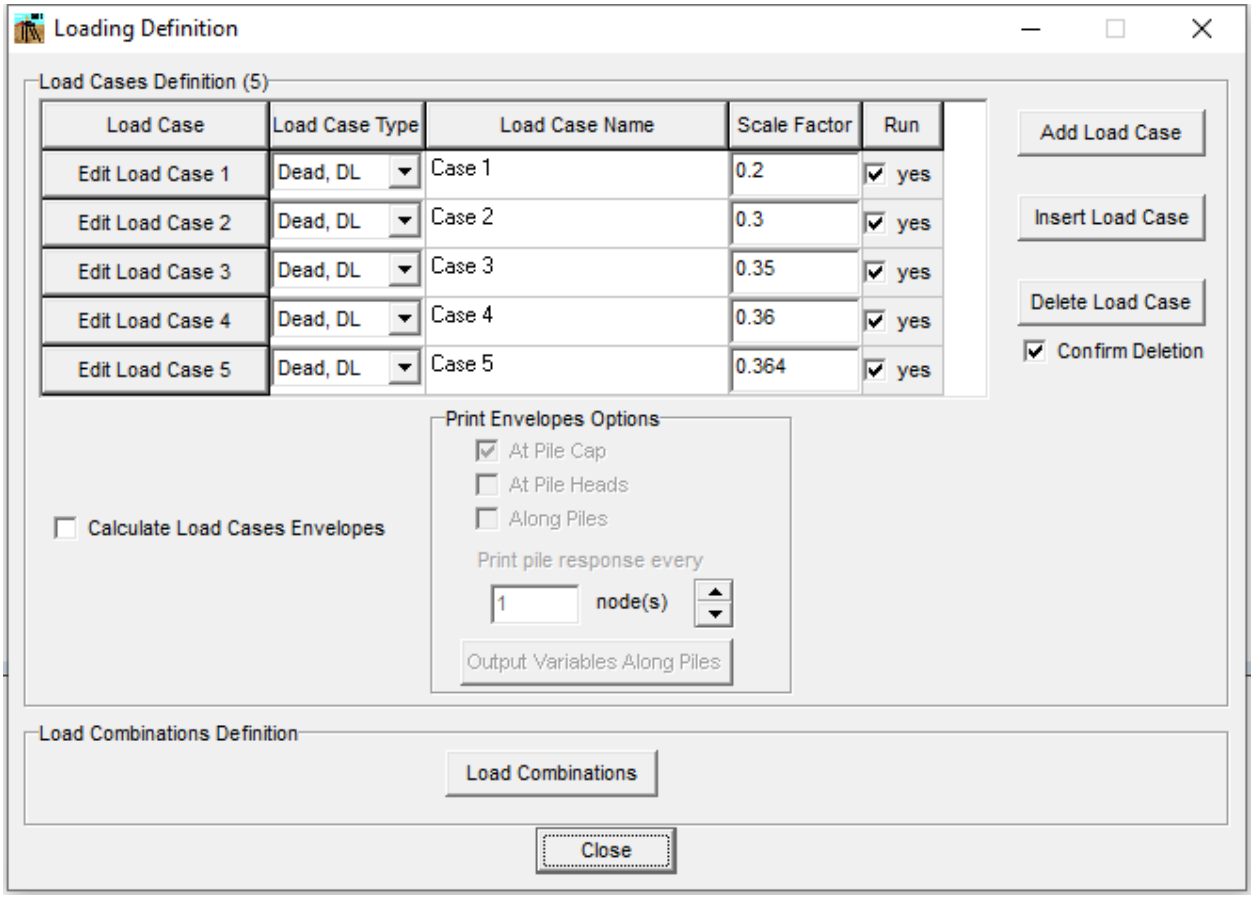


Figure 3.96 Loading Definition Window

Load Cases Definition

The maximum number of load cases that can be defined is limited to 100.

- **Add Load Case.** Add a new load case at the end of the list.
- **Insert Load Case.** Add a new load case after the selected one.
- **Delete Load Case.** Delete the current load case.

- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a load case.

For each load case (row) the following columns are available:

- **Load Case**. Click on the **Edit Load Case** button to open the **Edit Load Case** window.
- **Load Case Type**. Select the load type consistent with the defined load case. This is very useful for defining load combinations. The following types can be selected in GROUP 2022 as a load case.
    - Dead, DL
    - Live, LL
    - Seismic, EQ
    - Impact, IM
    - Wind, WN
    - Water, WT
    - Ice, IC
    - Soils, Hsl
    - Roof Live, Lr
    - Rain, Rn
    - Snow, Sn
    - Temperature, Tm
    - Special, Sp
- **Load Case Name**. Enter the name associated with the load case.
- **Scale Factor**. Enter the scale factor to be applied to the load case.
- **Scale Factor**. Check on this option to analyze the load case.

If two or more load cases are defined the program can compute the load case envelopes (maximum and minimum effects). Check the **Calculate Load Cases Envelopes** option to calculate the envelopes. Also, select the option for output of the calculated envelopes.

- *At pile cap*. Print the equivalent load and displacements of the pile cap at the origin.
- *At pile cap elements (Flexible Cap Version)*. Print the displacements at the nodes that belong to the flexible pile cap. For plate elements prints the internal forces and stresses at each integration point. For beam elements, prints the forces and displacements at some points along the beam.
- *At pile heads.* Print for each pile the pile top displacement and reactions in both the pile (local axes) and global (global axes) coordinate systems.

- *Along piles*. Print for each pile its deflection, moment, shear, soil reaction, stress and rigidity along the pile. If this option is selected the user may define the increment in nodes to print the response along the piles.

Click on the **Output Variables Along Piles** button to open the **Output Variables** window for defining which variables for results along the pile would like to be printed in the output file for the load cases envelopes.

Load Combinations Definition

Click on the **Load Combinations** button to open the **Load Combinations** window for defining load combinations. A check mark will show up if at least one load combination has been already defined. This button is disabled if no load cases have been defined.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.3.11.2 Edit Load Case Window

In this window the user can define load cases. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.97. For a model using a flexible pile cap (Flexible Cap Version), a sample window is shown in Figure 3.98.

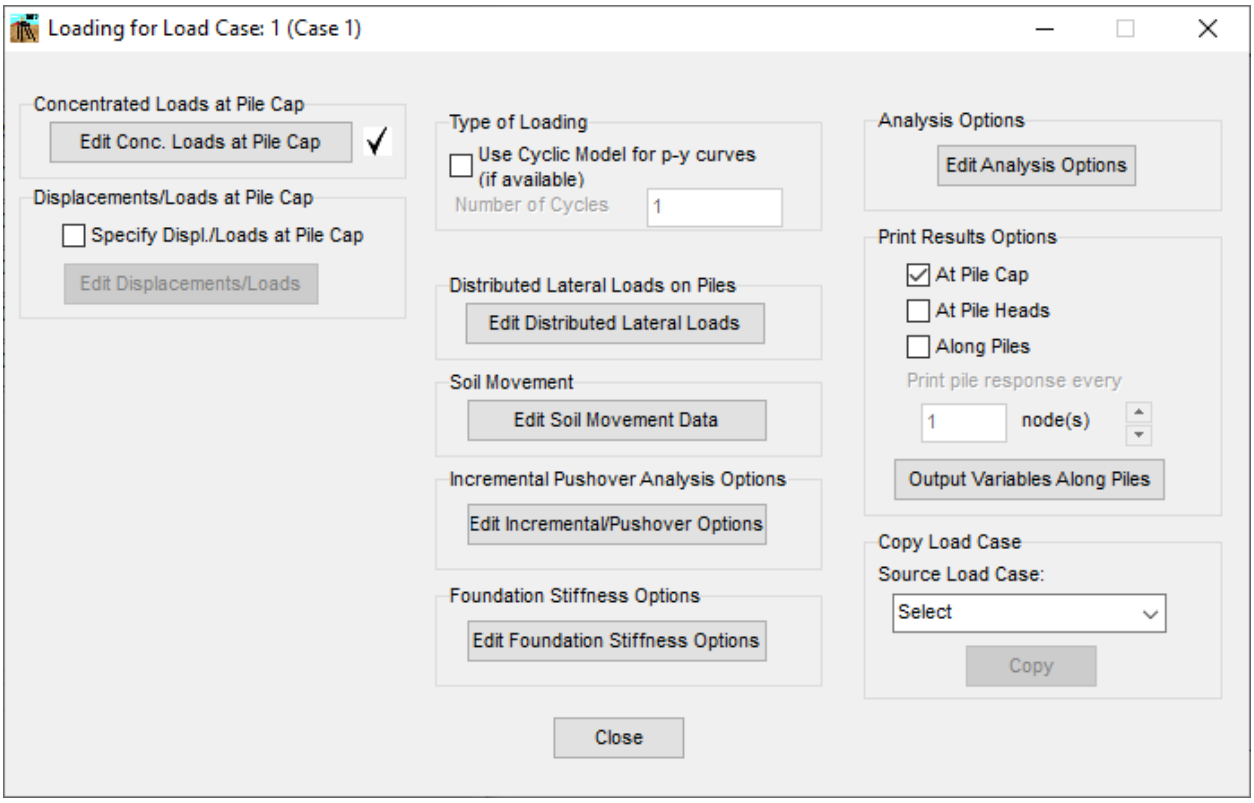


Figure 3.97 Edit Load Case Window

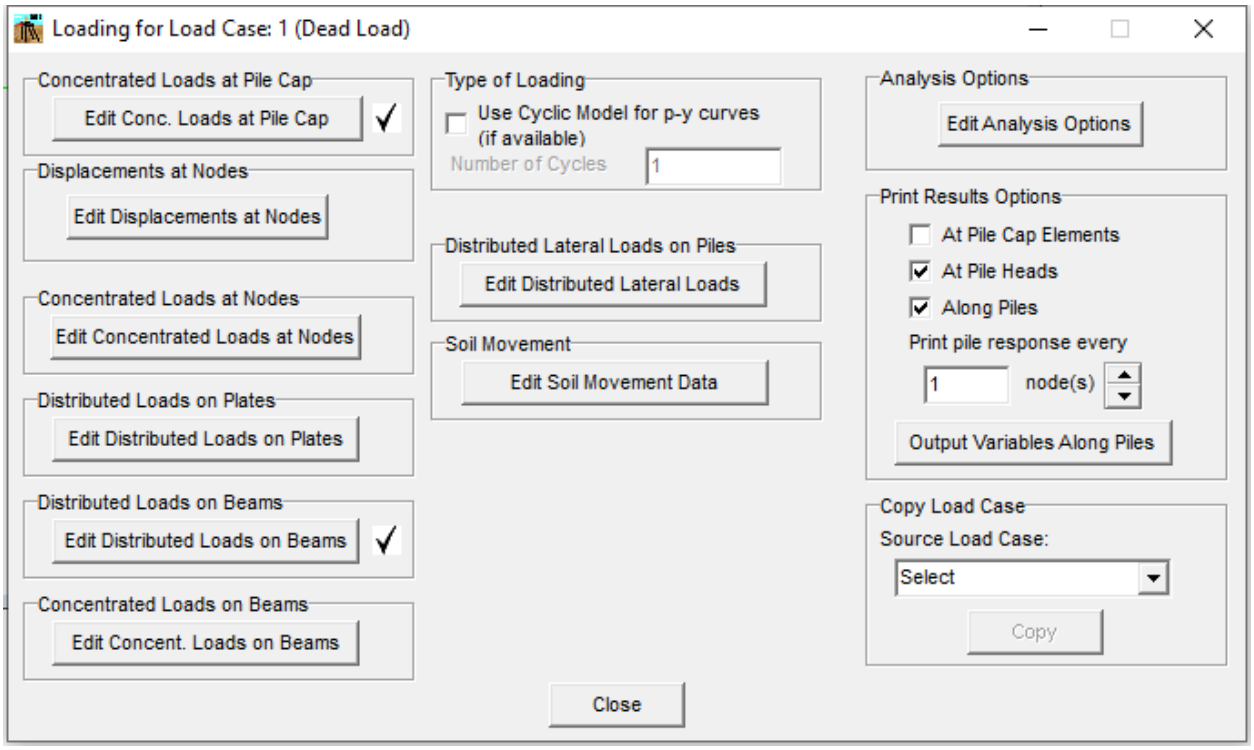


Figure 3.98 Edit Load Case Window for a model with a flexible pile cap -Flexible Cap Version

Concentrated Loads at Pile Cap

Click on the **Concentrated Loads at Pile Cap** button to open the **Concentrated Loads at Pile Cap** window for definition of concentrated loads acting at the piel cap. A check mark will appear if at least one concentrated load has been already defined for the current load case.

Displacement/Loads at Pile Cap

Select **Specify Displacements/Loads at Pile Cap** to enable the option to define specified displacements at pile cap. Then, click on the **Edit Displacements/Loads** button to open the **Specify Displacements/Loads** window for definition of the displacements that may be combined with loads at the pile cap. When this option is selected the concentrated load at pile cap option becomes inactive. Please note that the load case with definition of specified displacement/loads can not be included in any load combination. This option is not available if a model is defined using a flexible pile cap (Flexible Cap Version)

Type of Loading

Select **Use Cyclic Model por p-y curves if available** and enter the Number of Cycles to consider a Cyclic Loading. The type of loading is important to taking into account when analyzing piles under lateral loading. Further reference about the "Influence of Nature of Loading" is included in the Chapter "Soil Response Under Natural Loading" of the Technical Manual.

Cyclic loading is primarily used for storm, wave, or wind loading types. The main effect from cyclic loading is to reduce the soil resistance based on the soil criteria for cyclic p- curves, as described in the Technical Manual.

Analysis Options

Click on the **Edit Analysis Options** button to open the **Analysis Options** window for the definition of convergence parameters.

Distributed Lateral Loads on Piles

Click on the **Edit Distributed Lateral Loads** button to open the **Distributed Lateral Loads** window for definition of distributed lateral loads along the pile length. A check mark will appear if at least one distributed lateral load has been already defined for the current load case.

Soil Movement

Click on the **Edit Soil Movement Data** button to open the **Soil Movement** window for definition of soil movement along the piles. A check mark will appear if at least one soil movement data has been already defined for the current load case.

Foundation Stiffness Options

Click on the **Edit Foundation Stiffness Options** button to open the **Foundation Stiffness Options** window, so the user may ask the program to compute the flexibility and/or stiffness matrices for the current load case. A check mark will appear if a foundation flexibility and/or stiffness will be computed for the current load case. This option is not available for a load case if the Incremental/Pushover Analysis Option is defined. This option is not available if a model is defined using a flexible pile cap (Flexible Cap Version)

Incremental/PushOver Analysis Options

Click on the **Edit Incremental/PushOver Options** button to open the **Incremental/PushOver Analysis Options** window, so the user may ask the program to perform incremental/pushover analysis for the current load case. A check mark will appear if the incremental/pushover analysis will be performed for the current load case. This option is not available for a load case if the Foundation Stiffness Option is defined. This option is not available if a model is defined using a flexible pile cap (Flexible Cap Version)

Print Results Options

Select the option for outputting the results of the current load case.

- *At pile cap*. Print the equivalent load and displacements of the pile cap at the origin.
- *At pile cap elements (Flexible Cap Version)*. Print the displacements at the nodes that belong to the flexible pile cap. For plate elements prints the internal forces and stresses at each integration point. For beam elements, prints the forces and displacements at some points along the beam.
- *At pile heads.* Print for each pile the pile top displacement and reactions in both the pile (local axes) and global (global axes) coordinate systems.
- *Along piles*. Print for each pile its deflection, moment, shear, soil reaction, stress and rigidity along the pile. If this option is selected the user may define the increment in nodes to print the response along the piles.

Click on the **Output Variables Along Piles** button to open the **Output Variables** window for defining which variables for results along the pile would like to be printed in the output file for the current load case.

Copy Load Case

The option is helpful when there are load cases with few differences. So, the user may copy one into another and do the required changes as needed. Select the load case that is going to be the source (already defined), and then click on the **Copy** button to proceed with copying all the load case data to the current case. This option is available if at least two load cases have been defined.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

The following options are available for a model using a flexible pile cap (Flexible Cap Version)

Displacement at Nodes

Click on **Edit Displacements at Nodes** button to open the **Displacements at Nodes** window for definition of the displacements at the nodes that belong to the finite element mesh defining the flexible pile cap. Please note that the load case with definition of displacement at nodes can not be included in any load combination.

Concentrated Loads at Nodes

Click on **Edit Concentrated Loads at Nodes** button to open the **Concentrated Loads at Nodes** window for definition of concentrated loads at the nodes that belong to the finite element mesh defining the flexible pile cap.

Distributed Loads on Plates

Click on **Edit Distributed Loads on Plates** button to open the **Distributed Loads on Plates** window for definition of distributed loads on the plates that belong to the finite element mesh defining the flexible pile cap.

Distributed Loads on Beams

Click on **Edit Distributed Loads on Beams** button to open the **Distributed Loads on Beams** window for definition of distributed loads on the beams that belong to the finite element mesh defining the flexible pile cap.

Concentrated Loads on Beams

Click on **Edit Concent. Loads on Beams** button to open the **Concentrated Loads on Beams** window for definition of concentrated loads on the beams that belong to the finite element mesh defining the flexible pile cap.

3.3.11.3 Edit Concentrated Loads at Pile Cap Window

In this window the user can define the concentrated loads applied at the pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.97.

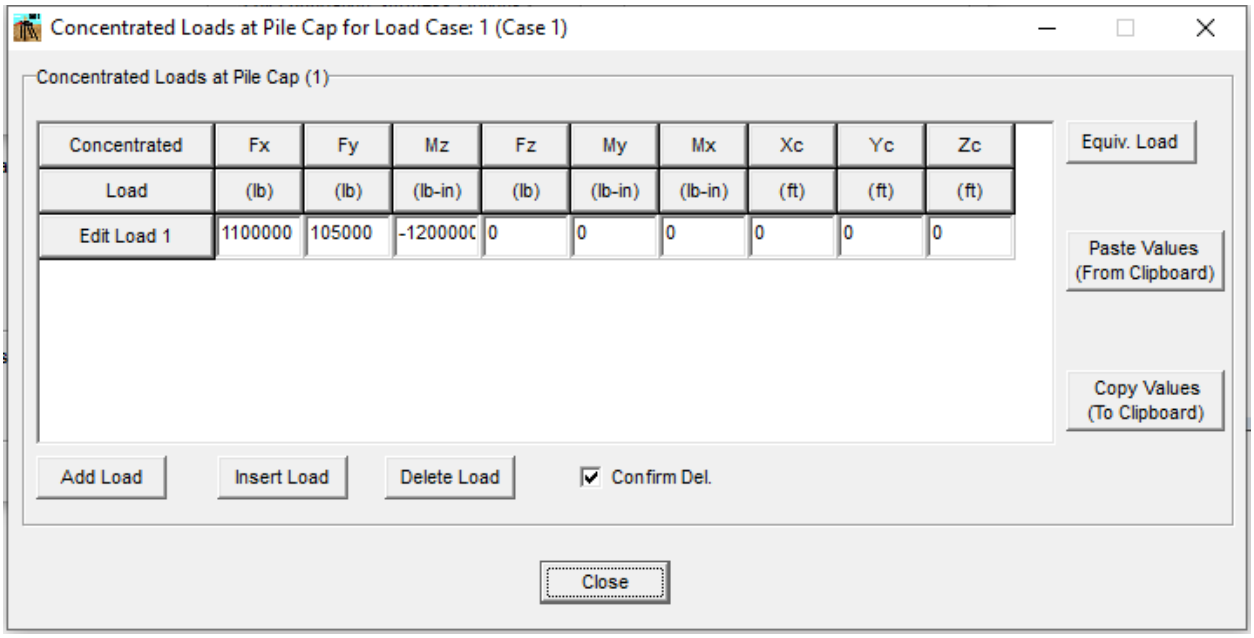


Figure 3.99 Concentrated Loads at Pile Cap Window

Concentrated Loads at Pile Cap

The maximum number of concentrated loads on a pile cap for each load case is limited to 100.

- **Add Load.** Add a new load at the end of the list.
- **Insert Load.** Add a new load below the currently selected one.
- **Delete Load.** Delete the selected load.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a load.

For each concentrated load (row) the following columns are available:

- **Concentrated Load**. Click on the **Edit Load** button to open the **Concentrated Load** window to enter the concentrated load and location. The concentrated load can also be entered in the next columns as described below.
    - **Fx**. Enter the applied force in the X direction (vertical).
    - **Fy**. Enter the applied force in the Y direction (horizontal).
    - **Mz**. Enter the applied moment in the Z direction.
    - **Fz**. Enter the applied force in the Y direction (horizontal). Available for 3D analysis.
    - **My**. Enter the applied moment in the Y direction. Available for 3D analysis.
    - **Mx**. Enter the applied moment in the X direction (torsional). Available for 3D analysis.

By clicking on the **Equiv. Load** button, the program will open the **Concentrated Load** window, which shows the equivalent concentrated load to be applied at the origin of coordinates. For a model with a rigid pile cap, this is the load to be applied for the analysis of the current load case. For a model with a flexible pile cap (Flexible Cap Version) each concentrated load will be projected to a location in the pile cap for the analysis.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated loads from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (5 for 2D models, and 9 for 3D models), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the concentrated loads to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.4 Concentrated Load Window

In this window the user may enter the concentrated load at the pile cap level. Also, this window shows the equivalent load at the origin of coordinates for the current load case.

For a 3-dimensional model, the user may enter six load components (i.e. vertical load in the X-direction, lateral load in the Y-direction, lateral load in the Z-direction, moment on the x-y plane, moment on the x-z plane, and torsional moment on the y-z plane). In addition, the user may enter the X, Y and Z coordinates of the location of the load. If the load is at the origin, X, Y and Z coordinates should be equal to zero. A sample of this window is shown in Figure 3.100.

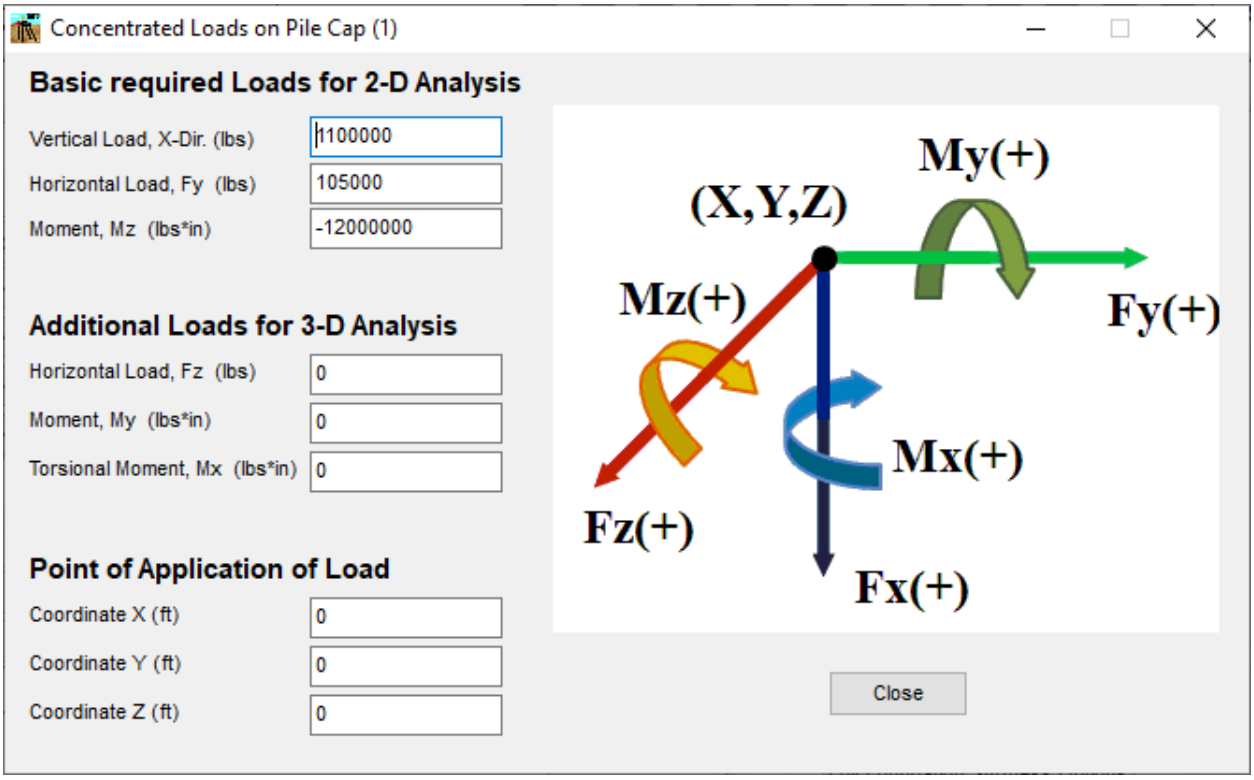


Figure 3.100 Concentrated Load Window. Loading for 3D Analysis

For a 2-dimensional model, the user may enter three load components (i.e. vertical load in the X-direction, lateral load in the Y-direction, and moment on the x-y plane). In addition, the user may enter the X and Y coordinates of the location of the load. If the load is at the origin, X and Y coordinates should be equal to zero. A sample of this window is shown in Figure 3.101.

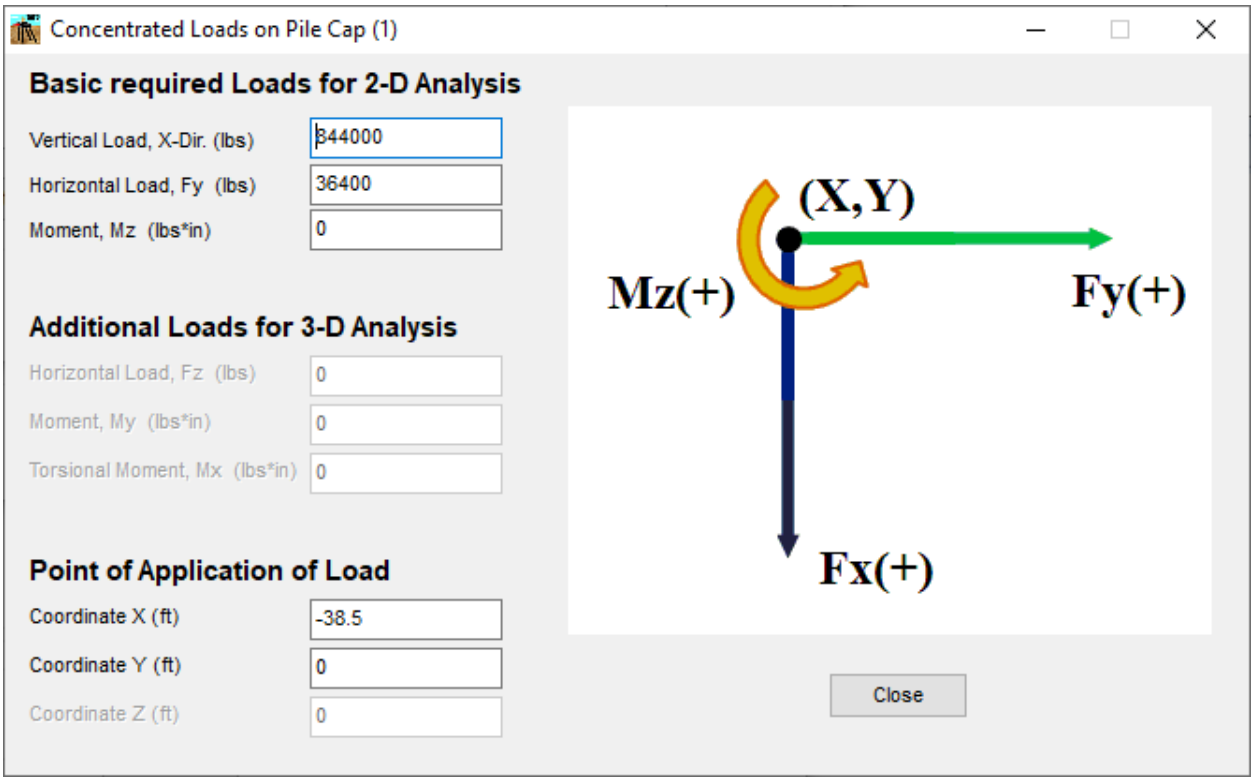


Figure 3.101 Concentrated Load Window. Loading for 2D Analysis

The right-handed Cartesian coordinate system as illustrated in the **Concentrated Load** window is used for both 2-D and 3-D models.

It should be noted that for 2-dimensional models, GROUP 2022 uses "positive" for moment acting counter-clockwise on the x-y plane in order to be consistent with the right-handed Cartesian coordinate system. It is different from previous versions (v4.0 and earlier). The user should check the sign convention if a data file created by previous versions of GROUP is loaded.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Concentrated Loads on Pile Cap** window.

3.3.11.5 Specified Displacements/Loads Window

In this window the user may enter specified displacements or concentrated load at the pile cap level. This window is not available for a model with a flexible pile cap (Flexible Cap Version).

For a 3-dimensional model, the user may enter a combination of 6 boundary conditions between displacements or loads at the pile cap (i.e. vertical load or vertical displacement in the X-direction, lateral load or lateral displacement in the Y-direction, lateral load or lateral displacement in the Z-direction, moment or rotation on the x-y plane, moment or rotation on the x-z plane, and torsional moment or rotation on the y-z plane). In addition, the user may enter the X, Y and Z coordinates of the location of the specified boundary condition (displacements and/or loads). If the boundary condition is applied at the origin, X, Y and Z coordinates should be equal to zero. A sample of this window is shown in Figure 3.102.

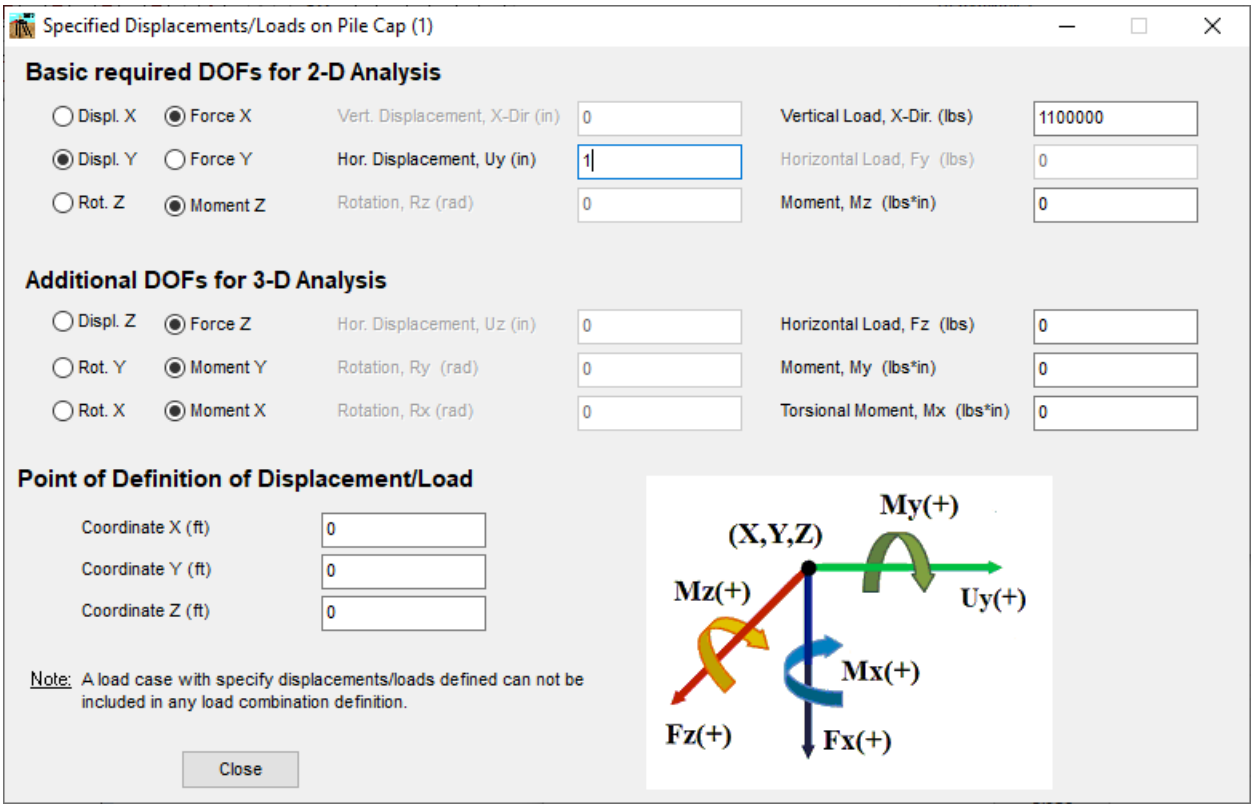


Figure 3.102 Specified Displacements/Loads Window. 3D Analysis Case

For a 2-dimensional model, the user may enter a combination of 3 boundary conditions between displacements or loads at the pile cap (i.e. vertical load or vertical displacement in the X-direction, lateral load or lateral displacement in the Y-direction, and moment or rotation on the x-y plane). In addition, the user may enter the X and Y coordinates of the location of the specified boundary condition (displacements and/or loads). If the boundary condition is applied at the origin, X and Y coordinates should be equal to zero. A sample of this window is shown in Figure 3.103.

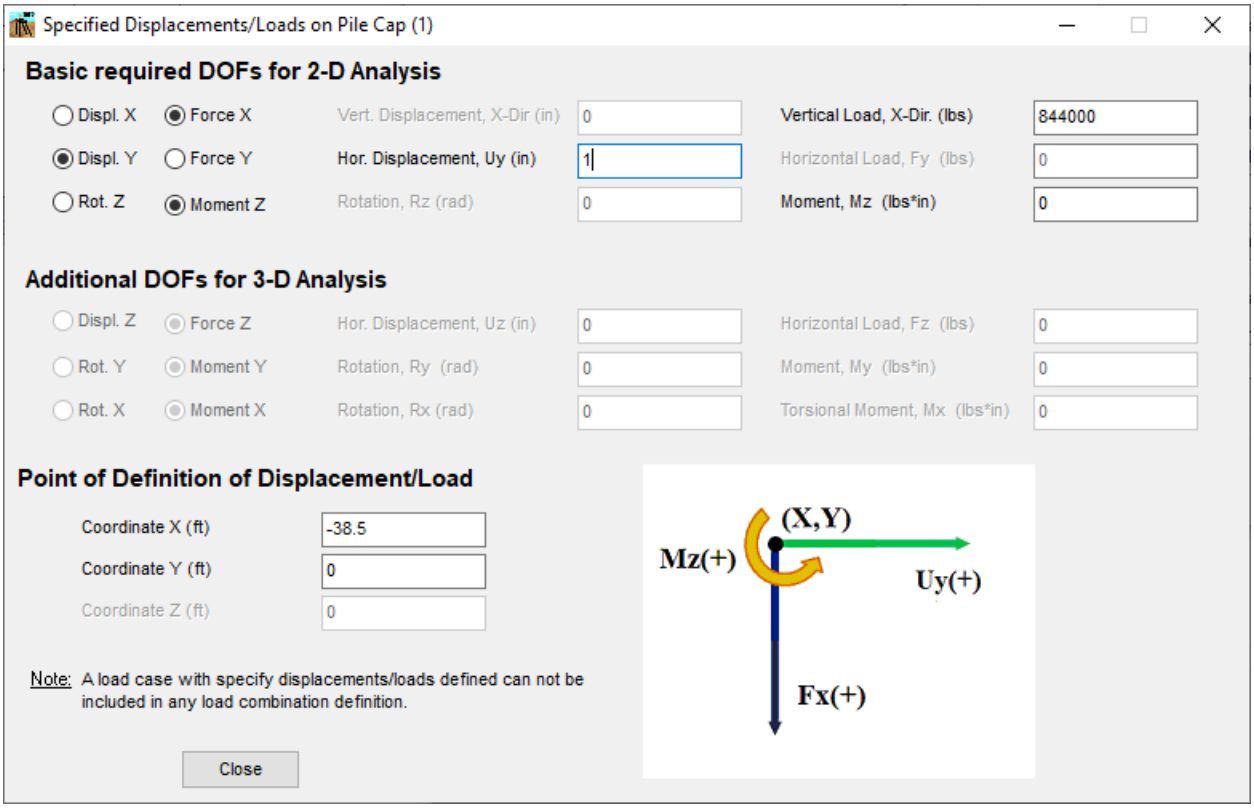


Figure 3.103 Specified Displacements/Loads Window. 2D Analysis Case

The right-handed Cartesian coordinate system as illustrated in the **Specified Displacement/Loads** window is used for both 2-D and 3-D models.

Please note that the load case that includes specified displacement/forces. It should be noted that for 2-dimensional models, GROUP 2022 uses "positive" for moment acting counter-clockwise on the x-y plane in order to be consistent with the right-handed Cartesian coordinate system. It is different from previous versions (v4.0 and earlier). The user should check the sign convention if a data file created by previous versions of GROUP is loaded.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.6 Displacement at Nodes Window -Flexible Cap Version

In this window the user can define displacement at nodes which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.104.

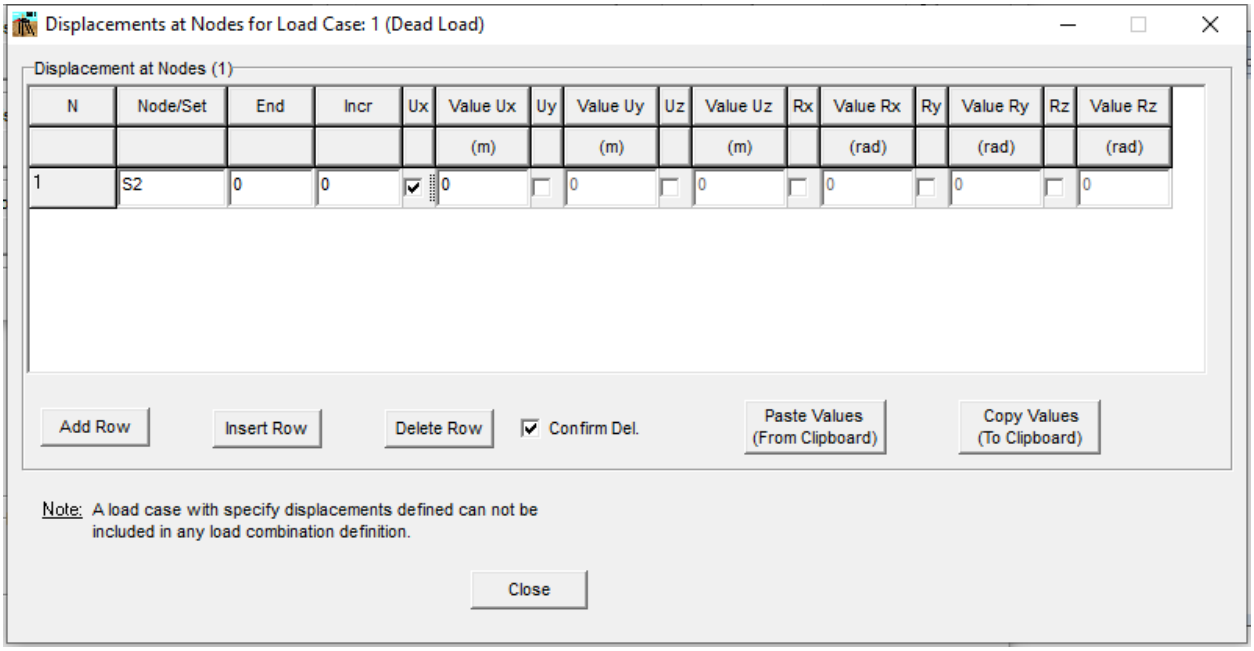


Figure 3.104 Displacements at Nodes Window - Flexible Cap Version

Displacement at Nodes

The maximum number of rows defining displacement at nodes for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.

- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Node/Set**. Enter the node number or the Node Set to which the specified displacement will be applied. If a Node Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of nodes, enter the highest node number to which the specified displacement will be applied. For a single node enter zero in this column.
- **Incr**. For a sequence of nodes, enter the increment on node number to get the intermediate node numbers on the sequence to which the specified displacement will be applied. For a single node enter zero in this column.
- **Ux**. Check this column if a specified displacement in the x direction is defined.
- **Value Ux**. Enter the value for the specified displacement in the x direction. If Ux is not checked, this value will be ignored.
- **Uy**. Check this column if a specified displacement in the y direction is defined.
- **Value Uy**. Enter the value for the specified displacement in the y direction. If Uy is not checked, this value will be ignored.
- **Uz**. Check this column if a specified displacement in the z direction is defined.
- **Value Uz**. Enter the value for the specified displacement in the z direction. If Uz is not checked, this value will be ignored.
- **Rx**. Check this column if a specified rotation in the x direction is defined.
- **Value Rx**. Enter the value for the specified rotation in the x direction. If Rx is not checked, this value will be ignored.
- **Ry**. Check this column if a specified rotation in the y direction is defined.
- **Value Ry**. Enter the value for the specified rotation in the y direction. If Ry is not checked, this value will be ignored.
- **Rz**. Check this column if a specified rotation in the z direction is defined.
- **Value Rz**. Enter the value for the specified rotation in the z direction. If Rz is not checked, this value will be ignored.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the displacement at nodes from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (15), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the specified displacements data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.7 Concentrated Loads at Nodes Window - Flexible Cap Version

In this window the user can define concentrated loads applied at nodes which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.105.

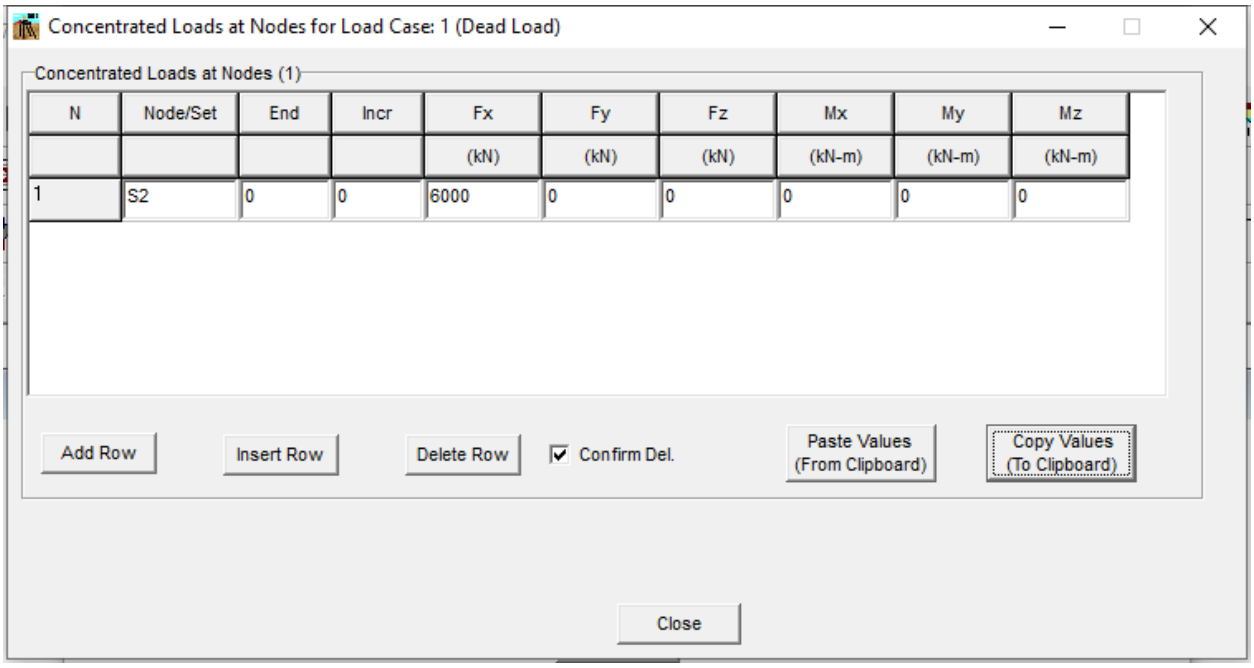


Figure 3.105 Concentrated Loads at Nodes Window -Flexible Cap Version

Concentrated Loads at Nodes

The maximum number of rows defining concentrated load at nodes for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.

- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Node/Set**. Enter the node number or the Node Set to which the concentrated loads will be applied. If a Node Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of nodes, enter the highest node number to which the concentrated load will be applied. For a single node enter zero in this column.
- **Incr**. For a sequence of nodes, enter the increment on node number to get the intermediate node numbers on the sequence to which the concentrated loads will be applied. For a single node enter zero in this column.
- **Fx**. Enter the value for the concentrated load in the x direction.
- **Fy**. Enter the value for the concentrated load in the y direction.
- **Fz**. Enter the value for the concentrated load in the z direction.
- **Mx**. Enter the value for the concentrated moment in the x direction.
- **My**. Enter the value for the concentrated moment in the y direction.
- **Mz**. Enter the value for the concentrated moment in the z direction.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated loads at nodes from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (9), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the concentrated loads data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.8 Distributed Loads on Plates Window - Flexible Cap Version

In this window the user can define distributed loads applied on plates which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.106.

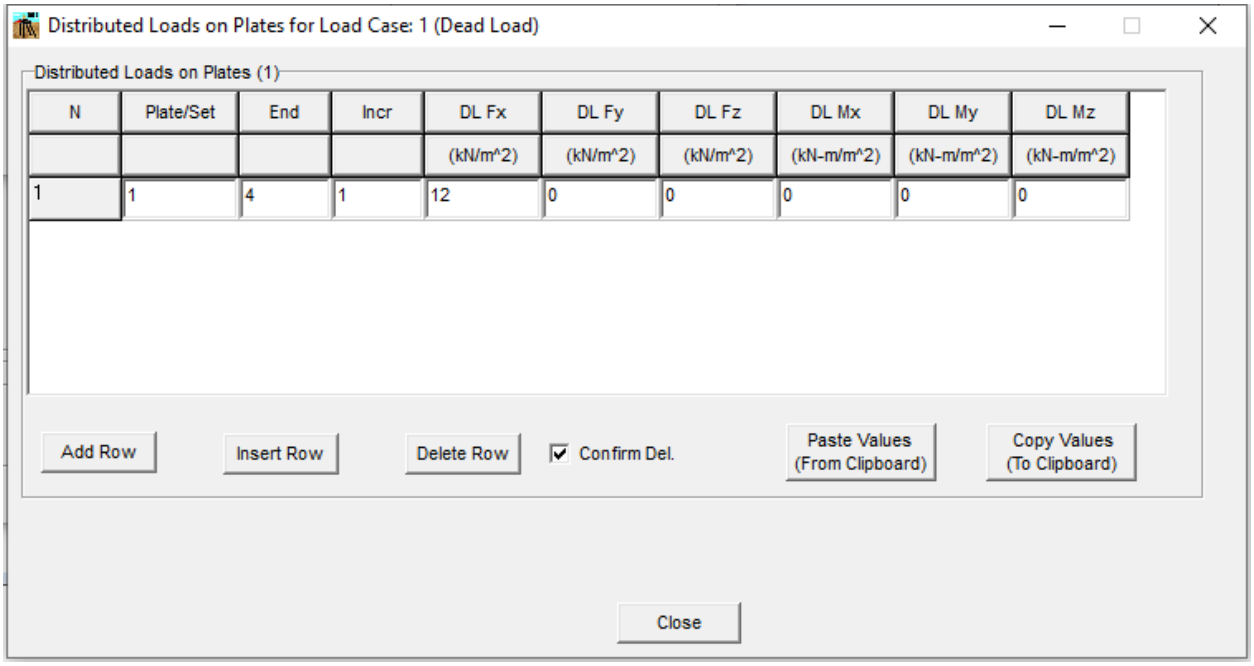


Figure 3.106 Distributed Loads on Plates Window - Flexible Cap Version

Distributed Loads on Plates

The maximum number of rows defining distributed load on plates for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Plate/Set**. Enter the plate number or the Plate Set to which the distributed loads will be applied. If a Plate Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of plates, enter the highest plate number to which the distributed load will be applied. For a single plate enter zero in this column.
- **Incr**. For a sequence of plates, enter the increment on plate number to get the intermediate plate numbers on the sequence to which the distributed load will be applied. For a single plate enter zero in this column.
- **DL. Fx**. Enter the value for the distributed load in the x direction.
- **DL. Fy**. Enter the value for the distributed load in the y direction.
- **DL Fz**. Enter the value for the distributed load in the z direction.
- **DL. Mx**. Enter the value for the distributed moment in the x direction.
- **DL. My**. Enter the value for the distributed moment in the y direction.
- **DL. Mz**. Enter the value for the distributed moment in the z direction.

By clicking on the **Paste Values (Clipboard)** button, the program will paste the distributed loads on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (9), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed loads data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.9 Distributed Loads on Beams Window - Flexible Cap Version

In this window the user can define distributed loads applied on beams which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.107.

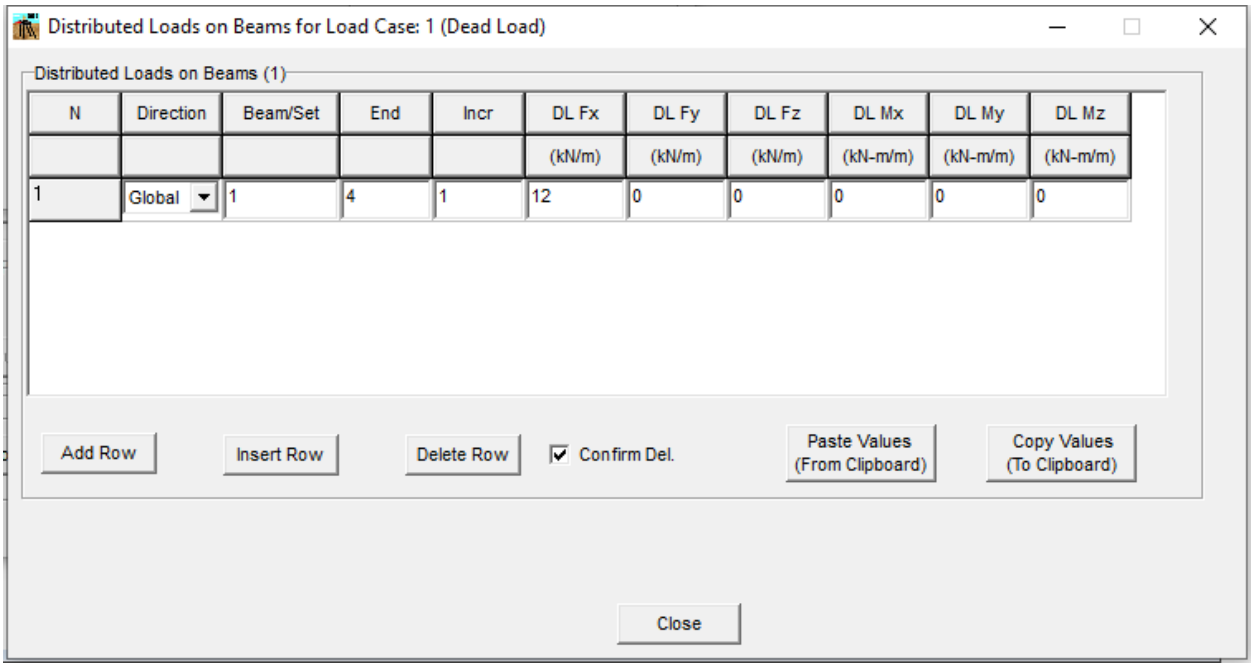


Figure 3.107 Distributed Loads on Beams Window - Flexible Cap Version

Distributed Loads on Beams

The maximum number of rows defining distributed load on beams for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Direction**. Select the direction in which the distributed load will be applied. The user can select along the local or global axes.
- **Beam/Set**. Enter the plate number or the Beam Set to which the distributed loads will be applied. If a Beam Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of beams, enter the highest beam number to which the distributed load will be applied. For a single beam enter zero in this column.

- **Incr**. For a sequence of beams, enter the increment on beam number to get the intermediate beam numbers on the sequence to which the distributed load will be applied. For a single beam enter zero in this column.
- **DL. Fx**. Enter the value for the distributed load in the x direction.
- **DL. Fy**. Enter the value for the distributed load in the y direction.
- **DL Fz**. Enter the value for the distributed load in the z direction.
- **DL. Mx**. Enter the value for the distributed moment in the x direction.
- **DL. My**. Enter the value for the distributed moment in the y direction.
- **DL. Mz**. Enter the value for the distributed moment in the z direction.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the distributed loads on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (10), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed loads data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.10 Concentrated Loads on Beams Window - Flexible Cap Version

In this window the user can define concentrated loads applied on beams which belong to the finite element mesh defining the flexible pile cap. The load case number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.108.

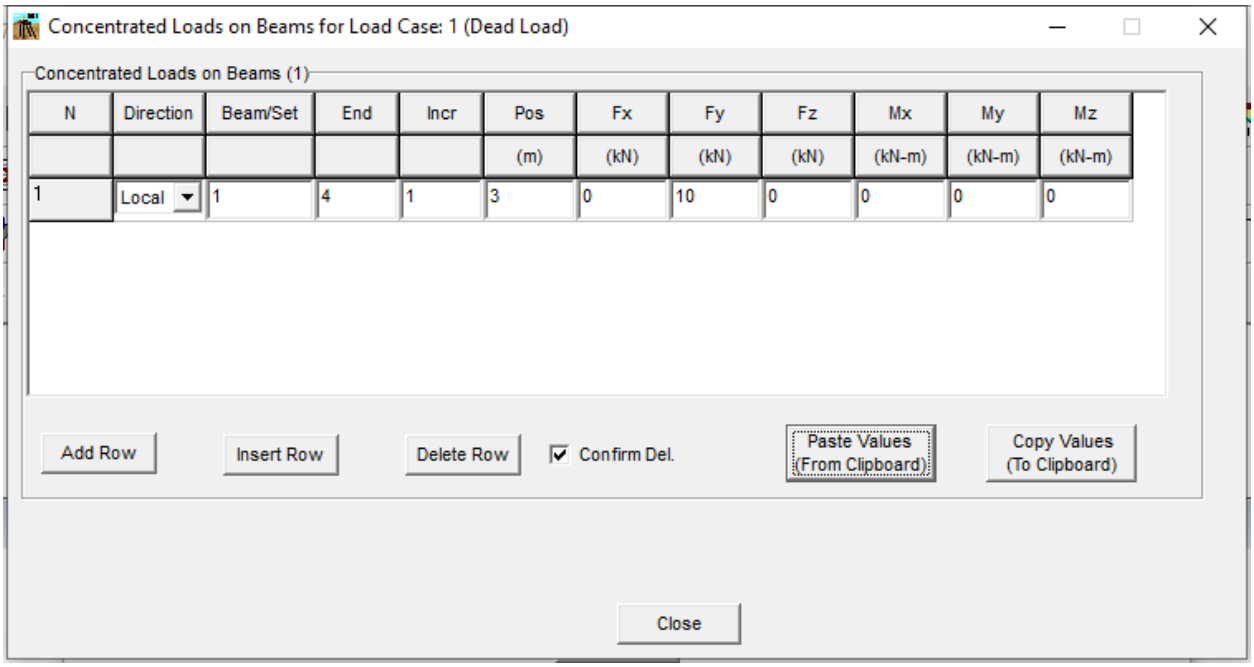


Figure 3.108 Concentrated Loads on Beams Window - Flexible Cap Version

Concentrated Loads on Beams

The maximum number of rows defining concentrated loads on beams for each load case is limited to 1000.

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the currently selected one.
- **Delete Row.** Delete the selected row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Direction**. Select the direction in which the concentrated load will be applied. The user can select along the local or global axes.
- **Beam/Set**. Enter the beam number or the Beam Set to which the distributed loads will be applied. If a Beam Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of beams, enter the highest beam number to which the concentrated load will be applied. For a single beam enter zero in this column.

- **Incr**. For a sequence of beams, enter the increment on beam number to get the intermediate beam numbers on the sequence to which the concentrated load will be applied. For a single plate enter zero in this column.
- **Pos**. Enter the position along the beam where the concentrated load is applied.
- **DL. Fx**. Enter the value for the concentrated load in the x direction.
- **DL. Fy**. Enter the value for the concentrated load in the y direction.
- **DL Fz**. Enter the value for the concentrated load in the z direction.
- **DL. Mx**. Enter the value for the concentrated moment in the x direction.
- **DL. My**. Enter the value for the concentrated moment in the y direction.
- **DL. Mz**. Enter the value for the concentrated moment in the z direction.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated loads on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (11), the program will not paste any values.

By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed loads data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Edit Load Case** window.

3.3.11.11 Distributed Lateral Loads Window

In this window the user may enter the distributed lateral loads along the piles. A sample of this window is shown in Figure 3.109.

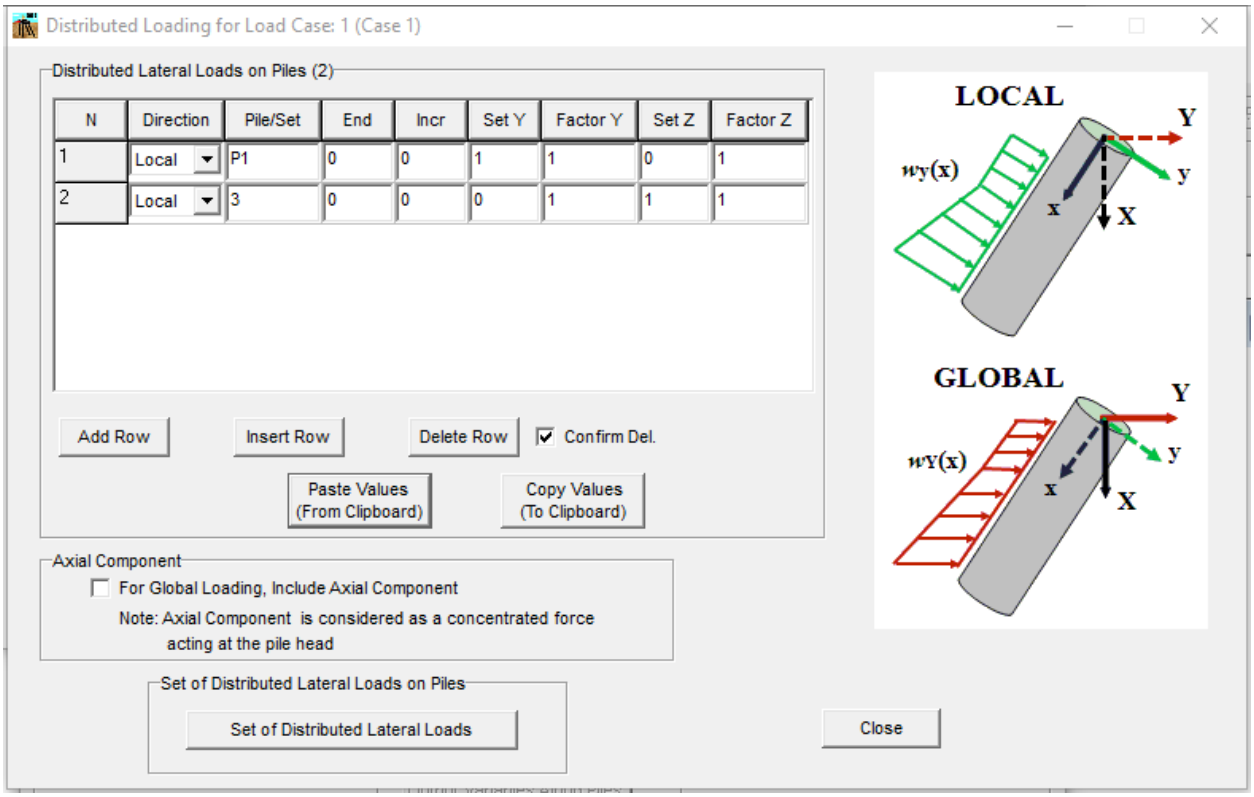


Figure 3.109 Distributed Lateral Loads Window

Distributed Lateral Loads on Piles

The maximum number of rows for distributed loads definition is limited to 100.

The following buttons may be available based in the number of load cases:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the current row.
- **Delete Row.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Direction**. Select the direction in which the distributed load will be applied. The user can select along the local (y and/or z) or global (Y and/or Z) axes. This option is useful when considering batter piles since the global and local axes are different.

- **Pile/Set**. Enter the pile number or the Pile Set to which the distributed load will be applied. If a Pile Set is entered, the values for End and Incr. will be ignored.
- **End**. For a sequence of piles, enter the highest pile number to which the distributed load will be applied. For a single pile enter zero in this column.
- **Incr**. For a sequence of piles, enter the increment on pile number to get the intermediate pile numbers on the sequence. For a single pile enter zero in this column.
- **Set Y**. Enter the set number of the distributed lateral load to be applied in the Y direction (either local or global).
- **Factor Y**. Enter the factor to be applied at the distributed lateral load applied in the Y direction (either local or global).
- **Set Z**. Enter the set number of the distributed lateral load to be applied in the Z direction (either local or global). Available for 3D analysis.
- **Factor Z**. Enter the factor to be applied at the distributed lateral load applied in the Z direction (either local or global). Available for 3D analysis.

Axial Component

The analysis of distributed loading on the Global axes is decomposed into the local axes, as shown in Figure 3.110. The distributed load in the axial direction (local axes x) is integrated and considered as a concentrated force acting at the pile head. Select the option **Include Axial Component** to consider the axial component of global loading into the analysis; otherwise, the axial component will not be included in the analysis.

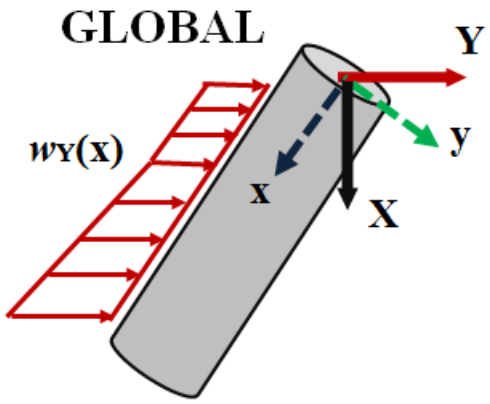
Set of Distributed Lateral Loads on Piles

Click on the **Set of Distributed Lateral Loads** button in order to open the **Set of Distributed Lateral Loads** window to define the distributed lateral loads along the pile.

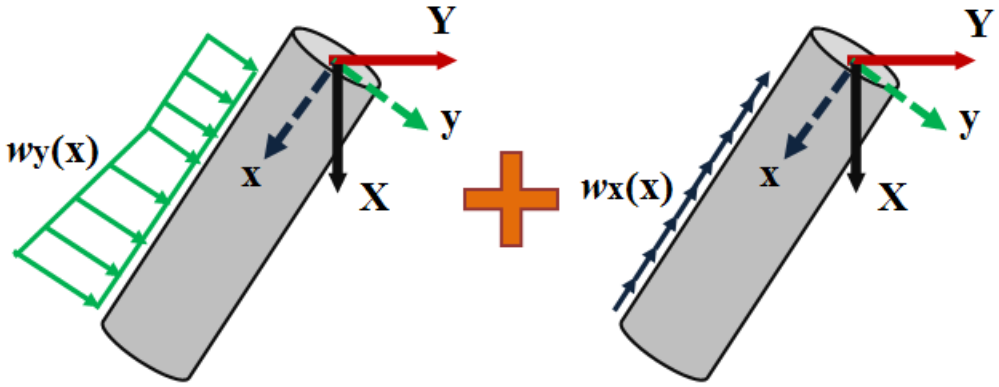
By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated loads on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (8 for 3D models and 6 for 2D models), the program will not paste any values. By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed loads data to the clipboard.

Close Button

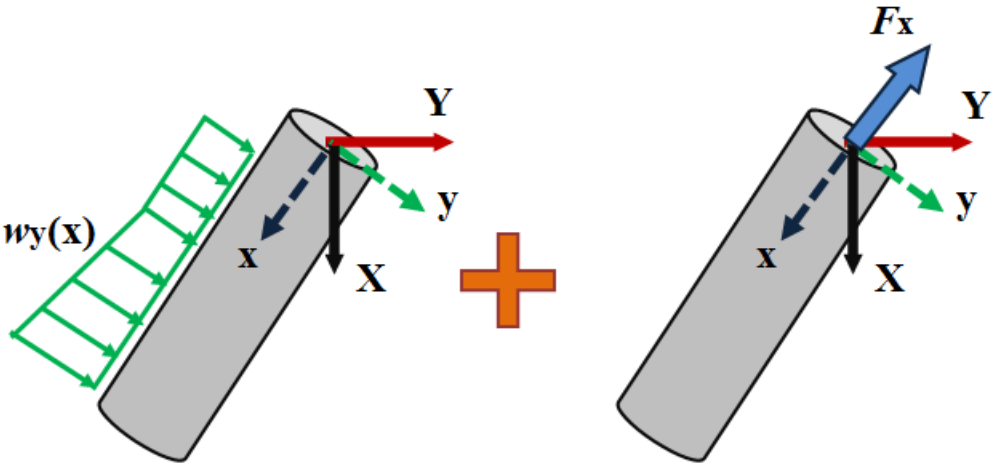
Click on the **Close** button to close the window and go back to the **Loading Definition** window.



a) Distributed Load along global axes Y



b) Equivalent load considering distributed loads along local axes y and local axes x



c) Equivalent load considering distributed load along local axes y and concentrated load at top of pile along local axes x (axial)

Figure 3.110 Equivalent distributed load in global and local axes

3.3.11.12 Soil Movement Window

In this window the user may enter the soil movement along the piles. A sample of this window is shown in Figure 3.111.

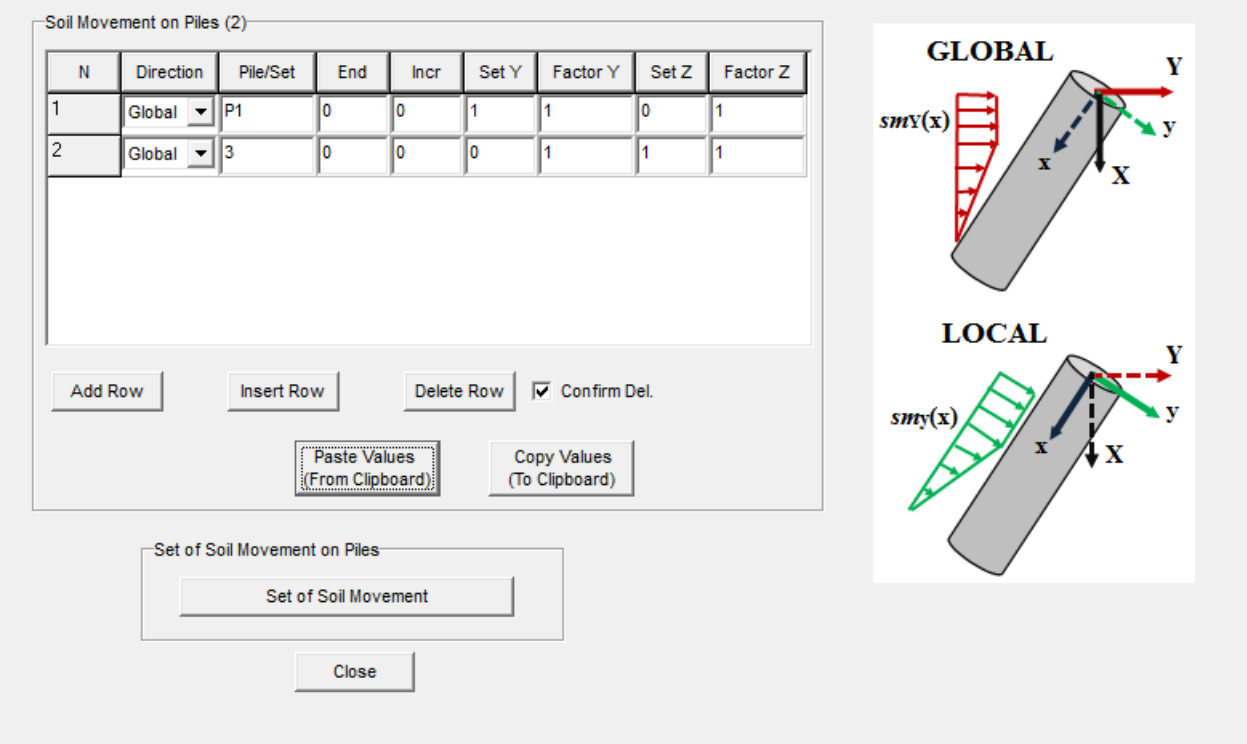


Figure 3.111 Soil Movement Window

Soil Movement on Piles

The maximum number of rows for soil movement definition is limited to 100.

The following buttons may be available based in the number of rows defined:

- **Add Row.** Add a new row at the end of the list.
- **Insert Row.** Add a new row below the current row.
- **Delete Row.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each row the following columns are available:

- **Direction**. Select the direction in which the soil movement will be applied. The user can select along the global (Y and/or Z) axes or local (y and/or z). This option is useful when considering batter piles since the global and local axes are different.
- **Pile/Set**. Enter the pile number or Pile Set to which the soil movement will be applied. It a Pile Set is entered the values for End and Incr. are ignored.
- **End**. For a sequence of piles, enter the highest pile number to which the soil movement will be applied. For a single pile enter zero in this column.
- **Incr**. For a sequence of piles, enter the increment on pile number to get the intermediate pile numbers on the sequence. For a single pile enter zero in this column.
- **Set Y**. Enter the set number of the soil movement set to be applied in the Y direction (either local or global).
- **Factor Y**. Enter the factor to be applied at the soil movement set applied in the Y direction (either local or global).
- **Set Z**. Enter the set number of the soil movement set to be applied in the Z direction (either local or global). Available for 3D analysis.
- **Factor Z**. Enter the factor to be applied at the soil movement set applied in the Z direction (either local or global). Available for 3D analysis.

Set of Soil Movement on Piles

Click on the **Set of Soil Movement** button to open the **Set of Soil Movement** window to define the distributed lateral loads along the pile.

By clicking on the **Paste Values (From Clipboard)** button, the program will paste the concentrated loads on plates from the clipboard. By using this option, the program will replace all the values already defined. If the number of columns in the clipboard is not equal to the columns needed (8 for 3D models and 6 for 2D models), the program will not paste any values. By clicking on the **Copy Values (To Clipboard)** button, the program will copy the distributed loads data to the clipboard.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

3.3.11.13 Incremental/PushOver Analysis Options Window

In this window the user may enter parameters to perform incremental/pushover analysis for the current load case. This window is not available for a model with a flexible pile cap (Flexible Cap Version).

Incremental/PushOver Analysis Options

The following options are available in this window:

- **Perform Incremental/PushOver Analysis.** If this option is checked, the program will perform incremental/pushover analysis for the current load case.
- **Number of Points.** Enter the number of points used to perform the incremental/pushover analysis. The minimum number of points is 1 and the maximum number of points is 25. The table for Factors of Loading will change according to the number of points defined.
- **Factors of Loading.** Enter the factors of loading at which the incremental/pushover analysis will be analyzed. The factors will be applied to the loads (or specify displacements) applied at the pile group.
- **Logarithmic.** Click on this option to generate loading factors based on arithmetic interpolation. The program uses the last entry in the table to generate other values.
- **Arithmetic.** Click on this option to generate loading factors based on logarithmic interpolation. The program uses the last entry in the table to generate other values.
- **Ux, Fx.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified displacement at the pile cap in the x direction or total load at the pile cap in the x direction.
- **Uy, Fy.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified displacement at the pile cap in the y direction or total load at the pile cap in the y direction.
- **Rot z, Mz.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified rotation at the pile cap in the z direction or total moment at the pile cap in the z direction.
- **Uz, Fz.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified displacement at the pile cap in the z direction or total load at the pile cap in the z direction. Only for 3D models
- **Rot y, My.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified rotation at the pile cap in the y direction or total moment at the pile cap in the y direction. Only for 3D models

- **Rot x, Mx.** If this option is checked, the program will keep constant for the incremental/pushover analysis the value assigned to specified rotation at the pile cap in the x direction or total moment at the pile cap in the x direction. Only for 3D models.
- **Distributed Loads.** If this option is checked, the program will keep constant for the incremental/pushover analysis the distributed loads applied at the piles.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

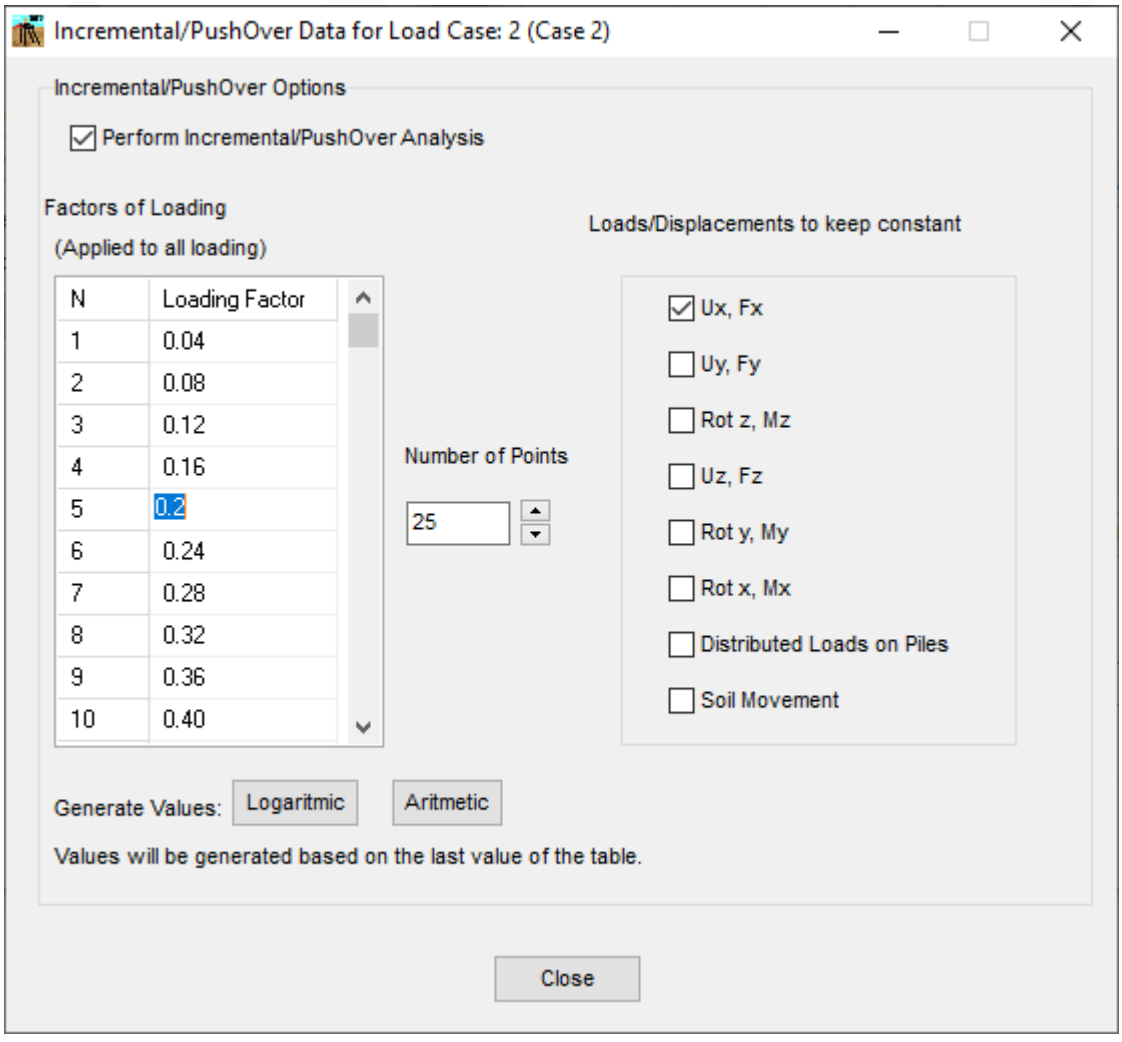


Figure 3.112 Foundation Stiffness Options Window based on Nonlinear Soil Response

3.3.11.14 Foundation Stiffness Options Window

In this window the user may enter parameters for estimating the flexibility matrix and/or the stiffness matrix of the pile group system. This window is not available for a model with a flexible pile cap (Flexible Cap Version).

The program provides two methods to estimate the stiffness matrix, the first one is based on the Nonlinear Soil Response and the second one in Equivalent Elastic Springs. The Foundations Stiffness Window depends on the method choose by the user

Method Based on Nonlinear Soil Response

A sample of this window is shown in Figure 3.113.

Foundation Stiffness Options

The following options are available in this window:

- **Generate Data for Foundation Stiffness.** If this option is checked, the program will compute the Flexibility Matrix for different levels of loads. See Figure 3.114 , Figure 3.115, and Figure 3.116 for estimation of The Flexibility Matrix in the two dimensional case.
- **Number of Points.** Enter the number of points used to compute the data for Foundation Stiffness. The minimum number of points is 1 and the maximum number of points is 10. The table for Factors of Loading will change according to the number of points defined.
- **Factors of Loading.** Enter the factors of loading at which the data for Foundation Stiffness will be generated. The factors will be applied to the concentrated loads at the pile cap. No distributed loads on piles will be included in order to estimate the Foundation Stiffness.
- **Logarithmic.** Click on this option to generate loading factors based on arithmetic interpolation. The program uses the last entry in the table to generate other values.
- **Arithmetic.** Click on this option to generate loading factors based on logarithmic interpolation. The program uses the last entry in the table to generate other values.
- **Compute Foundation Stiffness.** If this option is checked, the program will compute the Foundation Stiffness Matrix, based on the Flexibility Matrix estimated, for the different levels of loads. The Stiffness Matrix is defined as the inverse of the Flexibility Matrix.
- **Consider Only Diagonal Terms.** If this option is checked, the program will compute the Foundation Stiffness Matrix neglecting the out of diagonal terms. See Figure 3.122.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

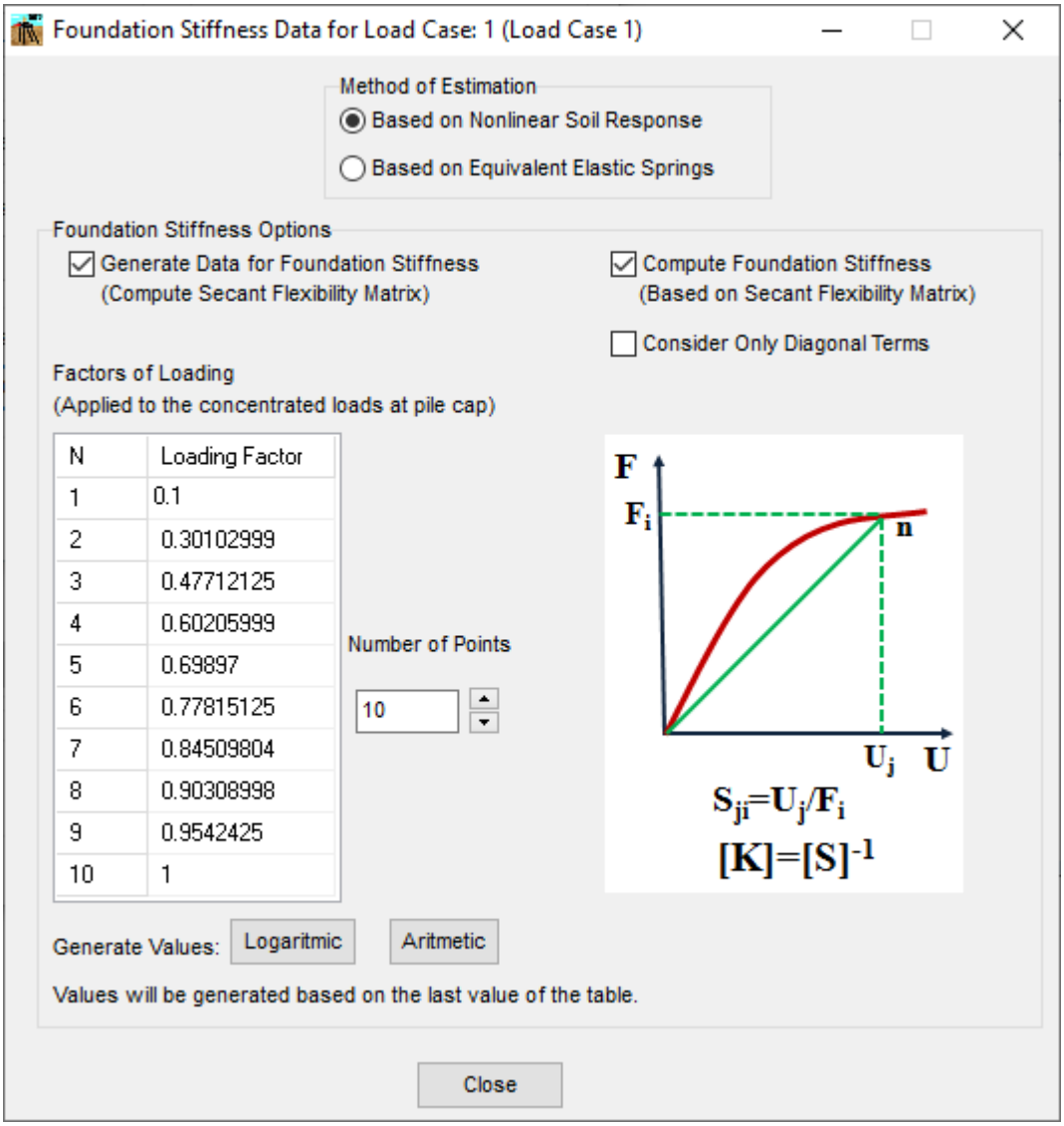


Figure 3.113 Foundation Stiffness Options Window based on Nonlinear Soil Response

Procedure to Estimate the Foundation Stiffness Based on Nonlinear Soil Response

The procedure to estimate the Foundation Stiffness Based on Nonlinear Soil Response can be summarized in the following steps:

1. **Estimation of Flexibility Coefficients**. The flexibility coefficients are estimating by performing an analysis applying a force in one degree of freedom and no force in all other degrees of freedom. By dividing the resultant displacements between the applied force, the flexibility components associated to the degree of freedom in which the load was applied the flexibility coefficients are estimated. See Figure 3.114 , Figure 3.115, and Figure 3.116 for the estimation of the flexibility components in the two dimensional case

2. **Estimation of Flexibility Matrix**. The Flexibility Matrix is obtained by filling out the flexibility coefficients estimated. The flexibility coefficients estimated while applying the force in one degree of freedom correspond to a complete row in the Flexibility Matrix. In Figure 3.117 a Flexibility Matrix for a 2-dimensional model is presented, while in Figure 3.118 the Flexibility Matrix for a 3-dimensional model is presented.
   Please note that in general the GROUP program represents the soil using nonlinear springs. For a nonlinear system, the symmetric coupling (off-diagonal) components may not have the same value, so the Flexibility Matrix obtained by the group program is non-symmetric. Only if the system is linear elastic the Flexibility Matrix will be symmetric.

3. **Estimation of the Stiffness Matrix**. The Stiffness Matrix is obtained by using the relationship between the Flexibility and Stiffness Matrices. The Stiffness Matrix is the inverse of the Flexibility Matrix.
   Please note that if the Flexibility Matrix is non-symmetric, the Stiffness Matrix is also non-symmetric.
   In Figure 3.120 a Flexibility Matrix for a 2-dimensional model is presented, while in Figure 3.121 the Flexibility Matrix for a 3-dimensional model is presented.

4. **Only Diagonal Terms**. If the user only wants to estimate a Stiffness Matrix considering only the diagonal terms, the program fills only the diagonal terms on the Flexibility Matrix, then the program inverts the Diagonal Flexibility Matrix to estimate the Diagonal Stiffness Matrix. Those matrices for the 3-dimensional case are presented in Figure 3.122.

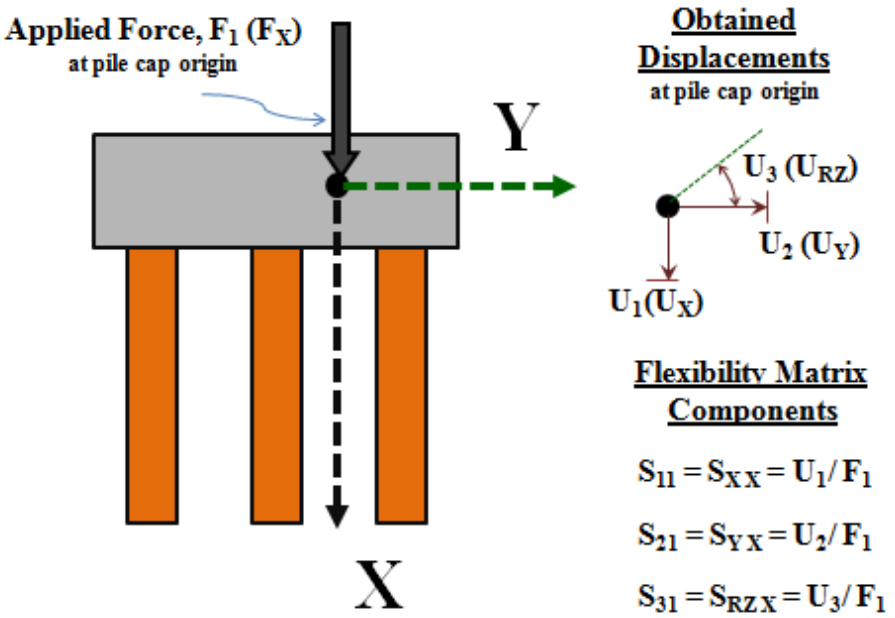


Figure 3.114 Flexibility Matrix Components due to a force in X direction (2D case)

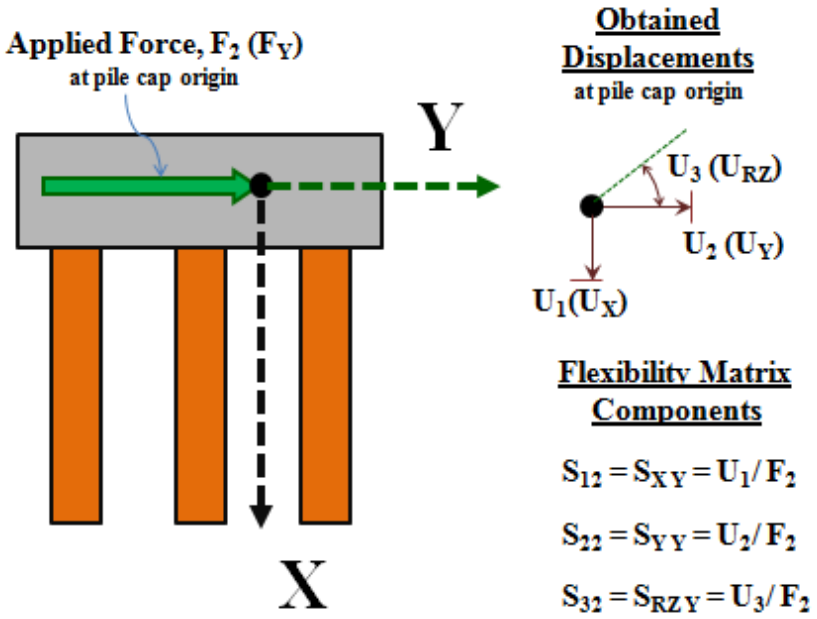


Figure 3.115 Flexibility Matrix Components due to a force in Y direction (2D case)

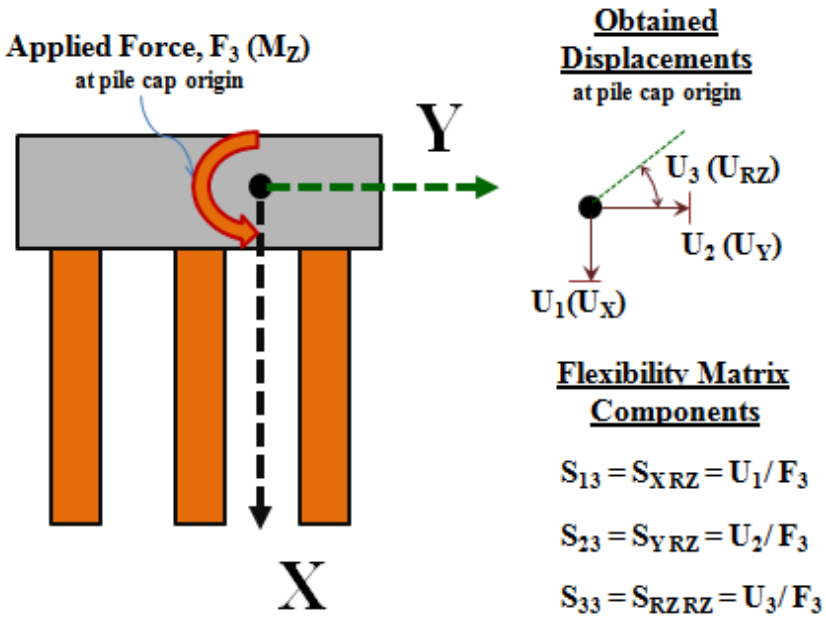


Figure 3.116 Flexibility Matrix Components due to a moment in Z direction (2D case)

$$[S]=\begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix}$$

$$[S]=\begin{bmatrix} S_{X\,X} & S_{X\,Y} & S_{X\,RZ} \\ S_{Y\,X} & S_{Y\,Y} & S_{Y\,RZ} \\ S_{RZ\,X} & S_{RZ\,Y} & S_{RZ\,RZ} \end{bmatrix}$$

Figure 3.117 Flexibility Matrix for a two-dimensional model

$$[S]=\begin{bmatrix} S_{11} & S_{12} & S_{13} & S_{14} & S_{15} & S_{16} \\ S_{21} & S_{22} & S_{23} & S_{24} & S_{25} & S_{26} \\ S_{31} & S_{32} & S_{33} & S_{34} & S_{35} & S_{36} \\ S_{41} & S_{42} & S_{43} & S_{44} & S_{45} & S_{46} \\ S_{51} & S_{52} & S_{53} & S_{54} & S_{55} & S_{56} \\ S_{61} & S_{62} & S_{63} & S_{64} & S_{65} & S_{66} \end{bmatrix}$$

$$[S]=\begin{bmatrix} S_{X\,X} & S_{X\,Y} & S_{X\,Z} & S_{X\,RX} & S_{X\,RY} & S_{X\,RZ} \\ S_{Y\,X} & S_{Y\,Y} & S_{Y\,Z} & S_{Y\,RX} & S_{Y\,RY} & S_{Y\,RZ} \\ S_{Z\,X} & S_{Z\,Y} & S_{Z\,Z} & S_{Z\,RX} & S_{Z\,RY} & S_{Z\,RZ} \\ S_{RX\,X} & S_{RX\,Y} & S_{RX\,Z} & S_{RX\,RX} & S_{RX\,RY} & S_{RX\,RZ} \\ S_{RY\,X} & S_{RY\,Y} & S_{RY\,Z} & S_{RY\,RX} & S_{RY\,RY} & S_{RY\,RZ} \\ S_{RZ\,X} & S_{RZ\,Y} & S_{RZ\,Z} & S_{RZ\,RX} & S_{RZ\,RZ} & S_{RZ\,RZ} \end{bmatrix}$$

Figure 3.118 Flexibility Matrix for a three-dimensional model

$$[K] = [S]^{-1}$$

Figure 3.119 Estimation of Stiffness Matrix from a Flexibility Matrix

$$[K] = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix}$$

$$[K] = \begin{bmatrix} K_{X\,X} & K_{X\,Y} & K_{X\,RZ} \\ K_{Y\,X} & K_{Y\,Y} & K_{Y\,RZ} \\ K_{RZ\,X} & K_{RZ\,Y} & K_{RZ\,RZ} \end{bmatrix}$$

Figure 3.120 Stiffness Matrix for a two-dimensional model

$$[K]=\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{bmatrix}$$

$$[K]=\begin{bmatrix} K_{X\,X} & K_{X\,Y} & K_{X\,Z} & K_{X\,RX} & K_{X\,RY} & K_{X\,RZ} \\ K_{Y\,X} & K_{Y\,Y} & K_{Y\,Z} & K_{Y\,RX} & K_{Y\,RY} & K_{Y\,RZ} \\ K_{Z\,X} & K_{Z\,Y} & K_{Z\,Z} & K_{Z\,RX} & K_{Z\,RY} & K_{Z\,RZ} \\ K_{RX\,X} & K_{RX\,Y} & K_{RX\,Z} & K_{RX\,RX} & K_{RX\,RY} & K_{RX\,RZ} \\ K_{RY\,X} & K_{RY\,Y} & K_{RY\,Z} & K_{RY\,RX} & K_{RY\,RY} & K_{RY\,RZ} \\ K_{RZ\,X} & K_{RZ\,Y} & K_{RZ\,Z} & K_{RZ\,RX} & K_{RZ\,RZ} & K_{RZ\,RZ} \end{bmatrix}$$

Figure 3.121 Stiffness Matrix for a three-dimensional model

$$[S]=\begin{bmatrix} S_{11} & 0 & 0 & 0 & 0 & 0 \\ 0 & S_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & S_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & S_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & S_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & S_{66} \end{bmatrix}$$

$$[K]=\begin{bmatrix} K_{11} & 0 & 0 & 0 & 0 & 0 \\ 0 & K_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & K_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & K_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & K_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & K_{66} \end{bmatrix}$$

Figure 3.122 Stiffness Matrix considering only diagonal terms for a three-dimensional model

Important Notes

The estimation of the flexibility components as well as the Stiffness Matrix may involve a numerous number of analyses, so it is recommended to be sure that the program gets a solution with the suggested level of load before running a case including the Stiffness Matrix Estimation.

If a force in a specific degree of freedom is not defined, the program will not estimate the flexibility components for that degree of freedom.

If the program can not get convergence for an analysis on a specific degree of freedom, the program will not estimate the flexibility components for that degree of freedom.

If the program does not get components for a specific degree of freedom, for numerical purposes it assumes a value of 1 for diagonal terms and 0 for off-diagonal terms. The values assumed for the program on those cases are only for completeness and to avoid numerical problems, those are not intended to replace more rigorous computations. User should proceed with caution if values are missing or there is no convergence in the solution.

Since the current procedure involves nonlinear behavior, both the Flexibility Matrix and Stiffness Matrix may be non-symmetric.

Method Based on Equivalent Elastic Springs

A sample of this window is shown in Figure 3.123.

Foundation Stiffness Options

The following options are available in this window:

- **Compute Foundation Stiffness.** If this option is checked, the program will compute the Stiffness Matrix for different levels of loads or displacements. See Figure 3.114 , Figure 3.115, and Figure 3.116 for estimation of The Stiffness Matrix in the two dimensional case.
- **Number of Points.** Enter the number of points used to compute the data for Foundation Stiffness. The minimum number of points is 1 and the maximum number of points is 10. The table for Factors of Loading will change according to the number of points defined.
- **Factors of Loading/Displacement.** Enter the factors of loading/displacement at which the Foundation Stiffness will be generated. The factors will be applied to the concentrated loads at the pile cap. No distributed loads on piles will be included in order to estimate the Foundation Stiffness.
- **Logarithmic.** Click on this option to generate loading factors based on arithmetic interpolation. The program uses the last entry in the table to generate other values.
- **Arithmetic.** Click on this option to generate loading factors based on logarithmic interpolation. The program uses the last entry in the table to generate other values.
- **Consider Axial Force in Piles.** If this option is checked, the program will include the contribution of the axial load in the Stiffness Matrix (second order effect).

- **Select level of loading.**
    - **Based on the Applied Load.** If this option is selected, the program will compute the Foundation Stiffness Matrix based on the defined load.
    - **Based on a Target Deflection.** If this option is selected, the program will scale the load until getting the target deflection. Some iterations may be needed.
    - **Scale only load associated with Deflection.** If this option is selected, the program will scale only the load associated with the target degree of freedom, all other loads will be kept constant throughout the iterations. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Ux.** Enter the value for a target deflection in the X direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Uy.** Enter the value for a target deflection in the Y direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Rot. z.** Enter the value for a target rotation in the Z direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Uz.** Enter the value for a target deflection in the Z direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Rot. x.** Enter the value for a target rotation in the X direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Rot. y.** Enter the value for a target rotation in the Y direction. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Tolerance (%).** Enter the value for specified tolerance while reaching the target deflection. This entry is available only if the option **Based on Target Deflection** is selected.
    - **Max. Iterations** Enter the maximum number of iterations allowed while reaching the target deflection. This entry is available only if the option **Based on Target Deflection** is selected.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

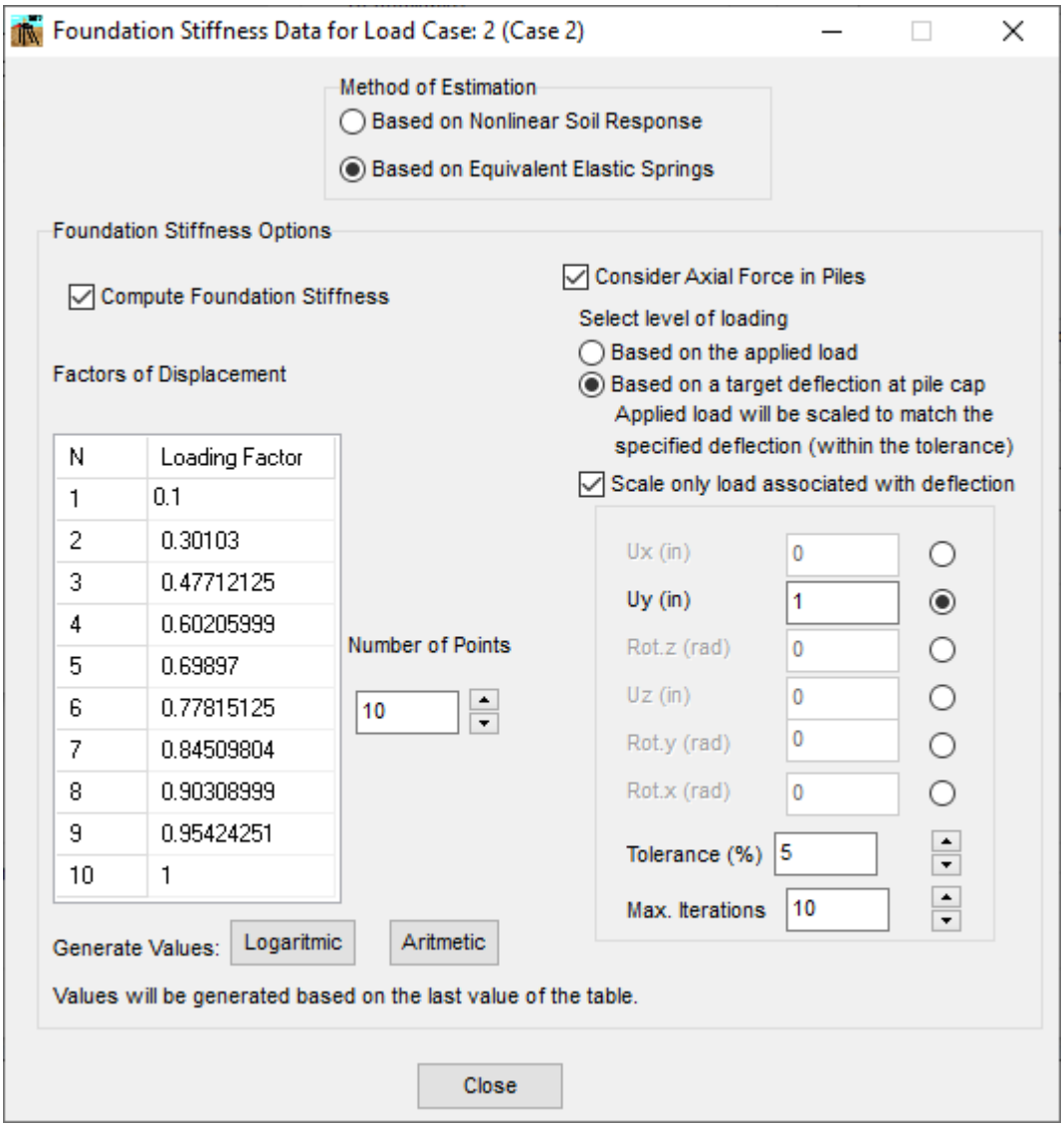


Figure 3.123 Foundation Stiffness Options Window based on Equivalent Elastic Springs

Procedure to Estimate the Foundation Stiffness Based on Equivalent Elastic Springs

The procedure to estimate the Foundation Stiffness Based on Equivalent Elastic Springs can be summarized in the following steps:

Perform analysis for a level of load or for a target displacement (may require iterations).

Estimate equivalent elastic lateral springs along the pile and axial springs at the top of pile.

Calculate equivalent elastic stiffness at each pile head by means of static condensation.

Compute equivalent elastic stiffness at pile cap using a kinematic approach.

1. **Analysis for Load or Target Displacement**. Perform an analysis for the defined level of load or for a specific target displacement. If a target displacement is desired, various analysis may be required until getting the target deflection within a tolerance.

2. **Estimation of Elastic Springs**. If a solution was obtained the soil behavior is represented using equivalent elastic springs. The equivalent elastic lateral springs along the pile are estimated by dividing the soil reaction between the lateral deflection times the increment on discretization along the pile. The equivalent elastic axial spring at the top of each pile is estimated by dividing the axial force at the top of the pile by the axial displacement at the top of the pile. For a three-dimensional model the equivalent elastic torsional spring at each pile is estimated by dividing the torsional moment at the top of the pile by the torsional rotation at the top of the pile.

3. **Static Condensation**. The lateral equivalent stiffness at each pile head is estimated based on the method of static condensation.

4. **Equivalent Stiffness for Pile Group**. The Equivalent Elastic Stiffness for the whole pile group is estimated by the kinematic method, considering the coordinates and direction of each pile.

<u>Important Notes</u>

The estimation of the Stiffness Matrix may involve a numerous number of analyses, so it is recommended to be sure that the program gets a solution with the suggested level of load before running a case including the Stiffness Matrix Estimation.

Since the current procedure involves equivalent linear springs, the Stiffness Matrix is symmetric.

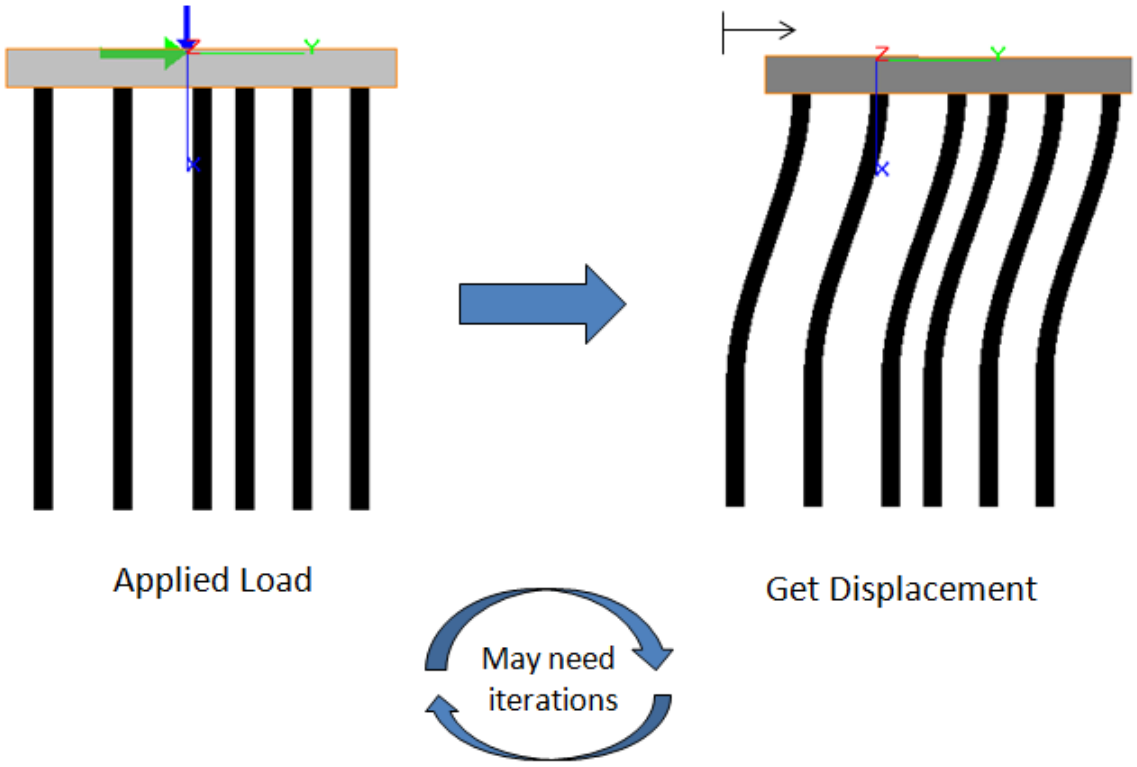


Figure 3.124 Analysis due to a load or target displacement

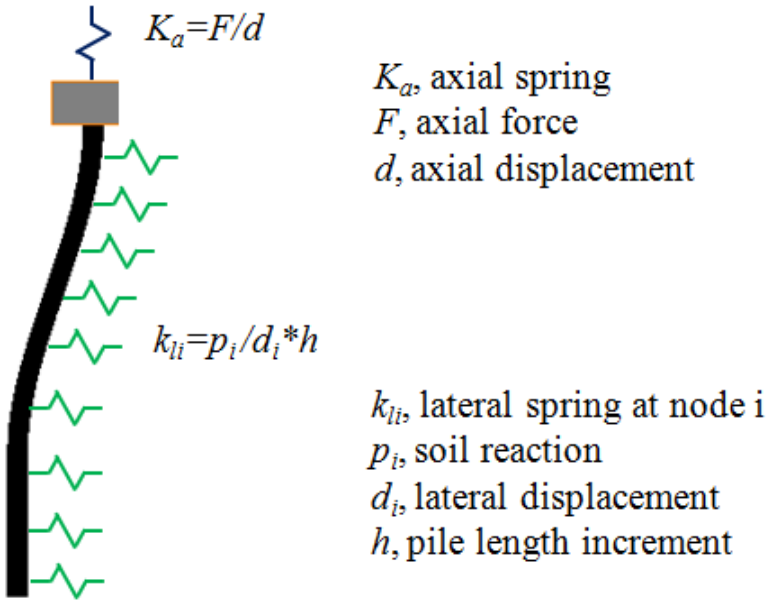
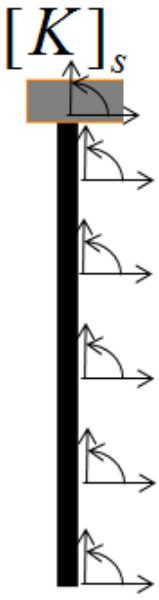


Figure 3.125 Estimation of equivalent elastic springs



$$[K][U]=[P]$$

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}\begin{bmatrix} U_1 \\ U_2 \end{bmatrix}=\begin{bmatrix} 0 \\ P \end{bmatrix}$$

All other DOFs

DOFs associated to pile head

$$\begin{bmatrix} K_{11}U_1+K_{12}U_2 \\ K_{21}U_1+K_{22}U_2 \end{bmatrix}=\begin{bmatrix} 0 \\ P \end{bmatrix}$$

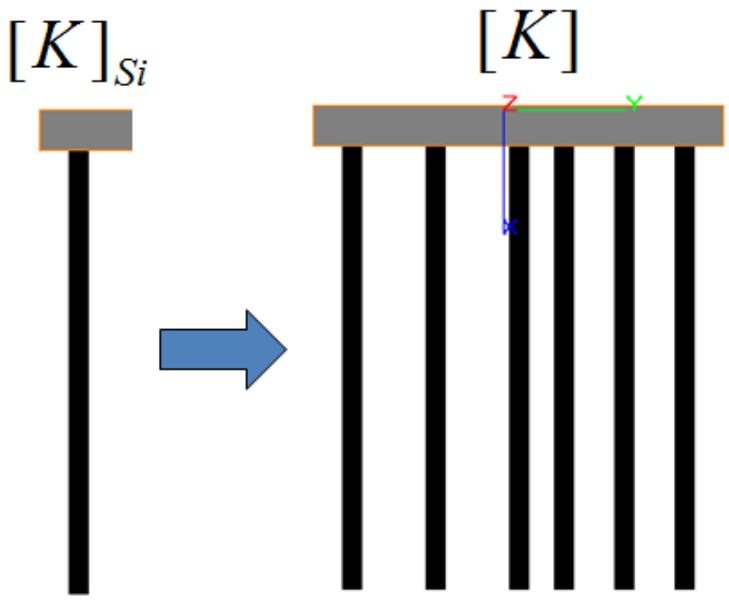
$$U_1 = K_{11}^{-1}K_{12}U_2$$

$$(K_{22}-K_{21}K_{11}^{-1}K_{12})U_2 = P$$

$$[K]_S = (K_{22}-K_{21}K_{11}^{-1}K_{12})$$

Note: Also Ks can be obtained by Gauss elimination after reducing all the DOFs not associated to the pile head.

Figure 3.126 Static Condensation to find Lateral Stiffness at Pile Head



$$[K]=\sum[K]_i$$

$$[K]_i=[R]_i^T[K]_{Si}[R]_i$$

Figure 3.127 Equivalent Elastic Stiffness for a Pile Group

3.3.11.15 Analysis Options Window

Several parameters for computation control are provided in this window. A sample of this window is shown in Figure 3.128.

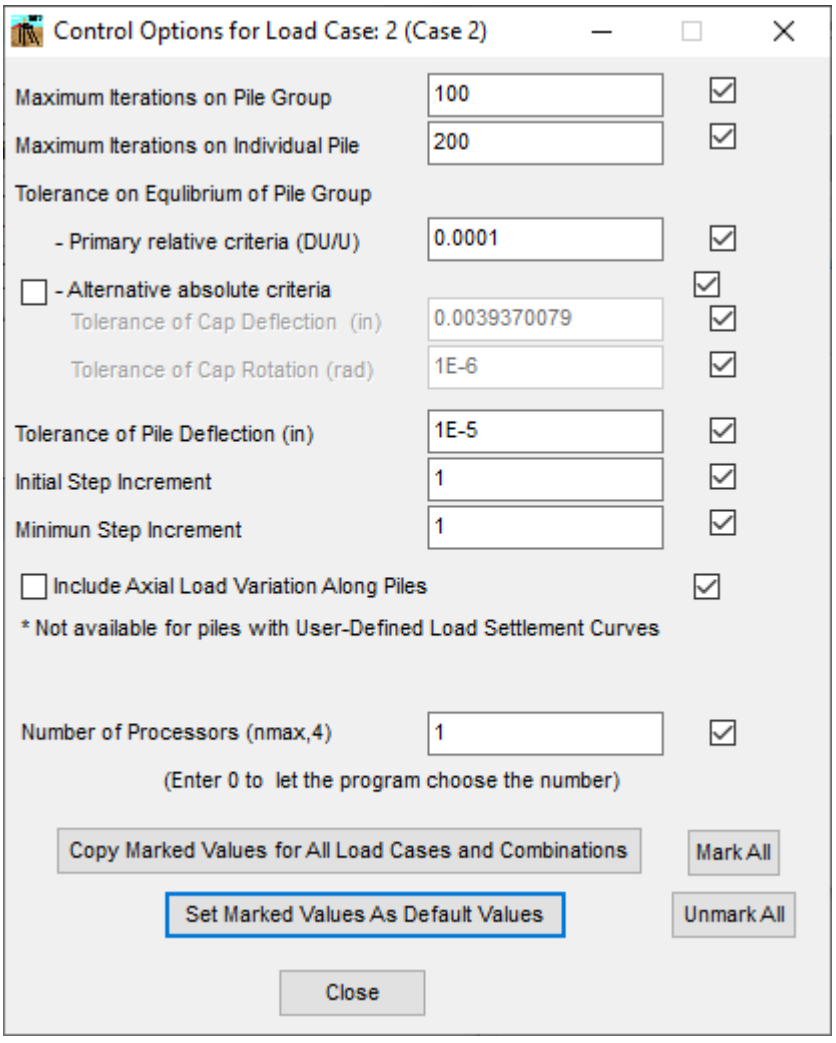


Figure 3.128 Analysis Option Window

A general description for the data needed under each computation parameter is listed below.

- *Maximum Iterations on Pile Group*. This is the maximum number of iterations allowed for the global behavior of the pile group. The default value is 100.
- *Maximum Iterations on Individual Pile*. This is the maximum number of iterations allowed for each individual pile. The default value is 100.
- *Tolerance on Equilibrium of Pile Group*.
    - *Primary relative criteria (DU/U).* This is the maximum tolerated relative error on pile cap deflections computed by two consecutive iterations for the global pile group. The default value is 0.0001.
    - *Alternative absolute criteria.* Check this option for using an absolute criteria to check the convergence on the pile cap. If checking this option, the following two values must be input:
    - *Tolerance of Cap Deflection*. This is the maximum tolerated difference on cap deflections computed by two consecutive iterations for the pile group. The default value is 0.0001 in (2.54×10^{-6} m).
    - *Tolerance of Cap Rotation*. This is the maximum tolerated difference on cap deflections computed by two consecutive iterations for the pile group. The default value is 1×10^{-6} rad.
    -
- *Tolerance on Pile Deflection*. This is the maximum tolerated difference on pile deflections computed by two consecutive iterations for individual piles. The default value is 0.0001 in (2.54×10^{-6} m).
- *Initial Step Increment*. This value is helpful if the program is not able to reach convergence by applying the total load. The user may use this value to apply the load in steps. At each step the program will apply an increment of load equal to this value times the total load. The default value is 1.
- *Minimum Step Increment*. This value is helpful if the program does not reach convergence by applying the total load. If the program does not reach convergence, the program will try to use a load increment factor equal to half of the current load increment factor. If the new increment factor is smaller than this value, the program will stop the analysis for the current load case. The default value is 1.
- *Include Axial Load Variation Along Piles*. Check this option, to use the axial load variation along piles for the analysis of the pile group. This option is only available if the program computes the axial load displacement curve internally, since the program computes the t-z and q-w curves internally. So, this option is not available in piles where a user defined axial load displacement curve is defined.

- *Number of Processors, nmax **n***. Enter the number of processors desired to be used for the analysis of the current load case. The program displays the maximum number of processors available. Leave the number as zero to let the program decided for the number of processors to be used for the analysis of the current load case. The default value is 1.

Save Marked Values As Default Values Button

Click on the **Set Marked Values As Default Values** button to override the default values by the ones marked defined in the current load case. The values not marked will not be overridden. The program will assign the default values for the definition of new load cases or combinations.

Copy Marked Values for All Loads and Combinations Button

Click on the **Copy Marked Values for All Loads and Combinations** button to override the values in all loads and combinations defined so far by the ones marked defined in the current load case. The values not marked will not be overridden.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

3.3.11.16 Output Variables Window

Define which variables for results along the piles will be printed in the output file. A sample of this window is shown in Figure 3.129.

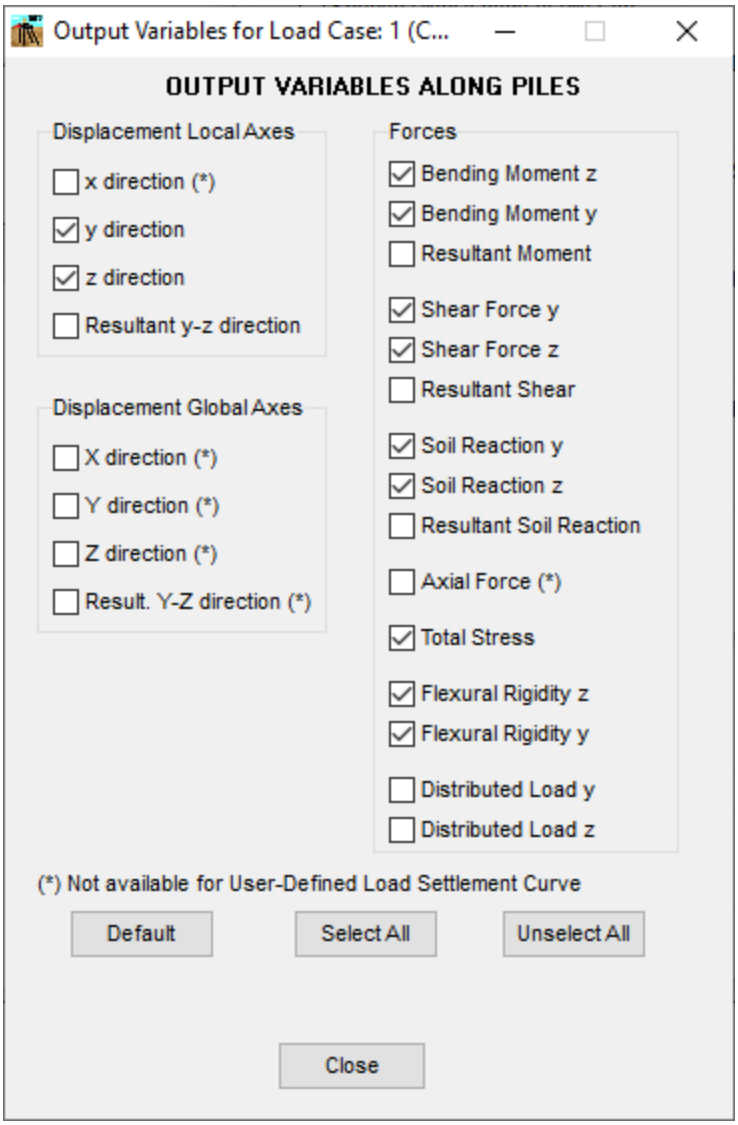


Figure 3.129 Output Variables Window

Mark the variable to be printed in the output file. Please note that some variables may not be available.

Default Button

Click on the **Default** button to mark the default values printed by default.

Select All Button

Click on the **Select All** button to mark all the variables.

Unselect All Button

Click on the **Unselect All** button to unmark all the variables.

Close Button

Click on the **Close** button to close the window and go back to the **Loading Definition** window.

3.3.11.17 Load Combinations Window

In this window the load combinations definition is carried out. A sample of this window is shown in Figure 3.130

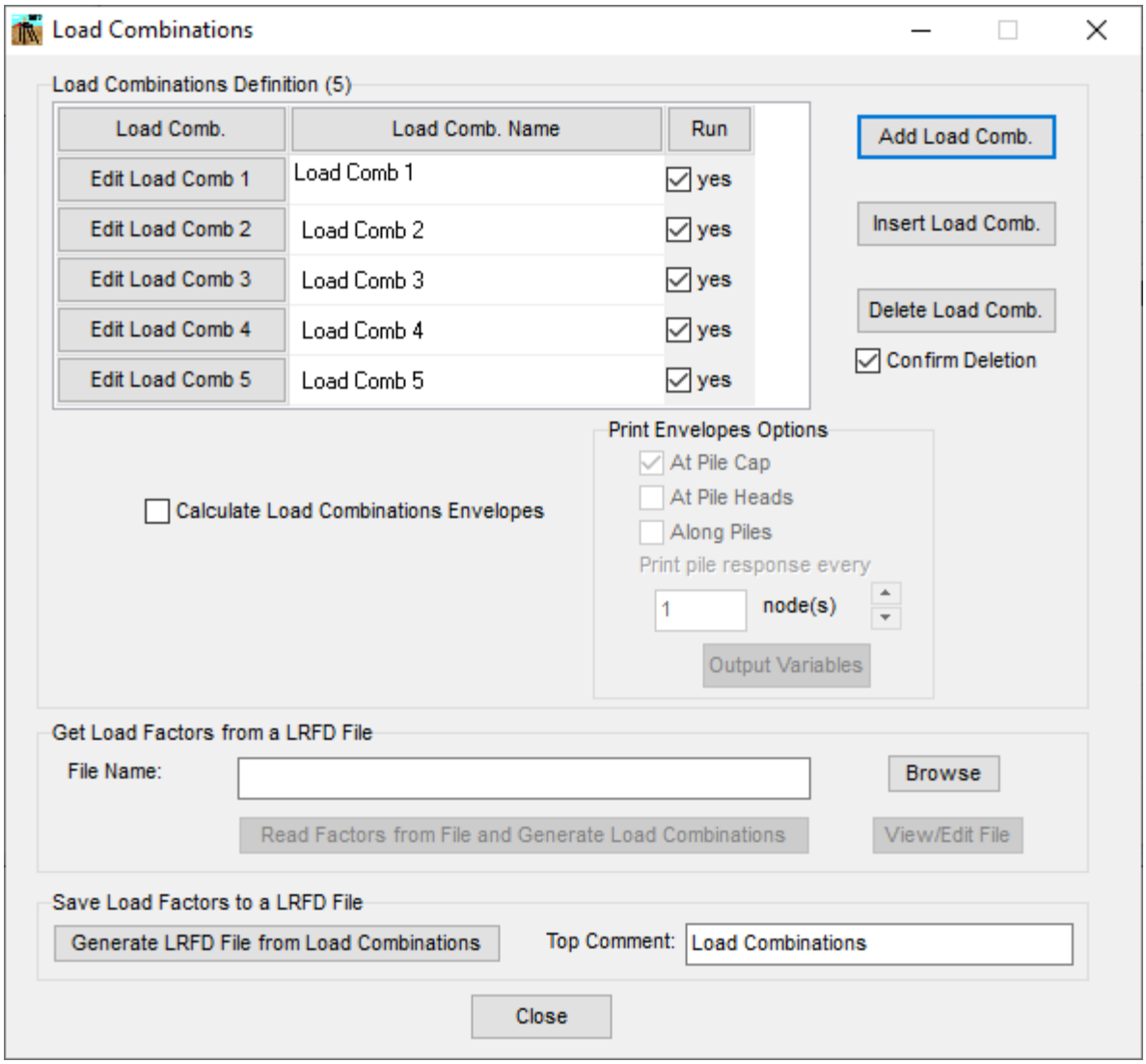


Figure 3.130 Load Combinations Window

Load Combinations Definition

The maximum number of load combinations is limited to 100

The following buttons may be available based on the number of load combinations:

- **Add Load Comb..** Add a new row at the end of the list.
- **Insert Load Comb.** Add a new row below the current row.
- **Delete Load Comb.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each load combination (row) the following columns are available:

- **Load Comb**. Click on the **Edit Load Comb** button to open the **Edit Load Combination** window for definition of load combinations factors and analysis options.
- **Load Comb Name**. In the second column enter the name of the load combination being defined.
- **Run**. Check this option to analyze the load combination.

If two or more load cases are defined the program can compute the load combination envelopes (maximum and minimum effects). Check the **Calculate Load Combinations Envelopes** option to calculate the envelopes. Also, select the option for output of the calculated envelopes.

- *Only at pile cap*. Print the equivalent load and displacements of the pile cap at the origin.
- *At pile cap and at pile heads.* Print the equivalent load and displacements of the pile cap at the origin. Also, for each pile print the pile top displacement and reactions in both pile coordinate system (local axes) and global coordinate system (global axes).
- *Pile cap, pile heads and along piles*. Print the equivalent load and displacements of the pile cap at the origin. Also, for each pile, print the pile top displacement and reactions in both pile coordinate system (local axes) and global coordinate system (global axes). In addition, for each pile, print deflection, moment, shear, soil reaction, stress and rigidity along the pile. If this option is selected the user may define the increment in nodes to print the response along the piles.

  Click on the **Output Variables** button to open the **Output Variables** window for defining which variables for results along the pile would like to be printed in the output file for the load combination envelopes.

Get Load Factors from a LRFD File

The following options are available:

- **File Name.** Enter the file name (extension .LRFD) which includes the factors for load and resistance.
- **Browse.** Click this button to navigate and select the file which includes the factors for load and resistance. If a file is selected, the file name entry will show the respective name of the file.
- **Read Factors from File and Generate Load Combinations.** Click this button to read the factors from the LRFD file and generate the load combinations. Load combinations will be generated based on the load factors read and the load case type.
- **View/Edit File.** Click this button to view and/or edit the LRFD file using the default word-processor. See Figure 3.131.

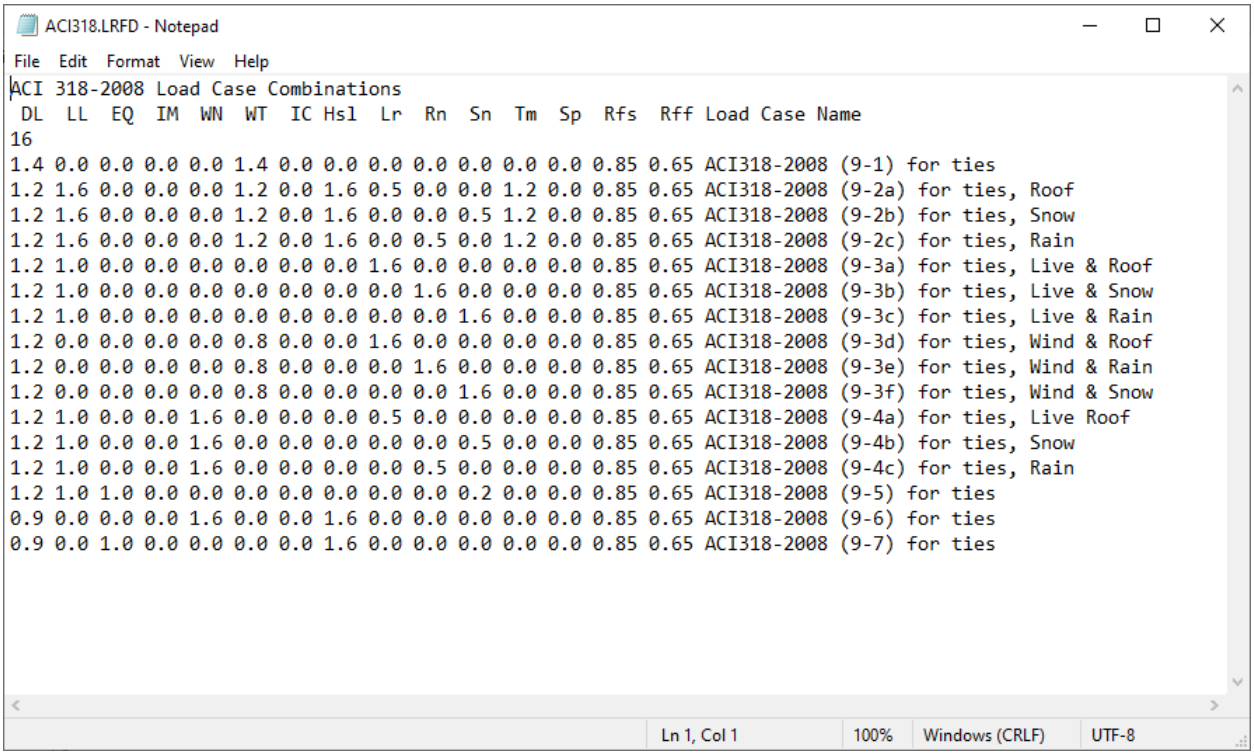


Figure 3.131 Sample of a LRFD File

Save Load Factors to a LRFD File

The following options are available:

- **Generate LRFD File from Load Combinations.** Click this button to generate an LRFD file based on the load combinations defined. The Save As window dialog will appear to save the file to a folder. Search for the folder and provide the name to save the file. If a file with the same name already exists, a confirmation message indicating so will appear. By clicking OK in the confirmation window, the input data file under the existing file name will be replaced with the current data.
- **Top Comment.** Enter the comment to be included in the first line of the generated LRFD file.

Close Button

Click on the **Close** button, to close the window and go back to the **Loading Definition** window.

3.3.11.18 Edit Load Combination Window

In this window the definition of each load combination is carried out. The load combination number and name are displayed at the top of the window. A sample of this window is shown in Figure 3.132

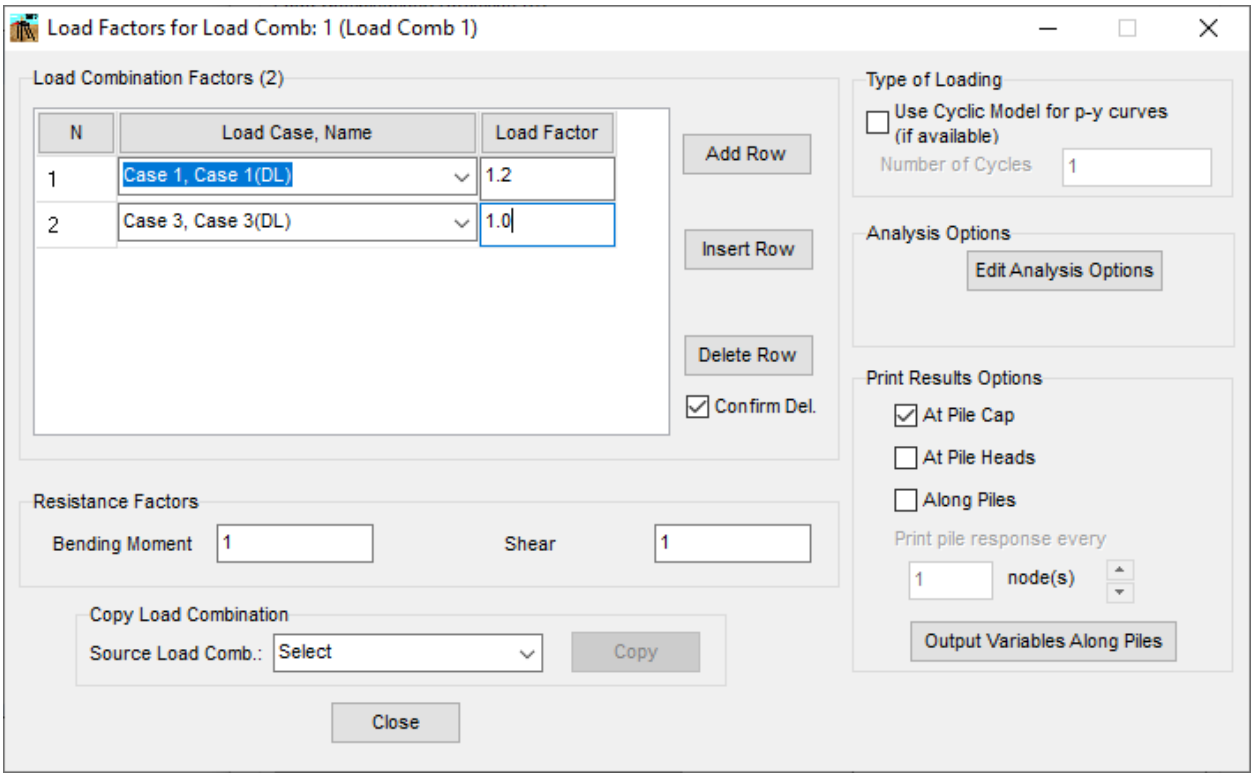


Figure 3.132 Edit Load Combination Window

Load Combination Factors

The maximum number of rows to define the load combination is limited to 100

The following buttons may be available based on the number of load cases:

- **Add Row.** To add a new row for load combination definition at the end of the list.
- **Insert Row.** To add a new row for load combination definition at the current row.
- **Delete Row.** To delete the current row.

- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a row.

For each load combination (row) the following columns are available:

- **Load Case, Name**. Select the Load Case to be part of the load combination.
- **Load Factor**. Enter the load factor associated to the load case.

Resistance Factors

Use to compute the demand capacity ratio on the load combinations:

- **Bending Moment.** Enter the resistance factor to be applied at the bending moment capacity.
- **Shear.** Enter the resistance factor to be applied at the shear capacity.

Type of Loading

Select **Use Cyclic Model por p-y curves if available** and enter the Number of Cycles to consider a Cyclic Loading. The type of loading is very important to take into account when analyzing piles under lateral loading. Further reference about the "Influence of Nature of Loading" is included in the Chapter "Soil Response Under Natural Loading" of the Technical Manual.

Cyclic loading is primarily used for storm, wave, or wind types of loading. The main effect from cyclic loading is to reduce the soil resistance based on the soil criteria for cyclic p- curves, as described in the Technical Manual.

Analysis Options

Click on the **Edit Analysis Options** button to open the **Analysis Options** window for the definition of convergence parameters for the analysis. The Analysis Option window is described under item *3.3.11.15 Analysis Options Window*

Print Results Options

Select the option for output of the results for the current load case.

- *Only at pile cap*. Print the equivalent load at origin and displacements of the pile cap at the origin.
- *At pile cap and at pile heads. Print the equivalent load at origin and displacements of the pile cap at the origin. Also, for each pile, print the pile top displacement and reactions in both pile coordinate system (local axes) and global coordinate system (global axes).*
- *Pile cap, pile heads and along piles*. Print the equivalent load at origin and displacements of the pile cap at the origin. Also, for each pile, print the pile top displacement and reactions in both pile coordinate system (local axes) and global coordinate system (global axes). In addition, for each pile, print deflection, moment, shear, soil reaction, stress and rigidity along the pile. If this

option is selected the user may define the increment in nodes to print the response along the piles.

Click on the **Output Variables** button to open the **Output Variables** window for defining which variables for results along the pile would like to be printed in the output file for the current load combination.

Copy Load Combination

The option is helpful for when there are load combinations that have few differences. So, the user may copy one into another and do the required changes as needed.

Select the load combination that is going to be the source (already defined), and then click on the **Copy** button to proceed with copying all the load combination data to the current combination.

This option is available if at least two load combinations have been defined.

Close Button

Click on the **Close** button, to close the window and go back to the **Load Combination** window.

3.3.12 Data - Soil Layers

This submenu allows the user to specify the different types of soil to be used for the automatic generation of soil resistance (*p-y*) curves and/or axial load-displacement curves and/or torque-rotation curves. GROUP 2022 will automatically generate the noted curves under default conditions. This submenu option is not needed if the user specifies external curves for soil-resistance, load-displacement and torque-rotation curves.

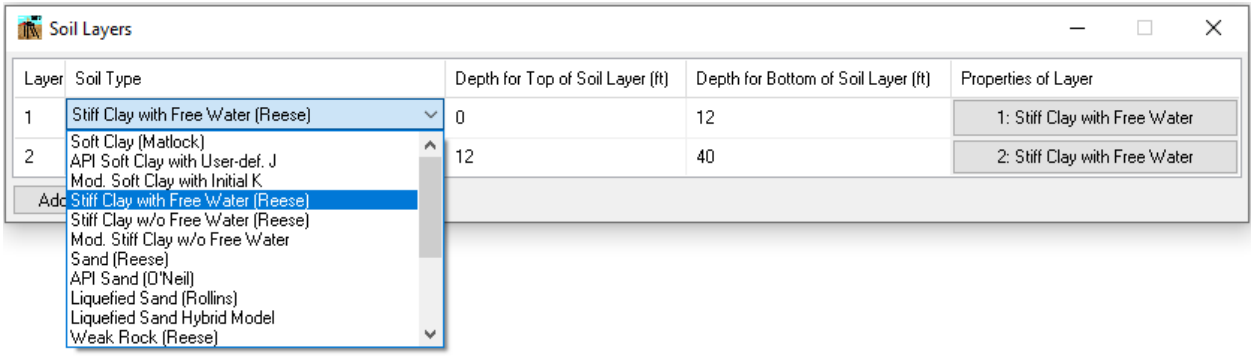


Figure 3.133 Soil Layers Window

The following buttons may be available based on the number of piles:

- **Add Row.** To add a new row for soil layer definition at the end of the list.
- **Insert Row.** To add a new row for soil layer definition at the current row.
- **Delete Row.** To delete the current row.

For each soil layer (row) the following columns are available:

- **Layer.** Sequential numbering automatically provided by the program for each soil layer. The maximum number of soil layers is limited to 20.
- **Soil Type.** There are 13 internal types of soil that the user can specify through the drop-down menu. Please refer to Chapter 4 of the *Technical Manual* for a general review of the theory of *p-y* curves in different types of soil. A review of load-displacement curves for different types of soil can be founded in Chapter 5 of the *Technical Manual*. The types of soil as well as the data input required for each particular soil are listed in Table 3.5 and Table 3.6 .
- **Depth for Top of Soil Layer.** Values for the top of the soil layer are entered according to the global origin of coordinates of the pile group. The origin of global coordinates is at the point of application of group loads, usually at the top of the pile cap. A positive value indicates a distance measured downward from the top of the pile cap. A negative value usually indicates a distance measured above the top of the pile cap (only used for permanently embedded pile caps).
- **Depth for Bottom of Soil Layer.** Values for the bottom of the soil layer are also entered according to the global origin of coordinates of the pile group. The coordinate of the bottom of each layer should always be equal to the coordinate of the top of the following layer.
- **Properties of Layer.** Click this button to enter the soil information about the layer. The parameters will depend on the soil type assigned.

Soil Type	Soil Parameters (UNITS)																				
	Effective Unit Weight	Undrained Cohesion, c	Strain Factor, ε_{50}	Friction Angle, ϕ	p-y Modulus, k	Uniaxial Comp. Strength, qu	Initial Modulus, of Rock Mass, Er	Strain Factor, k_{rm}	RQD	J	Type of Soil Test	Rock Type	SPT Blow-count	Cone Tip Res.	Residual. Strength	Poisson ratio	GSI	Dilat. /Intact Rock Mod.	Press. /Rock Mass Mod.	Ultimate Unit Site Friction	Ultimate Unit Tip Resis-tance
	(F/L^3)	(F/L^2)	(L/L)	(DEG)	(F/L^3)	(F/L^2)	(F/L^2)	(L/L)	(%)					(F/L^2)	(F/L^2)			(F/L^2)	(F/L^2)	(F/L^2)	(F/L^2)
Soft Clay (Matlock)	■	■	■																	■	■
API Soft Clay with User-def J	■	■	■							■										■	■
Mod. Soft Clay with Initial K	■	■	■		■					■										■	■
Stiff Clay with Free Water (Reese)	■	■	■		■															■	■
Stiff Clay w/o Free Water (Reese)	■	■	■																	■	■
Mod. Stiff Clay w/o Free Water	■	■	■		■															■	■
Sand (Reese)	■			■	■															■	■
API Sand (O'Neil)	■			■	■															■	■
Liquefiable Sand (Rollins)	■																			■	■
Liquefiable Sand Hybrid Model	■		■										■		■					■	■

Table 3.5 Soil Types and Parameters Needed per Soil Type

Soil Type	Soil Parameters (UNITS)																			
	Effective Unit Weight	Undrained Cohesion, c	Strain Factor, ε_{50}	Friction Angle, ϕ	p-y Modulus, k	Uniaxial Comp. Strength, qu	Initial Modulus, of Rock Mass, Er	Strain Factor, k_{rm}	RQD	J	Type of Soil Test	Rock Type	SPT Blow Count	Cone Tip Res.	Residual. Strength	Poisson ratio	GSI	Dilan. /Intact Rock Mod.	Press. /Rock Mass Mod.	Ultimate Unit Site Friction	Ultimate Unit Tip Resis-tance
	(F/L^3)	(F/L^2)	(L/L)	(DEG)	(F/L^3)	(F/L^2)	(F/L^2)	(L/L)	(%)					(F/L^2)	(F/L^2)			(F/L^2)	(F/L^2)	(F/L^2)	(F/L^2)
Weak Rock (Reese)	■					■	■	■	■											■	■
Strong Rock (Vuggy Limestone)	■					■														■	■
Piedmont Residual	■										■		■	■				■	■	■	■
Massive Rock	■					■						■				■	■	■	■	■	■
Loess	■													■						■	■
Silt (Cemented c-phi)	■	■	■	■	■															■	■
Elastic Subgrade	■																			■	■
User Defined p-y curves	■																			■	■

Table 3.6 Soil Types and Parameters Needed per Soil Type (continuation)

Description of the soil parameters as follows:

- **Effective Unit Weight.** Values of effective unit weight for each soil depth are entered in standard units of force per unit volume. The program will linearly interpolate for values between two consecutive soil depths.
- **Undrained Cohesion, c.** Values of undrained cohesion (undrained shear strength), for clays and silts at each depth are entered in standard units of force per unit area. The undrained shear strength is generally taken as half of the unconfined compressive strengths.
- **Strain Factor, ε50.** Values of strain at 50% of the maximum stress ϵ_{50} for clays at each soil depth are entered in standard units of strain. Table 3.7 provides values of ϵ_{50} for clays and Table 3.8 provides values for stiff clays.

Consistency of Clay	ϵ_{50}
Soft	0.020
Medium	0.010
Stiff	0.005

Table 3.7 Values of ε_{50} for Clays

Average Undrained Shear Strength (kPa)	ϵ_{50}
50-100	0.007
100-200	0.005
300-400	0.004

Table 3.8 Values of ε_{50} for Stiff Clays

- **Friction Angle, ϕ.** Values of the angle of internal friction, ϕ, for sands and/or silts at each soil depth are entered in degrees.
- ***p-y* Modulus, k.** This is the value for the constant k used in the equation $E_s = kx$. This constant is in units of force per cubic length and depends on the type of soil and lateral loading imposed on the pile group. It has two different uses: 1) to define the initial (maximum) value of E_s on internally generated *p-y* curves of submerged stiff clays and/or sands; and ii) to initialize the E_s array for the first iteration of pile analysis. Suggested values of the parameter k for sands are given in Table 3.9. Suggested values for the parameter k used for clays are given in Table 3.10.

Relative Density	Loose	Medium	Dense
Submerged Sand	20 lb/in^3	60 lb/in^3	125 lb/in^3
	5,430 kN/m^3	16,300 kN/m^3	33,900 kN/m^3
Sand Above WT	25 lb/in^3	90 lb/in^3	225 lb/in^3
	6,790 kN/m^3	24,430 kN/m^3	61,000 kN/m^3

Table 3.9 Soil-Modulus Parameter k for Sands

Clay Type	Avg. Undrained Shear Strength (c_u)	Static	Cyclic
Soft	1.74 – 3.47 psi	30 pci	
	250 – 500 psf		
	12 – 24 kPa	8,140 kPa/m	
Medium	3.47 – 6.94 psi	100 pci	
	500 – 1000 psf		
	24 – 48 kPa	27,150 kPa/m	
Stiff	6.94 – 13.9 psi	500 pci	200 pci
	1000 – 2000 psf		
	48 – 96 kPa	136,000 kPa/m	54,300 kPa/m
Very Stiff	13.9 – 27.8 psi	1000 pci	400 pci
	2000 – 4000 psf		
	96 – 192 kPa	271,000 kPa/m	108,500 kPa/m
Hard	27.8 – 55.6 psi	2000 pci	800 pci
	4000 – 8000 psf		
	192 – 383 kPa	543,000 kPa/m	217,000 kPa/m

Table 3.10 Soil Modulus Parameter k for Clays

- **Uniaxial Compressive Strength, qu.** This value is the unconfined compressive strength of rock at the specified depth. Values at elevations between the top and bottom elevations will be determined by linear interpolation
- **Initial Modulus of Rock Mass, E_r.** The mass modulus for weak rock should be entered for this value. This value may be measured in the field using an appropriate test or may be obtained from the product of the modulus reduction ratio and Young's modulus measured on intact rock specimens in the laboratory.
- **Strain Factor, k_{rm}.** This parameter typically ranges between 0.0005 and 0.00005. This value is requested for weak rock.
- **RQD.** This the Rock Quality Designation Value for weak Rock. The secondary structure of the weak rock is described using the Rock Quality Designation (RQD). Enter the value of RQD in percent for the weak rock
- **J.** This value is used for the API soft clay with user-defined J model and the Modified Soft Clay with Initial K model. The program does not perform error checking on the input value of J, the user should consider the advice by Matlock (refer to Chapter 4 of the Technical Manual) for selecting the J value.
- **Type of Soil Test.** This value is used for the Piedmont Residual model. There are four field testing methods: Dilatometer, Menard Pressuremeter, Standard Penetration Test and Cone Penetration Test.
- **Rock Type, Hoek Brown material index, m_i.** This value is used for the Massive Rock model. It is a dimensionless number ranging from 4 to 32 that depends on the type of rock.
- **SPT Blow Count.** This value is used for the Piedmont Residual model and the Liquified Sand Hybrid model.
- **Cone Tip Resistance.** This value is used for the Piedmont Residual model and the Loess model.
- **Residual Strength.** This value is used for the Liquified Sand Hybrid model.
- **Poisson's ratio.** This value is used for the Massive Rock model.
- **Geologic Strength Index, GSI.** This value is used for the Massive Rock model. It is a dimensionless number ranging from 0 to 100 that depends on the quality of the massive rock
- **Dilatometer Modulus.** This value is used for the Piedmont Residual model.
- **Pressuremeter Modulus.** This value is used for the Piedmont Residual model.
- **Intact Rock Modulus.** This value is used for the Massive Rock model.
- **Rock Mass Modulus.** This value is used for the Massive Rock model.
- **Ultimate unit side friction.** Values of maximum unit side friction are the maximum values of unit side resistance to axial loads, f_{max}. These values are used by the program in order to generate *t-z* curves internally. These values are ignored and should be left as zero if the user prefers to enter specific load-displacement curves. Values of maximum unit side friction should be entered in units of force per unit area. The user can also ask the program to compute an estimated value of tip resistance if zero is entered
- **Ultimate unit tip resistance.** Values of maximum unit tip resistance are the maximum values which can be developed for end bearing. These values are used by the program in order to generate *q-w* curves internally. These values are ignored and should be left as zero if the user

prefers to enter specific load-displacement curves. Values of maximum tip resistance should be entered in units of force per unit area. The user can also ask the program to compute an estimated value of tip resistance if zero is entered.

3.3.13 Data - Set of Soil Resistance (p-y) Curves

This submenu option allows the user to enter specific relationships of soil resistance (*p*) and lateral movement of the pile (*y*) for specified depths of each pile subgroup. This submenu option is inactive under normal conditions, because the program defaults to automatic generation of soil-resistance (*p-y*) curves. However, sometimes the user may prefer to enter specific *p-y* curves. These cases usually arise when local data for the soil response are available. The GROUP program cannot use a user-inputted *p-y* curve to calculate resistances against embedded pile caps.

To activate this submenu option, the user must check **Enter User-Defined Soil-Resistance (p-y) Curves** under the **Options** menu.

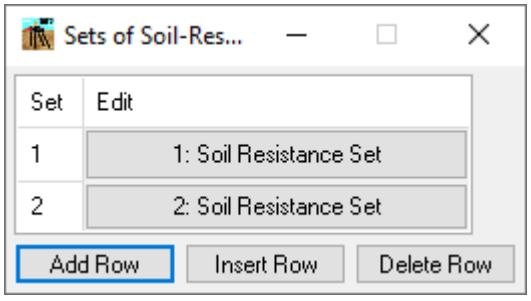


Figure 3.134 Set of Soil-Resistance Window

This window allows the user to manage the sets of soil-resistance curves. The user is allowed a maximum of fifty sets. Clicking on the **Soil Resistance Set** button in the **Edit** column allows the user to manage the curves within the set.

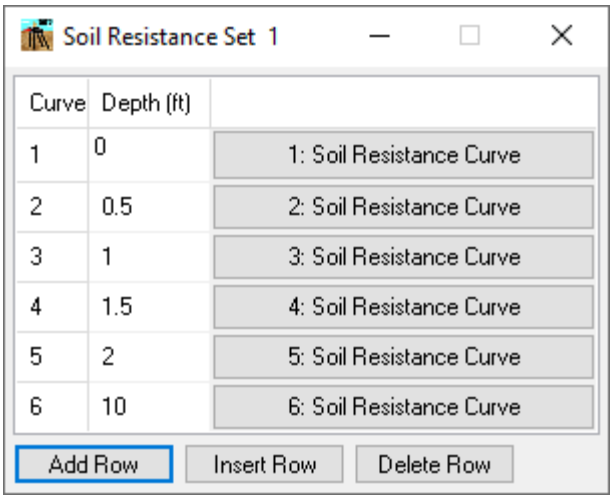


Figure 3.135 Soil-Resistance Set Window

Here the user should specify the depth of the soil resistance curves, each of which should represent the distance from the top of the pile to the depth where the *p-y* curve is being specified. Inputted *p-y* curves are linearly interpolated between two consecutive values of depth. The maximum number of depths in each soil-resistance set curve is limited to 20. Clicking on the **Soil Resistance Curve** button in the next column allows the user to edit the individual curve. The other curve sets (Axial Load-Displacement, Torque-Rotation, Distributed Lateral Loads, Nonlinear EI) are managed similarly.

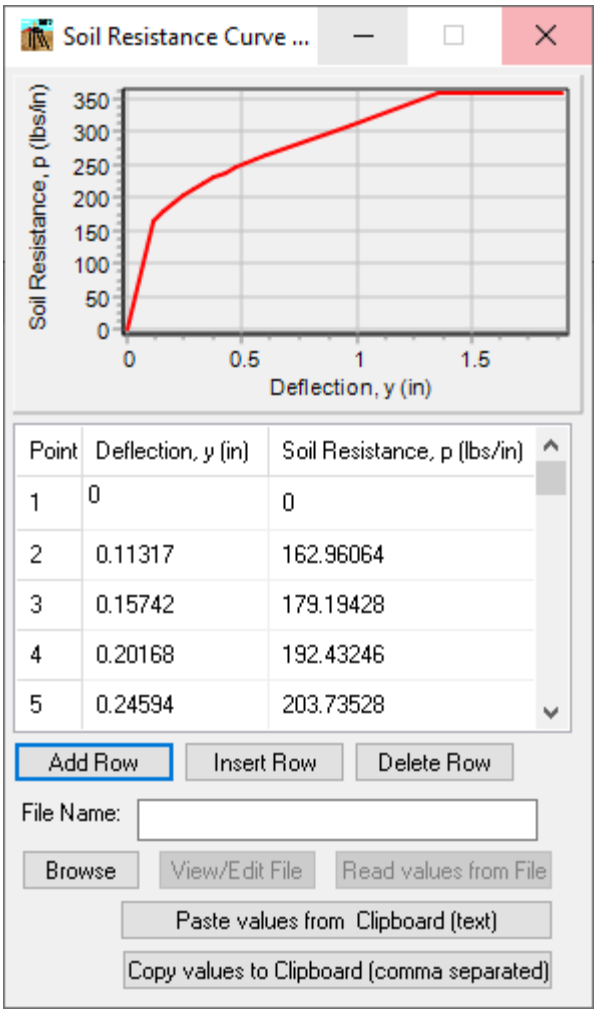


Figure 3.136 Soil-Resistance Curve Window

The following buttons may be available based on the number of piles:

- **Add Row.** To add a new row for definition of a p-y point at the end of the list.
- **Insert Row.** To add a new row for definition of a p-y point at the current row.
- **Delete Row.** To delete the current row.

For each p-y point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the soil-resistance curve being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each soil-resistance (*p-y*) curve is limited to 25. This column is also found in the other curve sets.

- **Soil Resistance, p.** Values of load transfer must be inputted in units of load per unit length. For reference, a review of the theory of "Soil Response for Lateral Loading" is included in Chapter 4 of the *Technical Manual*.
- **Deflection, y.** Values of lateral movement must be inputted in units of length. For reference, a review of the theory of "Soil Response for Lateral Loading" is included in Chapter 4 of the *Technical Manual*.

As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curve from a text file or from the clipboard (memory).
To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the p-y points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the p-y points will be populated based on the input data read and the chart will be updated.

3.3.14 Data - Sets of Axial Load-Displacement Curves

This submenu option allows the user to enter specific relationships of axial load versus displacement for the pile head of each pile subgroup. This submenu option is inactive under normal conditions, because the program defaults to automatic generation of load-displacement curves. However, sometimes the user may prefer to enter curves for axial load versus displacement at the pile head. These cases usually arise when data from loading tests are available.

To activate this submenu option, the user must check **Enter User-Defined Load-Displacement Curves** under the **Options** menu.

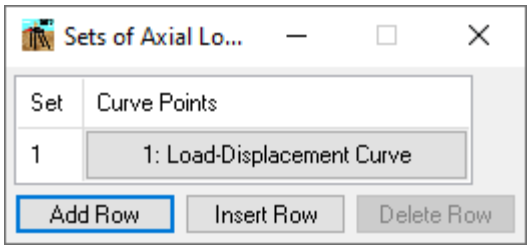


Figure 3.137 Set of Axial Load vs. Displacement Window

This window allows the user to manage the sets of axial load-displacement curves. The user is allowed a maximum of fifty sets. Clicking on the **Load-Displacement Curve** button allows the user to edit the individual curve. The other curve sets (Soil Resistance, Torque-Rotation, Distributed Lateral Loads, Nonlinear EI) are managed similarly.

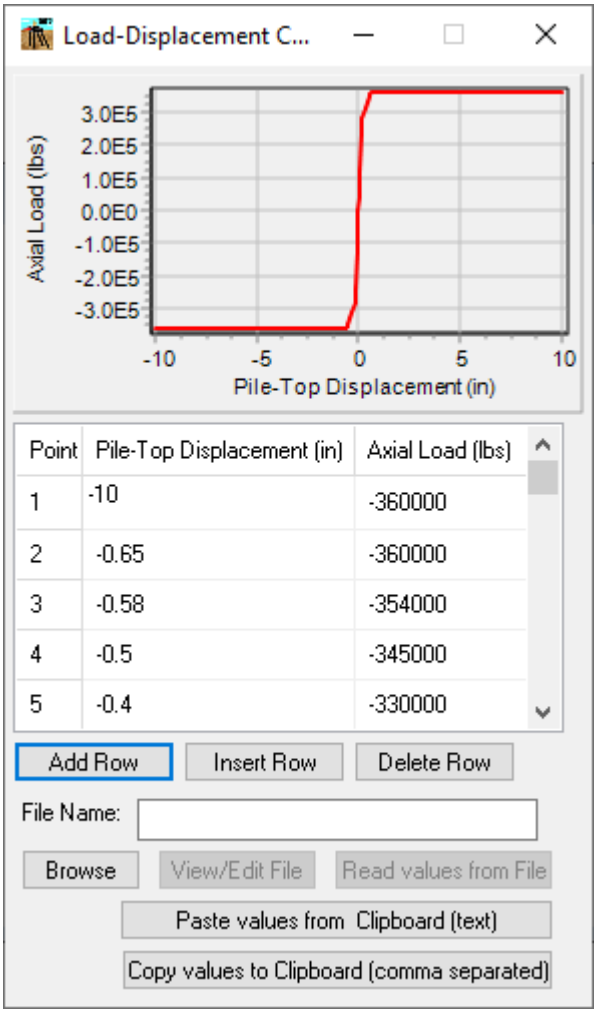


Figure 3.138 Axial Load-Displacement Curve Window

The **Axial Load-Displacement Curve** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each axial load-displacement point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the axial load-displacement curve being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each axial load-displacement curve is limited to 25.
- **Axial Load.** Values of axial load are entered in units of force with the positive direction being downwards (using the same sign convention as for the pile-group loads). For reference, a review of the theory of "Soil Response for Axial Loading" is included in Chapter 5 of the *Technical Manual*.
- **Pile-Top Displacement.** Values of displacement at pile top are entered in units of length with downward movements as positive. For reference, a review of the theory of "Soil Response for Axial Loading" is included in Chapter 5 of the *Technical Manual*.

As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curve from a text file or from the clipboard (memory).

To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.15 Data - Sets of Torque-Rotation Curves

This submenu option allows the user to enter specific relationships of torsional load versus rotation for the pile head of each pile for 3-dimensional models only. This submenu option is inactive under normal conditions, because the program defaults to automatic generation of torque-rotation curves. However, sometimes the user may prefer to enter curves for torque versus rotation at the pile head.

To activate this submenu option, the user must check **Enter User-Defined Torque-Rotation Curves** under the **Options** menu.

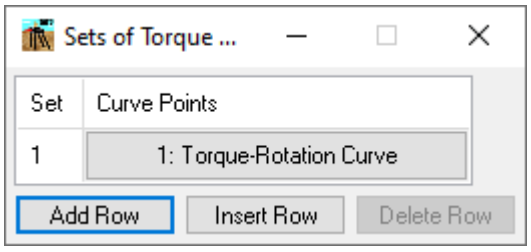


Figure 3.139 Set of Torque vs. Rotation Window

This window allows the user to manage the sets of torque-rotation curves. The user is allowed a maximum of fifty sets. Clicking on the **Torque-Rotation Curve** button allows the user to edit the individual curve. The other curve sets (Soil Resistance, Axial Load-Displacement, Distributed Lateral Loads, Nonlinear EI) are managed similarly.

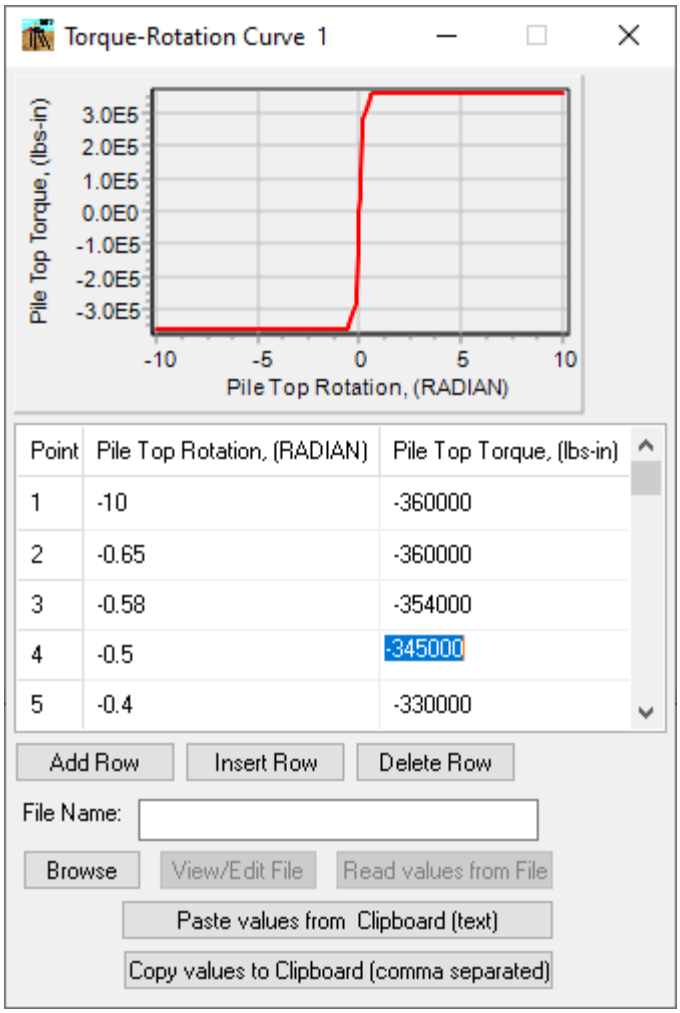


Figure 3.140 Torque-Rotation Curve Window

The **Torque-Rotation Curve** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each torque-rotation point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the torque-rotation curve being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each torque-rotation curve is limited to 25.
- **Pile Top Torque.** Values of torque are entered in units of force times length. For reference, a review of the theory of "Isolated Piles Subjected to Torque" is included in Chapter 3 of the *Technical Manual*.

- **Rotation:** Values of rotation at pile top are entered in units of radians. For reference, a review of the theory of "Isolated Piles Subjected to Torque" is included in Chapter 3 of the *Technical Manual*.

  As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

  The program also allows copying the entire curve from a text file or from the clipboard (memory).
  To copy the curve points from a file:
- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

  To copy the curve points from the clipboard:
- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.16 Data - Sets of Nonlinear EI

This submenu option allows the user to enter nonlinear moment-curvature data for specified pile sections. This submenu option is inactive under normal conditions. However, sometimes the user may want to enter nonlinear moment curvature data.

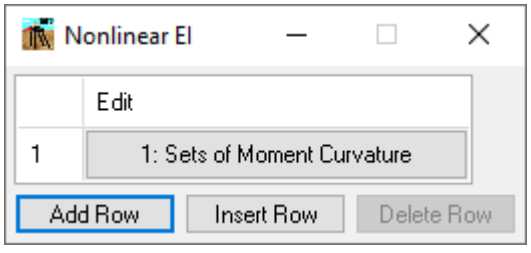


Figure 3.141 Set of Nonlinear EI Window

This window allows the user to manage the sets of nonlinear moment curvature curves. The user is allowed a maximum of fifty sets. Clicking on the **Sets of Moment Curvature** button in the **Edit** column allows the user to manage the curves within the set.

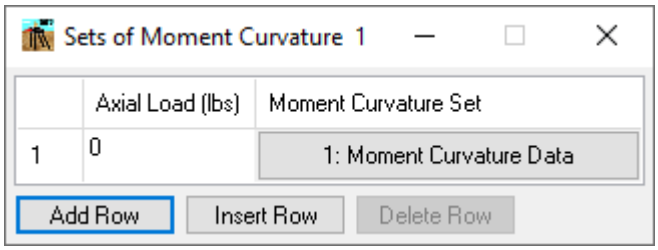


Figure 3.142 Moment Curvature Set Window

Here the user should specify the axial load of the moment curvature curve. Inputted moment curvature curves are linearly interpolated between two consecutive values of axial load. The maximum number of axial loads in each moment curvature set curve is limited to 20. Clicking on the **Moment Curvature Data** button in the next column allows the user to edit the individual curve. The other curve sets (Soil Resistance, Axial Load-Displacement, Torque-Rotation, Distributed Lateral Loads) are managed similarly.

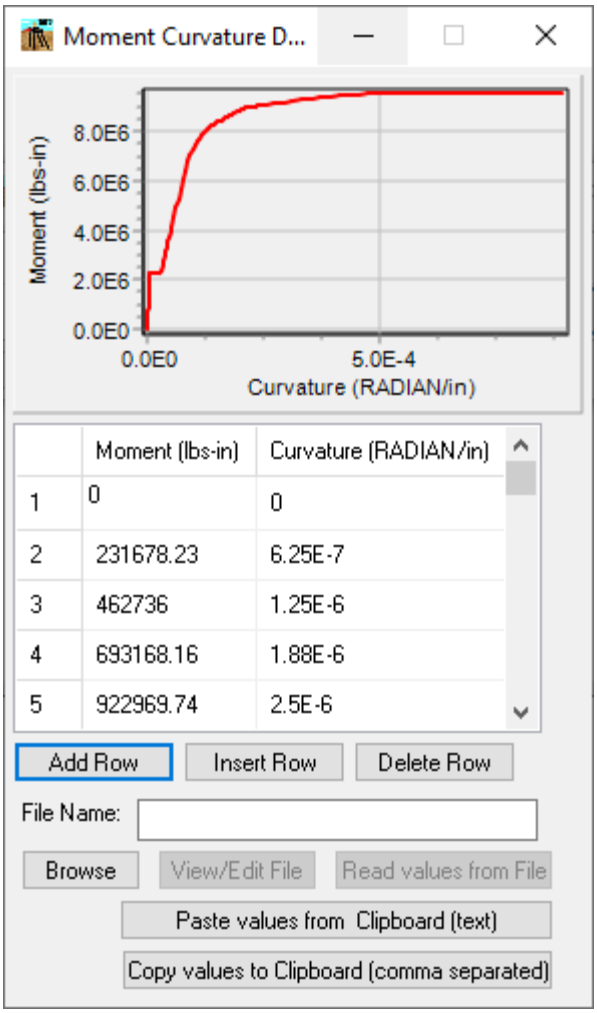


Figure 3.143 Moment Curvature Data Window

The **Moment Curvature Data** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each moment curvature point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the moment curvature data being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each moment curvature curve is limited to 25.
- **Moment.** Values of bending moment are entered in units of force times length.
- **Curvature.** Values of curvature are entered in units of radians per length.

As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curve from a text file or from the clipboard (memory).

To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.17 Data - Sets of Red. Factor Lat. Load-Along Depth

This submenu option allows the user to enter factors for lateral load along depth for each pile subgroup. This submenu option is inactive under normal conditions. However, sometimes the user would like to enter the factors along depth to simulate reduction on the soil response for various reasons.

To activate this submenu option, the user must check **Reduction Factors for Lateral Load along Depth** under the **Options** menu.

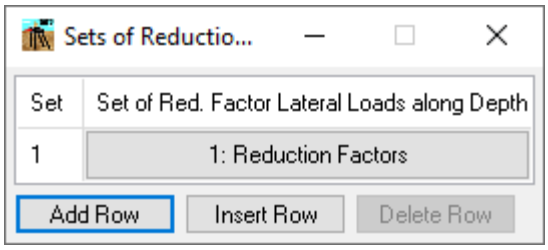


Figure 3.144 Set of Reduction Factors along Depth Window

This window allows the user to manage the sets of reduction factor for lateral load along depth. The user is allowed a maximum of one hundred sets. Clicking on the **Reduction Factors** button allows the user to edit the individual curve. The other curve sets (Soil Resistance, Torque-Rotation, Distributed Lateral Loads, Nonlinear EI) are managed similarly.

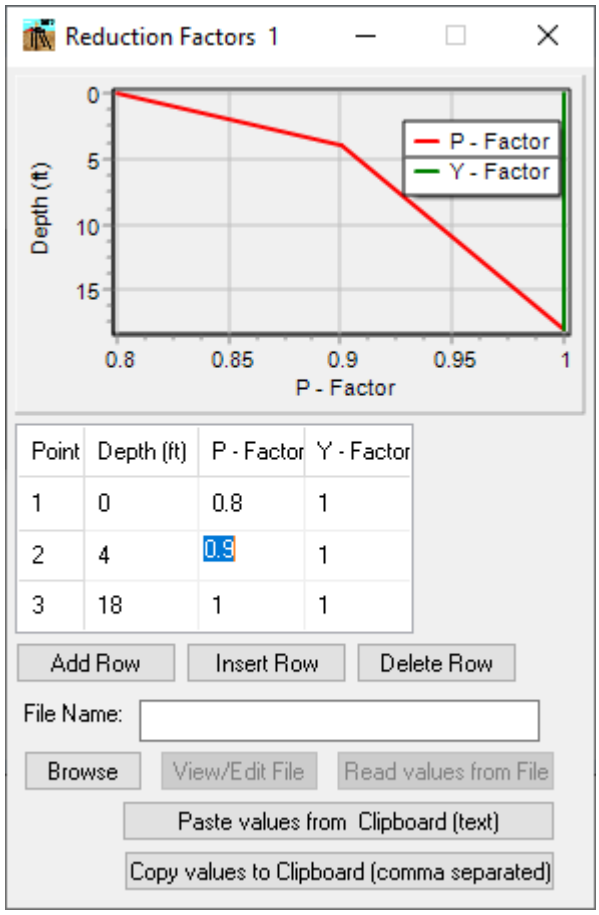


Figure 3.145 Reduction Factors Window

The **Reduction Factors** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the curve being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each reduction factor curve is limited to 50.
- **Depth.** Value of depth along pile at which the reduction factors are defined.
- **P-Factor.** Value of the reduction factor for the load at the specified depths.
- **Y-Factor.** Value of the reduction factor for displacement at the specified depths.

As the user enters the values, the curves will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curves from a text file or from the clipboard (memory).

To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curves. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curvse, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.18 Data - Sets of Soil Movement

This submenu option allows the user to enter distributed loads along the piles to be used for both 2-dimensional and 3-dimensional models. This submenu option is active. This window can be accessed also from the **Soil Movement** window

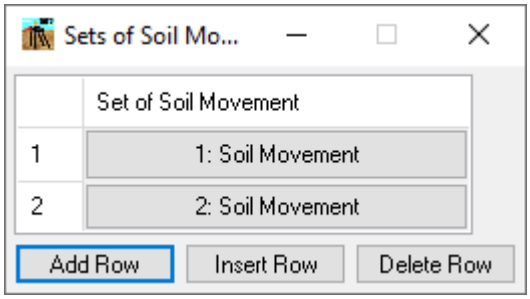


Figure 3.146 Set of Soil Movement Window

This window allows the user to manage the sets of soil movement. The user is allowed a maximum of two thousand sets. Clicking on the **Soil Movement** button allows the user to edit the individual distributed load. The other curve sets (Soil Resistance, Axial Load-Displacement, Torque-Rotation, Distributed Loads, Nonlinear EI) are managed similarly.

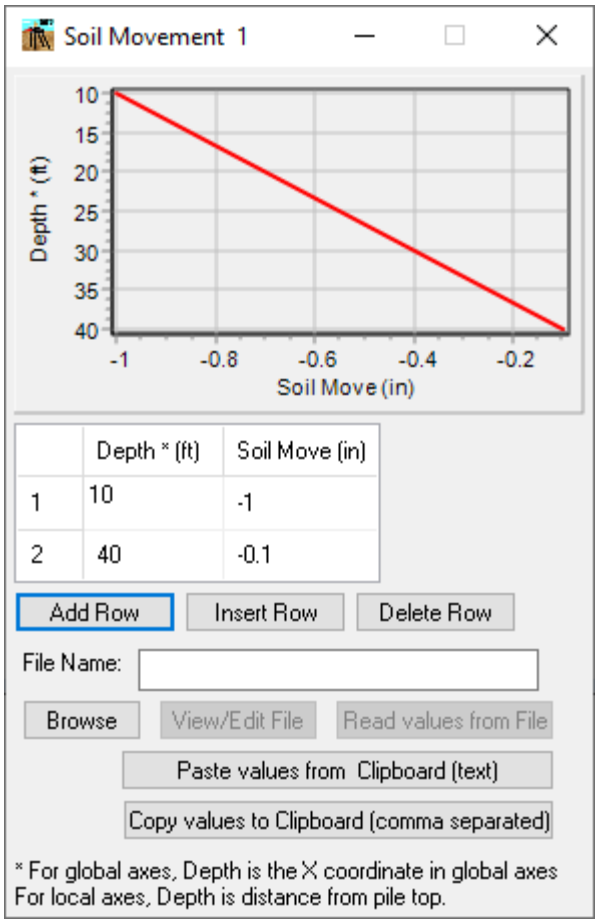


Figure 3.147 Soil Movement Window

The **Soil Movement** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each distributed load point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the soil movement data being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each distributed load set is limited to 25.
- **Depth.** Values of depths are entered in units of length. Each depth should represent the distance from the top of the pile to the depth where the soil movement is being specified.
- **Soil Move.** Values of soil movement entered in units of force per unit length.

As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curve from a text file or from the clipboard (memory).

To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.19 Data - Sets of Distributed Loads

This submenu option allows the user to enter distributed loads along the piles to be used for both 2-dimensional and 3-dimensional models. This submenu option is active. This window can be accessed also from the **Distributed Lateral Load** window

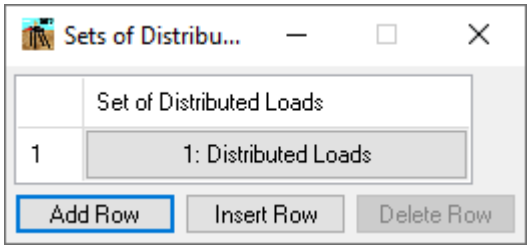


Figure 3.148 Set of Distributed Lateral Loads Window

This window allows the user to manage the sets of distributed lateral loads. The user is allowed a maximum of two thousand sets. Clicking on the **Distributed Loads** button allows the user to edit the

individual distributed load. The other curve sets (Soil Resistance, Axial Load-Displacement, Torque-Rotation, Nonlinear EI) are managed similarly.

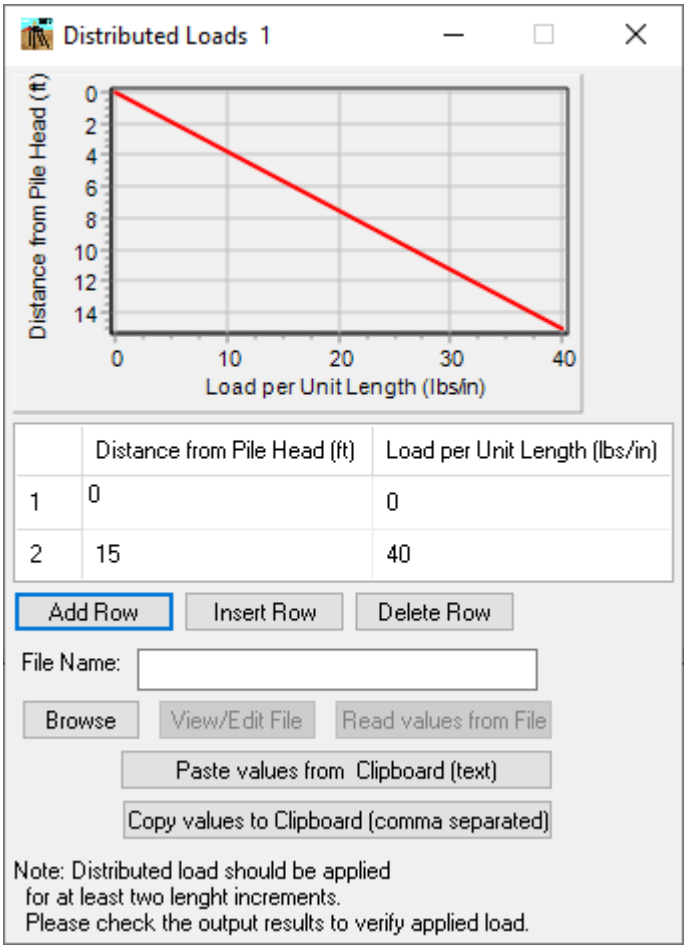


Figure 3.149 Distributed Load Window

The **Distributed Load** window is very similar to the **Soil Resistance Curve** window, so refer to the description of the Soil Resistance Window for operations. Specific features for this window are explained below:

For each distributed load point (row) the following columns are available:

- **Point.** This is a sequential number that is provided to each point of the distributed load data being defined. This number is automatically provided by the program as new rows are added. The maximum number of points in each distributed load set is limited to 25.

- **Depth.** Values of depths are entered in units of length. Each depth should represent the distance from the top of the pile to the depth where the load is being specified.
- **Load per Unit Depth.** Values of distributed load per unit depth are entered in units of force per unit length.

As the user enters the values, the curve will be displayed in the graph at the top of the window. If the chart is not updated since the last change, the user may need to move the cursor to another window to trigger window redraw.

The program also allows copying the entire curve from a text file or from the clipboard (memory).

To copy the curve points from a file:

- Click on **Browse** button to find the file with the information about the curve. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the points will be populated based on the input data read and the chart will be updated.

To copy the curve points from the clipboard:

- Copy the curve points to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the points will be populated based on the input data read and the chart will be updated.

3.3.20 Data- Sets of Data to Print p-y Curves

This submenu option allows the user to enter information for printing the internally generated p-y curves. This submenu option is inactive under normal conditions.

Only internally generated p-y curves may be printed by the program. Printed p-y curves are calculated expressly for the specified depths only for verification purposes. The number of depths that the user selects has no effects on the accuracy of the results because the p-y curves are calculated at each pile increment for internal computations.

In the output data file and graphics, the labels of depths of p-y curves are printed as depths below ground line. These labels will thus be different than requested depths of printout when the ground line is not at the same level as the origin of the global coordinate system.

If the user selects Pile-Cap Embedded under Pile Cap Options and also request to print an internally generated p-y curve within the height of the pile cap, then two different p-y curves will be printed: one corresponding to the p-y curve for the pile cap and the other corresponding to the p-y curve for the specified pile subgroup. The difference can easily be identified in the pile diameters. In addition, the p-y curve corresponding to the pile cap will usually be a stiffer curve.

To activate this submenu option, the user must check **Print p-y Curves** under the **Options** menu.

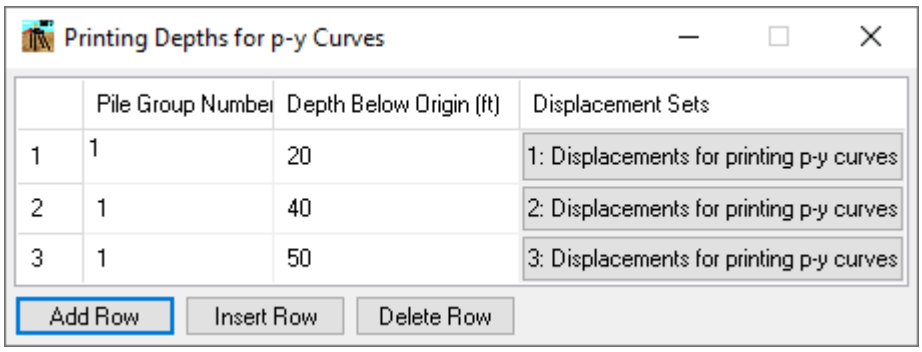


Figure 3.150 Printing Depths for p-y Curves Window

The following buttons may be available based on the number of desired p-y curves to print:

- **Add Row.** To add a new row for definition of a p-y print at the end of the list.
- **Insert Row.** To add a new row for definition of a p-y print at the current row.
- **Delete Row.** To delete the current row.

For each p-y print (row) the following columns are available:

- **Number.** This is a sequential number that is provided to each point of a p-y print being defined. This number is automatically provided by the program as new rows are added. The maximum number of p-y curves to print is limited to 50.
- **Pile Group Number.** Enter the number of the pile group associated with the p-y curve to print.
- **Depth Below Origin.** Values of depth must be inputted in units of length. Values for depth should be entered according to the global coordinate system. Values may be positive if downward from the origin or negative for soils above the pile cap.

- **Displacement Sets.** Click on the **Displacements for Printing p-y curves** button to enter the values of displacements at which the p-y curves are desired. GROUP will use default values if no displacements are entered.

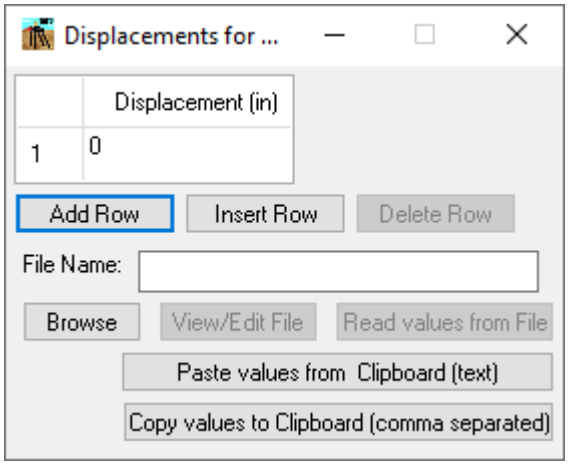


Figure 3.151 Displacements for p-y Curves Window

The following buttons may be available based on the number of displacements desired to print:

- **Add Row.** To add a new row for definition of a displacement at the end of the list.
- **Insert Row.** To add a new row for definition of a displacement at the current row.
- **Delete Row.** To delete the current row.

For each displacement (row) the following columns are available:

- **Number.** This is a sequential number that is provided to each displacement being defined. This number is automatically provided by the program as new rows are added. The maximum number of displacements to print is limited to 25.
- **Displacements.** Values of displacements must be inputted in units of length.

The program also allows copying the entire displacement list from a text file or from the clipboard (memory).
To copy the displacement list from a file:

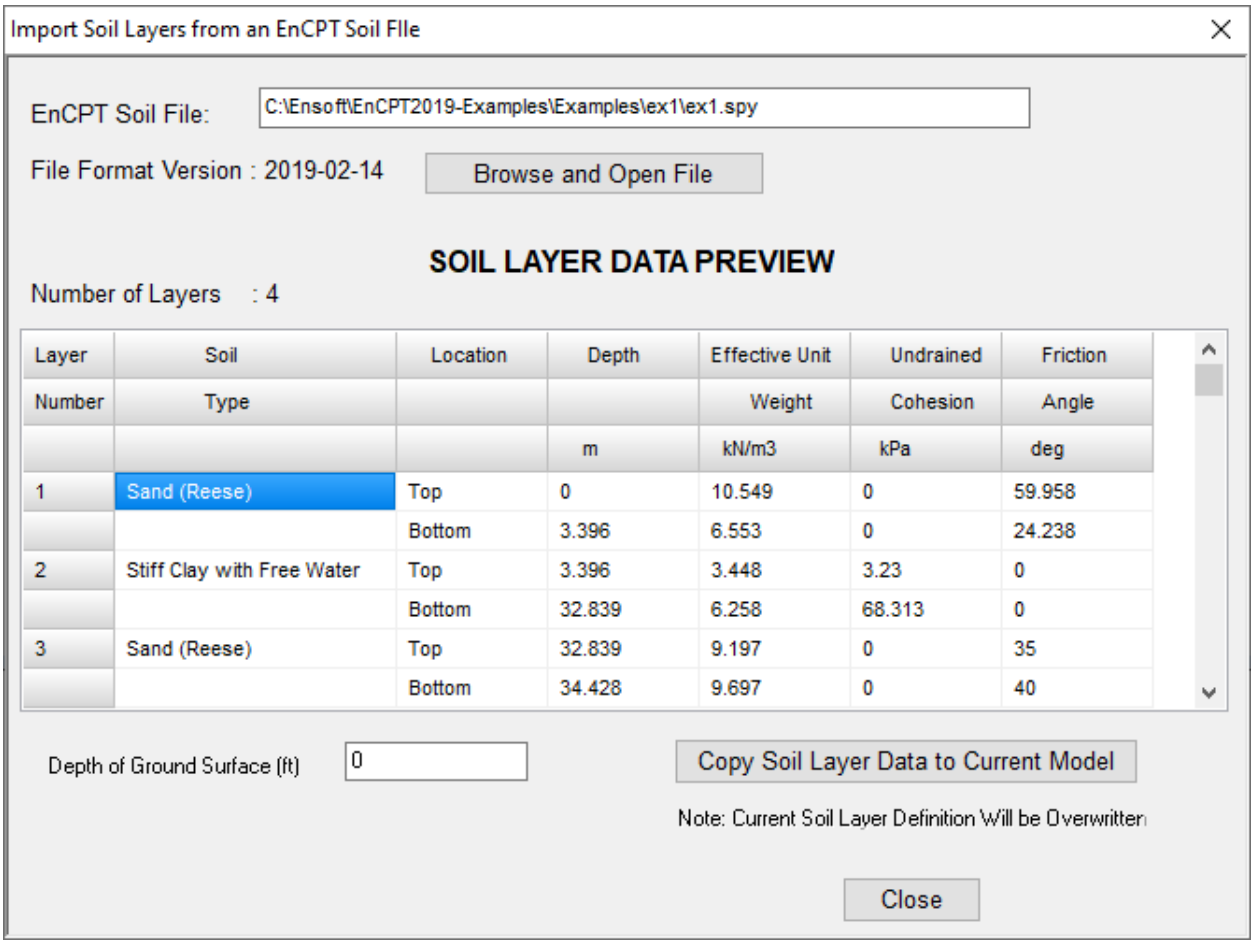
- Click on **Browse** button to find the file with the information about the displacement list. After selecting the file, the file name (including path) should appear in the File Name field.
- To review or edit the file click on the **View/Edit File** button. The text file will be open using the notepad editor.
- To copy the curve, click on the **Read Values from File**. If the copy is successful the displacement list will be populated based on the input data read.

To copy the displacement list from the clipboard:

- Copy the displacement list to the clipboard using a text editor or the original program. The information has to be in text format.
- Click on the **Paste values from Clipboard (text)** button to proceed with the copy. If the copy is successful, the displacement list will be populated based on the input data read.

3.3.21 Data – Import Soil Layers from an EnCPT Soil File

This submenu option is used to import soil layer generated by the EnCPT program.



Layer Number	Soil Type	Location	Depth	Effective Unit Weight	Undrained Cohesion	Friction Angle
			m	kN/m3	kPa	deg
1	Sand (Reese)	Top	0	10.549	0	59.958
		Bottom	3.396	6.553	0	24.238
2	Stiff Clay with Free Water	Top	3.396	3.448	3.23	0
		Bottom	32.839	6.258	68.313	0
3	Sand (Reese)	Top	32.839	9.197	0	35
		Bottom	34.428	9.697	0	40

Figure 3.152 Import Soil Layers from EnCPT Window

Click on the **Browse and Open File** button to navigate and select the output file generated for EnCPT which contains the soil layer data. If soil layer data is available in the file, the data will be displayed in the window.

Click on the **Copy Soil Layer Data to Current Model** button to transfer the data in memory to the internal tables of GROUP 2022. It should be noted that the program will overwrite the current definition of soil layering.

3.3.22 Data – Import Soil and Section Data from LPILE Model

This submenu option is used to import Soil and/or Section Data from an LPILE model (v2015/2016/2018/2019/2022).

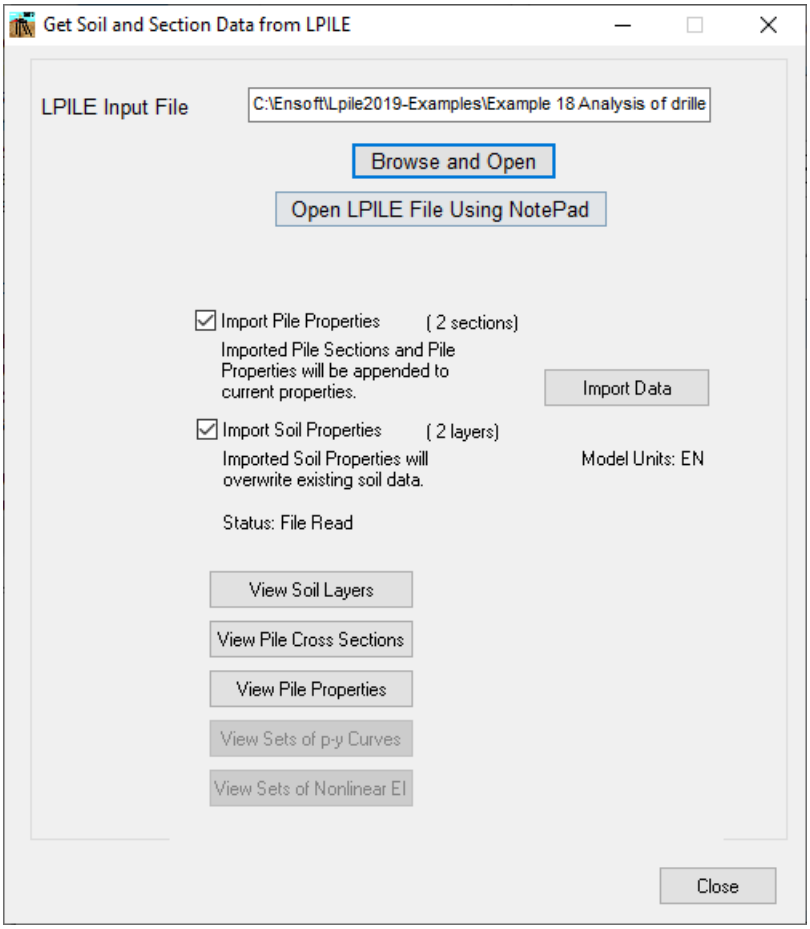


Figure 3.153 Import p-y curves from LPILE Window

Click on the **Browse and Open** button to navigate and select the input file for LPILE models (v 2015, 2018, 2019, 2022) which contains the input date. If the data is available, the information will be read into memory and a summary will be displayed.

The button **Open LPILE File Using Notepad** is enabled after a successful reading of the file into memory. Please click into this button to edit the file and check the data read by the program.

Click on the **Import Data** button to transfer the data in memory to the internal tables of GROUP 2022. Program will transfer the data that is checked. If the transferring of data was successful, a message indicating so will show up. It will also indicate the section number(s) where the section data were copied. For the soil data, the program will override the current soil layer defined. The program automatically will convert the data values to the current units being used on the GROUP model.

To verify the correct transfer of the data, click on the **View Soil Layers, View Pile Cross Sections** and/or **View Pile Properties** buttons.

Click on the **Close** button to Close the Window and go back to the main window of the program. Please note that the data will not be transferred to the GROUP tables if the user closes the window before clicking on the **Import Data** button.

3.3.23 Data – Import p-y Curves Generated by LPILE

This submenu option is used to import p-y curves generated by the LPILE program (v2012/2013/2015/2018/2019/2022). Actually, the user may be able to copy individual p-y curves using the option described item 3.3.13 (Data - Set of Soil Resistance (p-y) Curves). This option allows the user to transfer all the p-y curves generated by LPILE at once.

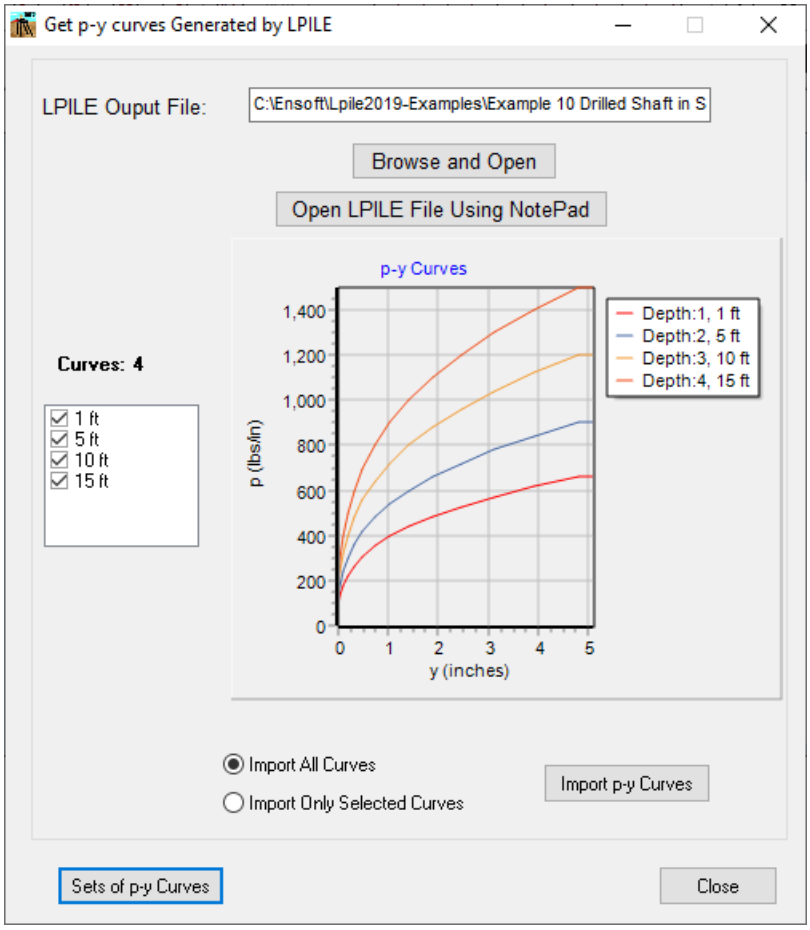


Figure 3.154 Import p-y curves from LPILE Window

Click on the **Browse and Open** button to navigate and select the output file generated for LPILE (v 2012/2013/2018/2019/2022) which contains the p-y curve data. If p-y curve is available in the file, the curves will be read into memory and plotted in the graph.

The button **Open LPILE File Using Notepad** is enabled after a successful reading of the file into memory. Please click into this button to edit the file and check the data read by the program.

Click on the **Import p-y Curves** button to transfer the data in memory to the internal tables of GROUP 2022. Program will transfer all the curves if the **Import All Curves** option is selected; otherwise it will transfer only the selected curves. If the transferring of data was successful, a message indicating so will show up. It will also indicate the set number where the p-y curves were copied. The program automatically will convert the p-y curves values to the current units being used on the GROUP model.

To verify the correct transfer of the data (or to edit p-y curve tables) click on the **Sets of p-y Curves** button. Please refer to item 3.3.13 (Data - Set of Soil Resistance (p-y) Curves) for a full description on the window.

Click on the **Close** button to Close the Window and go back to the main window of the program. Please note that the p-y curve data will not be transferred to the GROUP table if the user closes the window before clicking on the **Import p-y curves** button.

3.3.24 Data - Import Nonlinear EI Data Generated by LPILE

This submenu option is used to import Moment Curvature data generated by the LPILE program (v2012/2013/2015/2018/2019/2022). Actually, the user may be able to copy individual Moment-Curvature plots using the option described item 3.3.16 (Data - Sets of Nonlinear EI). This option allows the user to transfer all the moment curvature plots generated by LPILE at once.

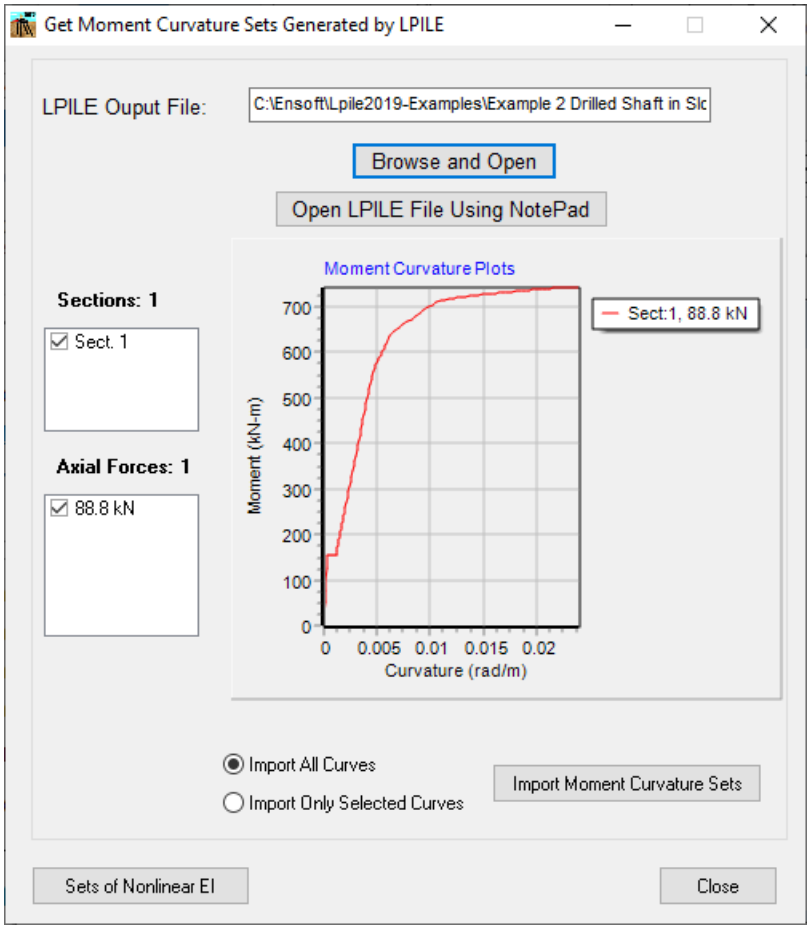


Figure 3.155 Import Nonlinear EI Data from LPILE Window

Click on the **Browse and Open** button to navigate and select the output file generated for LPILE (v 2012/2013/2018/2019/2022) which contains the Nonlinear EI data. If Moment curvature data is available in the file, the curves will be read into memory and plotted in the graph.

The button **Open LPILE File Using Notepad** is enabled after a successful reading of the file into memory. Please click into this button to edit the file and check the data read by the program.

Click on the **Import Moment Curvature Sets** button to transfer the data in memory to the internal tables of Group 2022. Program will transfer all the curves if the **Import All Curves** option is selected; otherwise it will transfer only the selected curves. If the transferring of data was successful, a message indicating so will show up. It will also indicate the set number(s) where the moment curvature data were copied. The program automatically will convert the moment curvature values to the current units being used on the GROUP model.

To verify the correct transfer of the data (or to edit p-y curve tables) click on the **Sets of Nonlinear EI** button. Please refer to item 3.3.16 (Data - Sets of Nonlinear EI) for a full description on the window.

Click on the **Close** button to Close the Window and go back to the main window of the program. Please note that the Moment Curvature data will not be transferred to the GROUP table if the user closes the window before clicking on the **Import p-y curves** button.

3.3.25 Data - Import Group Layout from Excel File

This submenu option is used to import a group layout from an excel file. The information required is consistent with the one listed in item 3.3.4 Data - Pile Group Properties and item 3.3.5 Data - Pile-Head Coordinates. User should take a look at the templates included in the examples folder to prepare a more complex pile group layout. Program does not check the consistency of the data in the excel file, so the file should be prepared with care to avoid crashing of the program. See item 3.3.27 Data - Export Group Layout to Excel File, for sheets required in the excel file.

3.3.26 Data - Import Loading from Excel File

This submenu option is used to import loading from an excel file. The information required is consistent with the one listed in item 3.3.11 Data - Loading. User should take a look at the templates included in the examples folder to prepare a more complex loading scheme. Program does not check the consistency of the data in the excel file, so the file should be prepared with care to avoid crashing of the program. See item 3.3.28Data - Export Loading to Excel File, for sheets required in the excel file.

3.3.27 Data - Export Group Layout to Excel File

This submenu option is used to export a group layout to an excel file. The information exported is consistent with the one listed in item 3.3.4 Data - Pile Group Properties and item 3.3.5 Data - Pile-Head Coordinates. User may use this option to generate a basic template in order to prepare a more complex pile group layout.

The following sheets are generated in the excel file

- **Parameters.** Include Version Number, Analysis Mode and Units.
- **3D-Layout.** Generated for 3D models. Include information for Pile Group Properties (item 3.3.4) and Pile Head Coordinates (item 3.3.5)
- **2D-Layout.** Generated for 2D models. Include information for Pile Group Properties (item 3.3.4) and Pile Head Coordinates (item 3.3.5)
- **Comments.** General comments for some columns in Layout sheet.

For a model using a flexible pile cap (Flexible Cap Version) the following additional sheets are generated

- **Node_Coordinates.** Include the node connectivity information (item 3.3.7.1)
- **Plate_Connectivity.** Include the information about plate connectivity (item 3.3.8.1) and plate offsets (item 3.3.8.2)
- **Beam_Connectivity.** Include the information about beam connectivity (item 3.3.9.1), beam offsets (item 3.3.9.2), beam end zones (item 3.3.9.3) and beam rotation (item 3.3.9.4)
- **Conc.Springs_Nodes.** Include information for Concentrated Springs at Nodes (item 3.3.10.1)
- **Dist.Springs_Plates.** Include information for Distributed Springs on Plates (item 3.3.10.2)
- **Dist.Springs_Beams.** Include information for Distributed Springs on Beams (item 3.3.10.3)

3.3.28 Data - Export Loading to Excel File

This submenu option is used to export loading to an excel file. The information exported is consistent with the one listed in item 3.3.11 Data - Loading. User may use this option to generate a basic template in order to prepares a more complex loading scheme.

The following sheets are generated in the excel file

- **Parameters.** Include Version Number, Analysis Mode and Units.
- **Load Cases.** Include definition for the load cases (item 3.3.11.1), parameters for the load case (item 3.3.11.2), and analysis options (item 3.3.11.15). The sheet is always created, but it may be blank if no load cases are defined.
- **Conc.Loads-3D.** Generated for 3D models. Include information for definition of concentrated loads at pile cap (item 3.3.11.4). Information for each load case can be separated by a blank line. The sheet is always created for a 3D model, but it may be blank if no concentrated loads are defined in any load case.
- **Conc.Loads-2D.** Generated for 2D models. Include information for definition of concentrated loads at pile cap (item 3.3.11.4). Information for each load case can be separated by a blank

line. The sheet is always created for a 2D model, but it may be blank if no concentrated loads are defined in any load case.

- **Spec.Disp.-3D.** Generated for 3D models. Include information for definition of specified displacements at pile cap (item 3.3.11.5). Information for each load case can be separated by a blank line. The sheet is always created for a 3D model, but it may be blank if no specified displacements are defined in any load case.
- **Spec.Disp.-2D.** Generated for 2D models. Include information for definition of specified displacements at pile cap (item 3.3.11.5). Information for each load case can be separated by a blank line. The sheet is always created for a 2D model, but it may be blank if no specified displacements are defined in any load case.
- **Stiffness.** Include information for definition of foundation stiffness at pile cap (item 3.3.11.14). Information for each load case can be separated by a blank line. The sheet is always created, but it may be blank if no stiffness option is defined in any load case.
- **Incremental.** Include information for definition of incremental/pushover analysis (item 3.3.11.14). Information for each load case can be separated by a blank line. The sheet is always created, but it may be blank if no incremental/pushover analysis is defined in any load case.
- **Dist.Load-3D.** Generated for 3D models. Include information for definition of distributed loads along the piles (item 3.3.11.11). Information for each load case can be separated by a blank line. The sheet is always created for a 3D model, but it may be blank if no distributed loads are defined in any load case.
- **Dist.Load-2D.** Generated for 2D models. Include information for definition of distributed loads along the piles (item 3.3.11.11). Information for each load case can be separated by a blank line. The sheet is always created for a 2D model, but it may be blank if no distributed loads are defined in any load case.
- **Soil Move.-3D.** Generated for 3D models. Include information for definition of soil movement along the piles (item 3.3.11.11). Information for each load case can be separated by a blank line. The sheet is always created for a 3D model, but it may be blank if no soil movement are defined in any load case.
- **Soil Move.-2D.** Generated for 2D models. Include information for definition of soil movement along the piles (item 3.3.11.11). Information for each load case can be separated by a blank line. The sheet is always created for a 2D model, but it may be blank if no soil movement are defined in any load case.
- **Load Comb.** Include definition for the load combinations (item 3.3.11.17), parameters for the load combination (item 3.3.11.18), and analysis options (item 3.3.11.15). The sheet is always created, but it may be blank if no load combinations are defined.
- **Comb-Factors.** Include definition for each load combination (item 3.3.11.18). The sheet is always created, but it may be blank if no combinations are defined in any load combination.
- **Comments.** General comments for this file.

For a model using a flexible pile cap (Flexible Cap Version) the following additional sheets are generated

- **Spec.Disp_Nodes.** Include information for Specified Displacements at Nodes (item 3.3.11.63.3.11.73.3.10.1)
- **Conc.Loads_Nodes.** Include information for Concentrated Loads at Nodes (item 3.3.11.73.3.10.1)
- **Dist.Loads_Plates.** Include information for Distributed Loads on Plates (item 3.3.11.8)
- **Dist.Loads_Beams.** Include information for Distributed Loads on Beams (item 3.3.11.9)
- **Conc.Loads_Beams.** Include information for Concentrated Loads on Beams (item 3.3.11.10)

3.4 Edit Menu (Editing Objects Using Graphics Tools)

To activate the options under the **Edit** menu, the **3D View** window must be open.

.

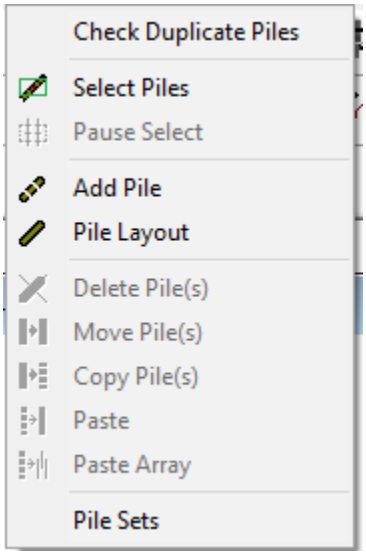


Figure 3.156 Edit Menu with 3D View Active

.

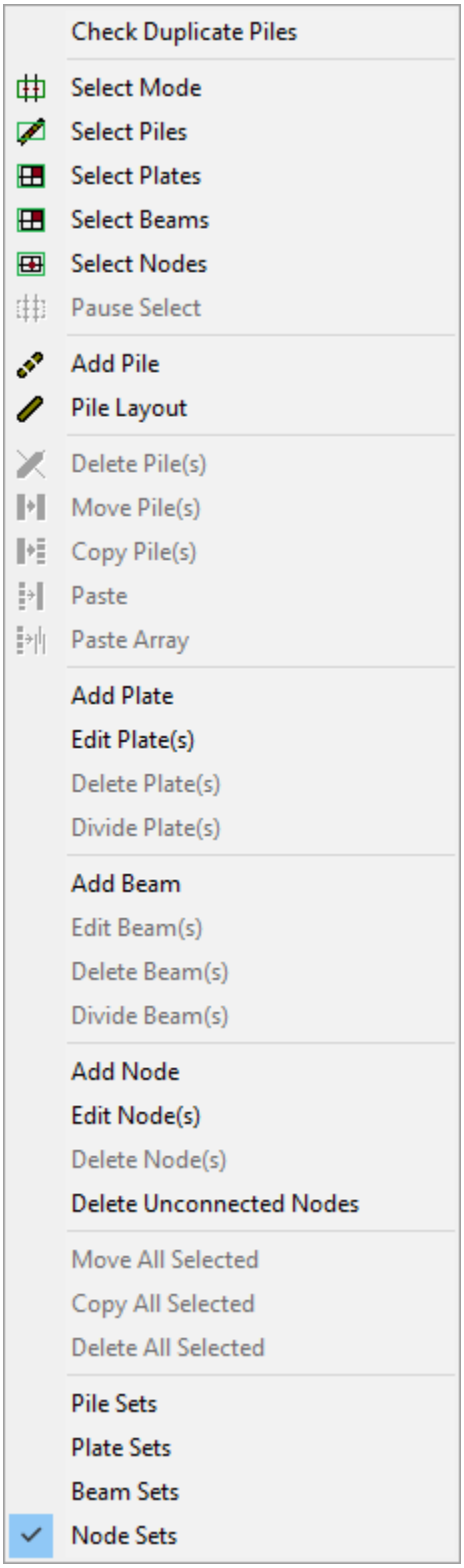


Figure 3.157 Edit Menu with 3D View Active - Flexible Cap Version

3.4.1 Edit - Select Piles

Activate **Select Piles** by clicking on **Select Piles** in the **Edit** menu. Note that the **Select Piles** option is checked. Also Select Piles can be activated by clicking on the **Select Piles** tool button.

Then, left click the desired pile to select it, and the selected pile will be shown in red. The status bar (at the bottom of the screen) will indicate the number of selected piles.

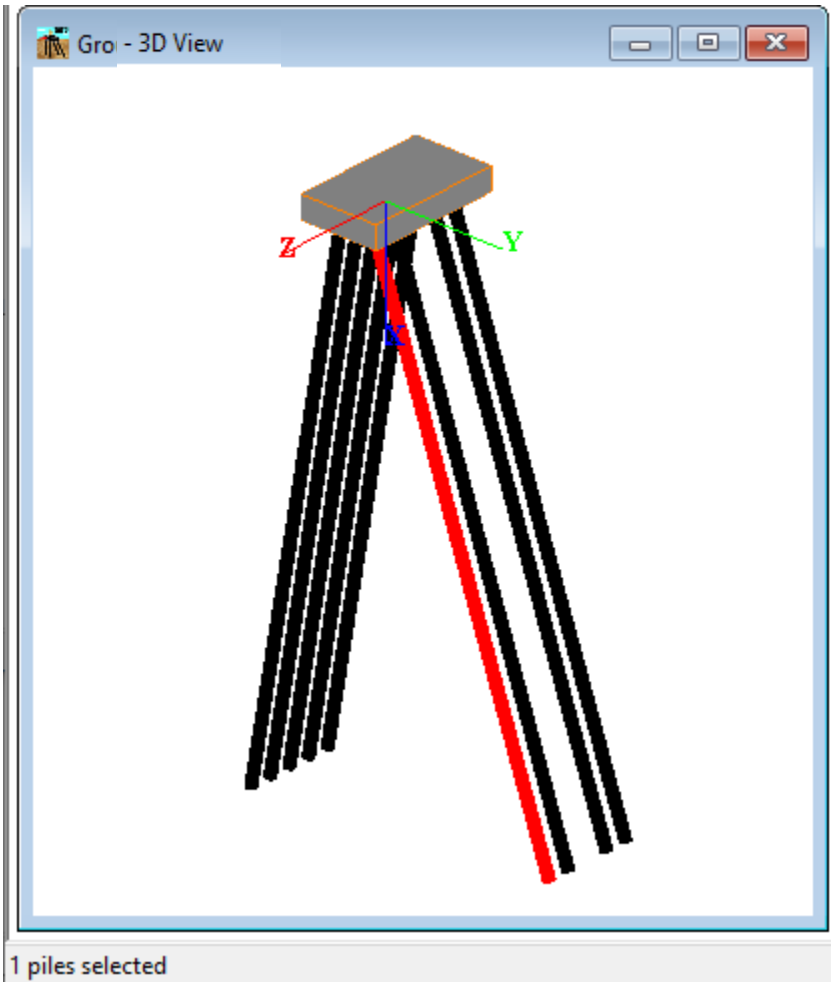


Figure 3.158 Selection by clicking

Selection can also be made using a box, by left clicking and dragging the mouse. Again, the selected piles will be shown in red.

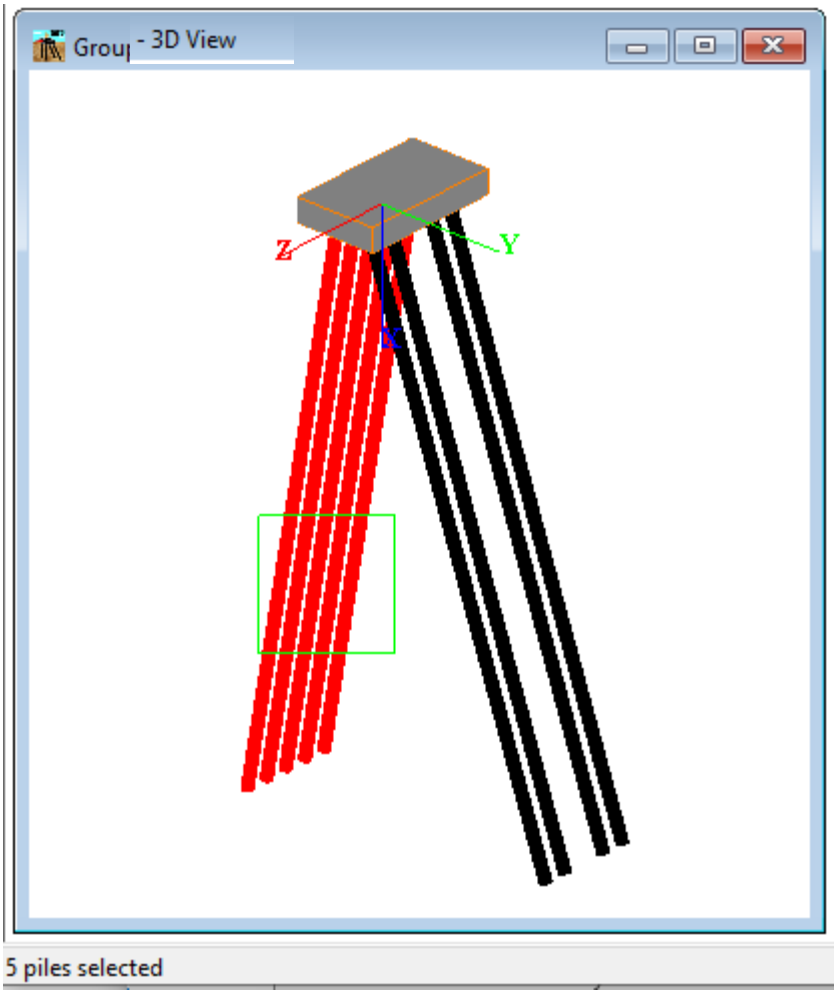


Figure 3.159 Selection using a box

To add piles to the selection, hold the **Shift** or **Ctrl** keys while selecting the new piles. To remove piles from the selection, press the **Ctrl** key and click on the pile(s) you wish to deselect.

To deactivate the **Select Piles,** click on **Select Piles** in the **Edit** menu. Note that the **Select Piles** option is not checked

3.4.2 Edit - Select Mode - Flexible Cap Version

Use this option to activate selection for all the objets in the model. This option include **Select Piles, Select Plates, Select Beams** and **Select nodes**. Also Select Mode can be activated by clicking on the **Select Mode** tool button.

To deactivate the **Select Mode,** click on **Select Mode** in the **Edit** menu. Note that the **Select Mode** option is not checked

3.4.3 Edit - Select Plates - Flexible Cap Version

Activate **Select Plates** by clicking on **Select Plates** in the **Edit** menu. Note that the **Select Plates** option is checked. Also Select Plates can be activated by clicking on the **Select Plates** tool button.

Then, left click the desired plate to select it, and the selected plate will be shown in purple. The status bar (at the bottom of the screen) will indicate the number of selected plates.

Selection can also be made using a box, by left clicking and dragging the mouse. The selected plates will be shown in purple.

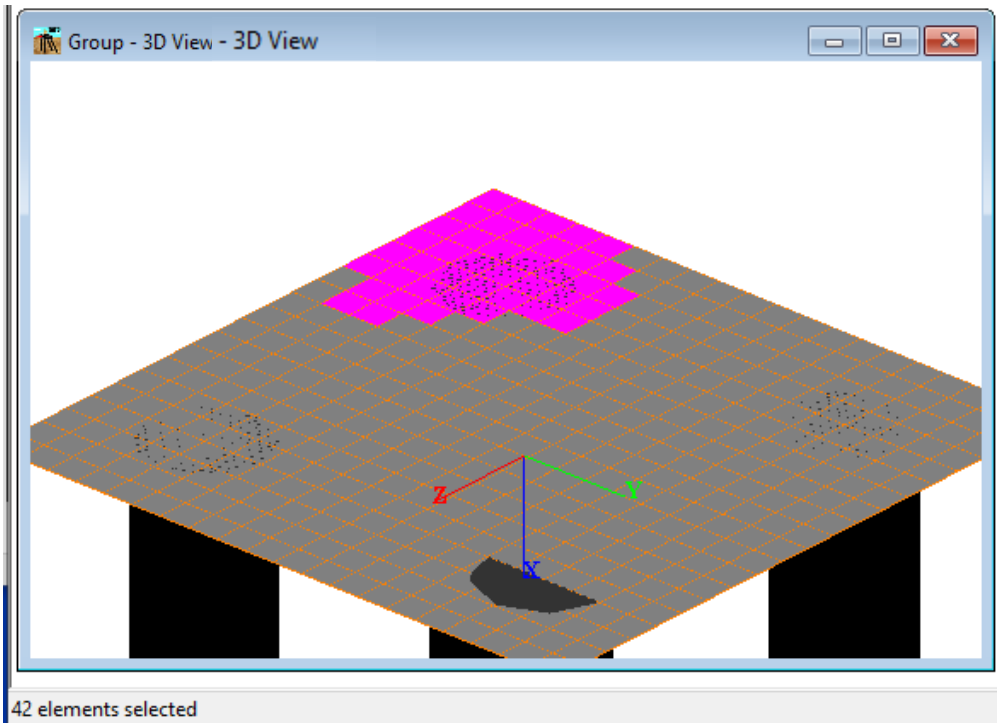


Figure 3.160 Selection of plates - Flexible Cap Version

To add plates to the selection, hold the **Shift** or **Ctrl** keys while selecting the new plates. To remove plates from the selection, press the **Ctrl** key and click on the plate(s) you wish to deselect.

To deactivate the **Select Plates,** click on **Select Plates** in the **Edit** menu. Note that the **Select Plates** option is not checked

3.4.4 Edit - Select Beams - Flexible Cap Version

Activate **Select Beams** by clicking on **Select Beams** in the **Edit** menu. Note that the **Select Beams** option is checked. Also Select Beams can be activated by clicking on the **Select Beams** tool button.

Then, left click the desired plate to select it, and the selected beam will be shown in red. The status bar (at the bottom of the screen) will indicate the number of selected beams.

Selection can also be made using a box, by left clicking and dragging the mouse. The selected beams will be shown in red.

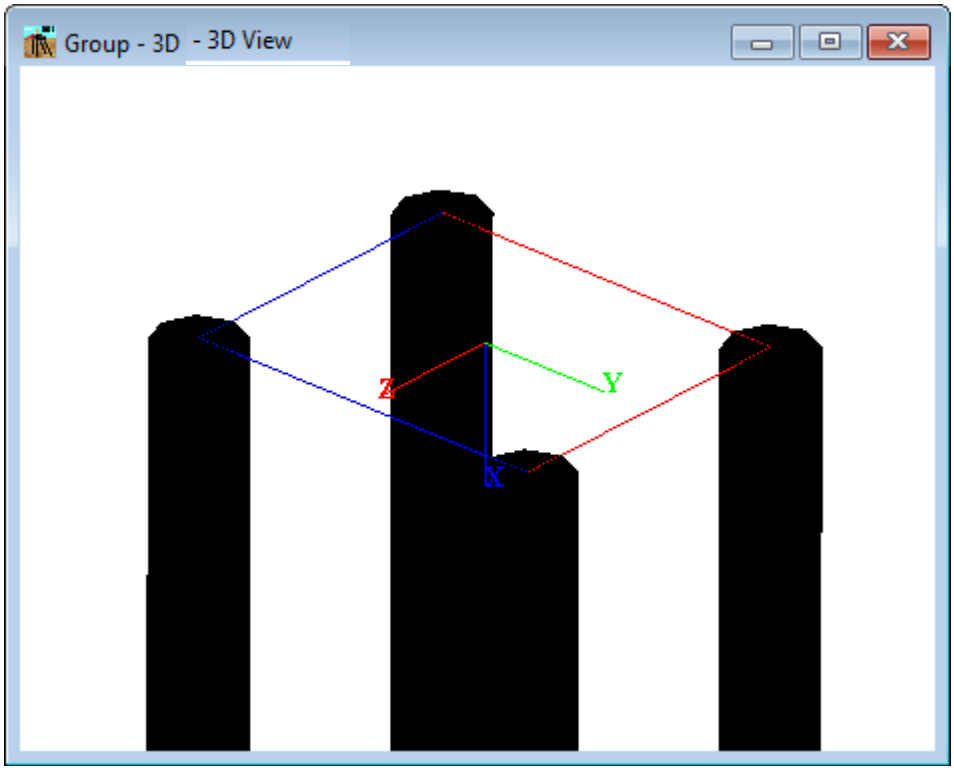


Figure 3.161 Selection of beams - Flexible Cap Version

To add beams to the selection, hold the **Shift** or **Ctrl** keys while selecting the new beams. To remove beams from the selection, press the **Ctrl** key and click on the beam(s) you wish to deselect.

To deactivate the **Select Beams,** click on **Select Beams** in the **Edit** menu. Note that the **Select Beams** option is not checked

3.4.5 Edit - Select Nodes - Flexible Cap Version

Activate **Select Nodes** by clicking on **Select Nodes** in the **Edit** menu. Note that the **Select Nodes** option is checked. Also Select nodes can be activated by clicking on the **Select nodes** tool button.

Then, left click the desired node to select it, and the selected node will be shown in light blue. The status bar (at the bottom of the screen) will indicate the number of selected nodes.

Selection can also be made using a box, by left clicking and dragging the mouse. The selected nodes will be shown in light blue.

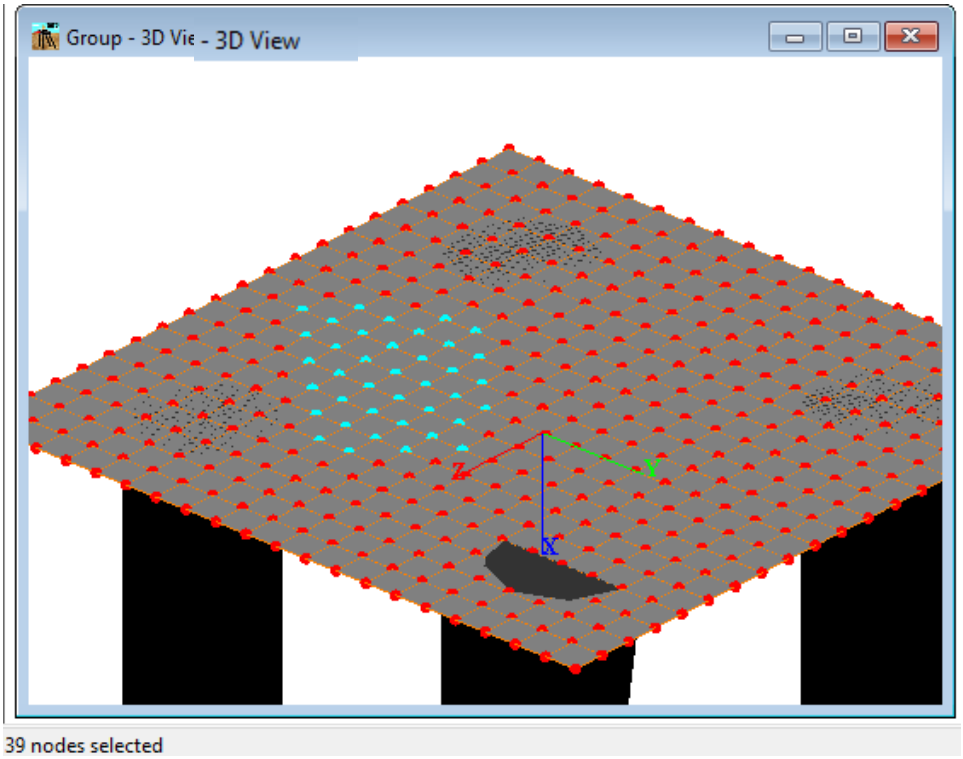


Figure 3.162 Selection of nodes - Flexible Cap Version

To add nodes to the selection, hold the **Shift** or **Ctrl** keys while selecting the new nodes. To remove nodes from the selection, press the **Ctrl** key and click on the node(s) you wish to deselect.

To deactivate the **Select Nodes,** click on **Select Nodes** in the **Edit** menu. Note that the **Select Nodes** option is not checked

Note, that the Selection can be performed for Piles, Plates, Beams and Nodes at the same time.

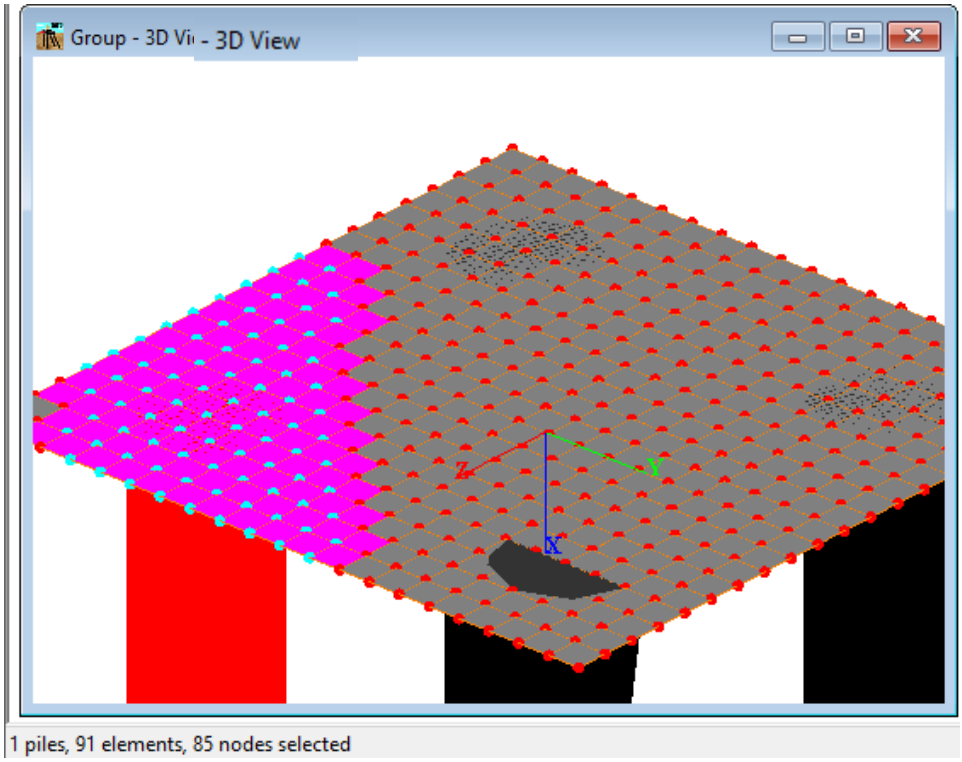


Figure 3.163 Selection of piles, plates and nodes - Flexible Cap Version

3.4.6 Edit – Pause Select

Activate **Pause Select** by clicking on **Pause Select** in the **Edit** menu. Note that the **Pause Select** option is checked. Also pause select can be activated by clicking on the **Pause Select** tool button.

The Pauses Select option is enabled after the Select Piles (or Select Plates or Select Beams or Select Nodes for the Flexible Cap Version) is activated. Be activating this option the current selection will be kept in memory, allowing to perform view transformations like translation rotation and zooming (which are deactivated while any Selection is activated). After getting the required view transformation deactivate the Pause Select option to continue with selection.

To deactivate the **Pause Select,** click on **Pause Select** in the **Edit** menu. Note that the **Pause Select** option is not checked

3.4.7 Edit -Add Pile

Select the **Add Pile** item from the **Edit** menu or click the **Add Pile** tool button. Then the **New Pile** window will appear. The window will vary between 2D and 3D models.

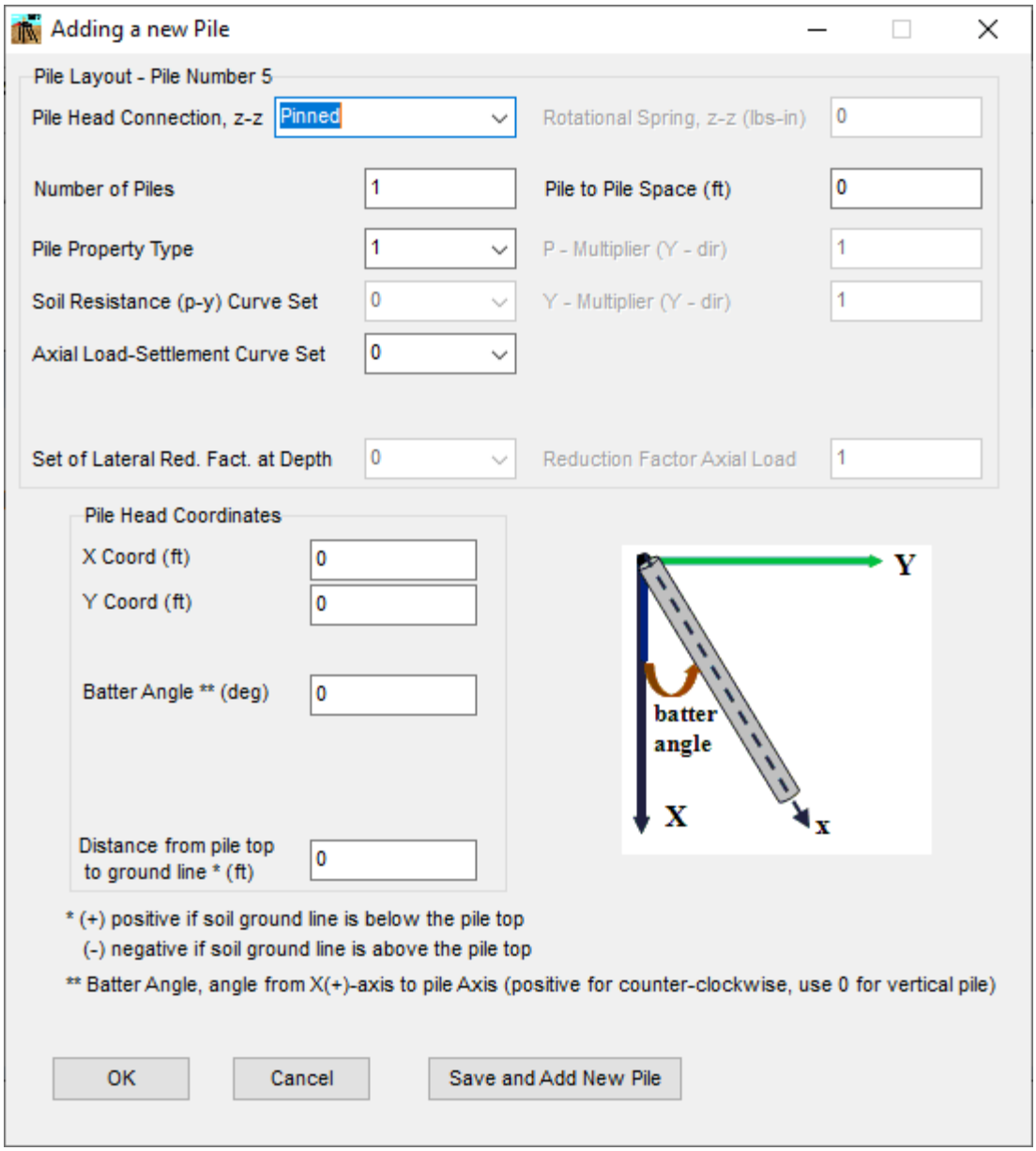


Figure 3.164 New Pile Window. 2D Model

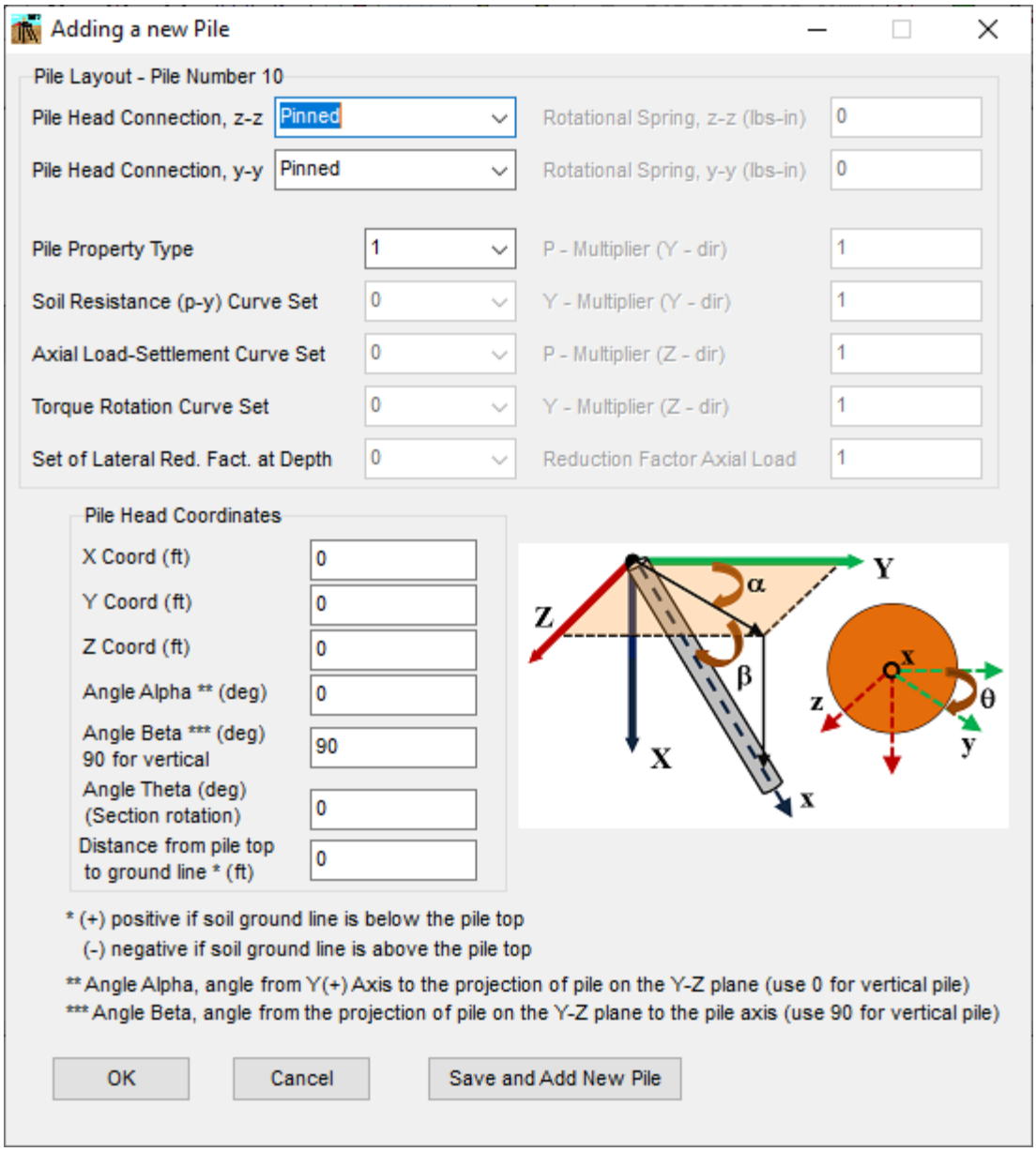


Figure 3.165 New Pile Window. 3D Model

Fill in the necessary information and click the **OK** button to save and close the window, or click the **Save and Add New Pile** button to save the information and return to the same window to add an additional pile, or click the **Cancel** button to cancel the process and close the window. Refer to Data - Pile Group Properties and Data - Pile Head Coordinates for a description of the different input parameters.

3.4.8 Edit - Pile Layout

Select the **Pile Layout** item from the **Edit** menu or click the **Pile Layout** tool button. The **Pile Layout** window will appear (this window is similar to the **New Pile** window). In this window the user may change the input parameters.

The **Pile Layout** window will vary slightly whether there are some piles selected or not

If no piles are selected the window will show up the input parameters of the current pile (starting with pile number 1). The user may navigate throughout the different piles using the **Previous** and **Next** buttons displayed at the bottom of the window. Also, the user may jump to a pile number by selecting the Pile Number at the **Select Pile Number** pull-down menu and then clicking the **Go** button. The current pile will be highlighted in the **3D View** window with green color.

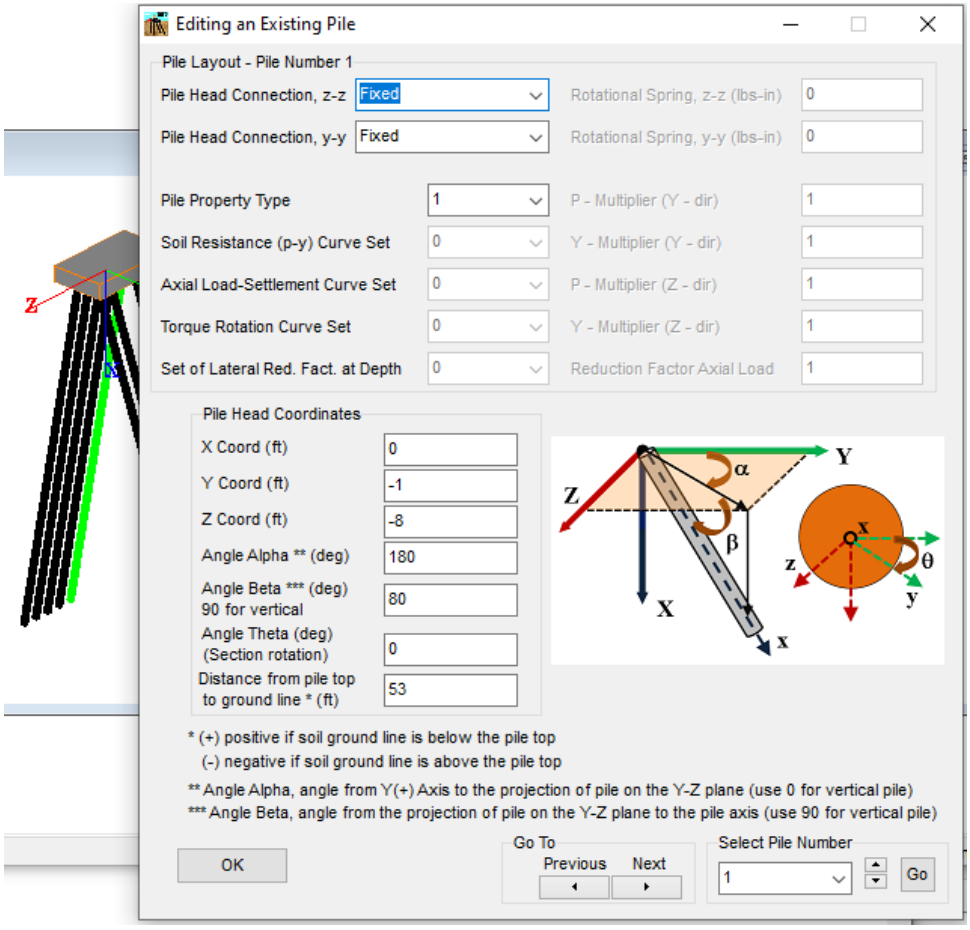


Figure 3.166 Pile Layout Window. No piles Selected

If piles are selected the window will show up the input information of the selected piles (which are highlighted in the **3D View** window in red color). In this window the input parameters that are similar to all the selected piles are displayed in white background, while the input parameters that are not the same for all the selected piles are shown in gray background. Any change in the input parameters will be assigned for all the piles selected.

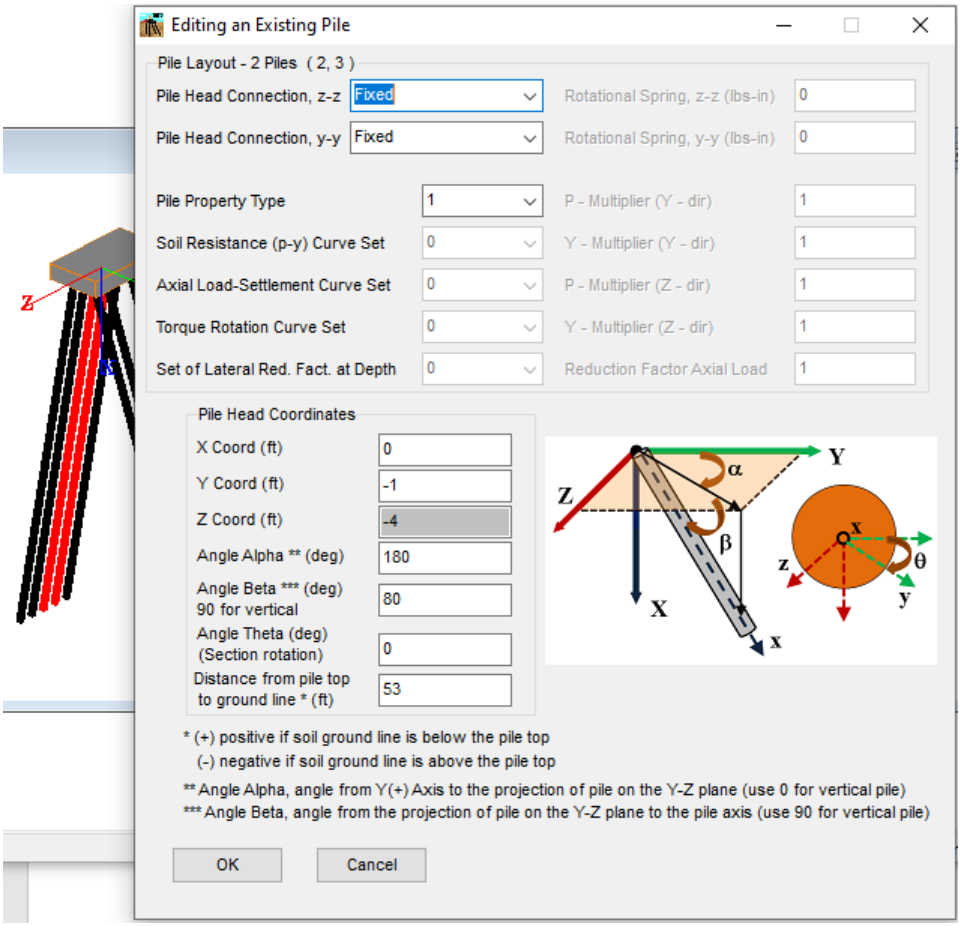


Figure 3.167 Pile Layout Window. Piles Selected

3.4.9 Edit - Delete Piles

Please note that this option is enabled after a selection has been performed. Delete piles by choosing **Delete Pile(s)** under the **Edit** menu, or by clicking the **Delete Pile** tool button. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process.

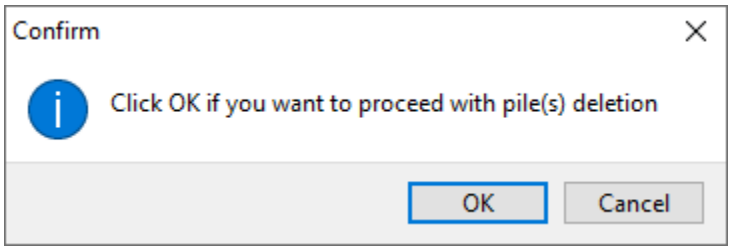


Figure 3.168 Window for confirmation of deletion

Please note that pile numbering may be rearranged after deletion of piles.

3.4.10 Edit - Move Piles

Please note that this option is enabled after copying has been performed. Move the selected piles, by choosing **Move Pile(s)** from the **Edit** menu or by clicking the **Move Pile(s)** tool button. The **Move Options** window will then appear.

By default, the **Absolute Coordinates** option is selected. The reference point is the pile head position of the first selected pile. The program will estimate the increment by the difference between the target and the reference point

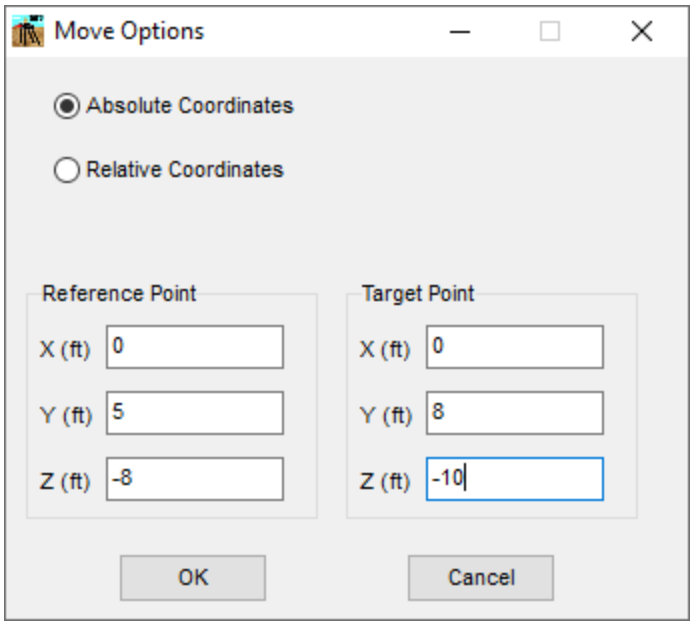


Figure 3.169 Move Options Window. Absolute Coordinates

If the **Relative Coordinates** option is selected, then the user must provide the coordinate increment.

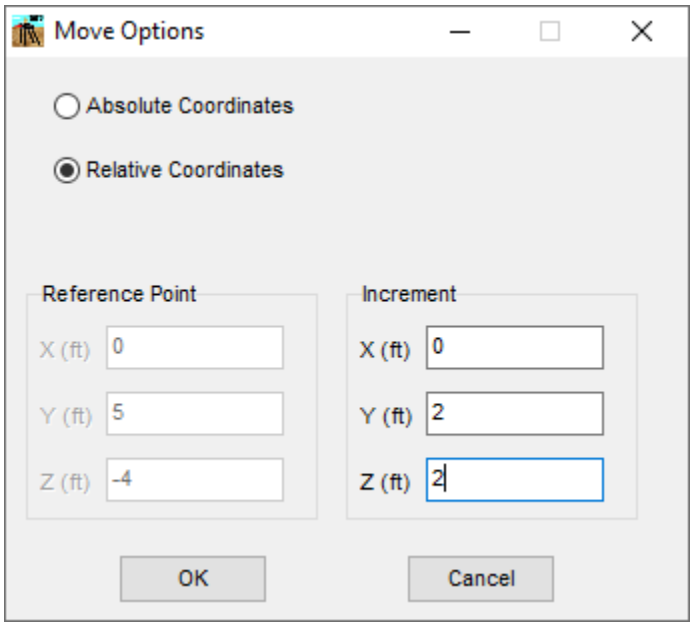


Figure 3.170 Move Options Window. Relative Coordinates

Fill in the necessary fields and click “OK” to finish the moving process, or click “Cancel” to cancel the process.

3.4.11 Edit - Copy Piles

Please note that this option is only enabled after a selection has been performed. Copy the selected piles to the buffer by choosing **Copy** from the E**dit** menu or by clicking the **Copy Pile** tool button.

3.4.12 Edit - Paste Pile

Please note that this option is enabled after copying has been performed. Paste the selected piles, by choosing **Paste** from the **Edit** menu or by clicking the **Paste Pile** tool button. The **Paste Options** window will then appear.

By default, the **Absolute Coordinates** option is selected. The reference point is the pile head position of the first selected pile. The program will estimate the increment by the difference between the target and the reference point

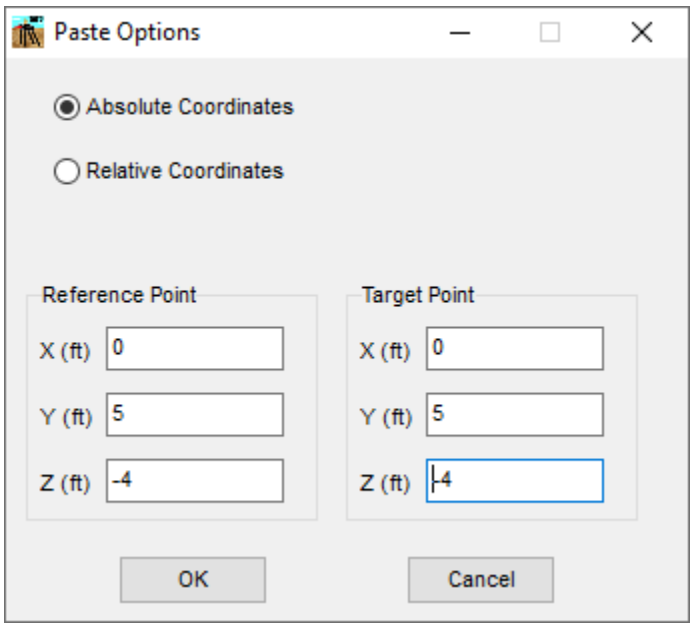


Figure 3.171 Paste Options Window. Absolute Coordinates

If the **Relative Coordinates** option is selected, then the user must provide the coordinate increment.

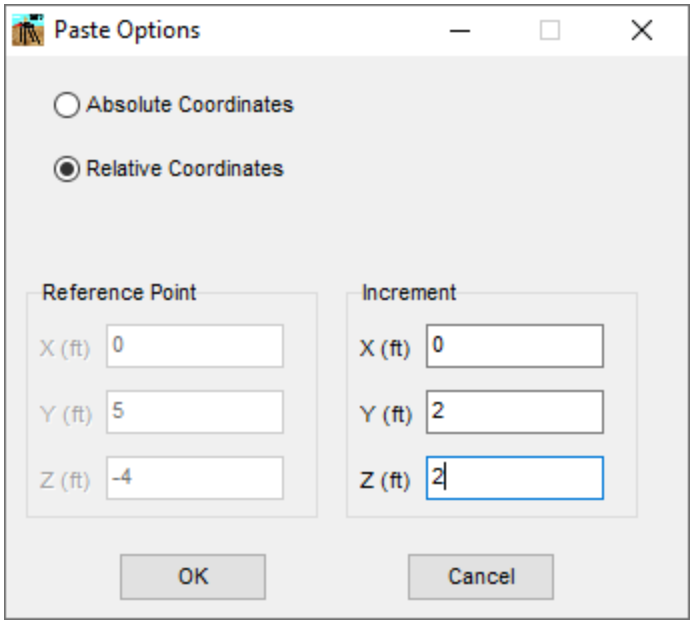


Figure 3.172 Paste Options Window. Relative Coordinates

Fill in the necessary fields and click “OK” to finish the copying and pasting process, or click “Cancel” to cancel the process.

3.4.13 Edit - Paste Array

Please note that this option is enabled after the copying process has been performed. Paste using an array of the selected piles, by selecting **Paste Array** from the Edit menu, or by clicking the **Paste Array** tool button. The **Paste Array Options** window will then appear.

Fill in the necessary fields and click “OK” to finish the copying and pasting process, or click “Cancel” to cancel the process.

Two paste array schemes are available. Rectangular array and Polar Array

For the rectangular array, the space increment in X, Y and Z directions and the number of rows to be copied in X, Y and Z directions are required

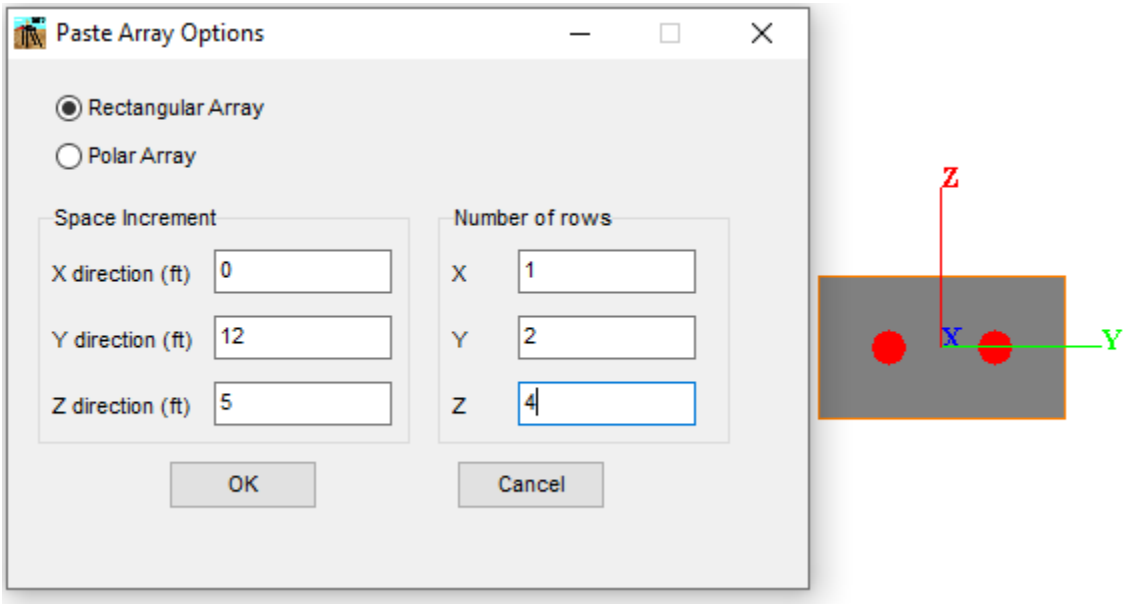


Figure 3.173 Paste Array Options Window. Rectangular array

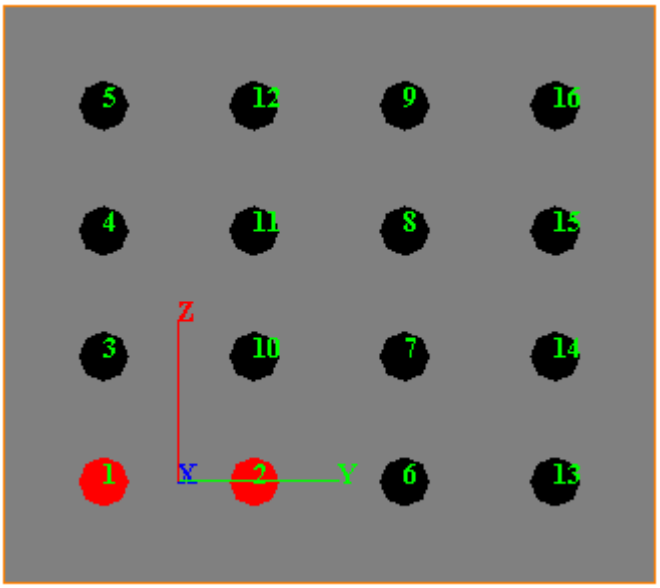


Figure 3.174 Pile layout after paste using rectangular array

For the polar array, the center of generation point (X, Y and Z), the number of lines to be generated, and the angle increment are required

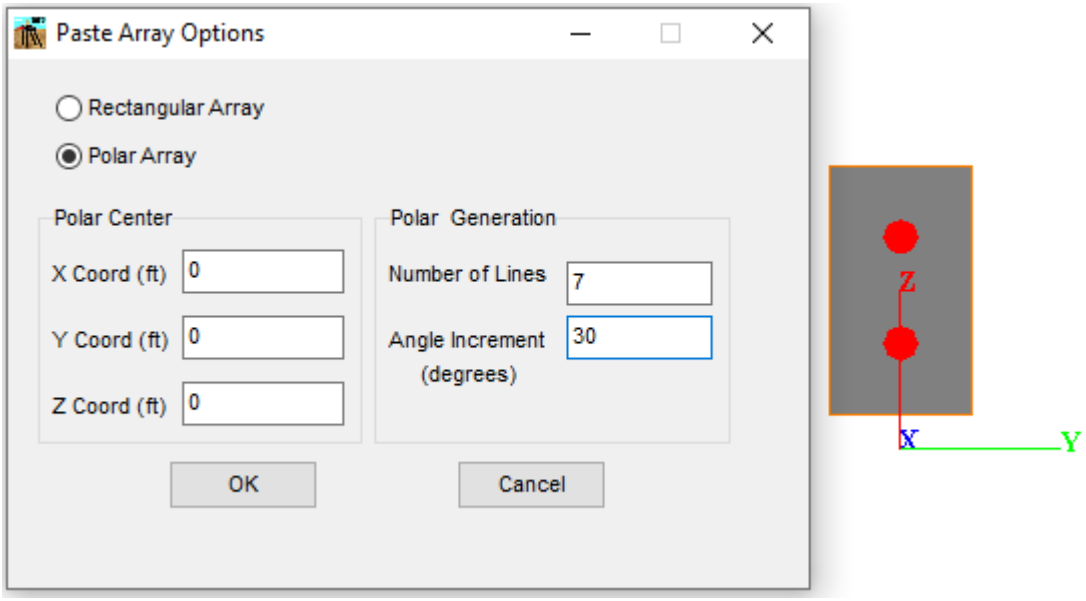


Figure 3.175 Paste Array Options Window. Polar array

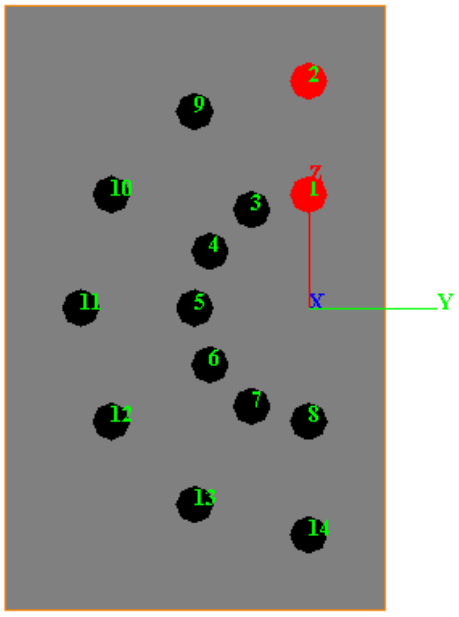


Figure 3.176 Pile layout after paste using polar array

3.4.14 Edit -Add Plate - Flexible Cap Version

Select the **Add Plate** item from the **Edit** menu. Then the **New Plate** window will appear.

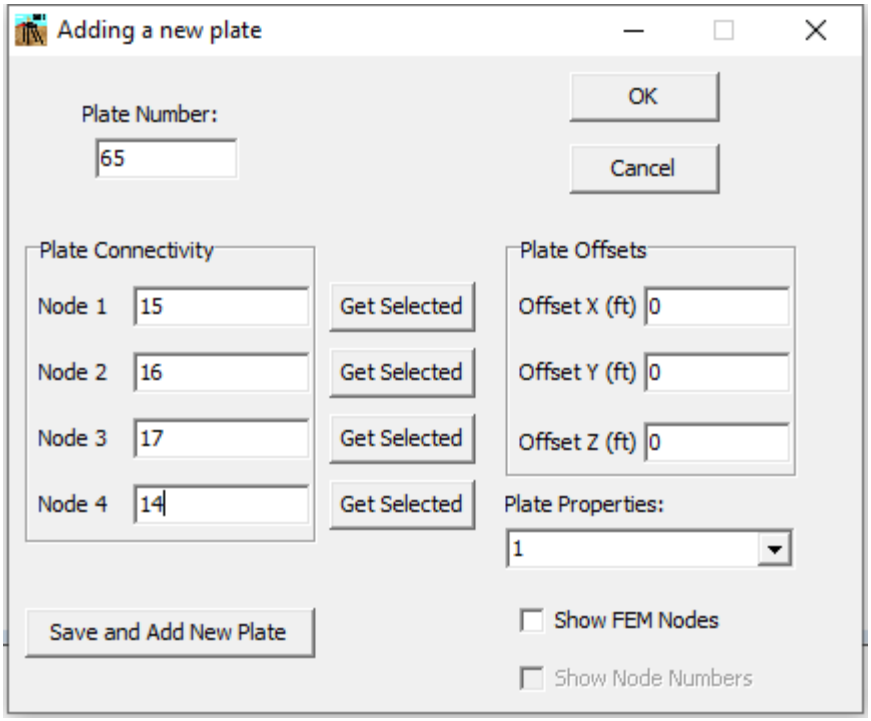


Figure 3.177 New Plate Window - Flexible Cap Version

Fill in the necessary information and click the **OK** button to save and close the window, or click the **Save and Add New Plate** button to save the information and return to the same window to add an additional plate, or click the **Cancel** button to cancel the process and close the window. Please note that the program may get the selected node by clicking on the **Get Selected** button.

3.4.15 Edit – Edit Plate

Select the **Edit Plate** item from the **Edit** menu. The **Edit Plate** window will appear (this window is similar to the **New Plate** window). In this window the user may change the input parameters.

The **Edit Plate** window will vary slightly whether there are some plates selected or not

If no plates are selected the window will show up the input parameters of the current plate (starting with plate number 1). The user may navigate throughout the different plates using the arrows at the right of Plate Number. Also, the user may jump to a plate number by selecting the Plate Number at the combo box under **Plate Number**. The current plate will be highlighted in the **3D View** window with light blue color.

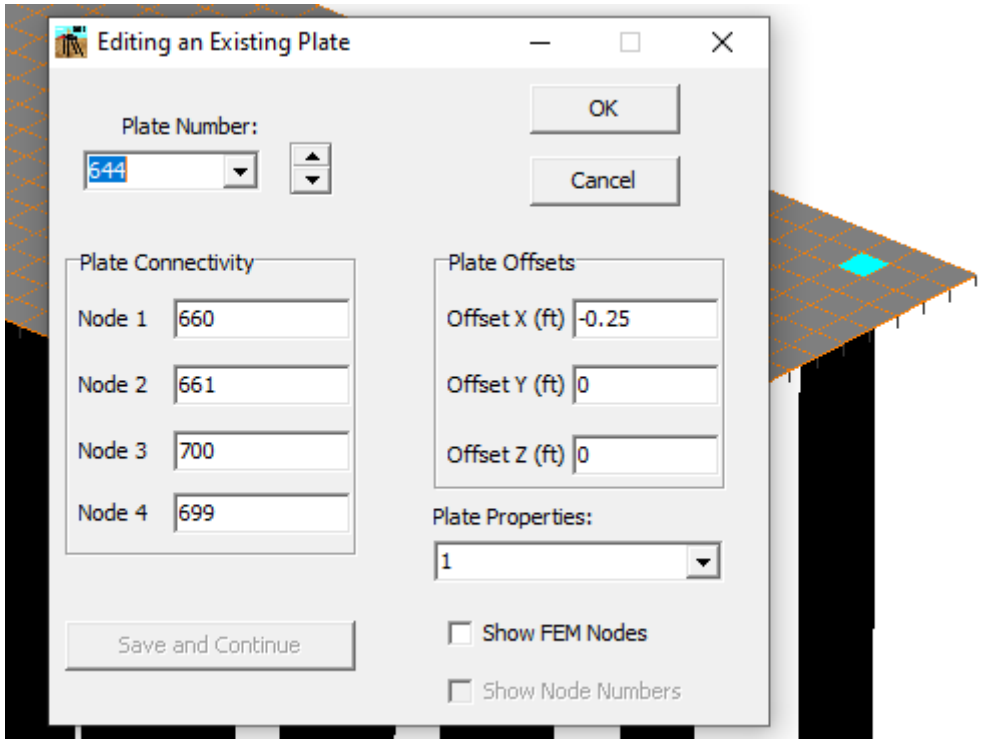


Figure 3.178 Edit Plate Window. No plates Selected - Flexible Cap Version

If plates are selected the window will show up the input information of the selected plates (which are highlighted in the **3D View** window in purple color). In this window the input parameters that are similar to all the selected plates are displayed in white background, while the input parameters

that are not the same for all the selected plates are shown in gray background. Any change in the input parameters will be assigned for all the plates selected.

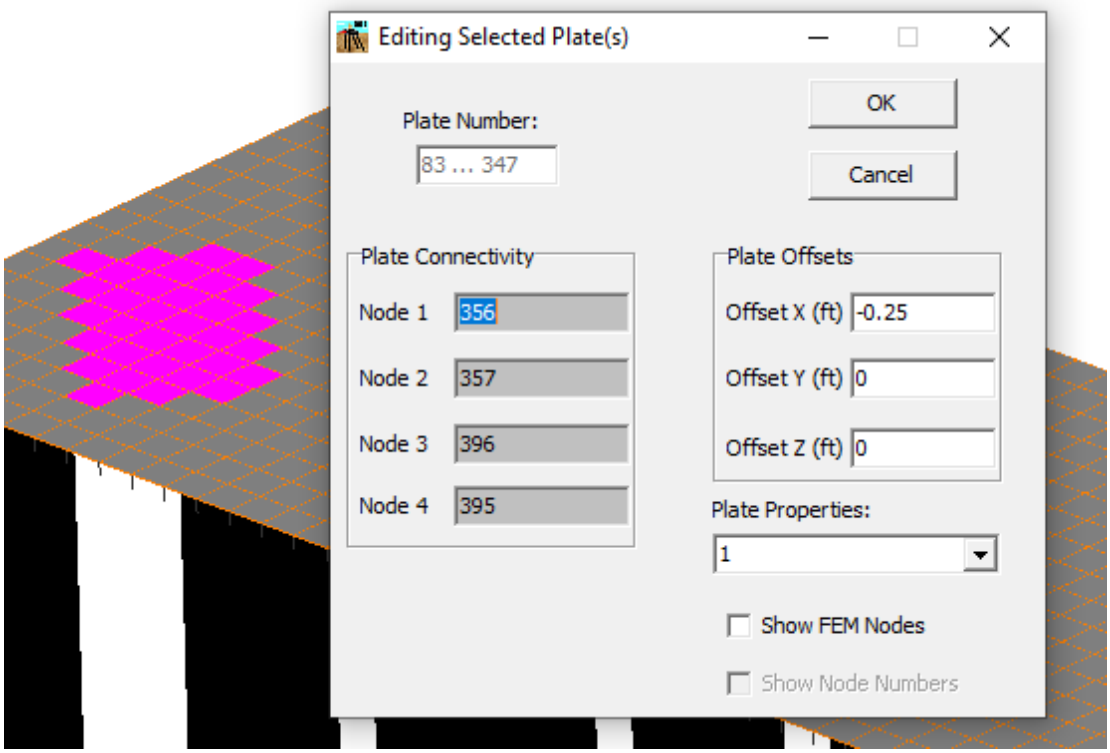


Figure 3.179 Edit Plate Window. Plates Selected - Flexible Cap Version

3.4.16 Edit - Delete Plates (Flexible Cap Version)

Please note that this option is enabled after a selection has been performed. Delete plates by choosing **Delete Plate(s)** under the **Edit** menu. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process.

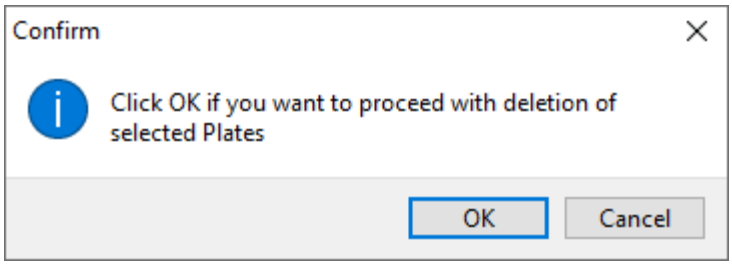


Figure 3.180 Window for confirmation of plates deletion - Flexible Cap Version

3.4.17 Edit - Divide Plates - Flexible Cap Version

Please note that this option is enabled after a selection has been performed. By selecting this option, the **Divide Plates** window will appear.

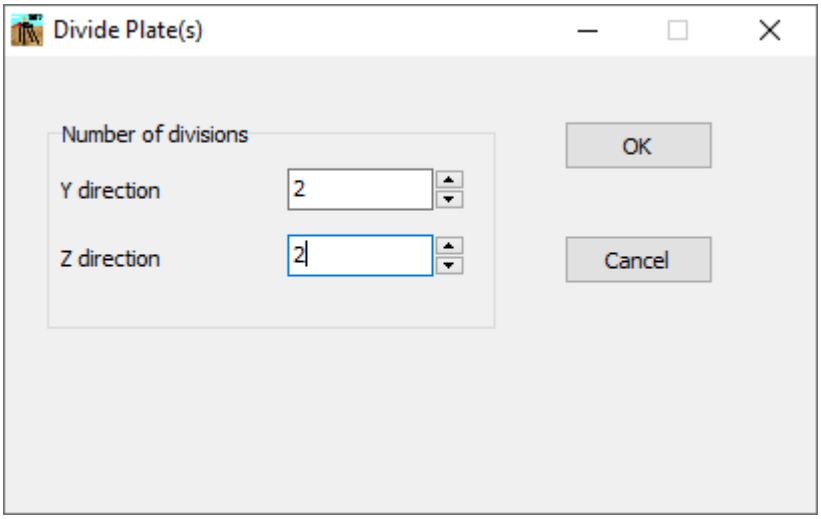
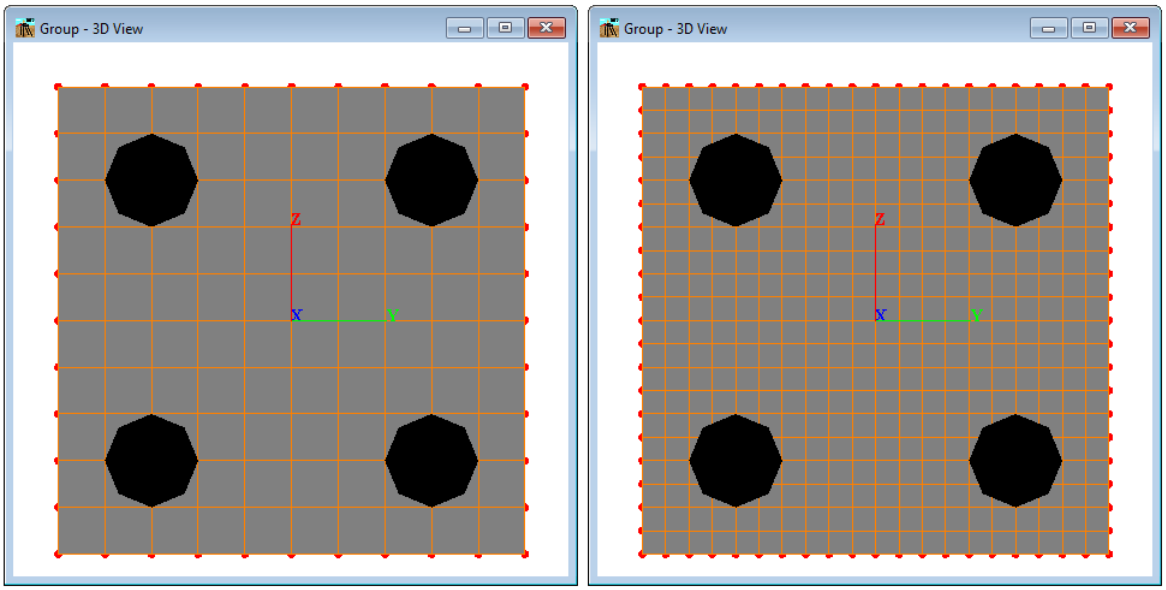


Figure 3.181 Divide Plates Window - Flexible Cap Version

Enter the number of divisions on the local Y and Z directions (refer to Figure 3.81 for axis definition and node numbering) and click “OK” to finish the plate division process, or click “Cancel” to cancel the process.

The program will divide only the selected plates; however, it is recommended to divide all the plates that belong to the mesh to guarantee mesh compatibility among all the plates.



a) Original Mesh b) After Divide Plates

Figure 3.182 Divide Plates option - Flexible Cap Version

3.4.18 Edit -Add Beam - Flexible Cap Version

Select the **Add Beam** item from the **Edit** menu. Then the **New Beam** window will appear.

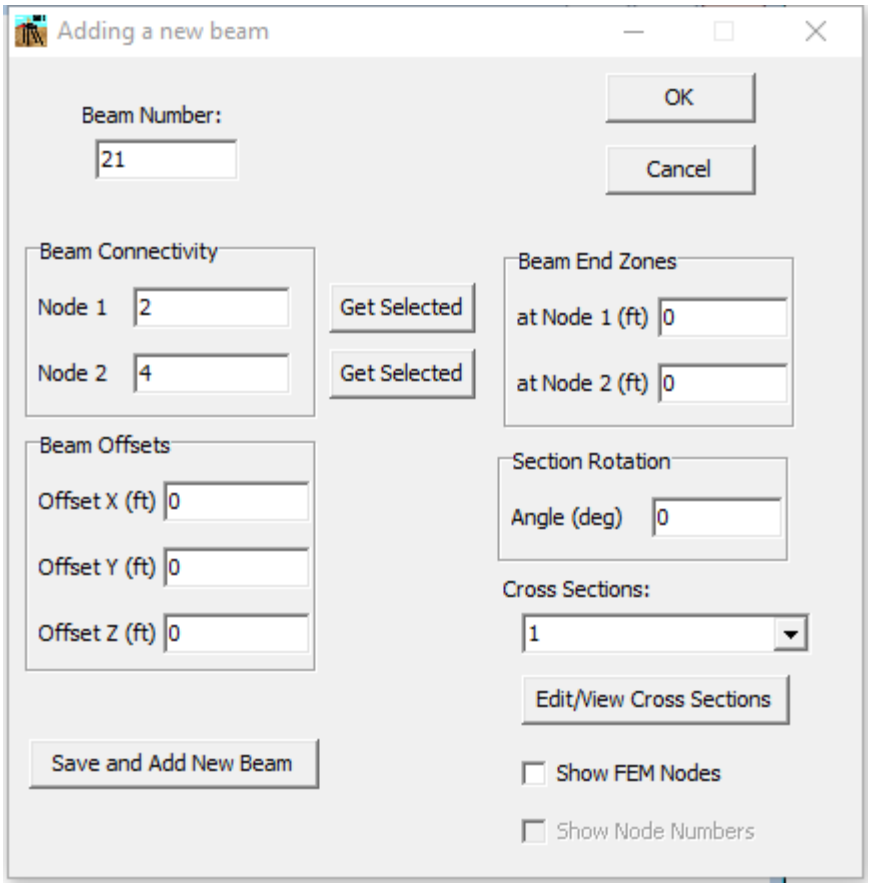


Figure 3.183 New Beam Window - Flexible Cap Version

Fill in the necessary information and click the **OK** button to save and close the window, or click the **Save and Add New Beam** button to save the information and return to the same window to add an additional beam, or click the **Cancel** button to cancel the process and close the window. Please note that the program may get the selected node by clicking on the **Get Selected** button.

3.4.19 Edit – Edit Beam

Select the **Edit Beam** item from the **Edit** menu. The **Edit Beam** window will appear (this window is similar to the **New Beam** window). In this window the user may change the input parameters.

The **Edit Beam** window will vary slightly whether there are some beams selected or not

If no beams are selected the window will show up the input parameters of the current beam (starting with beam number 1). The user may navigate throughout the different beams using the arrows at the right of Beam Number. Also, the user may jump to a beam number by selecting the Beam Number at the combo box under **Beam Number**. The current beam will be highlighted in the **3D View** window with yellow color.

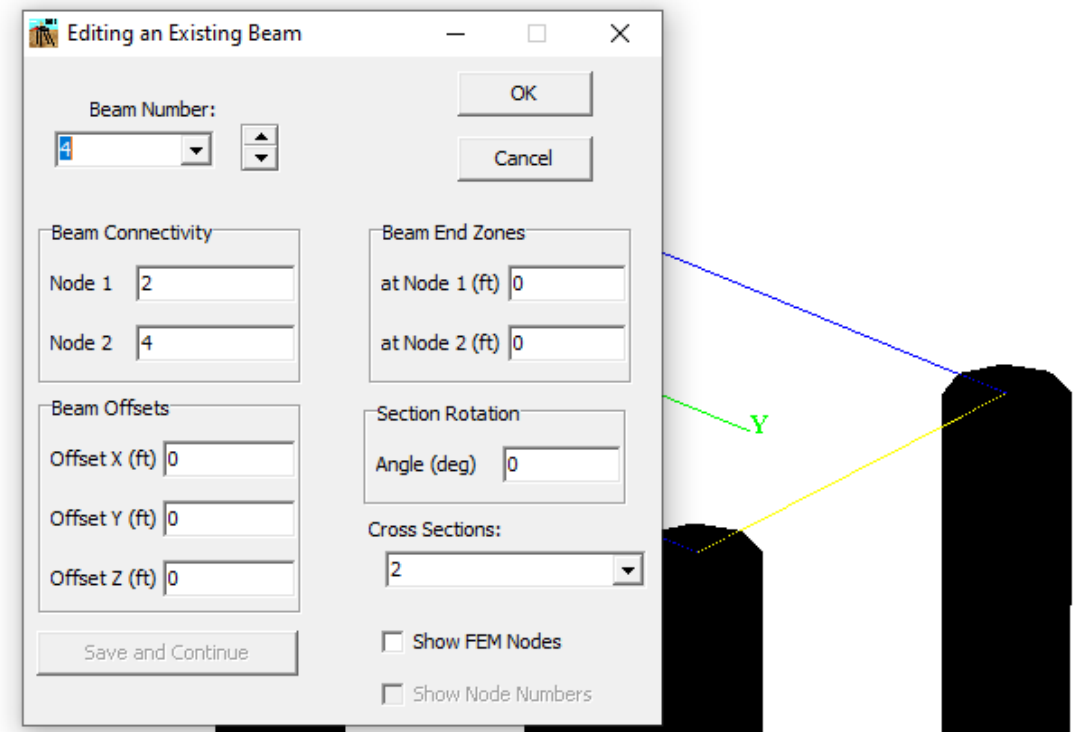


Figure 3.184 Edit Beam Window. No beams Selected - Flexible Cap Version

If beams are selected the window will show up the input information of the selected beams (which are highlighted in the **3D View** window in purple color). In this window the input parameters that are similar to all the selected plates are displayed in white background, while the input parameters that are not the same for all the selected plates are shown in gray background. Any change in the input parameters will be assigned for all the plates selected.

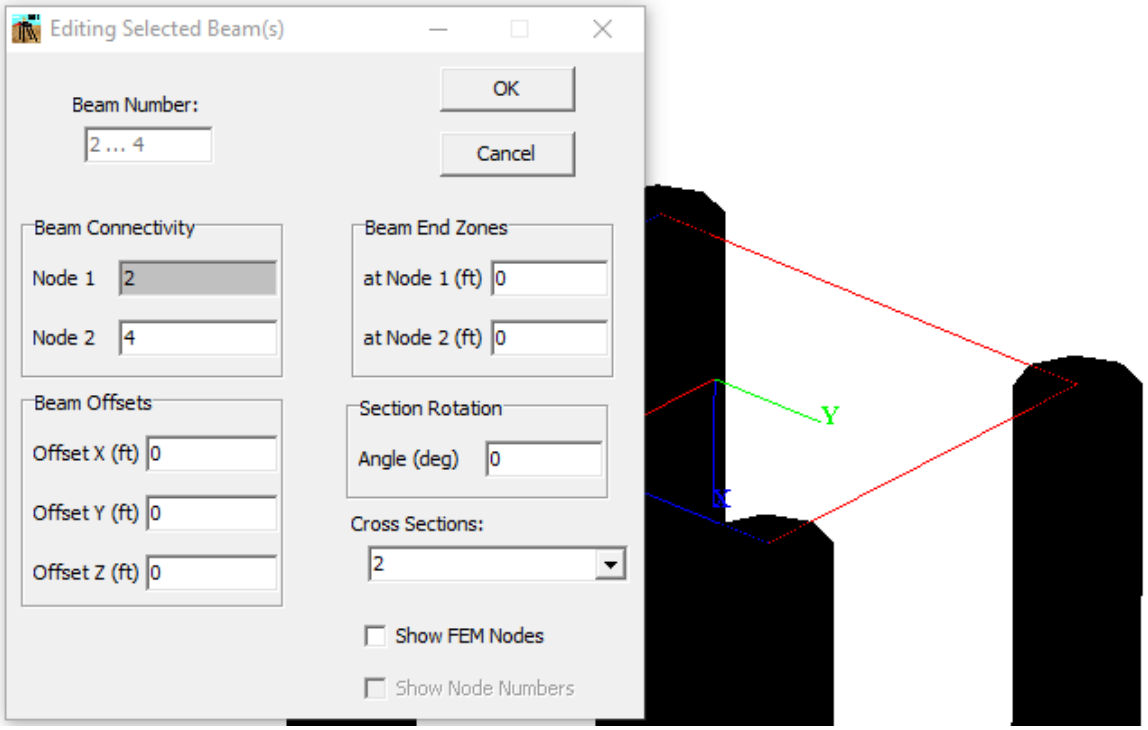


Figure 3.185 Edit Beam Window. Beams Selected - Flexible Cap Version

3.4.20 Edit - Delete Beams (Flexible Cap Version)

Please note that this option is enabled after a selection has been performed. Delete beams by choosing **Delete Beam(s)** under the **Edit** menu. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process.

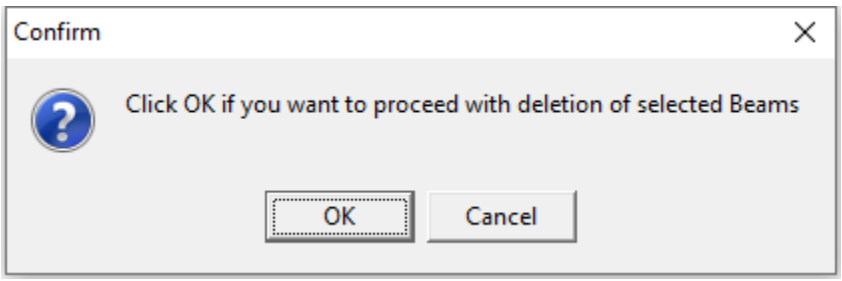


Figure 3.186 Window for confirmation of beams deletion - Flexible Cap Version

3.4.21 Edit - Divide Beams - Flexible Cap Version

Please note that this option is enabled after a selection has been performed. By selecting this option, the **Divide Beams** window will appear.

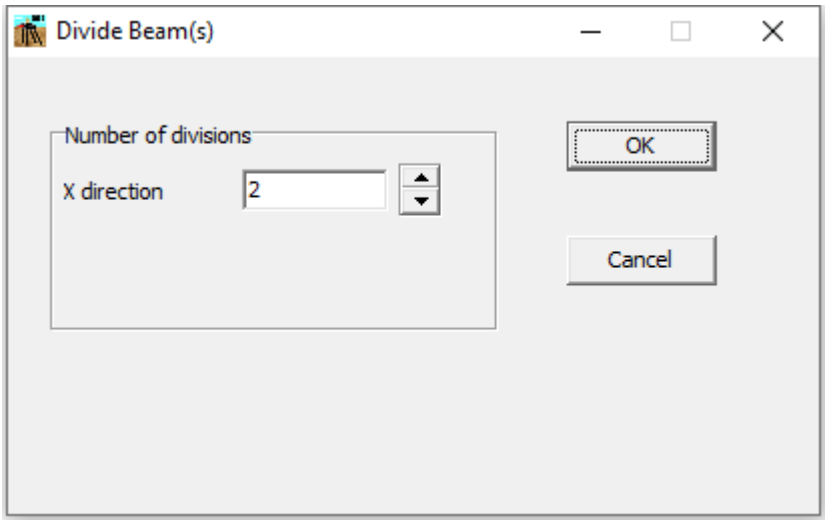
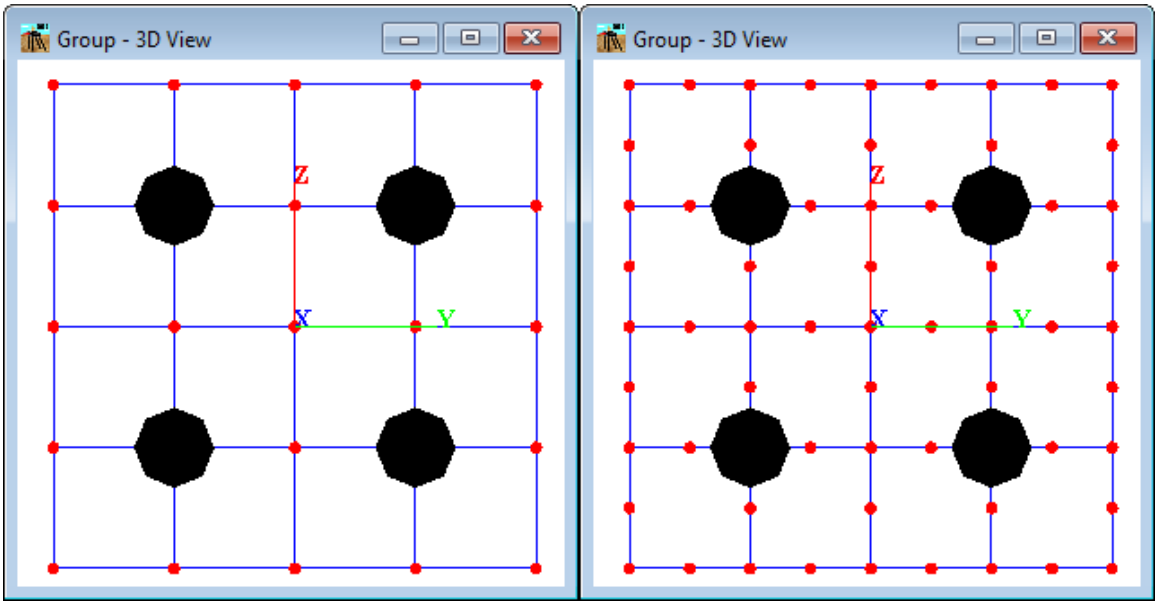


Figure 3.187 Divide Beams Window - Flexible Cap Version

Enter the number of divisions on the local X directions (refer to Figure 3.86 for axis definition and node numbering) and click “OK” to finish the beam division process, or click “Cancel” to cancel the process.



b) Original Mesh b) After Divide beams

Figure 3.188 Divide Beams option - Flexible Cap Version

3.4.22 Edit -Add Node - Flexible Cap Version

Select the **Add Node** item from the **Edit** menu. Then the **New Node** window will appear.

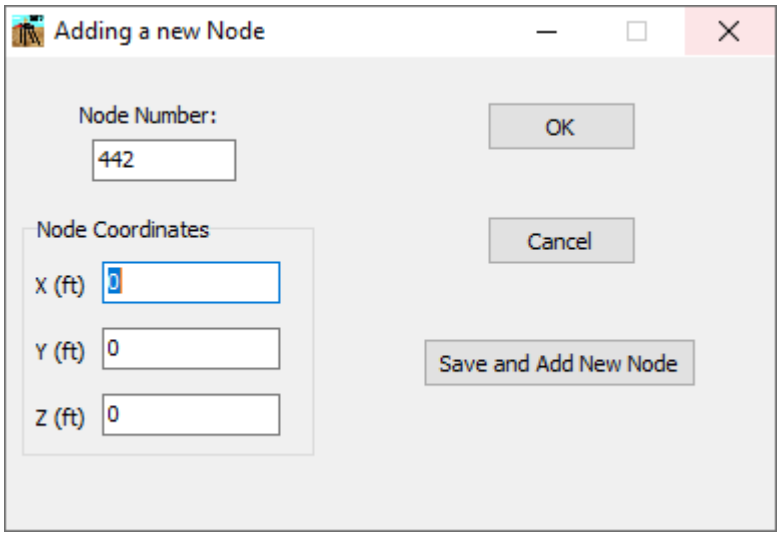


Figure 3.189 New Node Window - Flexible Cap Version

Fill in the necessary information and click the **OK** button to save and close the window, or click the **Save and Add New Node** button to save the information and return to the same window to add an additional plate, or click the **Cancel** button to cancel the process and close the window.

3.4.23 Edit – Edit Node

Select the **Edit Node** item from the **Edit** menu. The **Edit Node** window will appear (this window is similar to the **New Node** window). In this window the user may change the input parameters.

The **Edit Node** window will vary slightly whether there are some nodes selected or not

If no nodes are selected the window will show up the input parameters of the current node (starting with node number 1). The user may navigate throughout the different nodes using the arrows at the right of Node Number. Also, the user may jump to a node number by selecting the Node Number at the combo box under **Node Number**. The current node will be highlighted in the **3D View** window with yellow color.

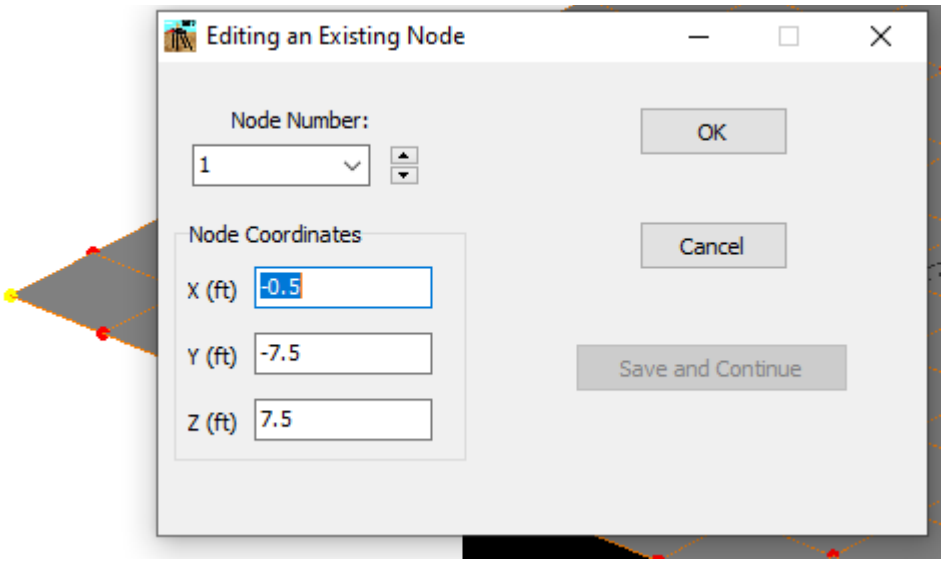


Figure 3.190 Edit Node Window. No nodes Selected - Flexible Cap Version

If nodes are selected the window will show up the input information of the selected nodes (which are highlighted in the **3D View** window in light blue color). In this window the input parameters that are similar to all the selected nodes are displayed in white background, while the input parameters that are not the same for all the selected nodes are shown in gray background. Any change in the input parameters will be assigned for all the nodes selected.

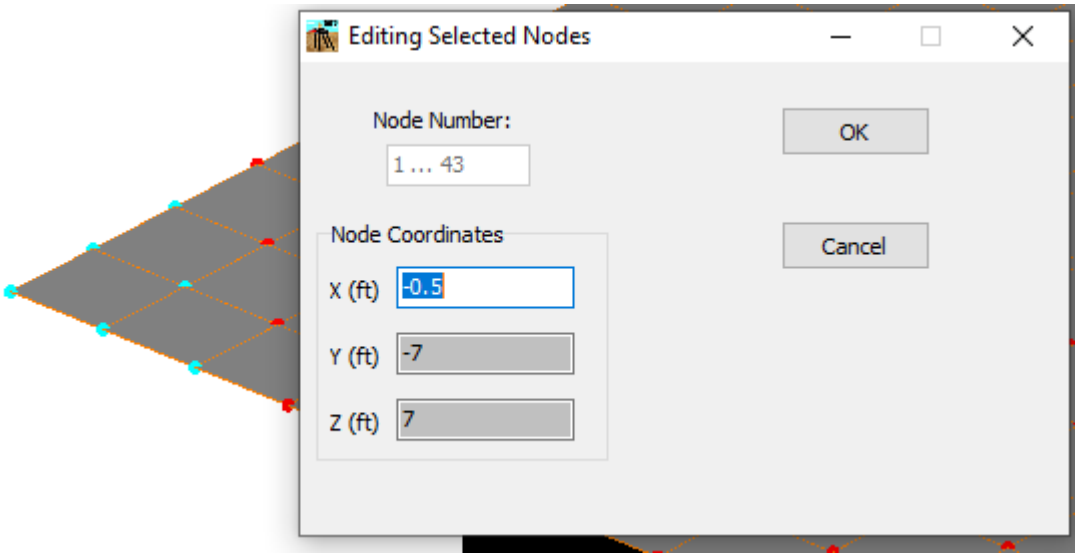


Figure 3.191 Edit Node Window. Nodes Selected - Flexible Cap Version

3.4.24 Edit - Delete Nodes - Flexible Cap Version

Please note that this option is enabled after a selection has been performed. Delete nodes by choosing **Delete Node(s)** under the **Edit** menu. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process. Note that only unconnected nodes will be deleted.

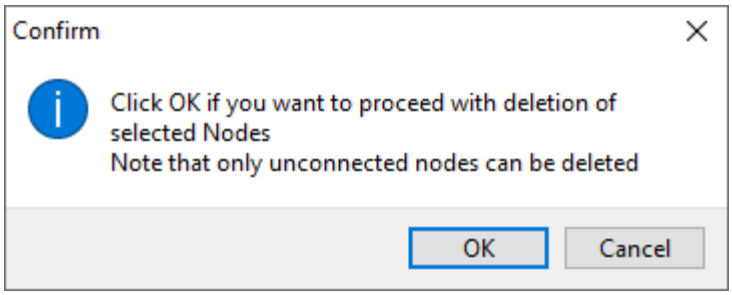


Figure 3.192 Window for confirmation of nodes deletion - Flexible Cap Version

3.4.25 Edit - Delete Unconnected Nodes - Flexible Cap Version

Delete unconnected nodes by choosing **Delete Unconnected Node(s)** under the **Edit** menu. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process.

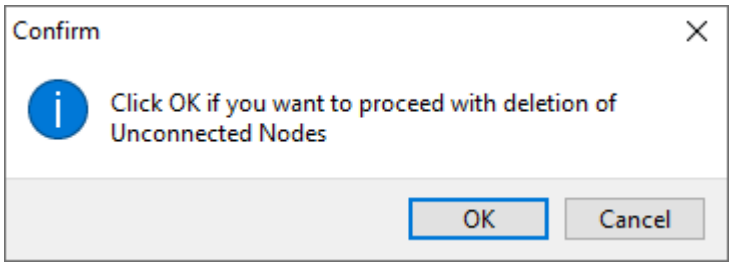


Figure 3.193 Window for confirmation of unconnected nodes deletion - Flexible Cap Version

3.4.26 Edit – Move All Selected – Flexible Cap Version

Please note that this option is enabled after selection of pile(s) node(s), plate(s) and/or beam(s). has been performed. Move the selected objects, by choosing **Copy Node(s), Plate(s) and/or (Beam(s)** from the **Edit** menu. The **Move All Selected** window will then appear.

By default, the **Absolute Coordinates** option is selected. Users should enter values for reference and target point. The program will estimate the increment by the difference between the target and the reference point.

If the **Relative Coordinates** option is selected, then the user must provide the coordinate increment.

Fill in the necessary fields and click "OK" to finish the moving process, or click "Cancel" to cancel the process.

Please note that some objects like beam and plates elements may not be moved if all the objects were not selected.

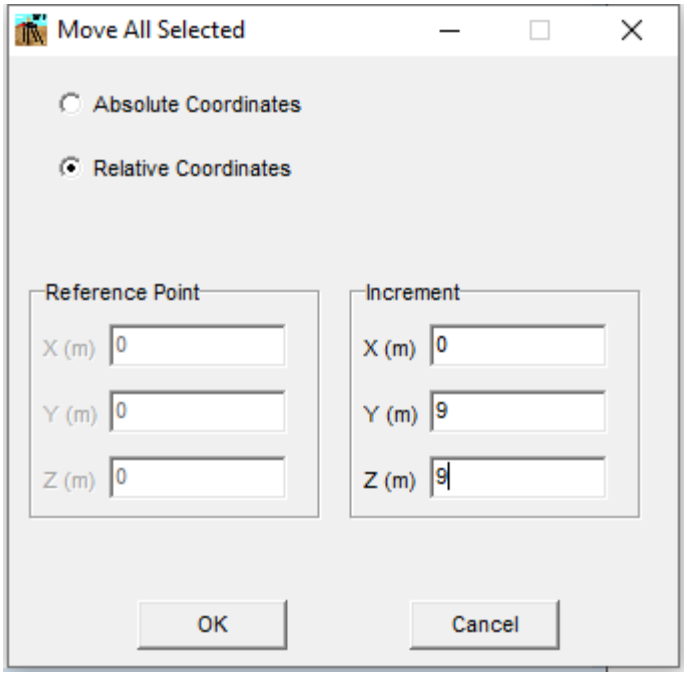


Figure 3.194 Move All Selected Window – Flexible Cap Version

3.4.27 Edit – Move All Selected – Flexible Cap Version

Please note that this option is enabled after selection of node(s), plate(s) and/or beam(s). has been performed. Copy the selected objects, by choosing **Copy Pile(s), Node(s), Plate(s) and/or (Beam(s)** from the **Edit** menu. The **Copy All Selected** window will then appear.

By default, the **Absolute Coordinates** option is selected. Users should enter values for reference and target point. The program will estimate the increment by the difference between the target and the reference point.

If the **Relative Coordinates** option is selected, then the user must provide the coordinate increment.

Fill in the necessary fields and click "OK" to finish the copying process, or click "Cancel" to cancel the process.

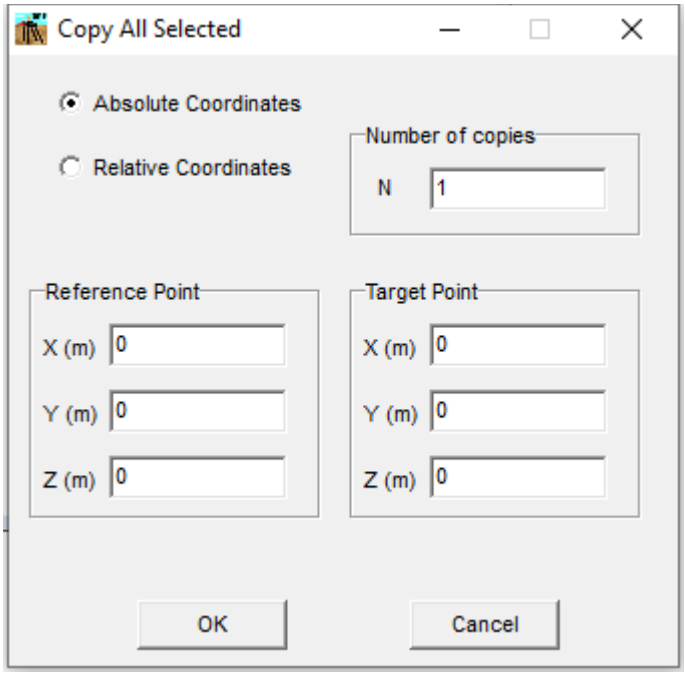


Figure 3.195 Copy All Objects Window – Flexible Cap Version

3.4.28 Edit - Delete All Selected - Flexible Cap Version

Please note that this option is enabled after a selection has been performed. Delete all selected by choosing **Delete All Selected** under the **Edit** menu. Then, a window for confirmation of the deletion will appear. Click "OK" to complete the deletion, or click "Cancel" to cancel the deletion process. Note that all elements (piles, plates and/or beams and maybe nodes selected will be deleted.

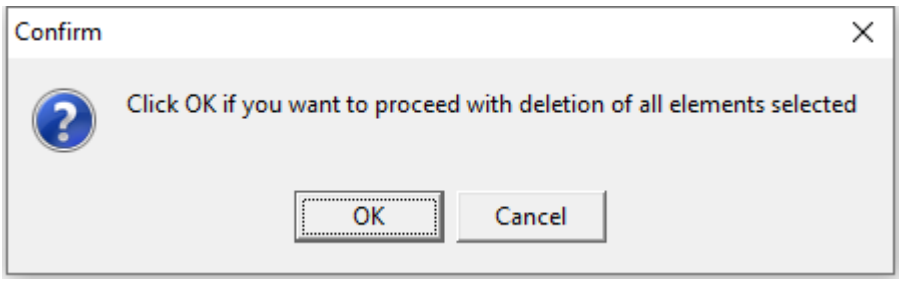


Figure 3.196 Window for confirmation of all selected elements deletion - Flexible Cap Version

3.4.29 Edit - Pile Sets

This submenu option allows to define pile sets that can be used to assign loading along the piles or to be used for selection

By selecting this option, the **Pile Sets Definition** window will appear

3.4.29.1 Pile Sets Definition Window

In this window the pile sets definition is carried out. A sample of this window is shown in Figure 3.197

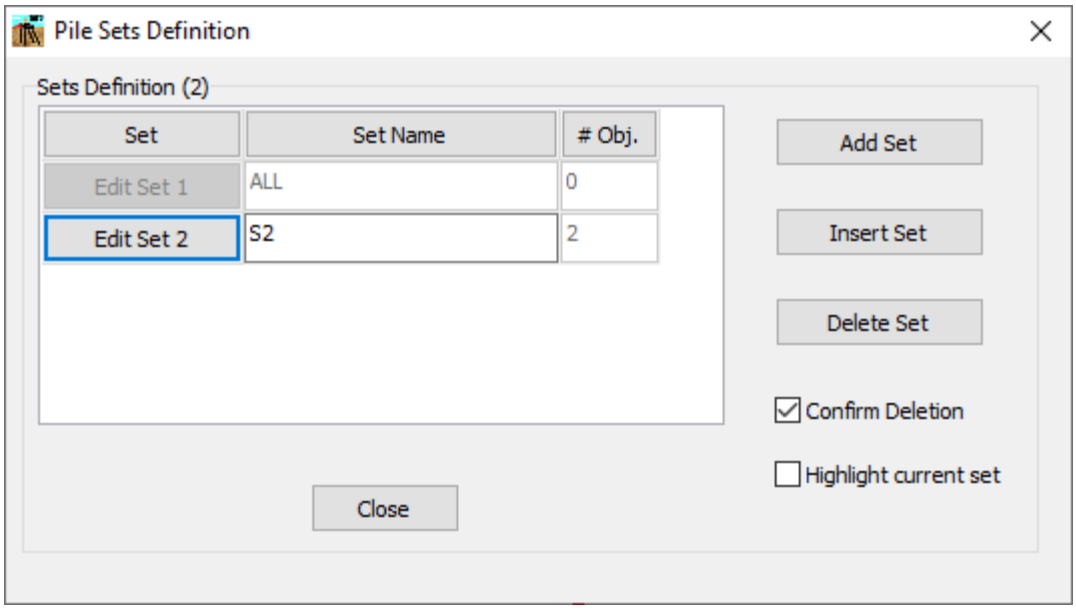


Figure 3.197 Pile Sets Definition Window

Sets Definition

- **Add Set.** Add a new row at the end of the list.
- **Insert Section.** Add a new row above the current row.
- **Delete Section.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a section.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

For each set (row) the following columns are available:

- **Edit Set**. Click on the **Edit Set** button to open the **Edit Pile Set** window for assigning the piles that will belong to the pile set being defined.
- **Set Name**. In the second column enter the name associated with the set being defined. The name should start with a letter and should not contain blanks. The Name ALL is reserved for set 1 and correspond to all the piles in the model.
- **# Obj**. In the third column the number of objects currently assigned to the set is displayed.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.4.29.2 Edit Pile Set Window

In this window the user can define the piles that belong to the current set. The set number is displayed at the top of the window. A sample of this window is shown in Figure 3.198.

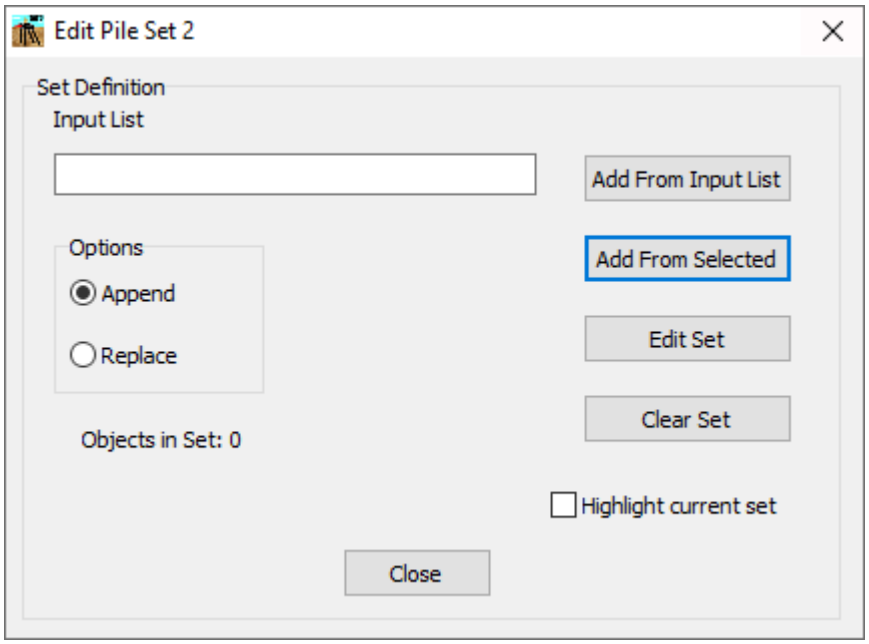


Figure 3.198 Edit Pile Set Window

Set Definition

- **Add From Input List.** Click on this button to add pile numbers to the pile set from the numbers entered in the Input List field. Entries in the Input List field are separated by the comma symbol ",", and a sequence is indicated by the column symbol ":". For instance, the sequence "1:5" will indicate the list 1,2,3,4,5 and the sequence "1:5:2" will indicate the sequence 1,3,5.
- **Add From Selected.** Click on this button to add object to the pile set from the selected piles.
- **Edit Set**. Click on this button to open a dialog to edit the pile set being defined.
- **Clear Set.** Click on this button to clear all the current entries of the pile set.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

Options

- **Append.** Select this option if new entries will be added to the current entries in the set.
- **Replace.** Select this option if new entries will be replacing the current entries in the set.

Close Button

Click on the **Close** button to close the window and go back to the Pile Sets Definition window.

3.4.30 Edit - Plate Sets - Flexible Cap Version

This submenu option allows to define plate sets that can be used to assign springs or loading on plates or to be used for selection

By selecting this option, the **Plate Sets Definition** window will appear

3.4.30.1 Plate Sets Definition Window

In this window the plate sets definition is carried out. A sample of this window is shown in Figure 3.199

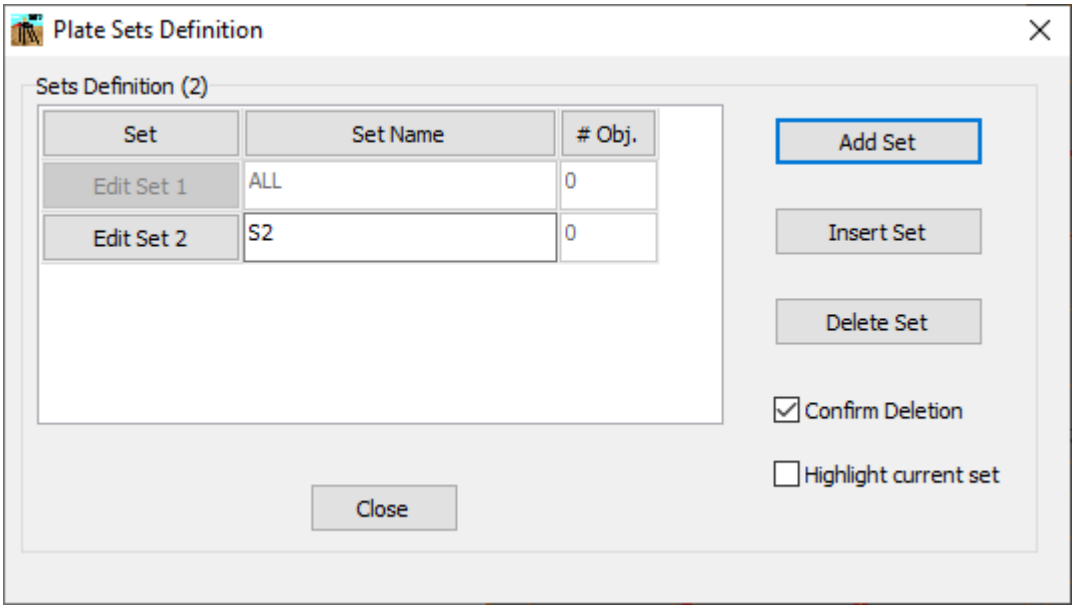


Figure 3.199 Plate Sets Definition Window - Flexible Cap Version

Sets Definition

- **Add Set.** Add a new row at the end of the list.
- **Insert Section.** Add a new row above the current row.
- **Delete Section.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a section.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

For each set (row) the following columns are available:

- **Edit Set**. Click on the **Edit Set** button to open the **Edit Plate Set** window for assigning the plates that will belong to the plate set being defined.
- **Set Name**. In the second column enter the name associated with the set being defined. The name should start with a letter and should not contain blanks. The Name ALL is reserved for set 1 and correspond to all the plates in the model.
- **# Obj**. In the third column the number of objects currently assigned to the set is displayed.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.4.30.2 Edit Plate Set Window

In this window the user can define the plates that belong to the current set. The set number is displayed at the top of the window. A sample of this window is shown in Figure 3.200.

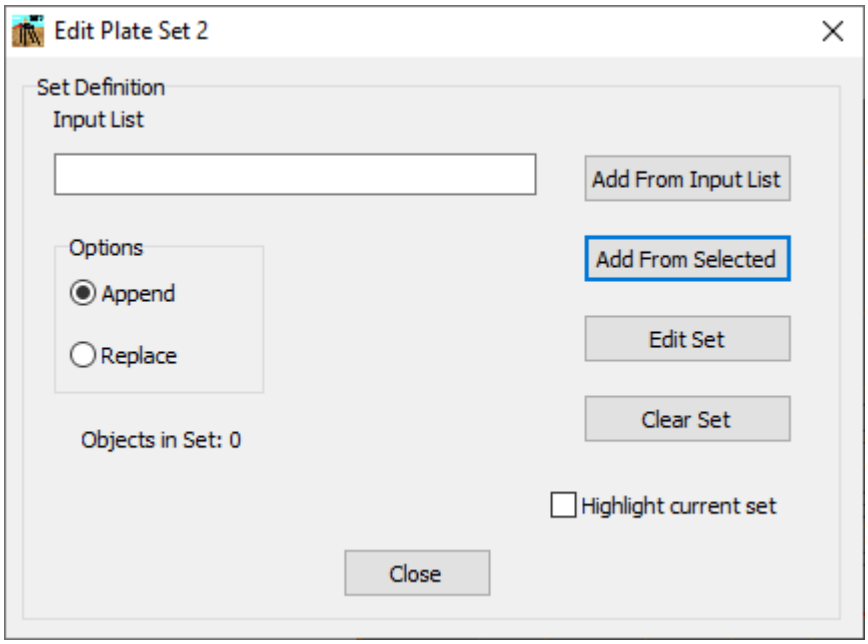


Figure 3.200 Edit Plate Set Window - Flexible Cap Version

Set Definition

- **Add From Input List.** Click on this button to add plate numbers to the plate set from the numbers entered in the Input List field. Entries in the Input List field are separated by the comma symbol ",", and a sequence is indicated by the column symbol ":". For instance, the sequence "1:5" will indicate the list 1,2,3,4,5 and the sequence "1:5:2" will indicate the sequence 1,3,5.
- **Add From Selected.** Click on this button to add object to the plate set from the selected plates.
- **Edit Set**. Click on this button to open a dialog to edit the plate set being defined.
- **Clear Set.** Click on this button to clear all the current entries of the plate set.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

Options

- **Append.** Select this option if new entries will be added to the current entries in the set.
- **Replace.** Select this option if new entries will be replacing the current entries in the set.

Close Button

Click on the **Close** button to close the window and go back to the Plate Sets Definition window.

3.4.31 Edit - Beam Sets - Flexible Cap Version

This submenu option allows to define plate sets that can be used to assign springs or loading on beams or to be used for selection

By selecting this option, the **Beam Sets Definition** window will appear

3.4.31.1 Beam Sets Definition Window

In this window the beam sets definition is carried out. A sample of this window is shown in Figure 3.201

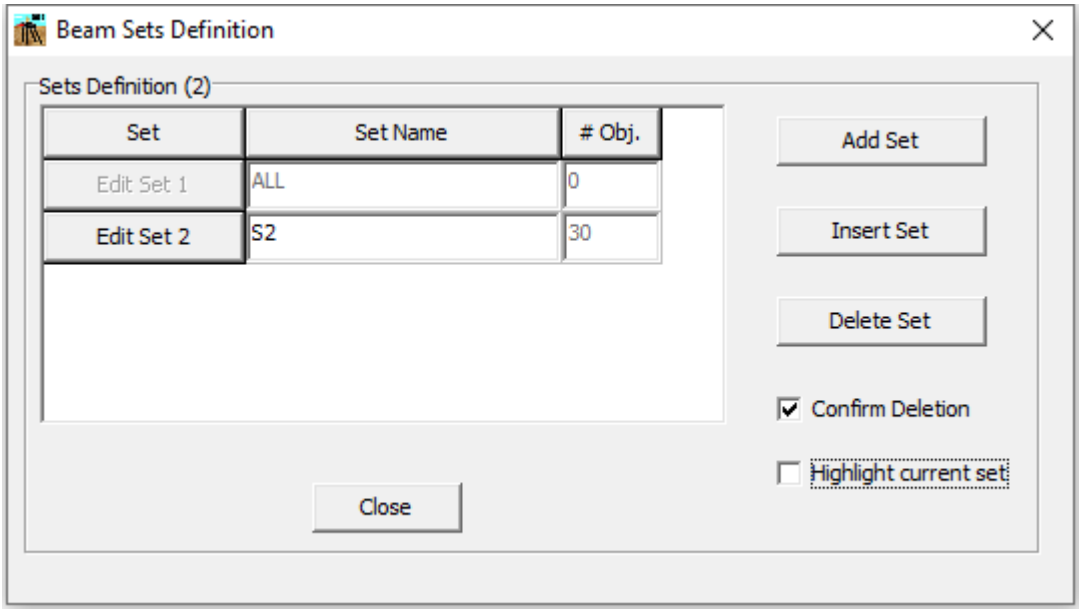


Figure 3.201 Beam Sets Definition Window - Flexible Cap Version

Sets Definition

- **Add Set.** Add a new row at the end of the list.
- **Insert Section.** Add a new row above the current row.
- **Delete Section.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a section.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

For each set (row) the following columns are available:

- **Edit Set**. Click on the **Edit Set** button to open the **Edit Plate Set** window for assigning the plates that will belong to the plate set being defined.
- **Set Name**. In the second column enter the name associated with the set being defined. The name should start with a letter and should not contain blanks. The Name ALL is reserved for set 1 and correspond to all the beams in the model.
- **# Obj**. In the third column the number of objects currently assigned to the set is displayed.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.4.31.2 Edit Beam Set Window

In this window the user can define the beams that belong to the current set. The set number is displayed at the top of the window. A sample of this window is shown in Figure 3.200.

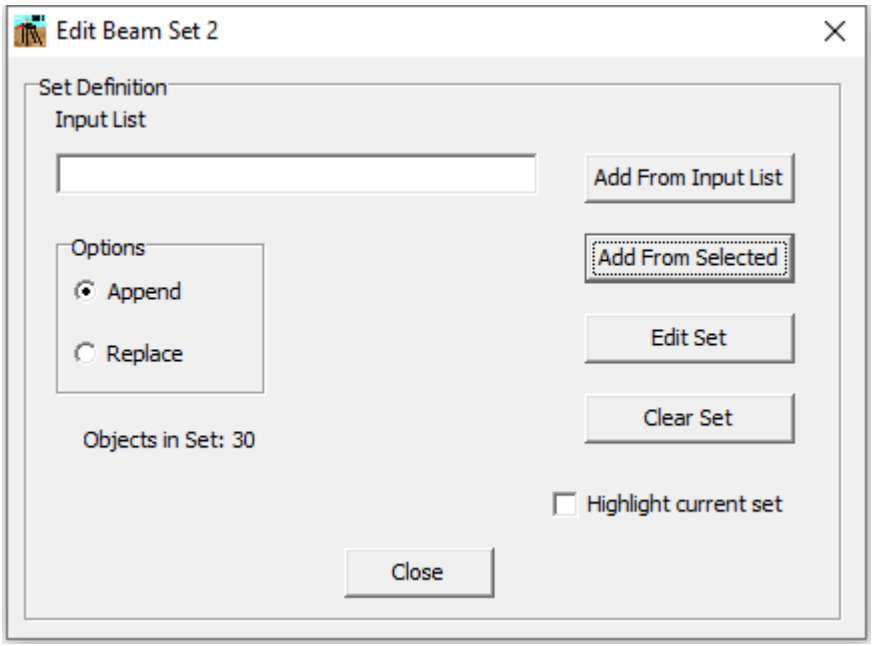


Figure 3.202 Edit beam Set Window - Flexible Cap Version

Set Definition

- **Add From Input List.** Click on this button to add beam numbers to the plate set from the numbers entered in the Input List field. Entries in the Input List field are separated by the comma symbol ",", and a sequence is indicated by the column symbol ":". For instance, the sequence "1:5" will indicate the list 1,2,3,4,5 and the sequence "1:5:2" will indicate the sequence 1,3,5.
- **Add From Selected.** Click on this button to add object to the beam set from the selected beams.
- **Edit Set**. Click on this button to open a dialog to edit the beam set being defined.
- **Clear Set.** Click on this button to clear all the current entries of the beam set.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

Options

- **Append.** Select this option if new entries will be added to the current entries in the set.
- **Replace.** Select this option if new entries will be replacing the current entries in the set.

Close Button

Click on the **Close** button to close the window and go back to the Beam Sets Definition window.

3.4.32 Edit - Node Sets - Flexible Cap Version

This submenu option allows to define node sets that can be used to assign loading along the nodes or to be used for selection

By selecting this option, the **Node Sets Definition** window will appear

3.4.32.1 Node Sets Definition Window

In this window the node sets definition is carried out. A sample of this window is shown in Figure 3.203.

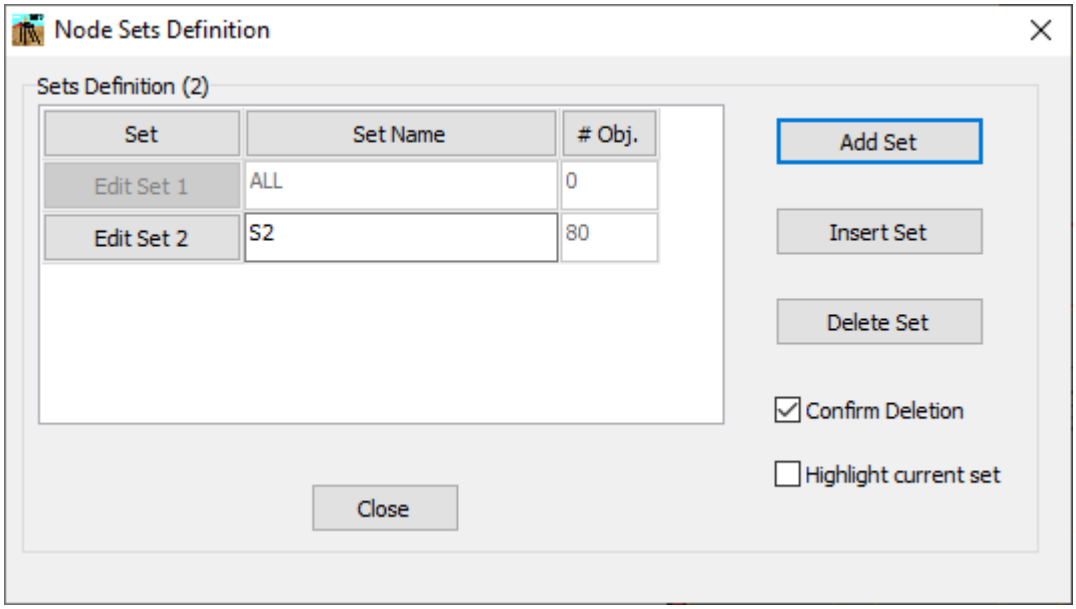


Figure 3.203 Node Sets Definition Window - Flexible Cap Version

Sets Definition

- **Add Set.** Add a new row at the end of the list.
- **Insert Section.** Add a new row above the current row.
- **Delete Section.** Delete the current row.
- **Confirm Deletion.** If this option is checked the program will require a confirmation before deleting a section.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

For each set (row) the following columns are available:

- **Edit Set**. Click on the **Edit Set** button to open the **Edit Node Set** window for assigning the nodes that will belong to the node set being defined.
- **Set Name**. In the second column enter the name associated with the set being defined. The name should start with a letter and should not contain blanks. The Name ALL is reserved for set 1 and correspond to all the piles in the model.
- **# Obj**. In the third column the number of objects currently assigned to the set is displayed.

Close Button

Click on the **Close** button to close the window and go back to the GROUP window.

3.4.32.2 Edit Node Set Window

In this window the user can define the nodes that belong to the current set. The set number is displayed at the top of the window. A sample of this window is shown in Figure 3.204.

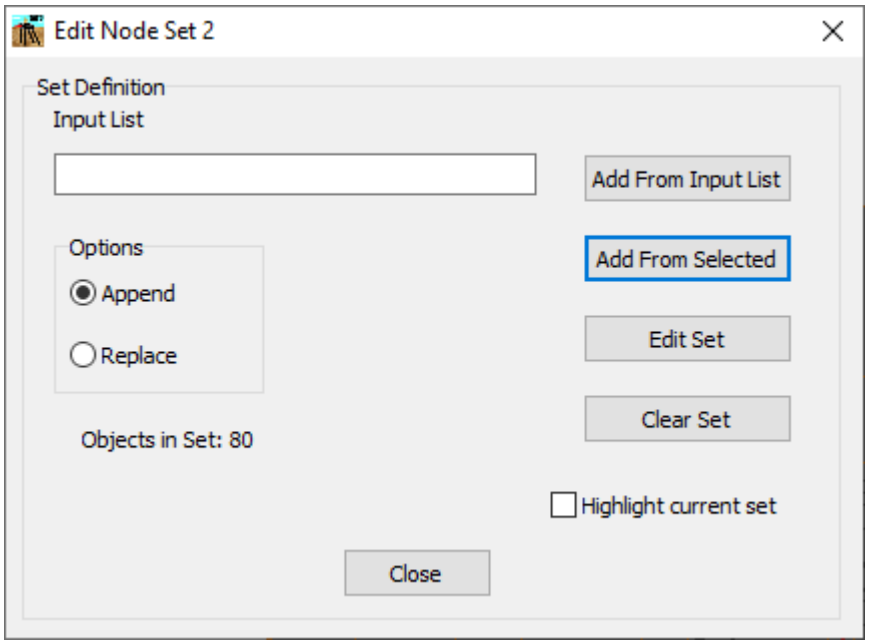


Figure 3.204 Edit Node Set Window - Flexible Cap Version

Set Definition

- **Add From Input List.** Click on this button to add node numbers to the node set from the numbers entered in the Input List field. Entries in the Input List field are separated by the comma symbol ",", and a sequence is indicated by the column symbol ":". For instance, the sequence "1:5" will indicate the list 1,2,3,4,5 and the sequence "1:5:2" will indicate the sequence 1,3,5.
- **Add From Selected.** Click on this button to add object to the node set from the selected nodes.
- **Edit Set**. Click on this button to open a dialog to edit the node set being defined.
- **Clear Set.** Click on this button to clear all the current entries of the node set.
- **Highlight Current Set.** If this option is checked the program will highlight (with a different color) the current set.

Options

- **Append.** Select this option if new entries will be added to the current entries in the set.
- **Replace.** Select this option if new entries will be replacing the current entries in the set.

Close Button

Click on the **Close** button to close the window and go back to the Node Sets Definition window.

3.5 Options Menu

This menu contains parameters that set preferences for the analytical run or problem description. Three types of submenu choices are available. Those ending with three periods (...) produce a sub-window for specification of further details. Those ending with a right arrow (▸) will show a drop-down list of possible choices. Finally, the selection of any other submenu option produces a check mark (✔) to indicate that it is active.

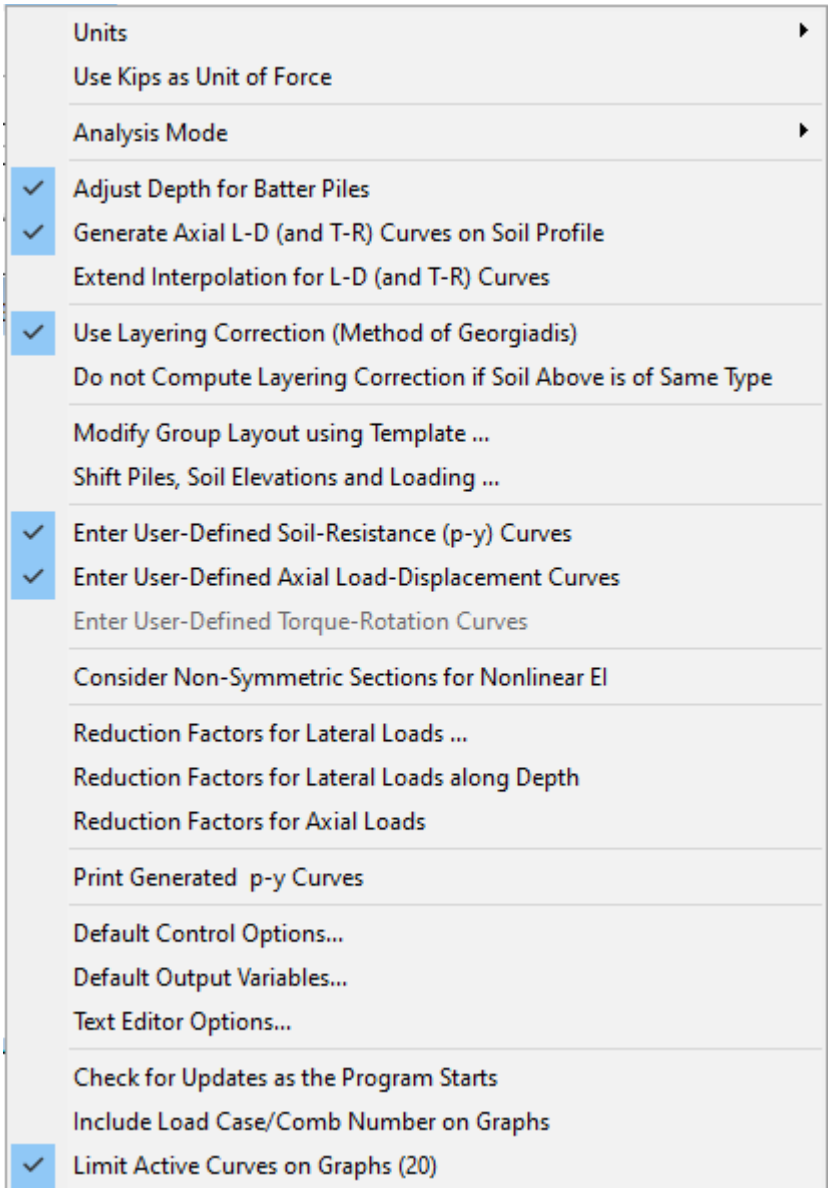


Figure 3.205 Options contained in the Options Menu

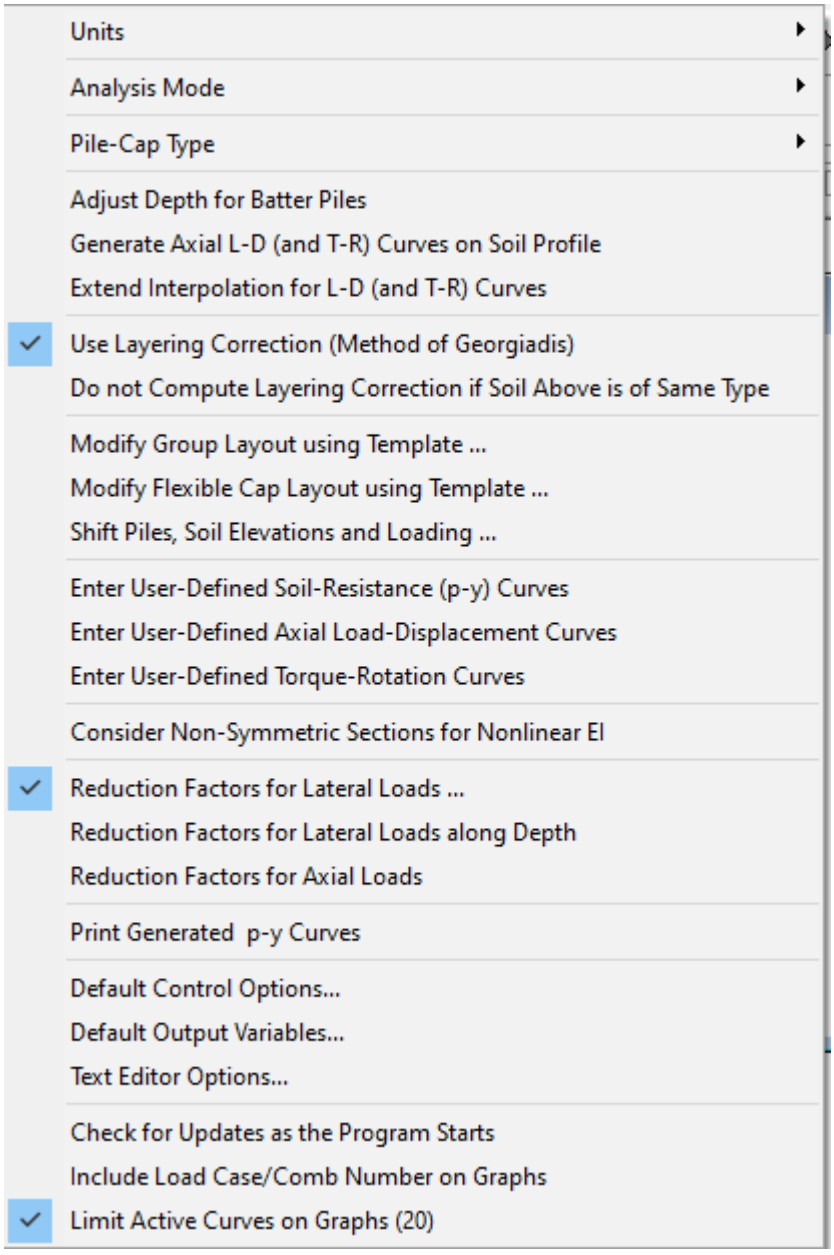


Figure 3.206 Options contained in the Options Menu - Flexible Cap Version

3.5.1 Options - Units

This option provides the user with a choice of specifying input data in either English Units or International Units (S.I. units).

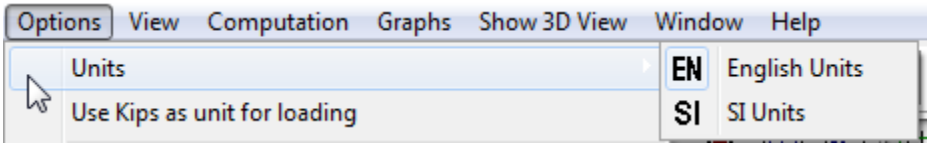


Figure 3.207 Units Option

In general, the following specific units of measurement will be used throughout the program:

Units	Length	Force
English (EN) (without kip option)	feet (ft) inches (in)	pounds (lbs)
English (EN) (with kip option)	feet (ft) inches (in)	kips (kip)
International (SI)	meters (m), millimeters (mm)	kilonewtons (kN)

Table 3.11 Unit systems

The user may change from one system of units to the other as many times as desired. The program will automatically convert all the appropriate input that was already specified for the user before the time of selection of a different system of units. Small conversion errors may appear if multiple changes of units are performed. The user must always check that the input data have been converted appropriately.

3.5.2 Options – Use Kips as Unit of Force

This option provides the user with a choice to use load in kips or pounds while using an English Units system. If the option is checked any variable related to loading (force) should be entered in kips, in the option is checked the loading (force) should be entered in pounds. The option files will be printed in a consistent manner.

The user may check or unchecked this option as many times as desired. The program will automatically convert all the appropriate input that was already specified for the user before the time of selection of a different option. Small conversion errors may appear if multiple changes of units are performed. The user must always check that the input data have been converted appropriately.

3.5.3 Options - Analysis Mode

GROUP allows the user to analyze the pile group either using 2-dimensional model or using 3-dimensional model. The program will use the 3-dimensional mode as the default condition.

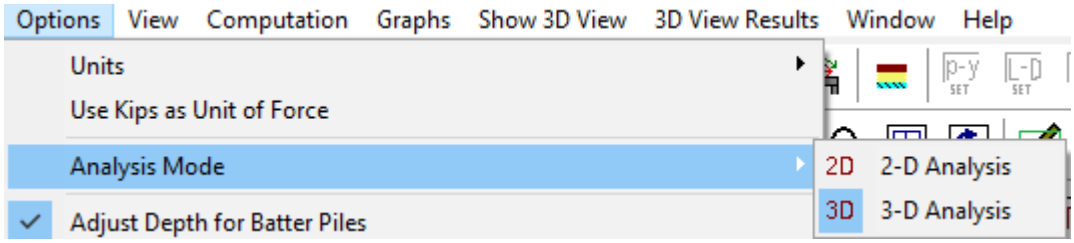


Figure 3.208 Analysis Mode Option

For the 2-dimensional model, the user enters only three load components (vertical load in the X-direction, lateral load in the Y-direction and bending moment in the Z-direction (X-Y plane). A two-dimensional model is recommended for conditions where piles are arranged symmetrically in a pile group. For the 3-dimensional analysis both loads and moments in the X, Y and Z directions can be entered.

Some menu options and windows will change accordingly to the analysis mode selected.

When the user changes the Analysis Mode, the program will ask if the user wants to convert the pile group layout to the corresponding new mode. For instance, the following window appears while changing from 3D Analysis to a 2D Analysis. By clicking "Yes", the pile layout will be converted; by clicking "No", the pile configuration will be lost and by clicking "Cancel" the Mode change will be cancelled. It is recommended to check the converted model for accuracy.

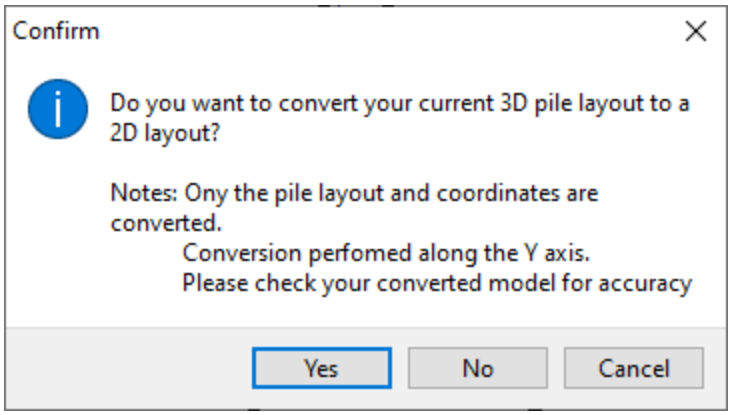


Figure 3.209 Message window to confirm conversion from 3D layout to a 2D layout

3.5.4 Options – Pile Cap Type - Flexible Cap Version

The standard version of the program GROUP 2022 consider the pile-cap using rigid assumptions. The flexible cap version of the program GROUP 2022 allows the user to analyze the pile cap either using a rigid model or a flexible model. The program will use the rigid model as the default condition. The flexible model is only available for 3-dimensional analysis.

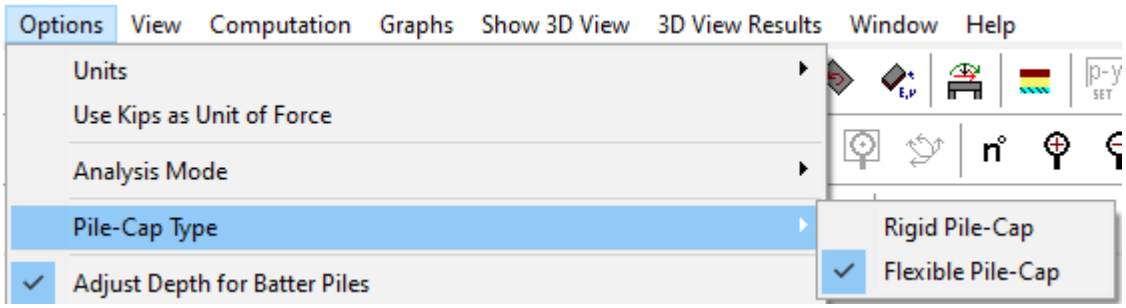


Figure 3.210 Pile-Cap Type Options - Flexible Cap Version

For the rigid model, the movement of the pile-cap can be represented by the movement of the origin of coordinates. The origin of coordinates has 3 degrees of freedom for a 2-dimensional analysis and 6 degrees of freedom for a 3-dimensional analysis.

For the flexible model, the pile-cap is discretized using a finite element model with 4 node plate elements and/or beam elements. Each node of the finite element mesh considers 6 degrees of freedom: 3 translations and 3 rotations. The piles are connected to the pile-cap through rigid elements.

3.5.5 Options – Adjust Depth for Batter Piles

This option will be checked under program default conditions. The check mark indicates that the program will use the vertical projection of the piles in order to estimate the soil resistance, p-y curves, t-z curves and q-w curves. If this option is unchecked the program will use the actual length of the pile to estimate soil resistance, as was performed in previous versions of GROUP.

3.5.6 Options – Generate Axial L-D (and T-R) curves on Soil Profile

This option will be checked under program default conditions. The check mark indicates that the program will generate the axial load versus displacement curves (and torque versus rotation curves) considering the soil profile for each pile and the batter angle. If this option is unchecked the program generate one curve per pile property, as was performed in previous versions of GROUP.

3.5.7 Extend Interpolation for L-D (and T-R) curves

This option will be checked under program default conditions. The check mark indicates that the program will extend the interpolation of the L-D (and T-R curves) towards the maximum defined. The program will use the value of load (torque) corresponding the to the maximum displacement (rotation) if the displacement (rotation) is larger than the maximum defined. The program will use the value of load (torque) corresponding the to the maximum displacement (rotation) if the displacement (rotation) is smaller than the minimum defined. If this option is not checked the program will use a value of zero in those events (as previous versions of GROUP)

3.5.8 Use Layering Correction (Method of Georgadis)

This option will be checked under program default conditions. The check mark indicates that the program will use layering correction. By unchecking this option, the program will not use a layering correction, so the equivalent depth of each layer will be equal to actual depth of the layer.

3.5.9 Do not use Layering Correction is soil above is of same type

This option will be unchecked under program default conditions. The check mark indicates that the program will not estimate layering correction if soil above of current layer is of same material. By unchecking this option, the program will use a layering correction even if the soil above is of same type.

3.5.10 Options - Modify Group Layout using Template

This option allows to modify the existing layout for a regular rectangular or circular pile group configuration. After selecting this option, the Modify Group Layout using Template Window will appear. The window is similar to the New from Template window described in item 3.2.2 (File - New from Template). Please refer to item 3.2.2 (File - New from Template) for a full description on the window. The new option in this window is the ability to define a new pile as shown in Figure 3.211 or select from a previous pile properties as shown in Figure 3.212.

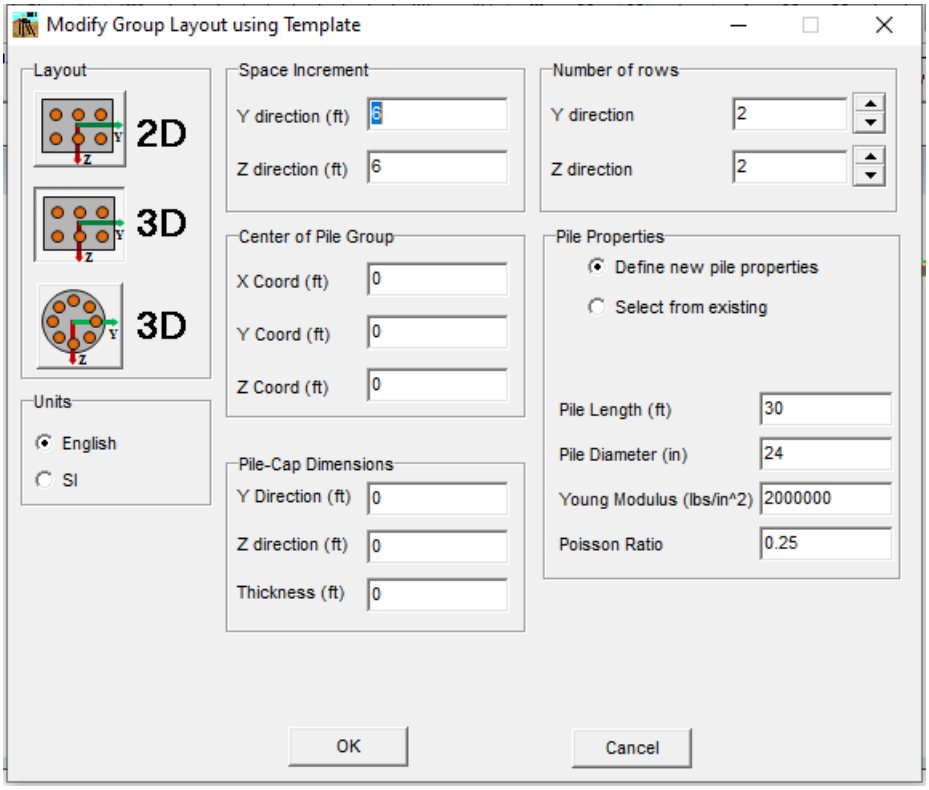


Figure 3.211 Modify Group Layout using Template Window. Defining new pile properties.

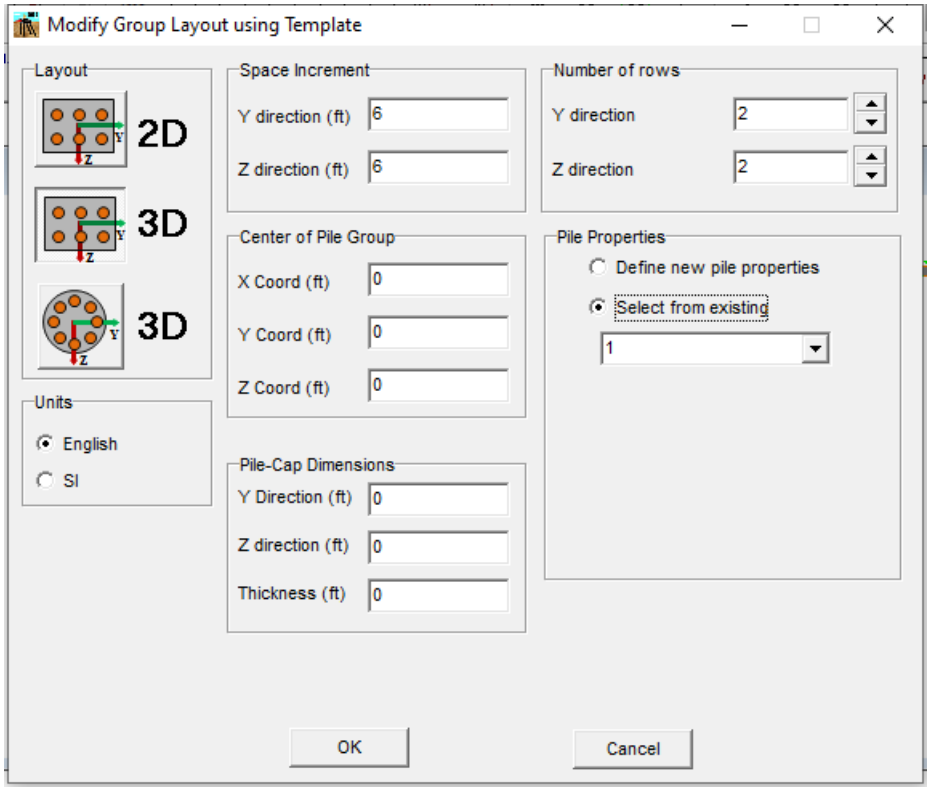


Figure 3.212 Modify Group Layout using Template Window. Using existing pile properties.

3.5.11 Options - Modify Group Layout using Template – Flexible Cap Version

This option allows to modify the existing layout for a regular rectangular or circular pile group configuration. After selecting this option, the Modify Group Layout using Template Window will appear. The window is similar to the New from Template window described in item 3.2.3 (File - New from Template – Flexible Cap Version). Please refer to item 3.2.3 (File - New from Template – Flexible Cap Version) for a full description on the window. The new option in this window is the ability to define a new pile as shown in Figure 3.213 Figure 3.211or select from a previous pile properties as shown in Figure 3.214.

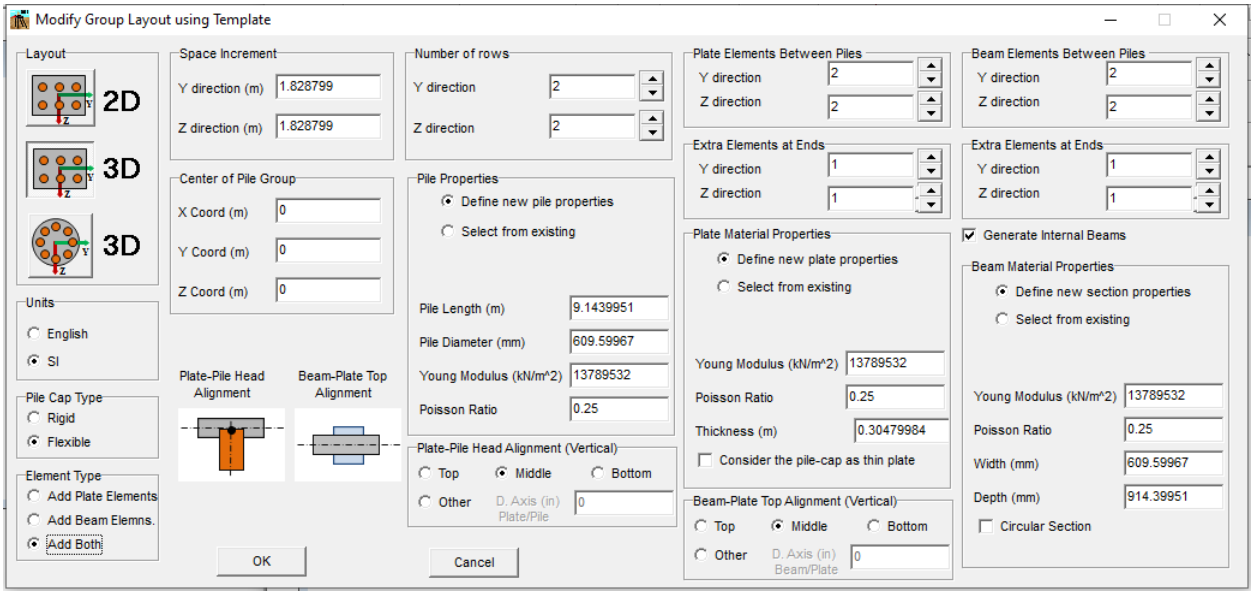


Figure 3.213 Modify Group Layout using Template Window. Defining new properties – Flexible Cap Version

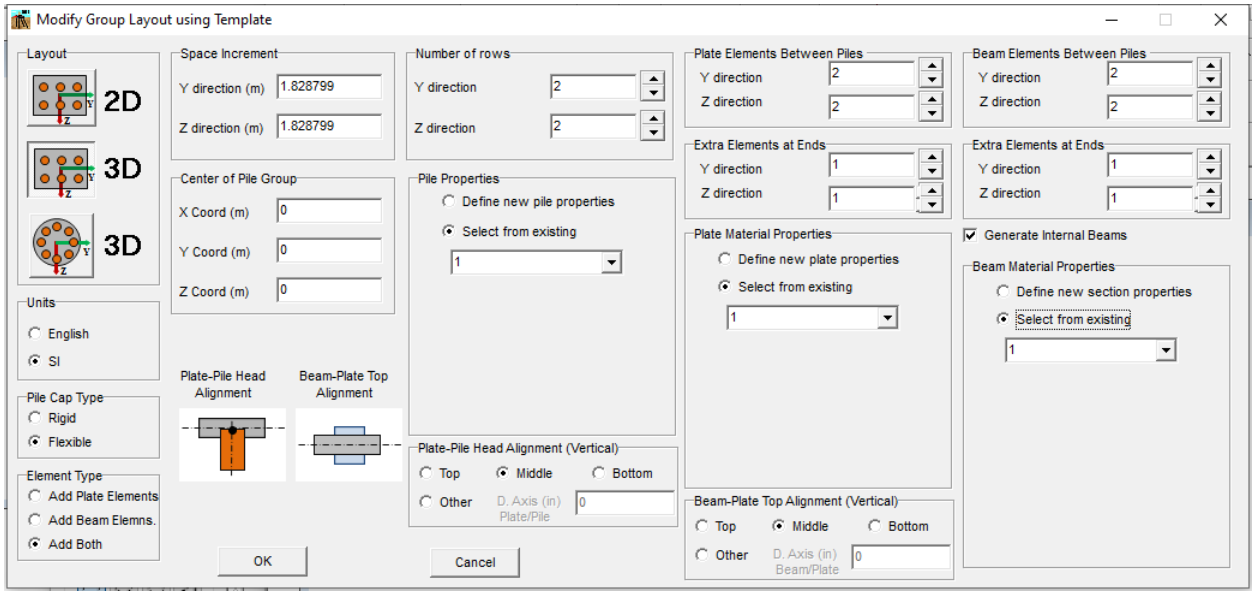


Figure 3.214 Modify Group Layout using Template Window. Using existing properties – Flexible Cap Version.

3.5.12 Options - Modify Flexible Cap Layout using Template - Flexible Cap Version

This option allows to modify the existing flexible pile-cap finite element mesh. This option will only define rectangular pile-caps. After selecting this option, the Modify Flexible Cap Layout using Template Window will appear.

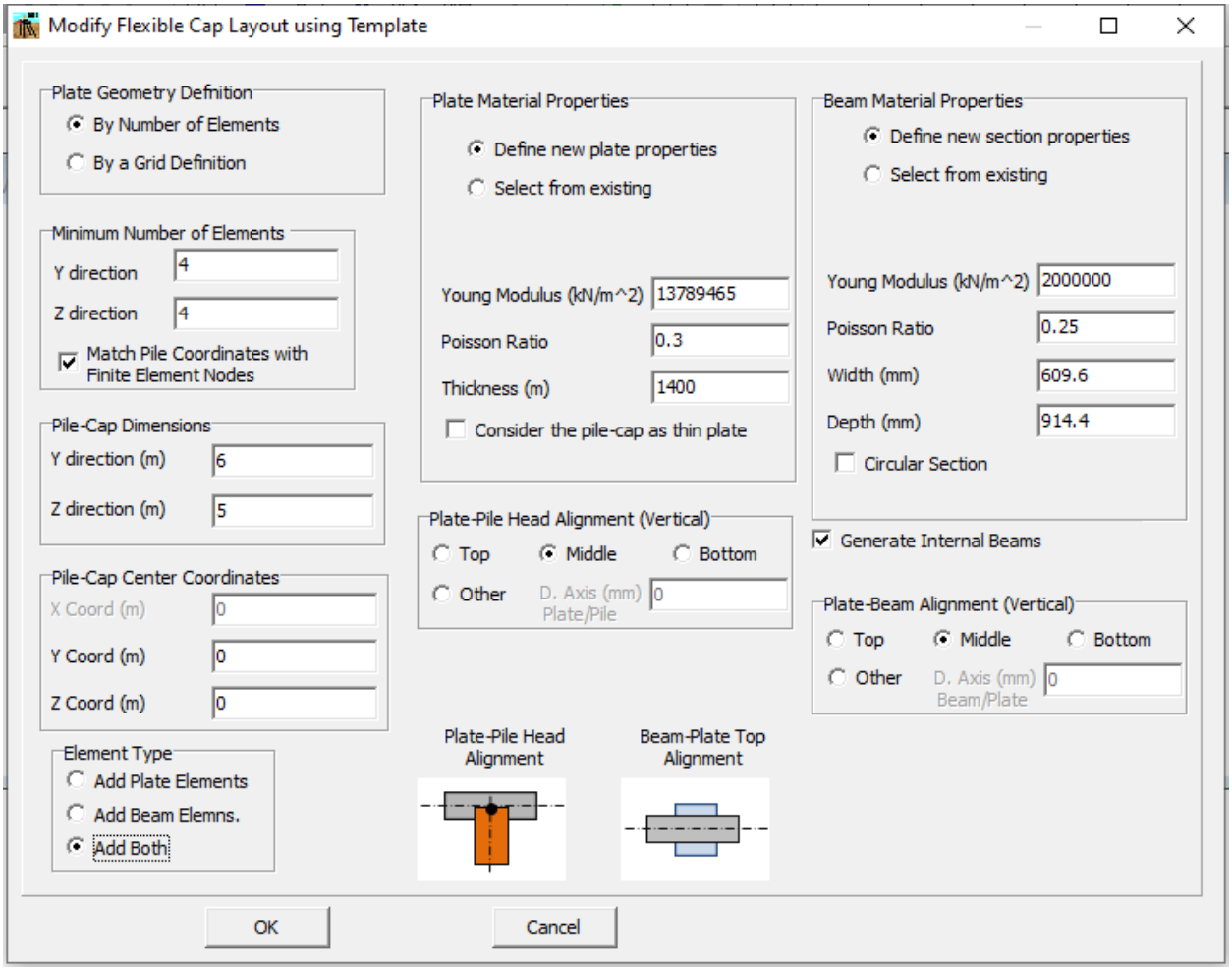


Figure 3.215 Modify Flexible Cap Layout using Template Window - Flexible Cap Version

The following parameters can be defined in the Modify Flexible Cap Using Template window:

- **Plate Geometry Definition**: Select between the options: By Number of Elements or By Grid Definition. Various entries may show up or hide based on the selection.

- **Minimum Number of Plate Elements**: Enter the Minimum Number of Plates for generation of the rectangular finite element mesh. Enter number of elements for Y and Z directions.
- **Match Pile Coordinates with Finite Element Nodes**: Check this option (default) to ensure a full compatibility between the finite element mesh of the pile-cap and pile head coordinates.
- **Pile-Cap Dimension**: Enter the dimensions of the rectangular pile-cap in the Y and Z directions.
- **Pile-Cap Center Coordinates**: Enter the coordinates for the center of the pile-cap (Y and Z coordinates).
- **Grid Definition**: Enter the coordinates for the coordinates that will define the finite element mesh for the Y and Z directions.

Other options of the window are similar to the New from Template window described in item 3.2.3 (File - New from Template – Flexible Cap Version). Please refer to item 3.2.3 (File - New from Template – Flexible Cap Version) for a description on those options.

Note that the pile-cap is considered to be horizontal and the X coordinate of all the nodes that belong to the pile cap will be estimated based on the X coordinate of the pile group (minimum X value for all piles). A rigid element will be used to connect each pile head with a vertical projected node (recommended for compatibility) or inside plate or inside beam that belongs to the finite element mesh defining the flexible pile-cap.

3.5.13 Options – Shift Piles and Soil Elevations

This option allows the user to move up (or down) the pile head coordinates, soil coordinates and loading on pile cap.

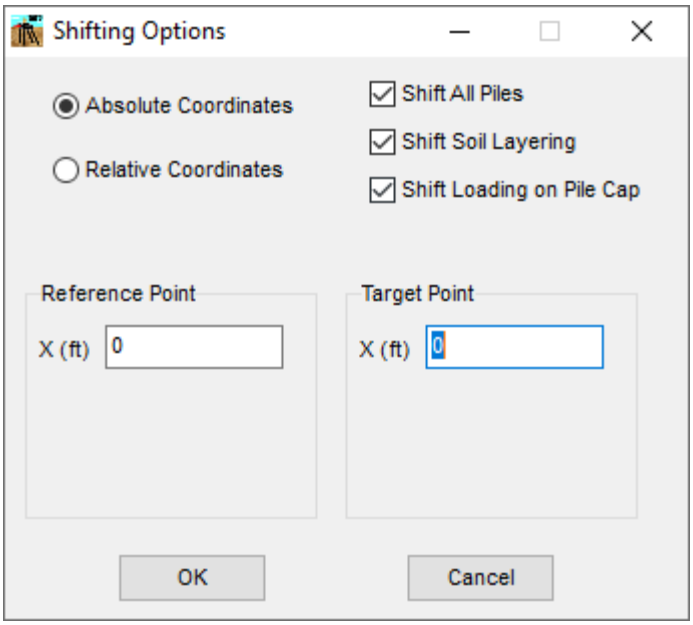


Figure 3.216 Shifting Options

For a model with a flexible pile-cap model (Flexible Cap Version). The vertical coordinates of the nodes that belong to the finite element mesh can also be shifted.

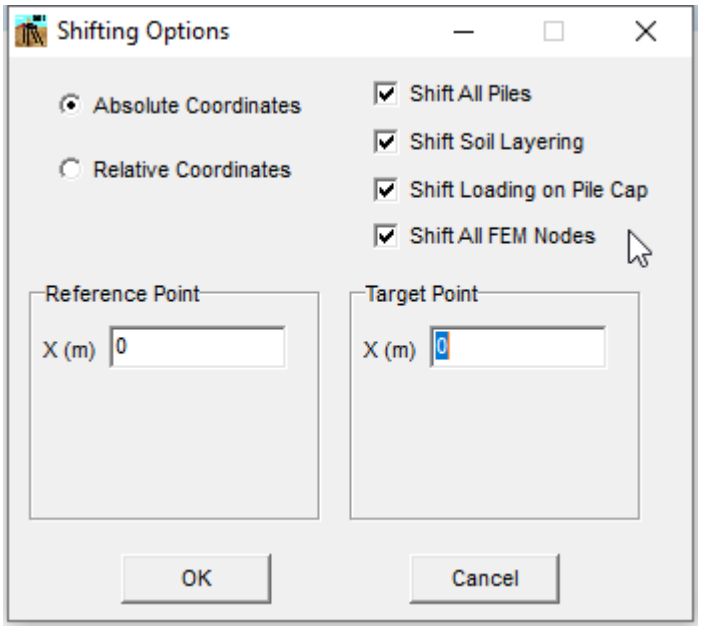


Figure 3.217 Shifting Options - Flexible Cap Version

Click OK to shift the checked data based on the values entered.

3.5.14 Options - Enter User-Defined Soil-Resistance (p-y) Curves

This option will be unchecked under program default conditions. The check mark indicates that the model may include User-Defined soil-resistance curves. If this option is check-marked the option **Sets of Soil-Resistance (p-y) Curves** in the **Data** menu is activated. Extra information will also need to be inputted in the **Soil-Resistance (p-y) Curve Set** column of the **Pile Groups** window under the **Data menu.**

3.5.15 Options - Enter User-Defined Load-Displacement Curves

This option will be unchecked under program default conditions. The check mark indicates that the program will accept User-Defined load-displacement curves. If this option is check-marked the option **Sets of Axial Load-Displacement Curves** in the **Data** menu is activated. Extra information will also need to be inputted in the **Axial Load-Settle. Curve Set** column of the **Pile Groups** window under the **Data menu.**

3.5.16 Options - Enter User-Defined Torque-Rotation Curves

This option is enabled for a 3-dimensional model. This option will be unchecked under program default conditions. The check mark indicates that the program will accept User-Defined torque-rotation curves. If this option is check-marked the option **Sets of Torque-Rotation Curves** in the **Data** menu is activated. Extra information will also need to be inputted in the **Torque-Rotation Curve Set** column of the **Pile Groups** (3D) window under the **Data menu.**

3.5.17 Options – Consider Non-Symmetric Sections for Nonlinear EI

This option will be unchecked under program default conditions. The check mark indicates that the program will be able to estimate nonlinear bending stiffness for non-symmetric sections, for this case the moment curvature may be non-symmetric and it will have positive and negative values. By unchecking this option, the program will be able to consider only symmetric sections for the estimation

of nonlinear bending stiffness, for this case the moment curvature estimated will be assumed to be symmetric and it will have only positive values.

3.5.18 Options - Reduction Factors for Lateral Loads

By selecting this option, the Reduction Factors for Lateral Loads Window will appear.

Under default conditions, group-reduction factors will not be considered for lateral loading.

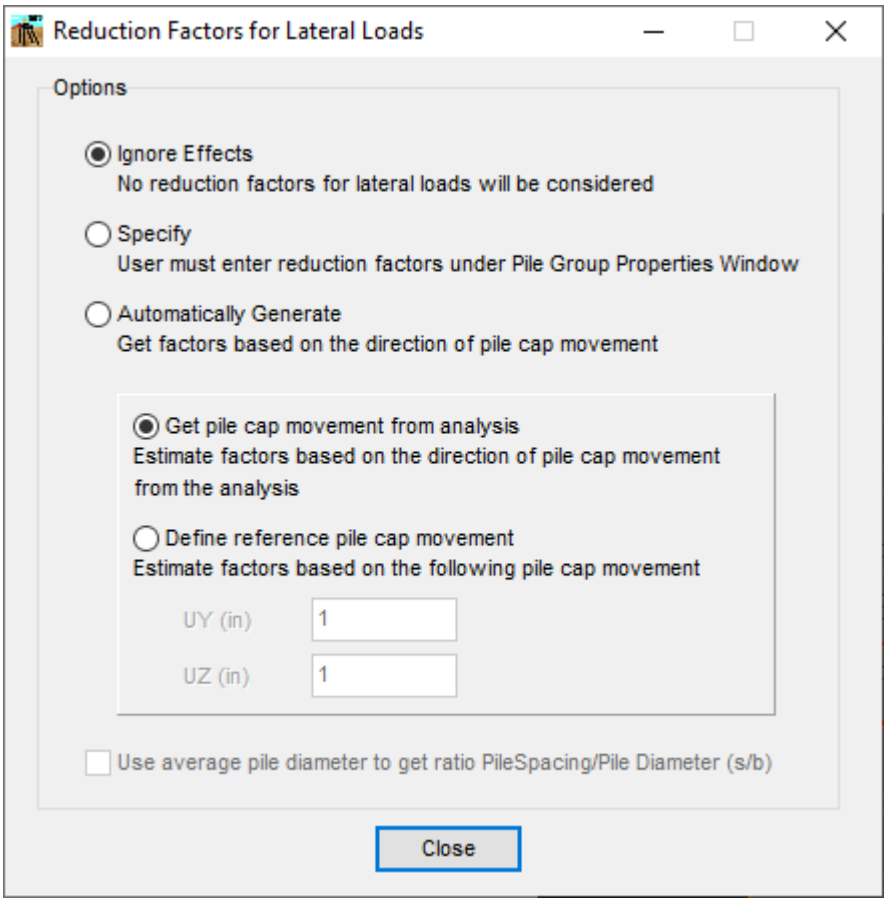


Figure 3.218 Reduction Factors for Lateral Loads Window

The user is provided with the following options:

- **Ignore Effects:** Indicates that the group effects of closely-spaced piles will be ignored

- **Specify:** Indicates that the user will directly specify the values of reduction factors for lateral loads. Those values are specified in the **P-Multiplier** and **Y-Multiplier** columns of the **Pile Groups** window under the **Data** menu**.**
- **Automatically Generate:** Indicates that the program will automatically generate reduction factors for piles in a closely-spaced group.
  For the latest two options the user can assign one of the next two sub-options:
  - **Get movement from analysis:** The program will get the direction of cap movement from the analysis. For a rigid cap model the program uses the movement (Y and Z) at the origin of coordinates. For a flexible cap model (Flexible Cap Version), the program uses the averaged movement from all the FEM nodes that belong to the pile cap.
  - **Define reference movement:** The direction of movement will be specified by the user in the UY and UZ fields. Check on this option if there are problems of convergence and the solution is moving back and forth between iterations. You can do an initial analysis using ignoring effects to get the direction of movement and then do the final analysis using this option.

Information about these factors are included in Chapter 6 in the *Technical Manual*, "Reduction Factor for Soil Resistance in a Closely-Spaced Pile Group".

3.5.19 Options - Reduction Factors for Lateral Loads Along Depth

Under default conditions, reduction factors for lateral loads along depth will not be considered. This option should be selected if the user wishes to consider a reduction factor for lateral loading along depth. If this option is check-marked the option **Sets of Red. Fact. Lateral Load along Piles** in the **Data** menu is activated. Extra information will also need to be inputted in the **Set of Lateral Red. Factor along Depth** column of the **Pile Groups** window under the **Data menu.**

3.5.20 Options - Reduction Factors for Axial Loads

Under default conditions, reduction factors for axial loads will not be considered for axial loading. This option should be selected if the user wishes to consider a reduction factor for axial loading. The reduction factors for axial loads will be specified by the user in the **Red. Factor Axial Load** column of the **Pile Groups** window under the **Data** menu**.**

Information about these factors are included in Chapter 7 in the *Technical Manual*, "Reduction Factors for Piles under Axial Loading in a Closely-Spaced Pile Group".

3.5.21 Options - Print Generated p-y Curves

If the user wishes to print p-y curves at selected depths for verification purposes, this option must be checked. If this option is check-marked, the option **Sets of Data to Print generated p-y curves** under the **Data** menu is activated.

3.5.22 Options - Default Control Options ...

By selecting this option, the Default Control Options Window will appear.

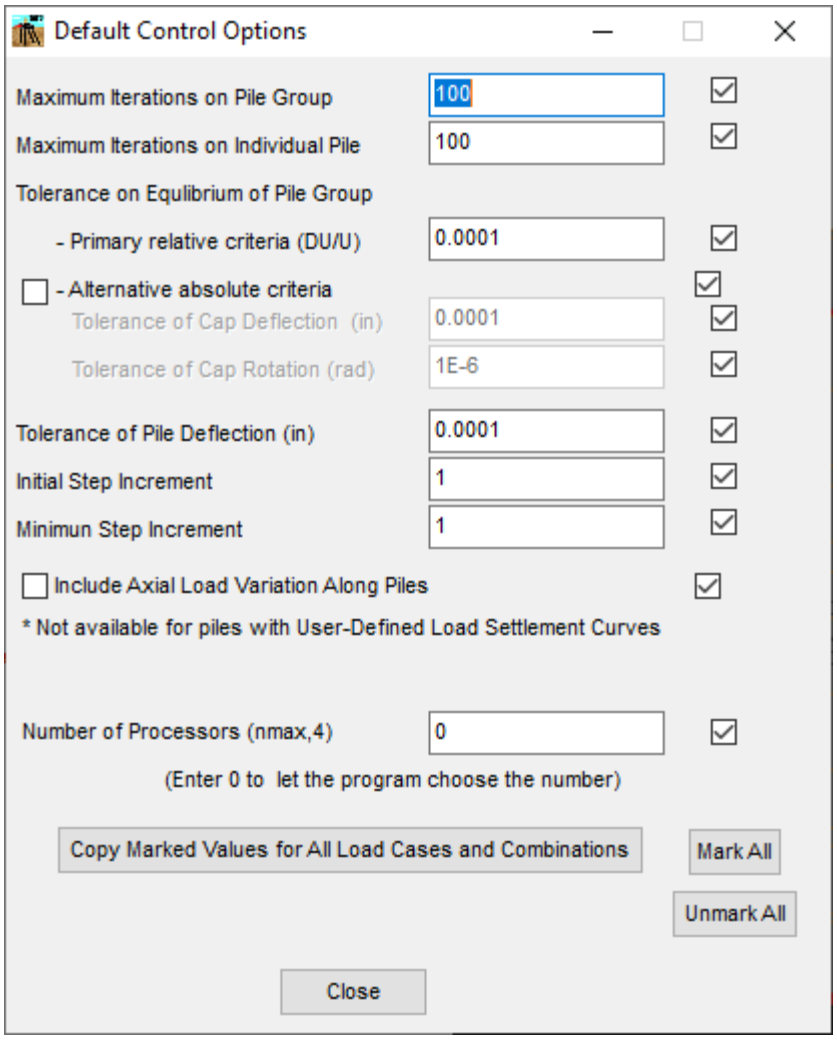


Figure 3.219 Default Control Options Window

A general description for the data needed under each computation parameter is listed below.

- *Maximum Iterations on Pile Group*. This is the maximum number of iterations allowed for the global behavior of the pile group. The default value is 100.
- *Maximum Iterations on Individual Pile*. This is the maximum number of iterations allowed for each individual pile. The default value is 100.
- *Tolerance on Equilibrium of Pile Group*.
  - *Primary relative criteria (DU/U).* This is the maximum tolerated relative error on pile cap deflections computed by two consecutive iterations for the global pile group. The default value is 0.0001.

- *Alternative absolute criteria.* Check this option for using an absolute criteria to check the convergence on the pile cap. If checking this option, the following two values must be input:
- *Tolerance of Cap Deflection*. This is the maximum tolerated difference on cap deflections computed by two consecutive iterations for the pile group. The default value is 0.0001 in (2.54×10^{-6} m).
- *Tolerance of Cap Rotation*. This is the maximum tolerated difference on cap deflections computed by two consecutive iterations for the pile group. The default value is 1×10^{-6} rad.
-

- *Tolerance on Pile Deflection*. This is the maximum tolerated difference on pile deflections computed by two consecutive iterations for individual piles. The default value is 0.0001 in (2.54×10^{-6} m).
- *Initial Step Increment*. This value is helpful if the program is not able to reach convergence by applying the total load. The user may use this value to apply the load in steps. At each step the program will apply an increment of load equal to this value times the total load. The default value is 1.
- *Minimum Step Increment*. This value is helpful if the program does not reach convergence by applying the total load. If the program does not reach convergence, the program will try to use a load increment factor equal to half of the current load increment factor. If the new increment factor is smaller than this value, the program will stop the analysis for the current load case. The default value is 1.
- *Include Axial Load Variation Along Piles*. Check this option, to use the axial load variation along piles for the analysis of the pile group. This option is only available if the program computes the axial load displacement curve internally, since the program computes the t-z and q-w curves internally. So, this option is not available in piles where a user defined axial load displacement curve is defined.
- *Number of Processors, nmax* ***n***. Enter the number of processors desired to be used for the analysis of the current load case. The program displays the maximum number of processors available. Leave the number as zero to let the program decided for the number of processors to be used for the analysis of the current load case. The default value is 1.

Copy Marked Values for All Loads and Combinations Button

Click on the **Copy Marked Values for All Loads and Combinations** button to override the values in all loads and combinations defined so far by the ones marked defined in the current load case. The values not marked will not be overridden.

Close Button

Click on the **Close** button to close the window and go back to the **Main** window.

3.5.23 Options - Default Output Variables ...

By selecting this option, the Default Output Variables Window will appear.

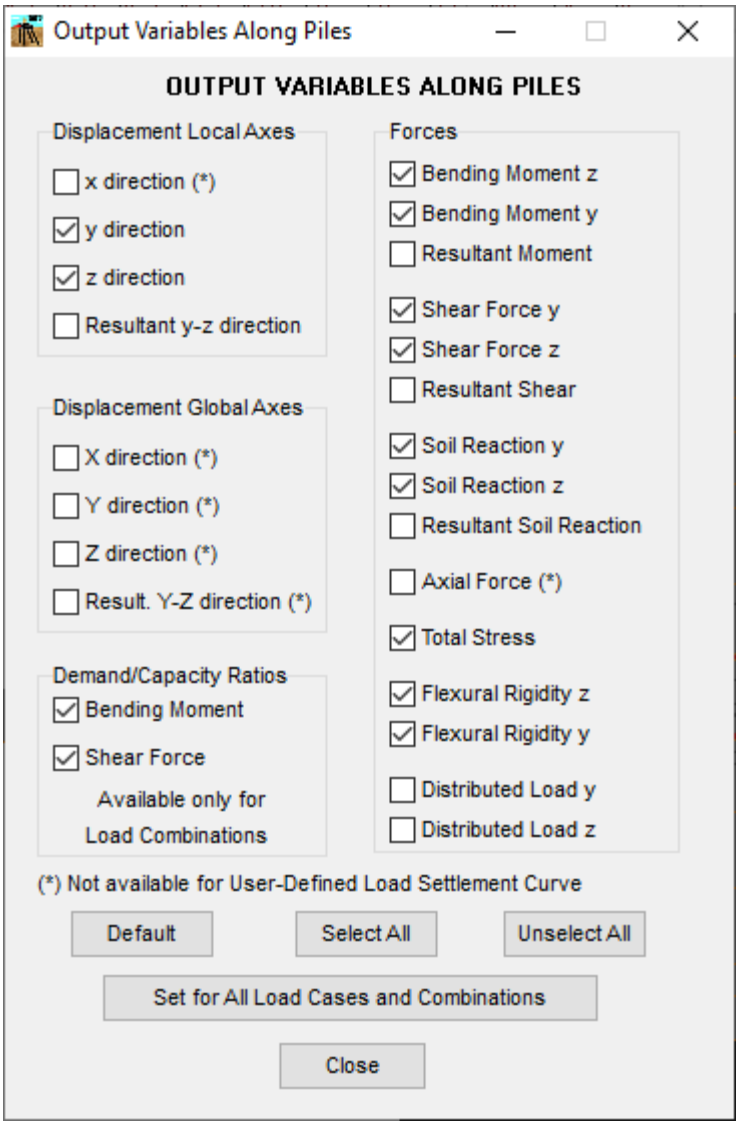


Figure 3.220 Output Variables Window

Mark the variable to be printed in the output file. Please note that some variables may not be available.

Default Button

Click on the **Default** button to mark the default values printed by default.

Select All Button

Click on the **Select All** button to mark all the variables.

Unselect All Button

Click on the **Unselect All** button to unmark all the variables.

Set for All Load Cases and Combinations Button

Click on the **Set for All Load Cases and Combinations** button to copy the default parameter values for all the load cases and combinations defined for the current model.

Close Button

Click on the **Close** button to close the window and go back to the **Main** window.

3.5.24 Options - Text Editor Options ...

By selecting this option, the Text Editor Options Window will appear.

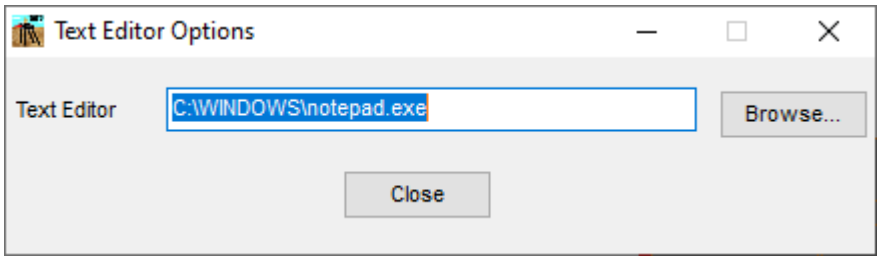


Figure 3.221 Text Editor Options Window

In the prompt that appears, the user should enter the complete path and command line for their preferred text editor or word processor. As a default, the command line c:\windows\notepad.exe sets Microsoft Notepad as the default text editor. This text editor will be used to **Edit Input File, Processor Run Notes, Output File,** and **Summary File** when called from the **Computation** menu.

3.5.25 Options - Check for Updates as the Program Starts

This option is used to instruct GROUP to check the ENSOFT website for updates to software when GROUP is opened. An active internet connection is required.

If the option is check-marked the following will appear next time the program starts

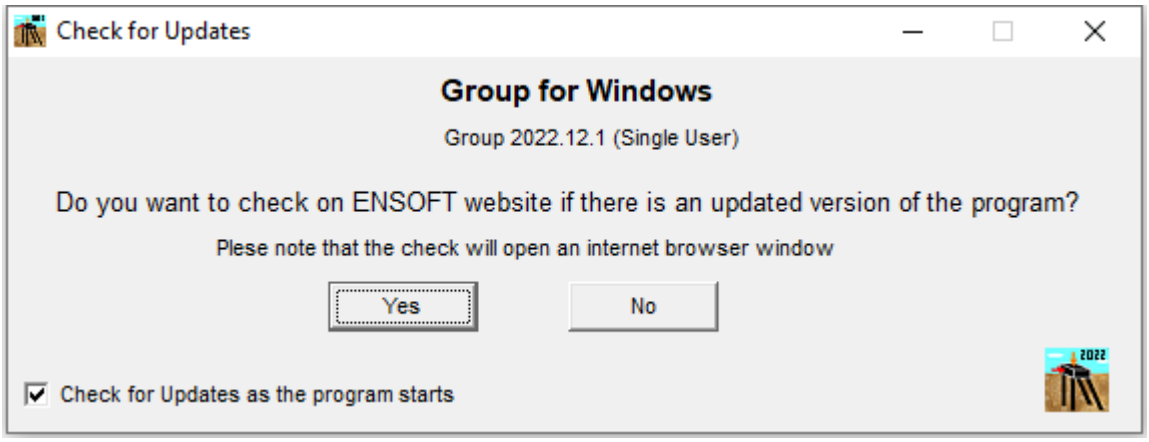


Figure 3.222 Message window to confirm checking for updates on ENSOFT site

Clicking “Yes” opens the user’s default internet browser to an Ensoft webpage listing update information specific to GROUP 2022.

3.5.26 Options - Include Load Case/Comb Number on Graphs

This option is used to instruct GROUP to include the Load Case/Comb. Number at the bottom of the displayed graphs.

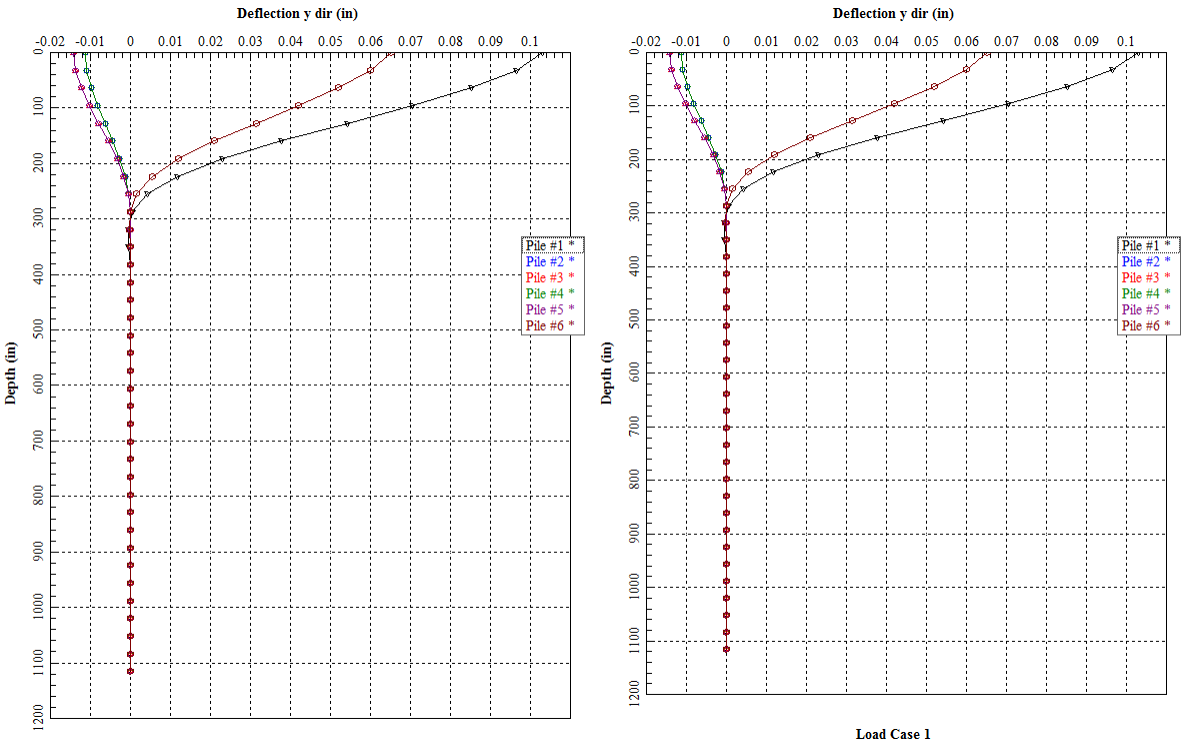


Figure 3.223 Effect of the Include Load Case/Comb Number on Graphs option.

3.5.27 Options - Limit Active Curves on Graphs

This option is used to instruct GROUP to limit the number of default active curves on graphs to 20. The reason to limit the number of curves is to display the plot faster. If the user wants to plots all the curves, just uncheck this option. Please note that while the graph is displayed the user can activate or deactivate curves using the Plot Menu.

3.6 View Menu

This menu is used to review, in proportional graphical form, the geometric shape and configuration for the modeled pile group.

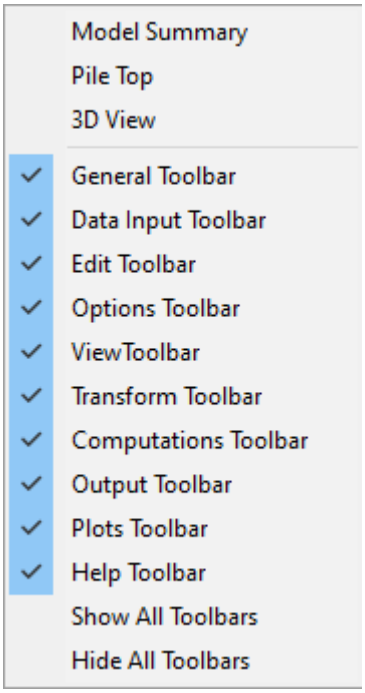


Figure 3.224 Options contained in the View Menu when 3D View is not active

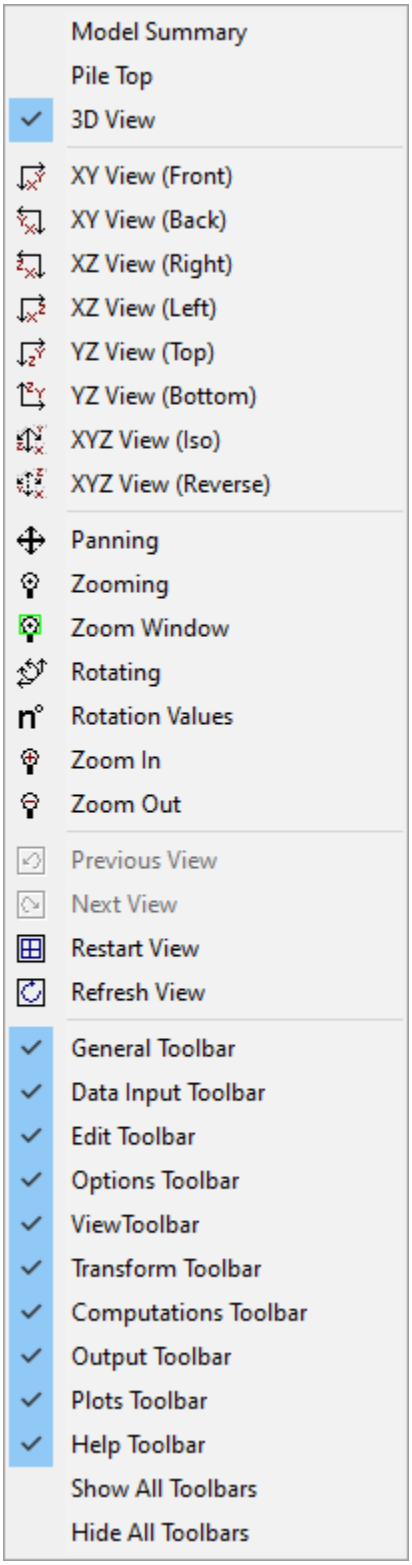


Figure 3.225 Options contained in the View Menu when 3D View is active

3.6.1 View - Pile Top

This option provides a plan view of the group model, as observed from the top.

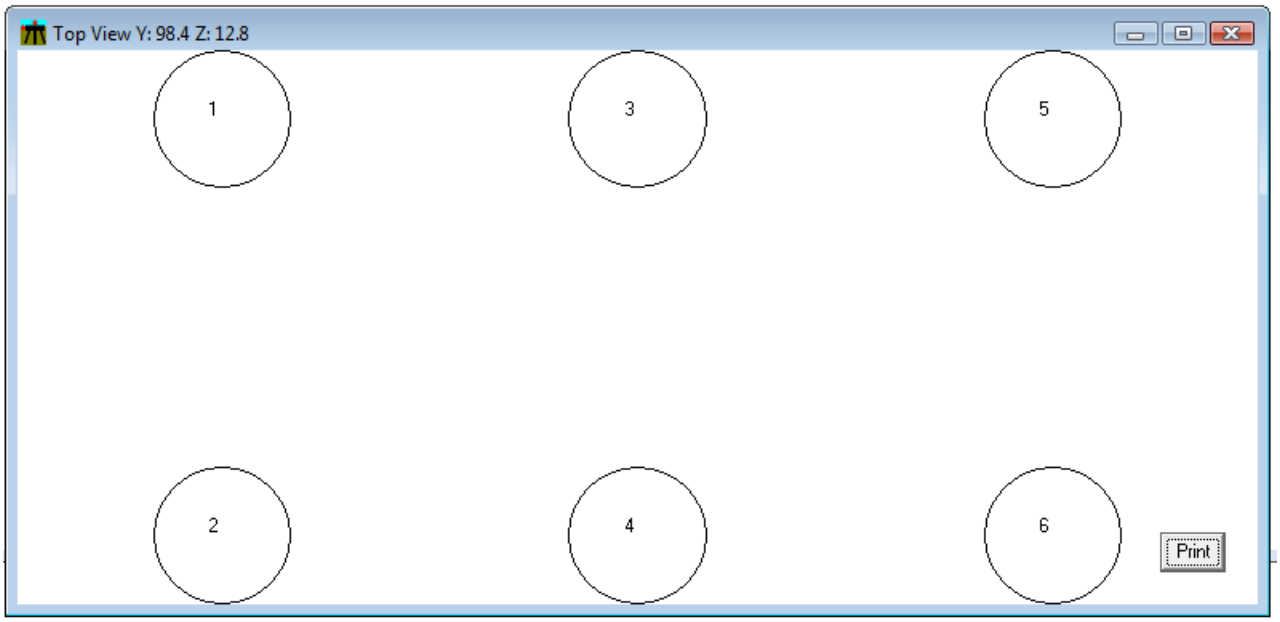


Figure 3.226 Sample of Pile Top View

3.6.2 View - 3D View

This option opens the **3D View** window that provides a 3-dimensional view of the group model.

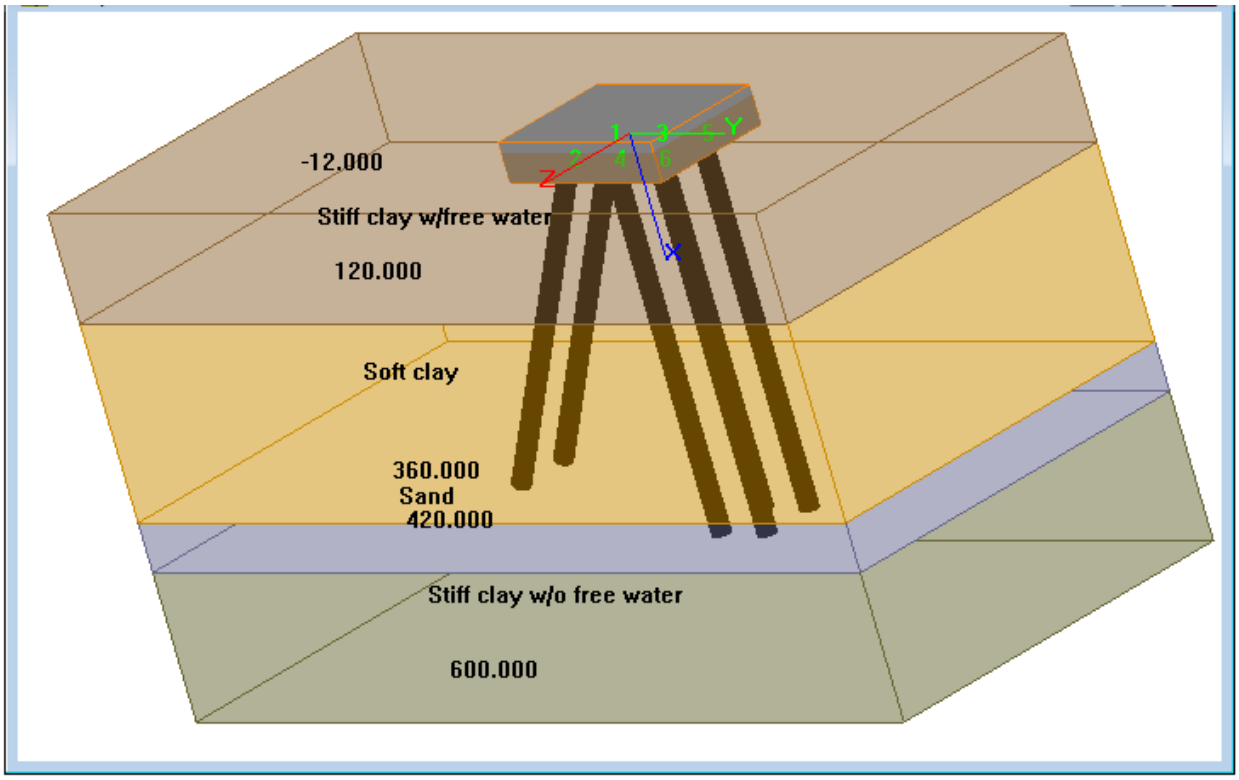


Figure 3.227 Sample of 3D View

3.6.3 View - XY View (Front)

This option switches the viewing angle in a direction perpendicular to the XY plane (Front View).

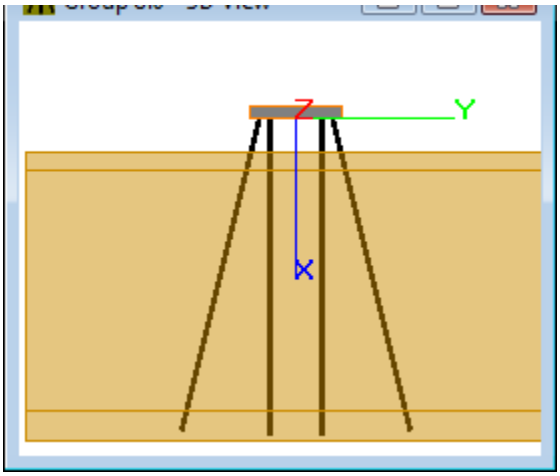


Figure 3.228 XY View (Front)

3.6.4 View - XY View (Back)

This option switches the viewing angle in a direction perpendicular to the XY plane (Back View).

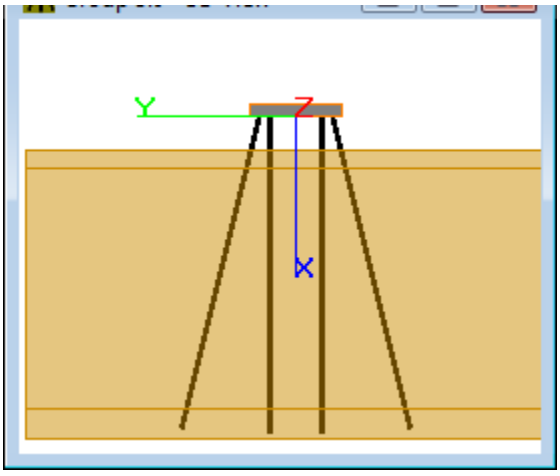


Figure 3.229 XY View (Back)

3.6.5 View - XZ View (Right)

This option switches the viewing angle in a direction perpendicular to the XZ plane (Right View).

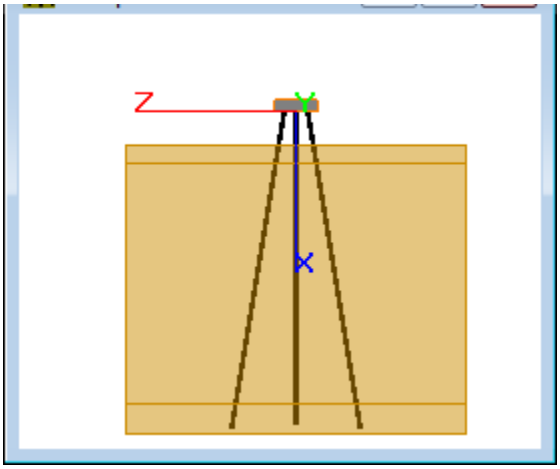


Figure 3.230 XZ View (Right)

3.6.6 View - XZ View (Left)

This option switches the viewing angle in a direction perpendicular to the XZ plane (Left View).

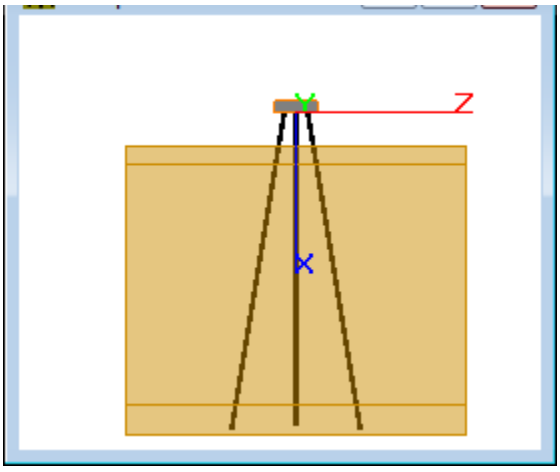


Figure 3.231 XZ View (Left)

3.6.7 View - YZ View (Top)

This option switches the viewing angle in a direction perpendicular to the YZ plane (Top View).

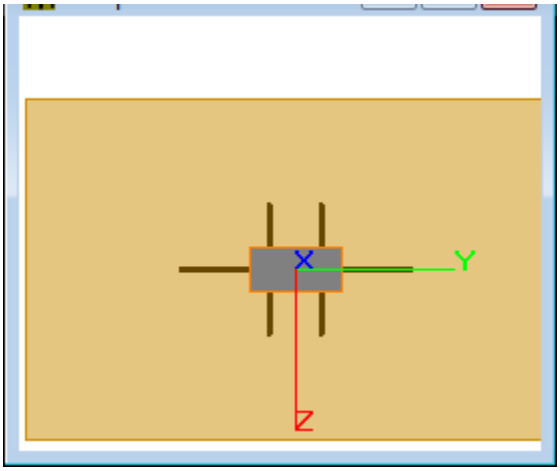


Figure 3.232 YZ View (Top)

3.6.8 View - YZ View (Bottom)

This option switches the viewing angle in a direction perpendicular to the YZ plane (Bottom View).

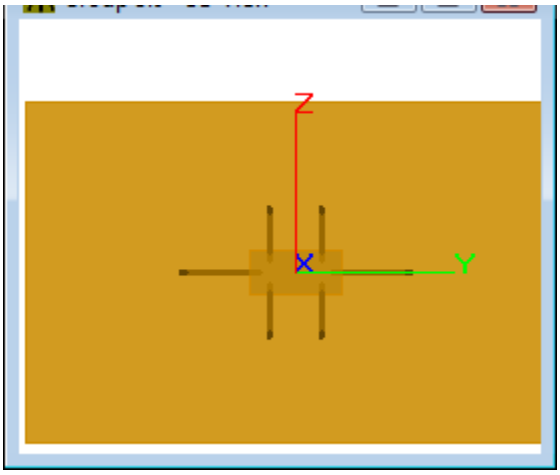


Figure 3.233 YZ View (Bottom)

3.6.9 View - XYZ View (Iso)

This option switches the viewing angle in to an isometric view (Iso View).

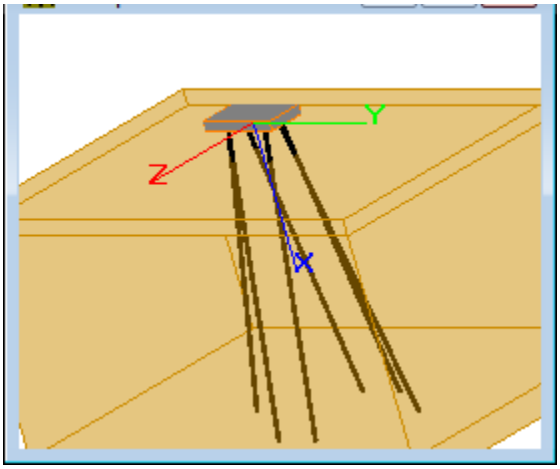


Figure 3.234 XYZ View (Iso)

3.6.10 View - XYZ View (Reversed)

This option switches the viewing angle to the back view (Reversed View).

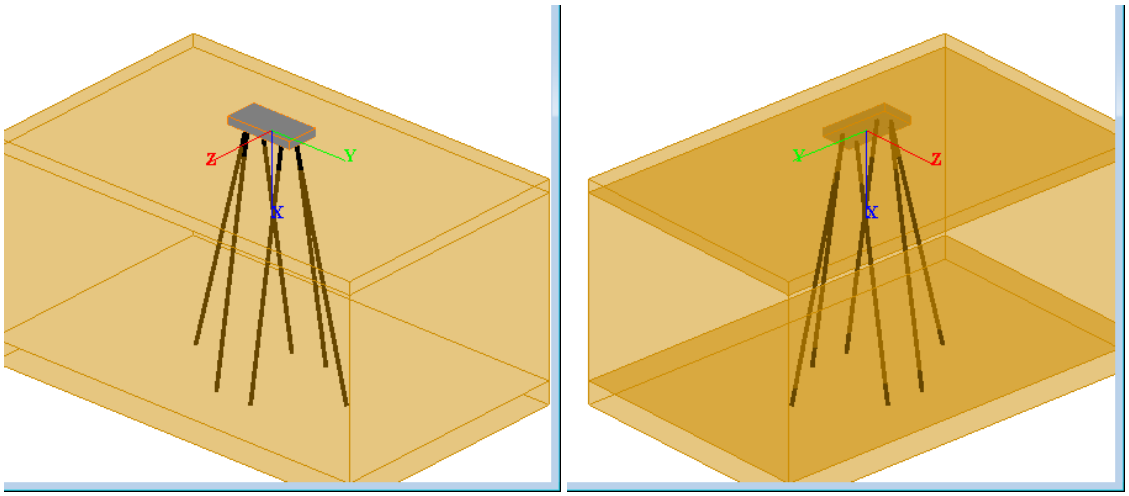


Figure 3.235 XYZ View (Reversed)

3.6.11 View - Panning

This option set the mouse movement to a translation of the viewing reference point.

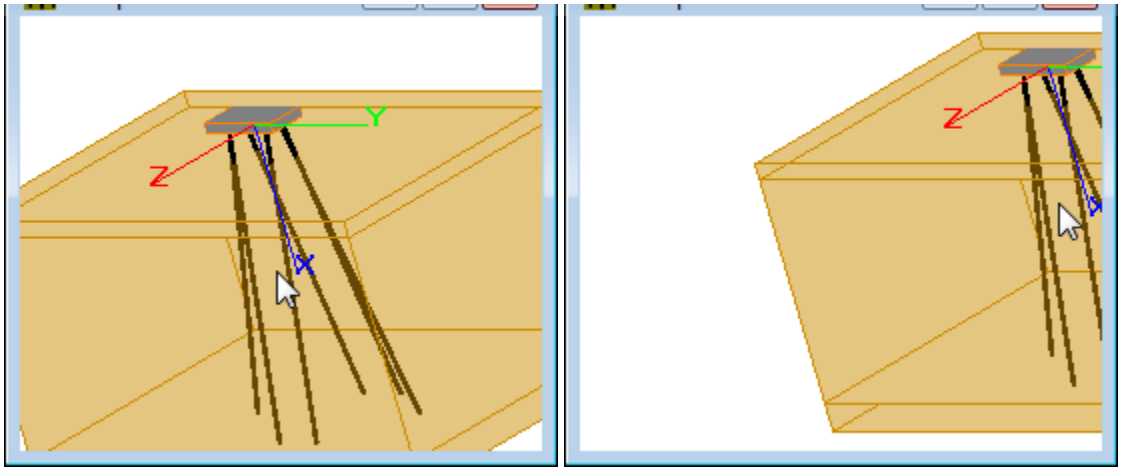


Figure 3.236 Panning in the 3D-View display

3.6.12 View - Zooming

This option set the mouse movement to enlarge or reduce the model displayed.

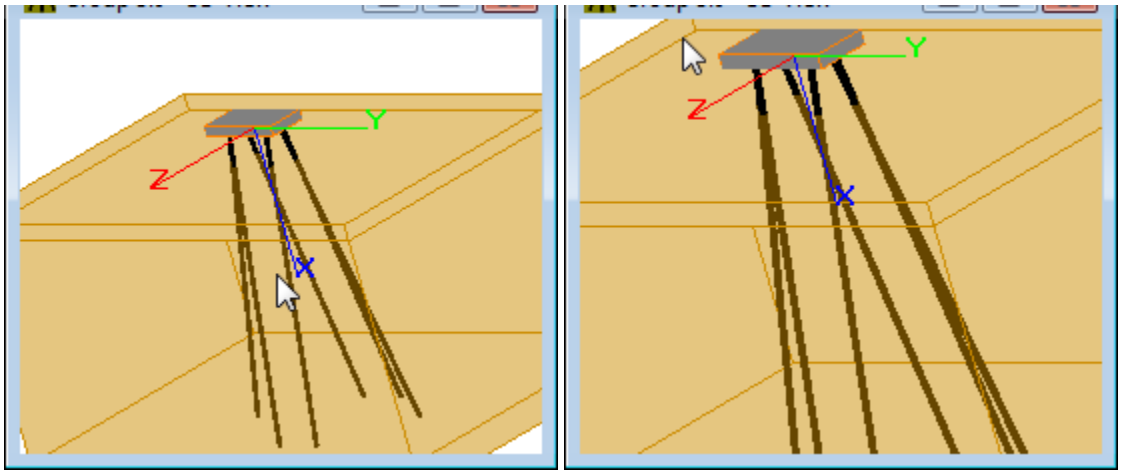


Figure 3.237 Zooming in the 3D-View display

3.6.13 View - Zoom Window

This option allows to enlarge the current view based on the box traced by the mouse movement.

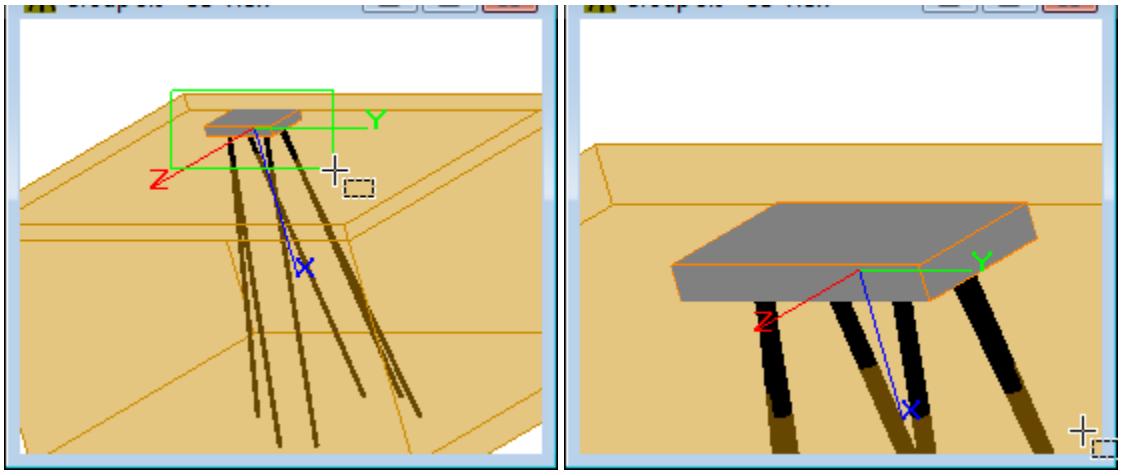


Figure 3.238 Zoom by defining a window in the 3D-View display

3.6.14 View - Rotating

This option set the mouse movement to a rotation of the viewing angle.

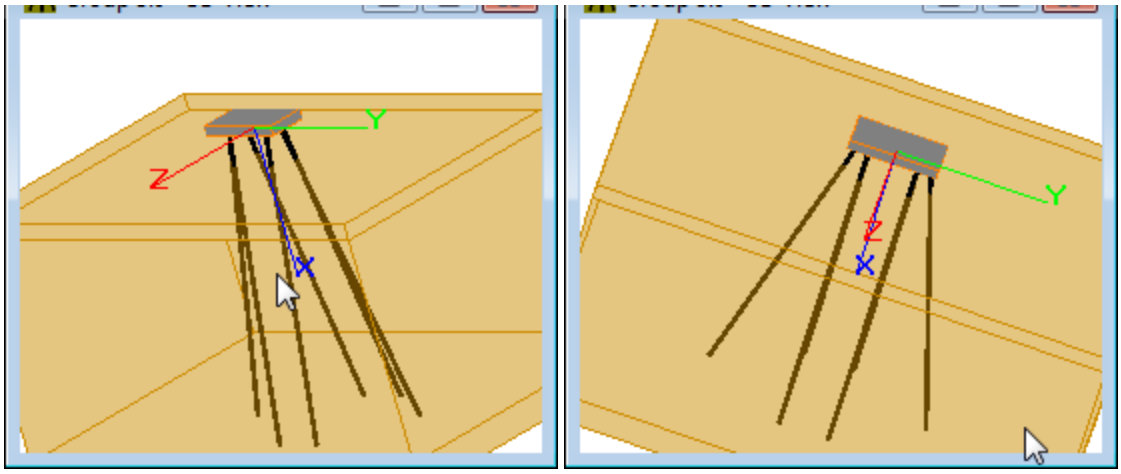


Figure 3.239 Rotating in the 3D View display

3.6.15 View - Rotation Values

This option allows the user to enter the rotation angles of the viewing angle.

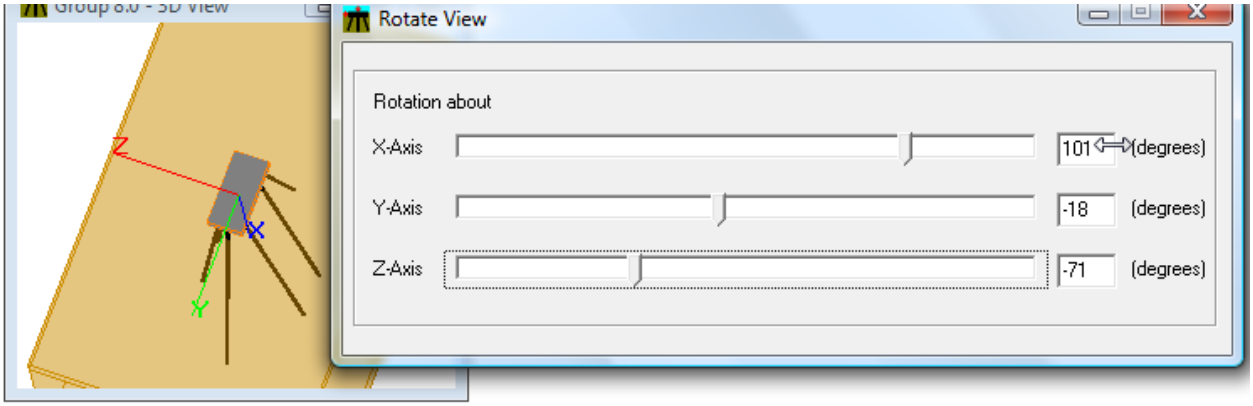


Figure 3.240 Rotating in the 3D View by defining the rotation angles

3.6.16 View - Zoom In

This option allows the user to enlarge the current view about 1.5 times.

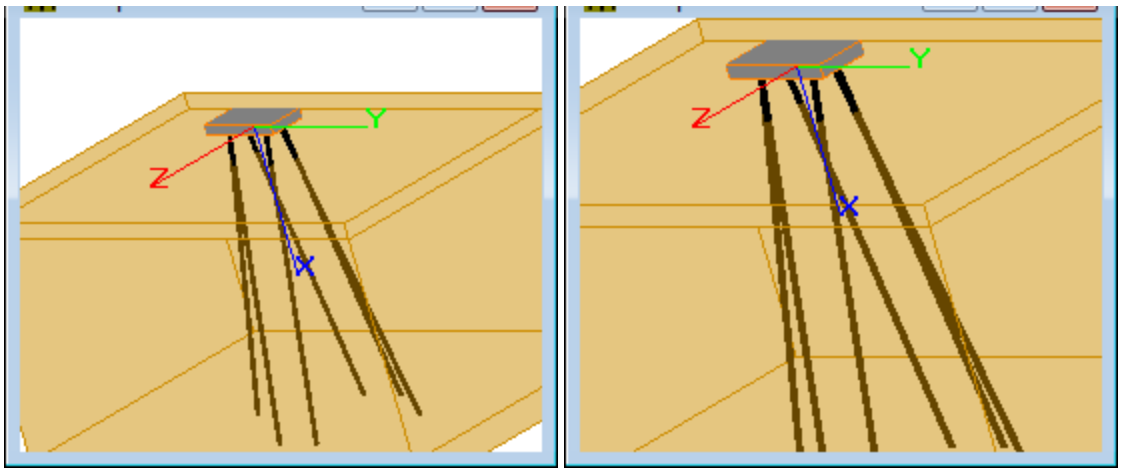


Figure 3.241 Effect of the Zoom-in option

3.6.17 View - Zoom Out

This option allows the user to reduce the current view about 1.5 times.

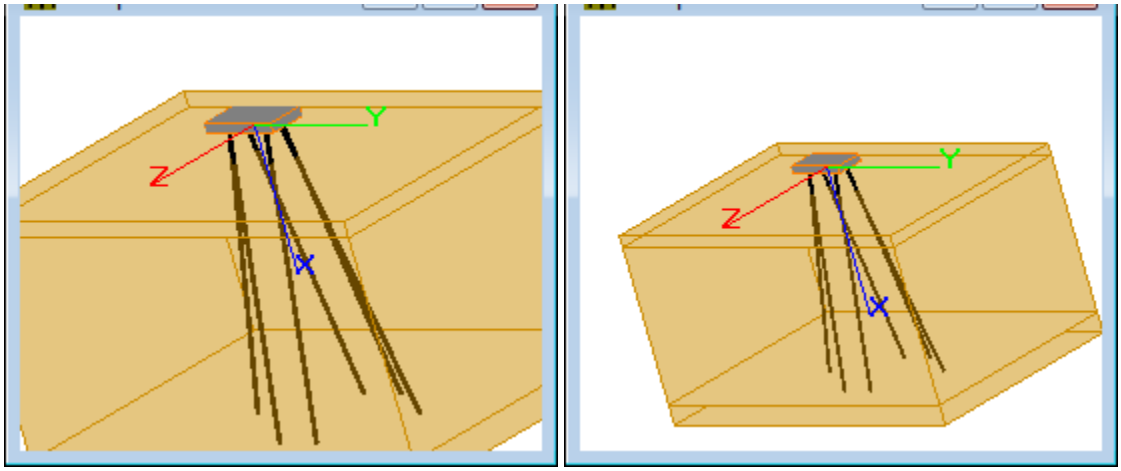


Figure 3.242 Effect of the Zoom-out option

3.6.18 View – Previous View

This option allows the display the previous view used if available.

3.6.19 View – Next View

This option allows the display the next view used if a previous view was selected.

3.6.20 View - Restart View

This option allows the user to reset all the panning, rotation and zooming performed and display the model using a view angle perpendicular to the XY plane (Front View).

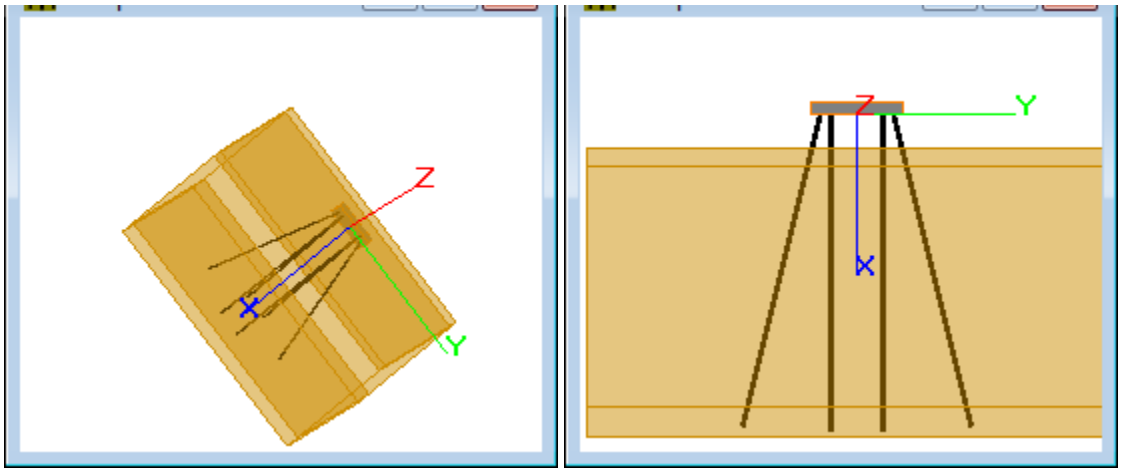


Figure 3.243 Effect of the Restart View option

3.6.21 View - Refresh View

This option allows the user to redraw the model. This option is useful to display the model after some changes have been made. Changes may not be displayed automatically in the 3D View window.

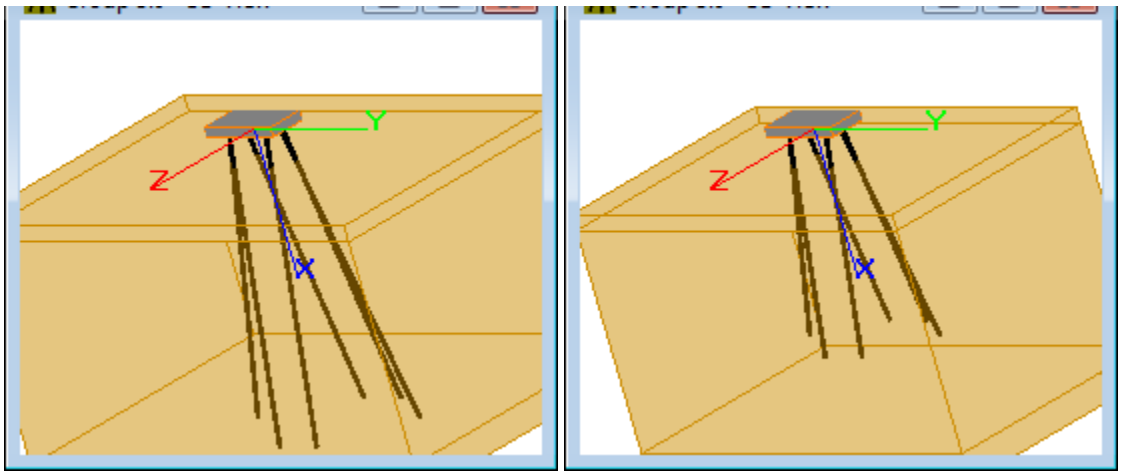


Figure 3.244 Effect of the Refresh View option, after reducing pile length

3.6.22 View - General Toolbar

This option allows the user to toggle on and off the General Toolbar. The General Toolbar is associated with the **File** Menu

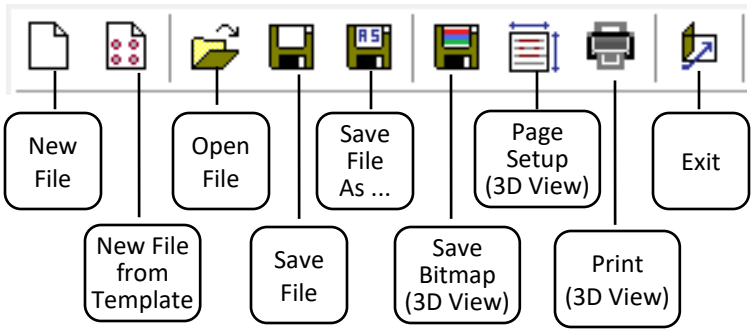


Figure 3.245 General Toolbar

3.6.23 View - Data Input Toolbar

This option allows the user to toggle on and off the Data Input Toolbar. The Data Input Toolbar is associated with the **Data** Menu.

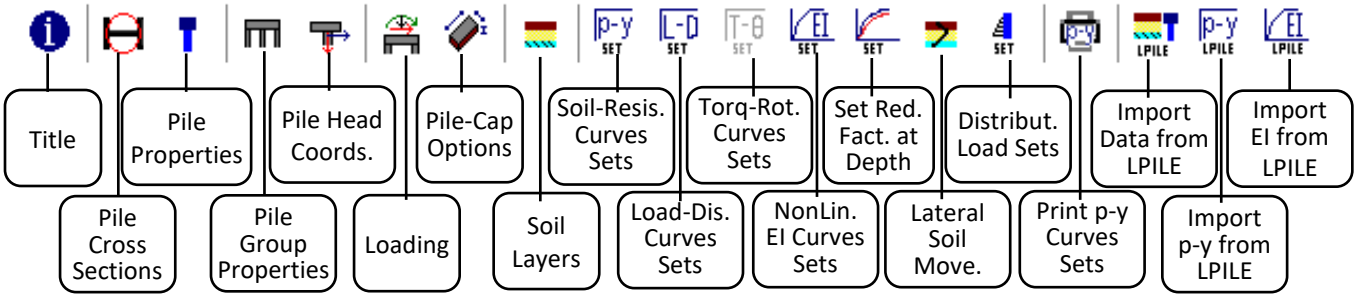


Figure 3.246 Data Input Toolbar

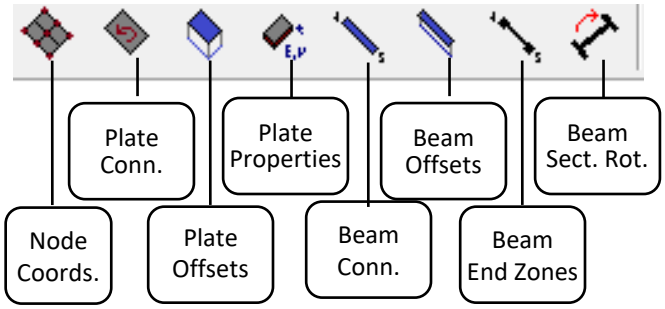


Figure 3.247 Data Input Toolbar. Additional Buttons for Flexible Cap Version

3.6.24 View - Edit Toolbar

This option allows the user to toggle on and off the Edit Toolbar. The Edit Pile Toolbar is associated with the **Edit** Menu

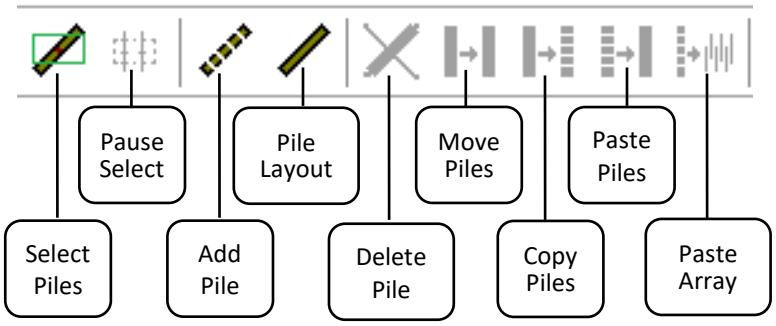


Figure 3.248 Edit Toolbar

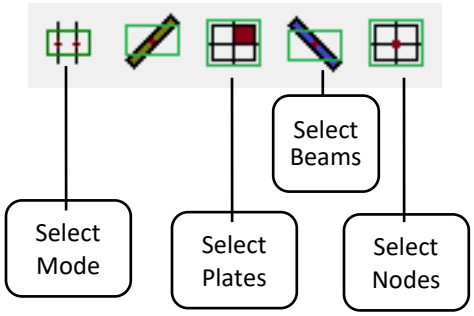


Figure 3.249 Edit Toolbar. Additional Buttons for Flexible Cap Version

3.6.25 View - Options Toolbar

This option allows the user to toggle on and off the Options Toolbar. The Options Toolbar is associated with the **Options** Menu

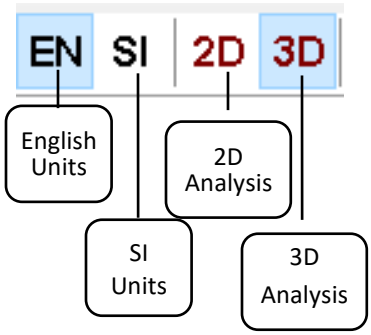


Figure 3.250 Data Input Toolbar

3.6.26 View - View Toolbar

This option allows the user to toggle on and off the View Toolbar. The View Toolbar is available if the **3D View** window is active and it is associated with the View Options in the **Options** Menu.

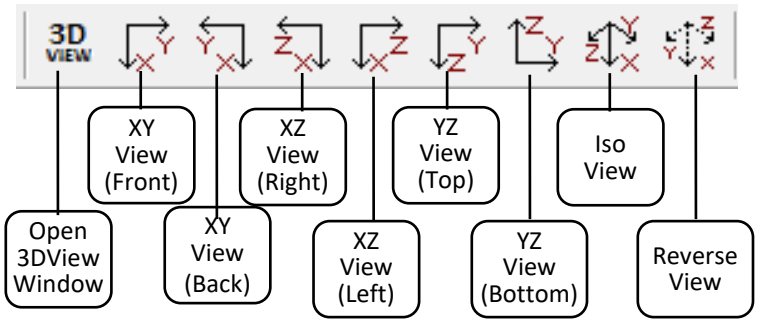


Figure 3.251 View Toolbar

3.6.27 View - Transform Toolbar

This option allows the user to toggle on and off the Transform Toolbar. The Transform Toolbar is available if the **3D View** window is active and it is associated with the Transform Options in the **Options** Menu.

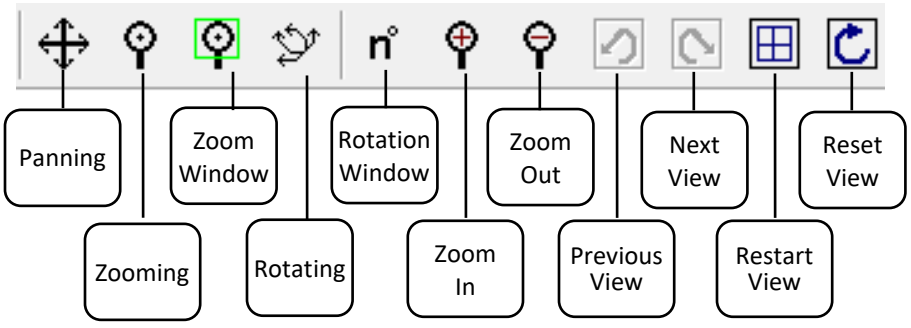


Figure 3.252 View Toolbar

3.6.28 View - Computations Toolbar

This option allows the user to toggle on and off the Computations Toolbar. The Computations Toolbar is associated with the **Computation** Menu

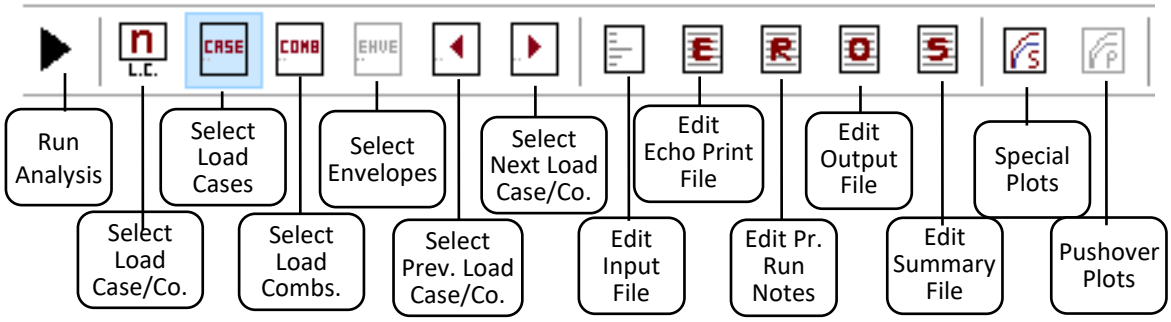


Figure 3.253 Computations Toolbar

3.6.29 View - Output Toolbar

This option allows the user to toggle on and off the Output Toolbar. The Output Toolbar is available if the **3D View** window is active and it is associated with the Output Options in the **Show 3D View** Menu

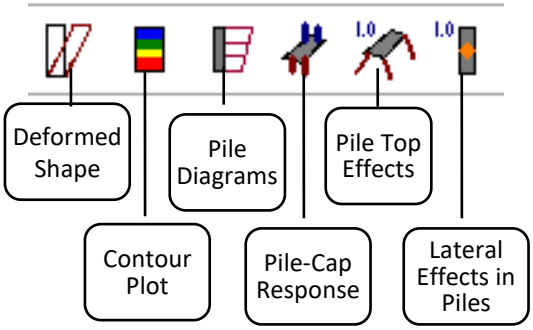


Figure 3.254 Output Toolbar

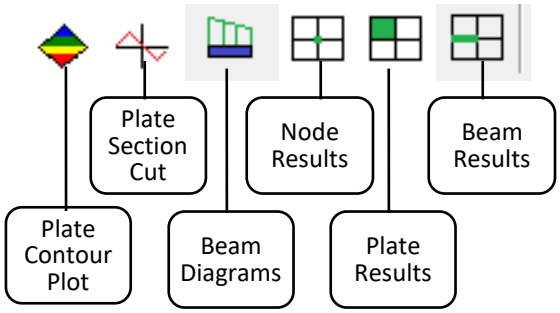


Figure 3.255 Output Toolbar. Additional Buttons for Flexible Cap Version

3.6.30 View - Plots Toolbar

This option allows the user to toggle on and off the Plots Toolbar. The Plots Toolbar is associated with the **Graphs** Menu

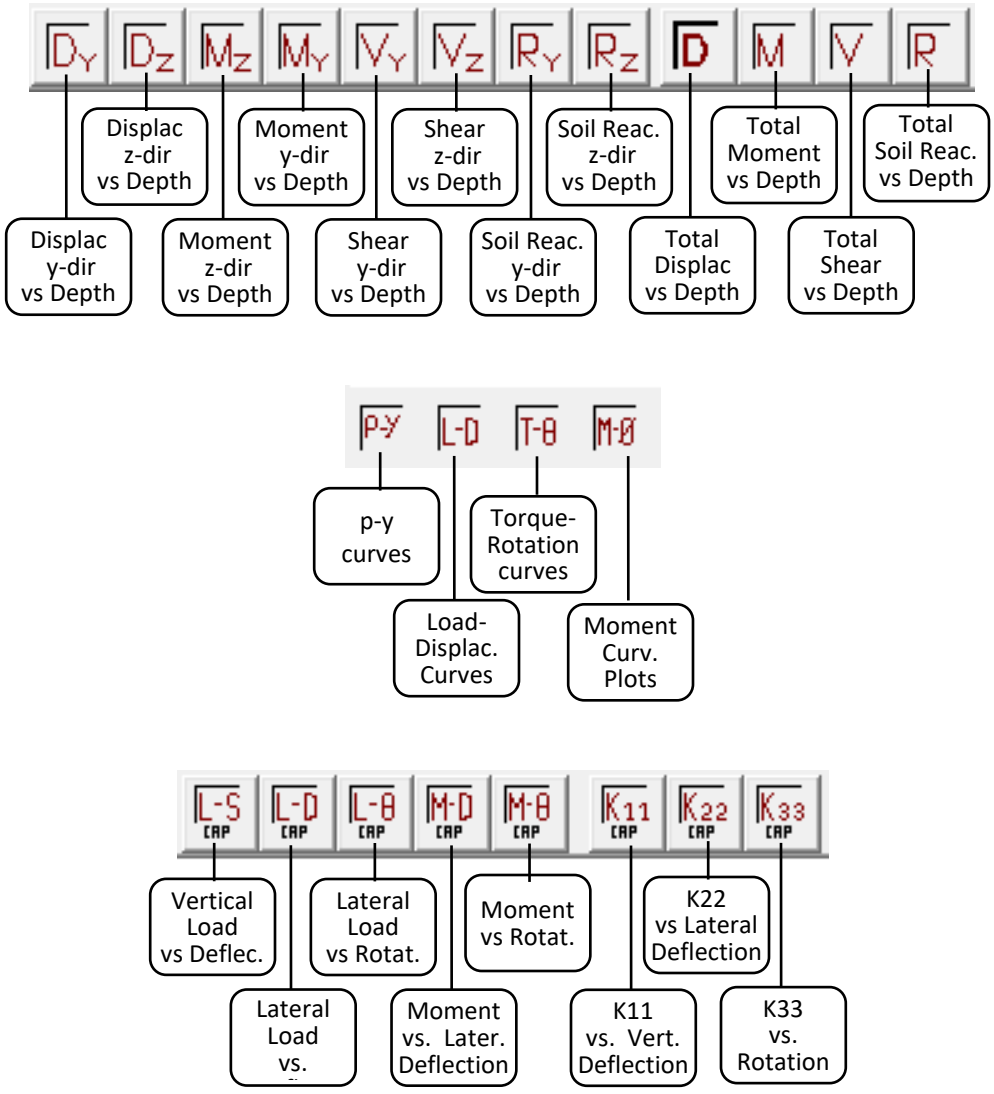


Figure 3.256 Plots Toolbar

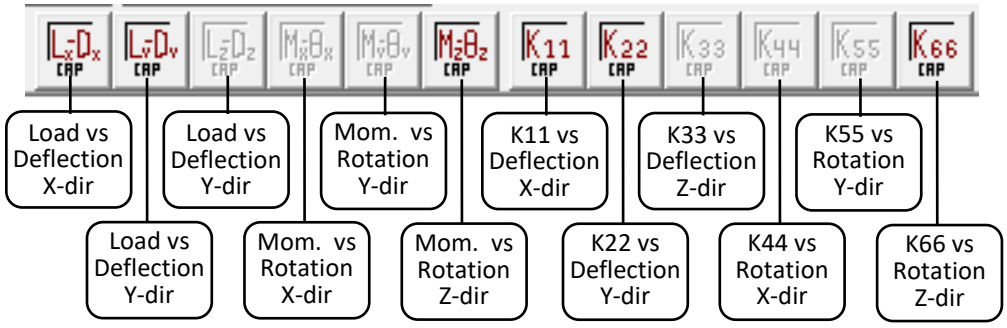


Figure 3.257 Plots Toolbar (Continuation)

3.6.31 View - Help Toolbar

This option allows the user to toggle on and off the Help Toolbar. The Help Toolbar is Associated with the **Help** Menu

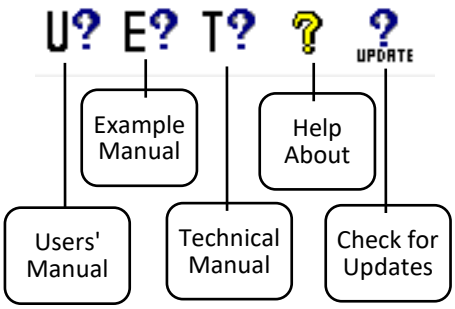


Figure 3.258 Help Toolbar

3.7 Show 3D View Menu

The **Show 3D View** menu is available when the **3D View** window is displayed. It provides to the user different options for display of the model.

This menu is used to review, in proportional graphical form, the geometric shape and configuration for the modeled pile group.

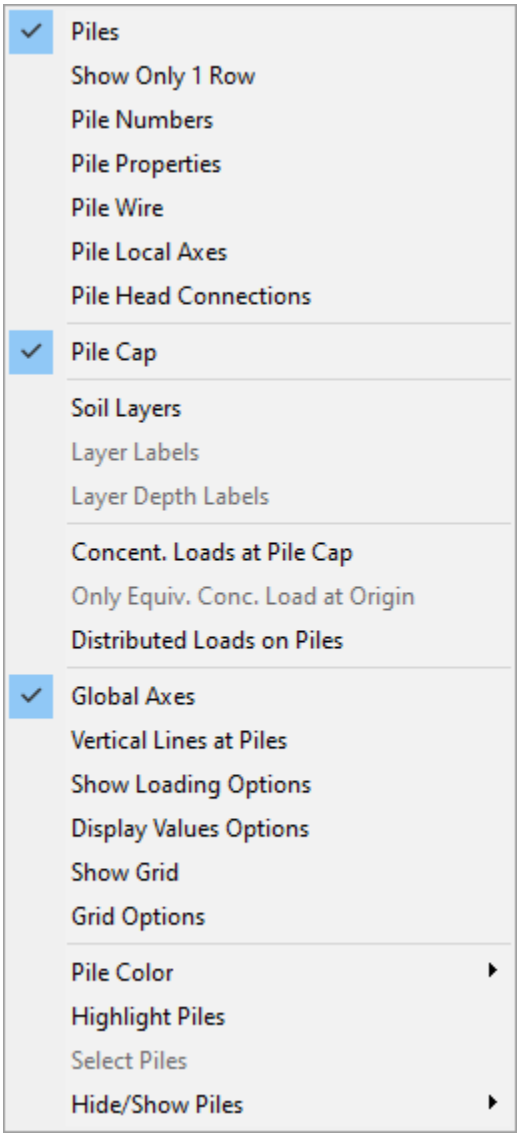


Figure 3.259 Options contained in the Show 3D View Menu

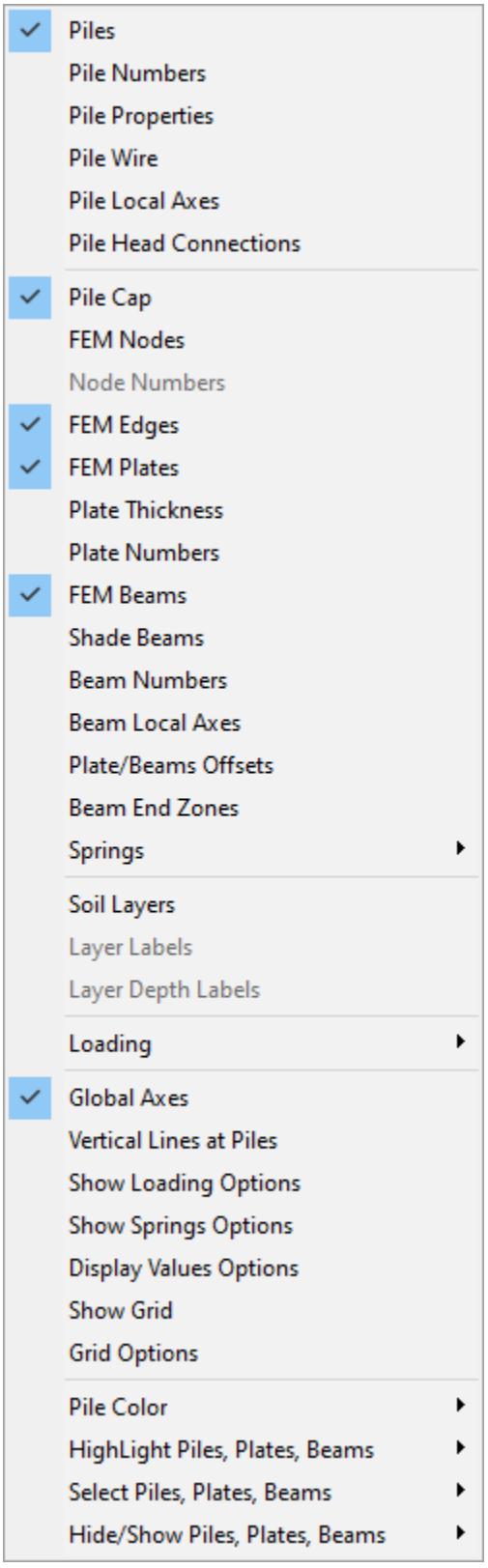
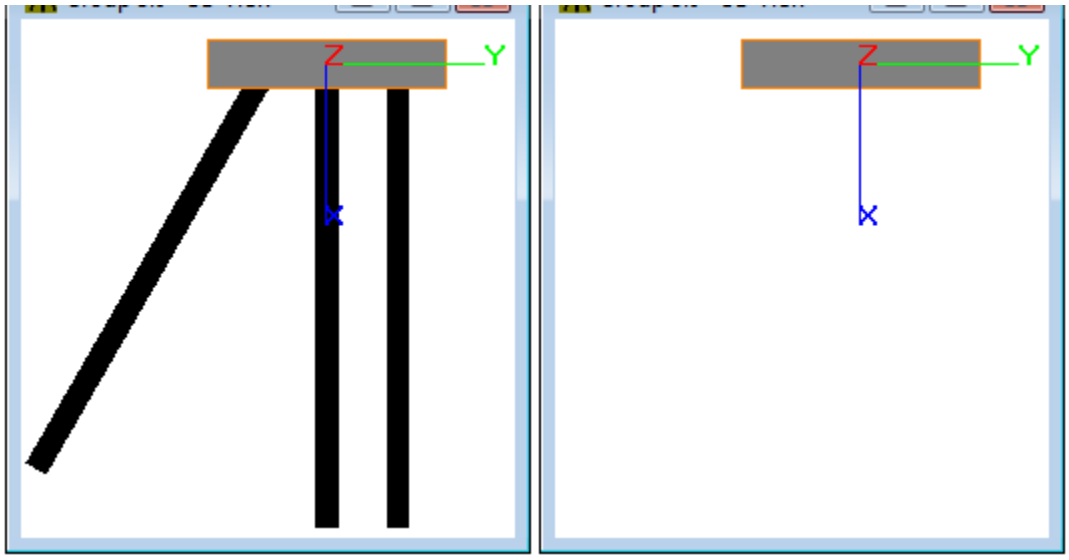


Figure 3.260 Options contained in the Show 3D View Menu – Flexible Cap Version

Below there is a description of the menu items

3.7.1 Show 3D View -Piles

This option allows the user to show the piles.



a) Piles option check-marked b) Piles option not check-marked

Figure 3.261 Effects on 3D model display using Piles option

3.7.2 Show 3D View -Show Only 1 Row

This option is available only in 2D models and allows the display on only one row of piles

3.7.3 Show 3D View -Pile Numbers

This option allows the user to show the pile numbers at each pile head.

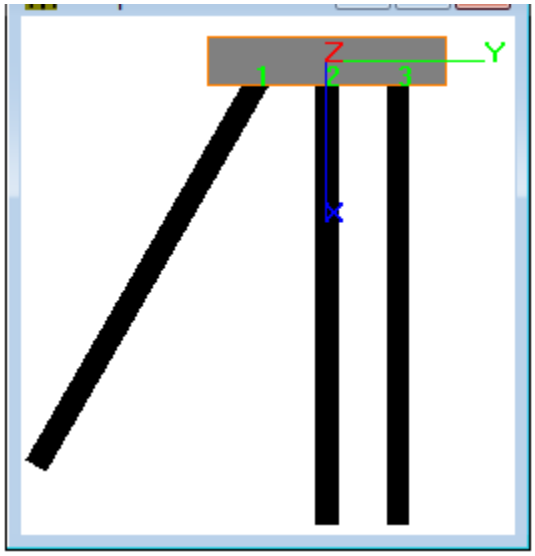


Figure 3.262 Effects on 3D model display using Pile Numbers option

3.7.4 Show 3D View -Pile Properties

This option allows the user to show the pile property index at each pile head.

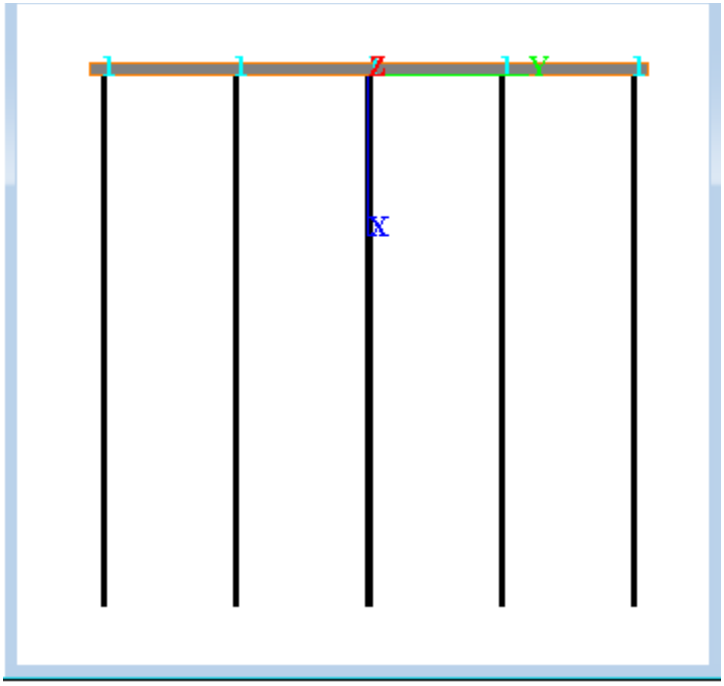
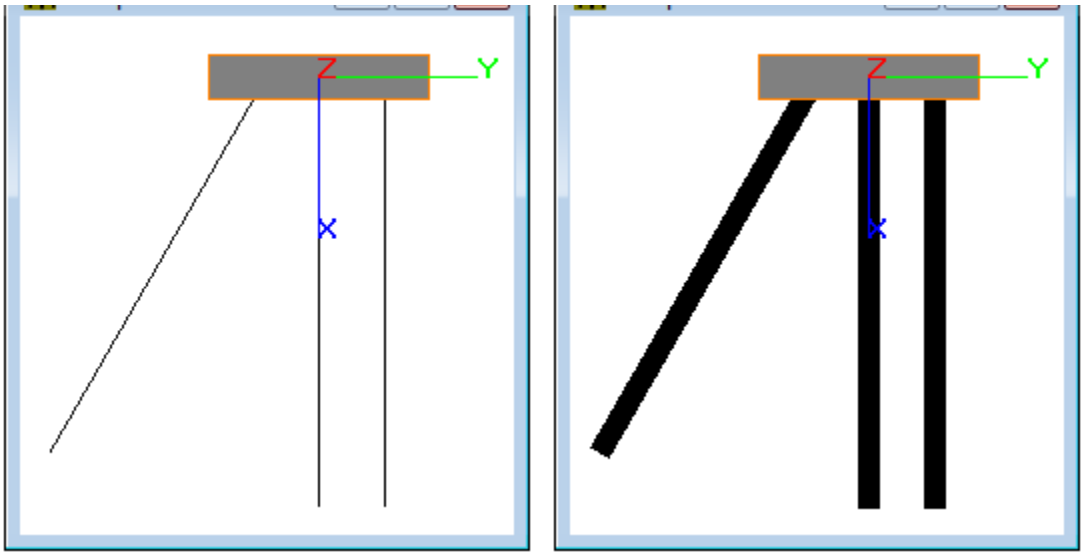


Figure 3.263 Effects on 3D model display using Pile Properties option

3.7.5 Show 3D View -Pile Wire/Pile Shade

This option allows the user to toggle between showing the piles as lines or as surfaces.



a) Pile Wire option b) Pile Shade option

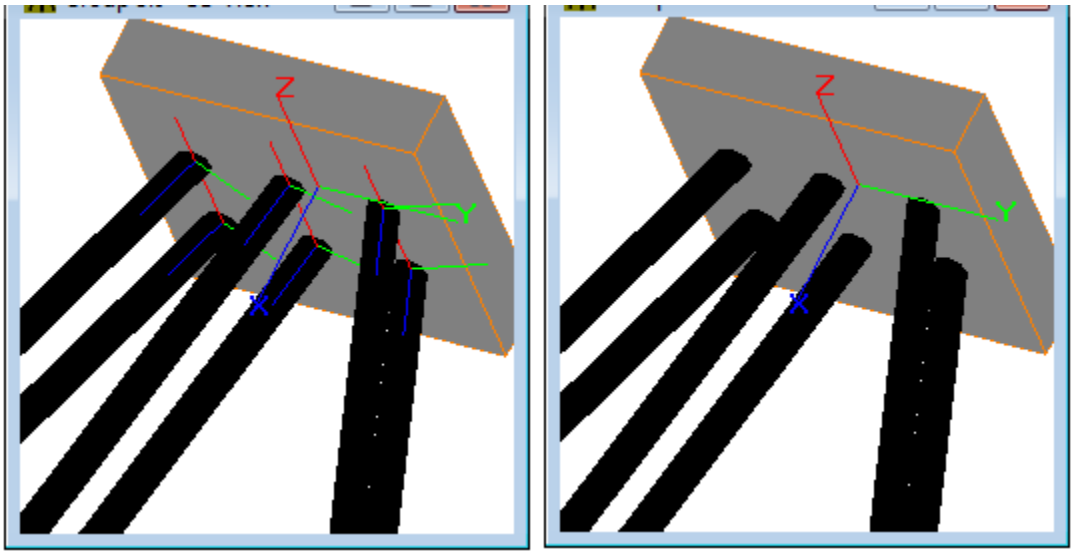
Figure 3.264 Effects on 3D model display using Pile Wire/Pile Shade options

3.7.6 Show 3D View -Pile Local Axes

This option allows the user to show the local axes at the top of each pile head.

The following color pattern is used:

- x axis, blue
- y axis, green
- z axis, red



a) Local Axes check-marked b) Local Axes not check-marked

Figure 3.265 Effects on 3D model display using Local Axes option

3.7.7 Show 3D View –Pile Head Connections

This option allows the user to show the symbols indicating the pile head connections at the top of each pile head.

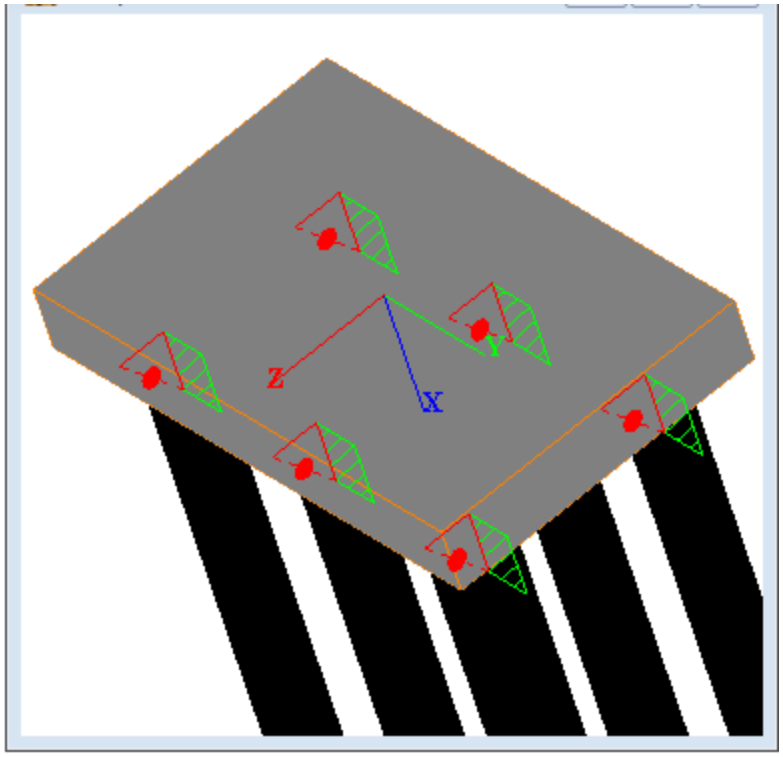
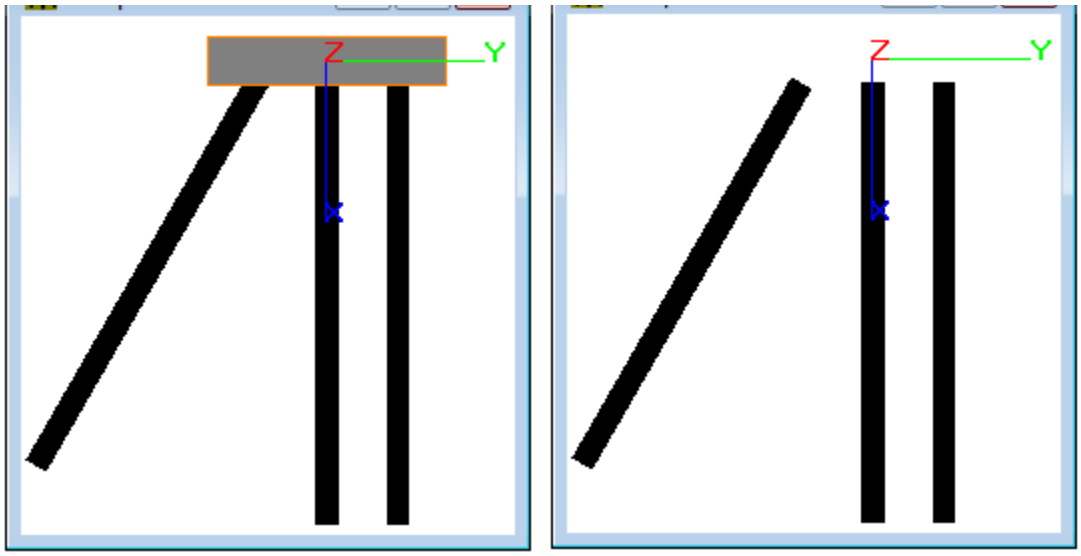


Figure 3.266 A 3D model display using Pile Head Connections option

3.7.8 Show 3D View -Pile Cap

This option allows the user to show the pile cap.

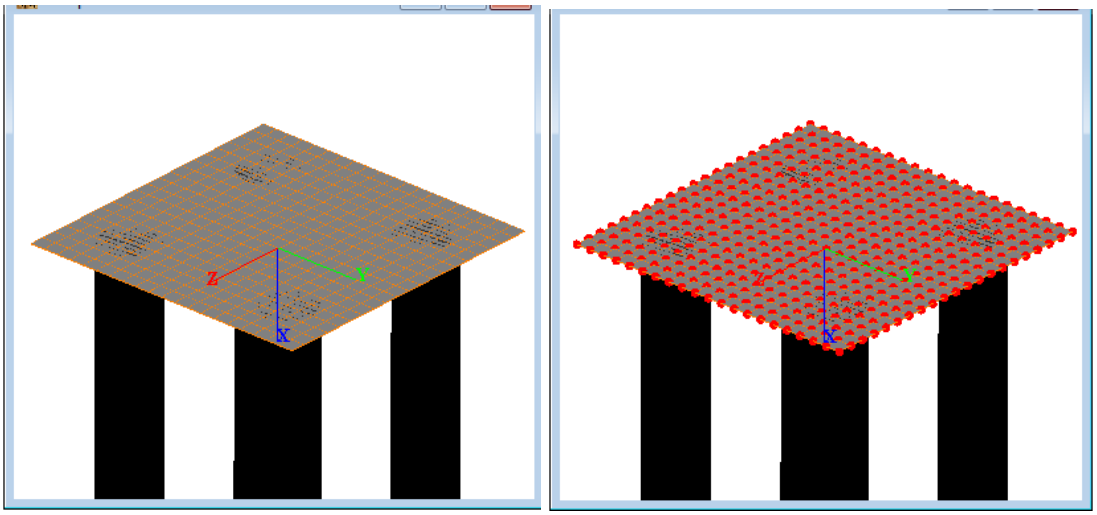


a) Pile Cap option check-marked b) Pile Cap option not check-marked

Figure 3.267 Effects on 3D model display using Pile Cap option

3.7.9 Show 3D View – FEM Nodes - Flexible Cap Version

This option allows the user to display the FEM nodes that belong to the finite element mesh defining the flexible pile-cap. Nodes will be draw with red color



a) FEM Nodes not check-marked b) FEM Nodes check-marked

Figure 3.268 Effects on 3D model display using FEM Nodes option. Flexible Cap Version

3.7.10 Show 3D View – Node Numbers - Flexible Cap Version

This option allows the user to display the node numbers. This option is enabled if the FEM Nodes are displayed.

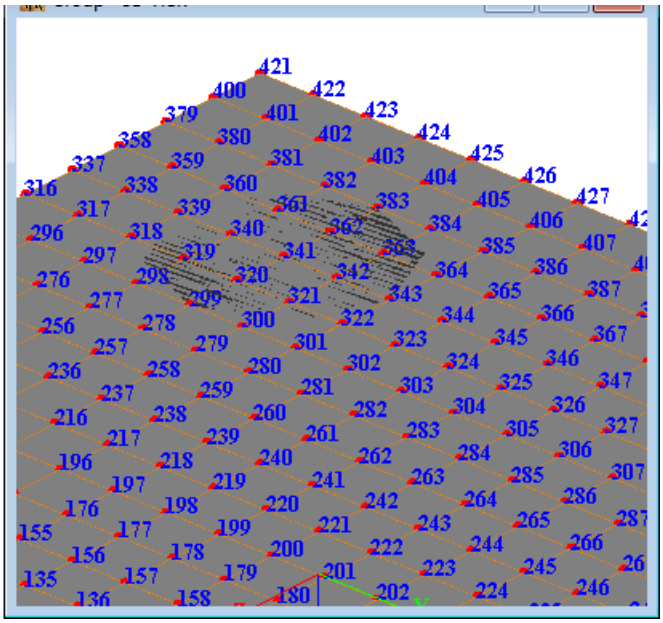
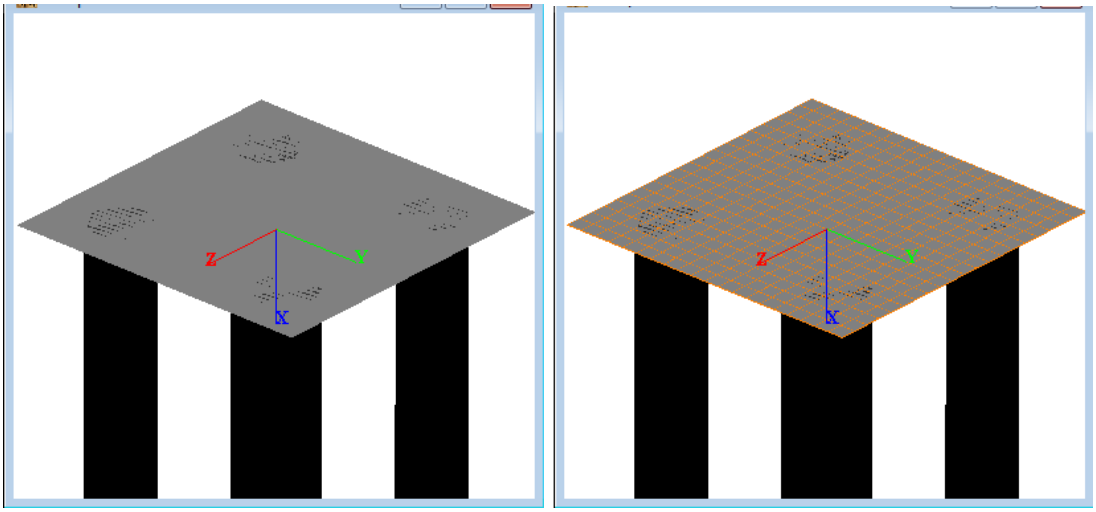


Figure 3.269 Node Numbers option. Flexible Cap Version

3.7.11 Show 3D View – FEM Edges - Flexible Cap Version)

This option allows the user to display the FEM Edges that belong to the finite element mesh defining the flexible pile-cap

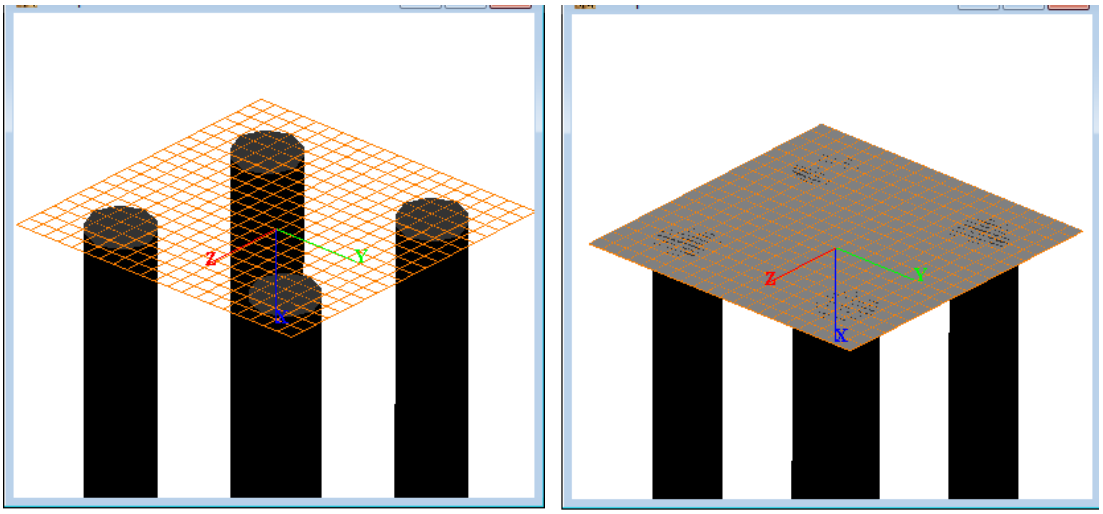


a) FEM Edges not check-marked b) FEM Edges check-marked

Figure 3.270 Effects on 3D model display using FEM Edges option - Flexible Cap Version

3.7.12 Show 3D View – FEM Plates - Flexible Cap Version

This option allows the user to display the FEM plates that belong to the finite element mesh defining the flexible pile-cap.

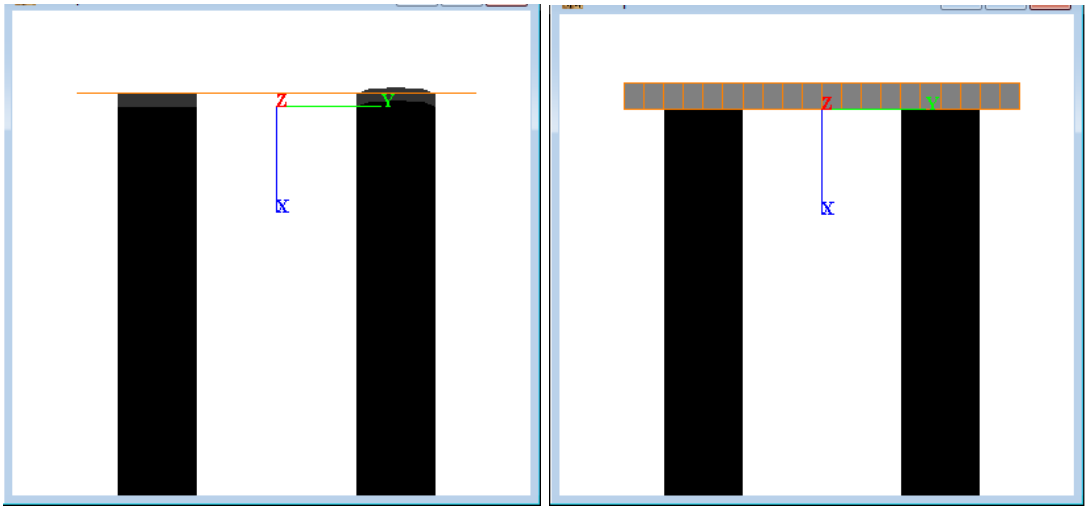


a) FEM Plates not check-marked b) FEM Plates check-marked

Figure 3.271 Effects on 3D model display using FEM Plates option - Flexible Cap Version

3.7.13 Show 3D View – Plate Thickness - Flexible Cap Version

This option allows the user to display the thickness of plate elements in the flexible pile-cap; otherwise plate elements are displayed with no thickness (default).



a) Plate Thickness not check-marked b) Plate Thickness check-marked

Figure 3.272 Effects on 3D model display using Plate Thickness option. Flexible Cap Version

3.7.14 Show 3D View – Plate Numbers - Flexible Cap Version

This option allows the user to display the plate numbers. This option is enabled if the FEM Plates are displayed.

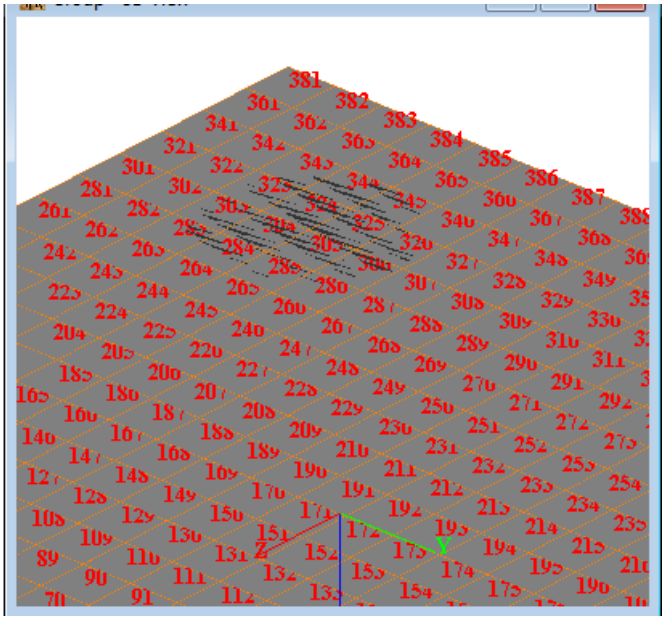
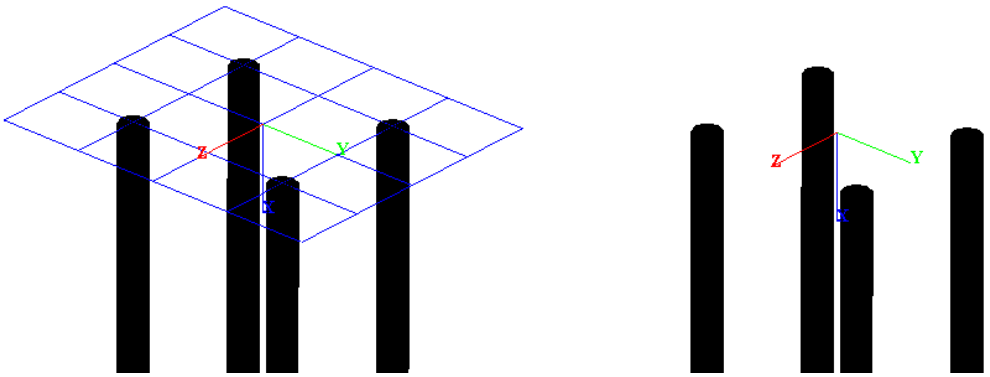


Figure 3.273 Plate Numbers option - Flexible Cap Version

3.7.15 Show 3D View – FEM Beams - Flexible Cap Version

This option allows the user to display the FEM beams that belong to the finite element mesh defining the flexible pile-cap.

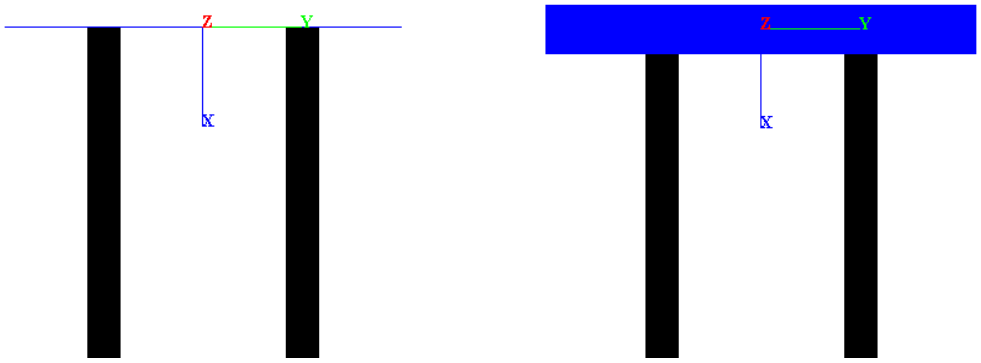


a) FEM Beams not check-marked b) FEM Beams check-marked

Figure 3.274 Effects on 3D model display using FEM Beams option - Flexible Cap Version

3.7.16 Show 3D View – Shade beams - Flexible Cap Version

This option allows the user to shade the beams elements in the flexible pile-cap; otherwise beam elements are displayed as lines (default).



a) Shade Beams not check-marked b) Shade Beams check-marked

Figure 3.275 Effects on 3D model display using Shade Beams option. Flexible Cap Version

3.7.17 Show 3D View – Beam Numbers - Flexible Cap Version

This option allows the user to display the beam numbers. This option is enabled if the FEM Beams are displayed.

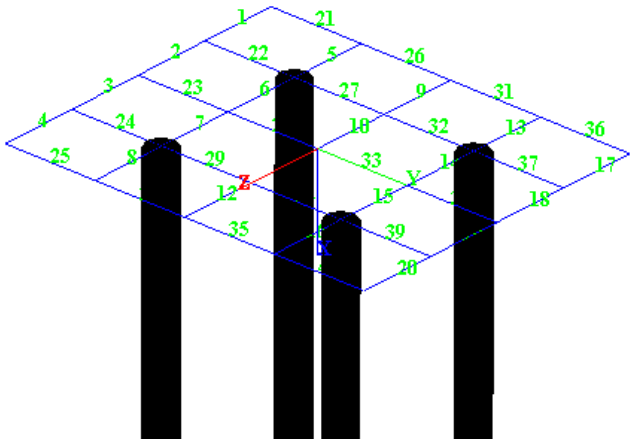


Figure 3.276 Beam Numbers option - Flexible Cap Version

3.7.18 Show 3D View – Beam Local Axes - Flexible Cap Version

This option allows the user to display the beam numbers. This option is enabled if the FEM Beams are displayed.

The following color pattern is used:

- x axis, black
- y axis, green
- z axis, red

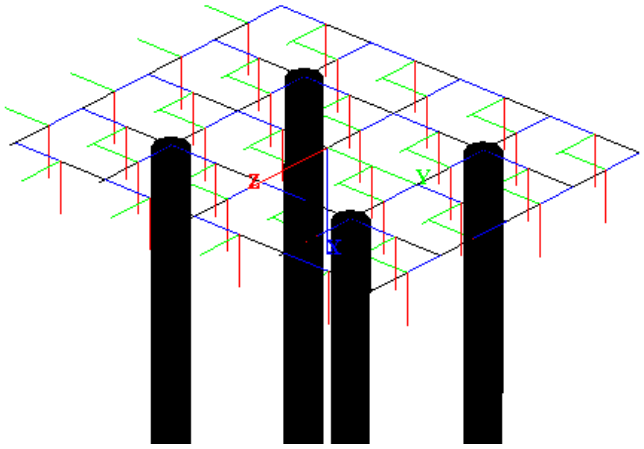
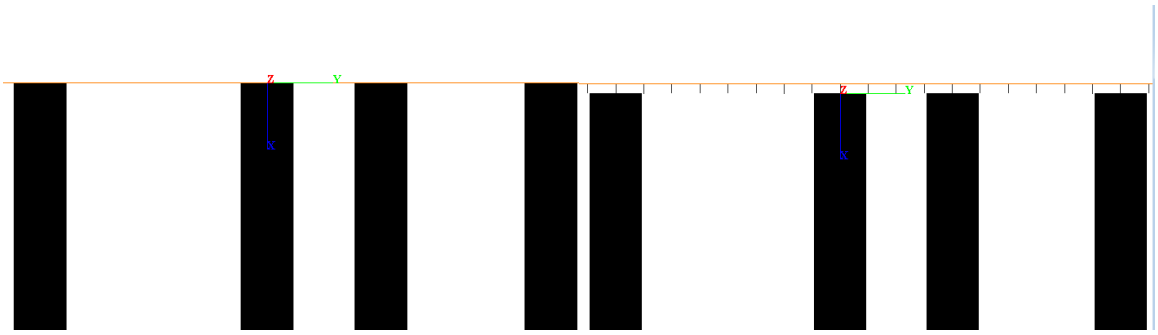


Figure 3.277 Beam Numbers option - Flexible Cap Version

3.7.19 Show 3D View – Plate/Beam Offsets - Flexible Cap Version

This option allows the user to display the offsets of the elements if they are defined. Program will display lines between the nodes and the corners of plate elements or ends of beam elements.

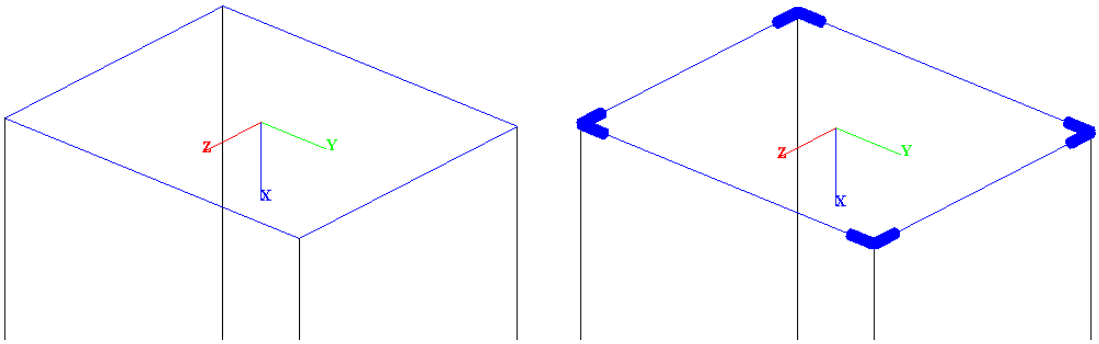


a) Plate/Beam Offsets not check-marked b Plate/Beam Offsets check-marked

Figure 3.278 **Plate/**Beam Offsets option - Flexible Cap Version

3.7.20 Show 3D View – Beam Local Axes - Flexible Cap Version

This option allows the user to display the end zones of the beams if they are defined.



a) Beam End Zones not check-marked b Beam End Zones check-marked

Figure 3.279 Beam Numbers option - Flexible Cap Version

3.7.21 Show 3D View – Concentrated Springs at Nodes (Flexible Pile Cap)

This option allows the user to show the concentrated springs at nodes.

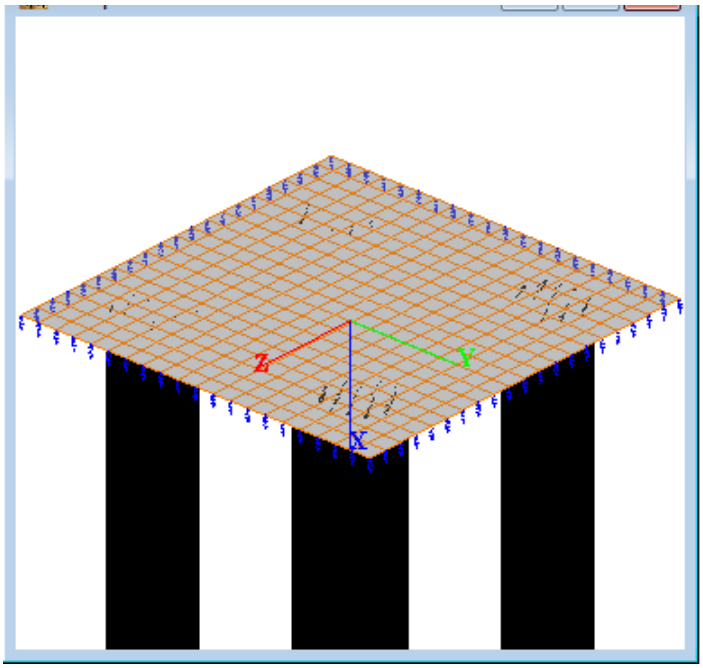


Figure 3.280 Concentrated Springs at Nodes option - Flexible Cap Version

3.7.22 Show 3D View - Distributed Springs on Plates (Flexible Pile Cap)

This option allows the user to show the distributed springs on plates.

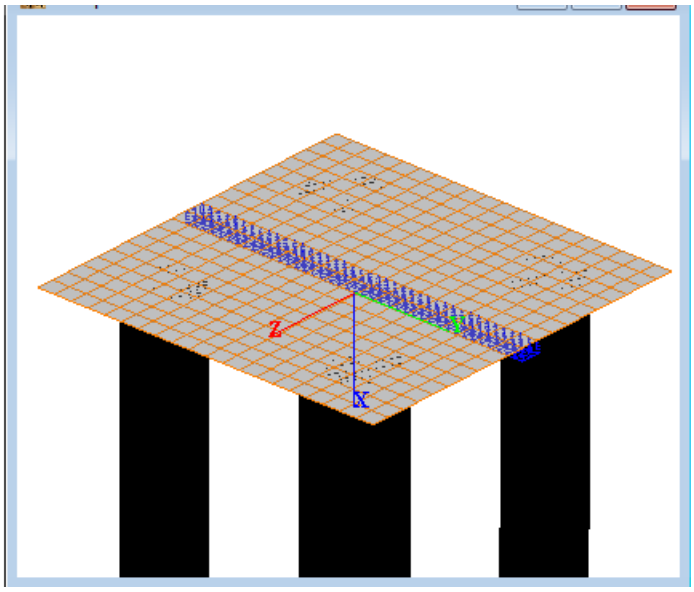


Figure 3.281 Distributed Springs on Plates option - Flexible Cap Version

3.7.23 Show 3D View - Distributed Springs on Beams (Flexible Pile Cap)

This option allows the user to show the distributed springs on beams.

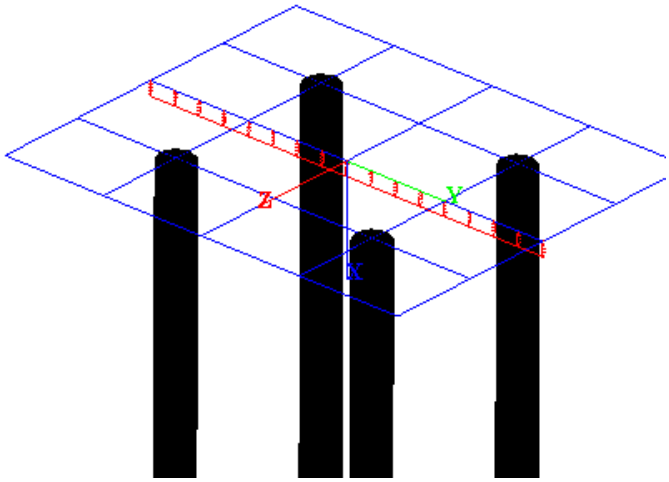


Figure 3.282 Distributed Springs on Plates option - Flexible Cap Version

3.7.24 Show 3D View -Soil Layers

This option allows the user to show the soil layers.

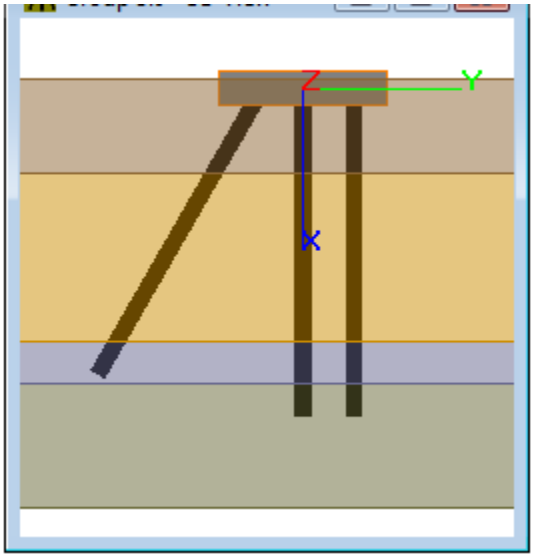


Figure 3.283 Effects on 3D model display using Soil Layers option

3.7.25 Show 3D View -Layer Labels

This option allows the user to show the layer labels.

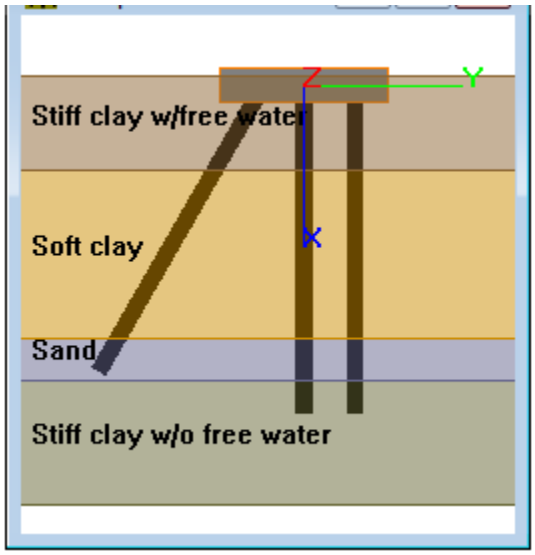


Figure 3.284 Effects on 3D model display using Layer Labels option

3.7.26 Show 3D View -Layer Depth Labels

This option allows the user to show the layer depth labels.

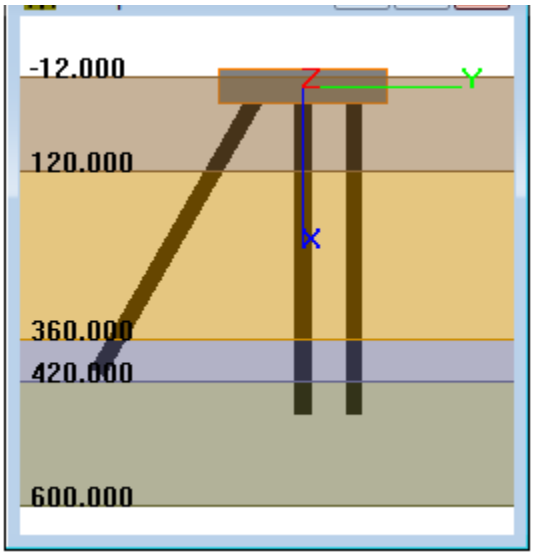


Figure 3.285 Effects on 3D model display using Layer Depth Labels option

3.7.27 Show 3D View - Concentrated Loads at Pile Cap

This option allows the user to show the concentrated loads at the pile cap for the current load case.

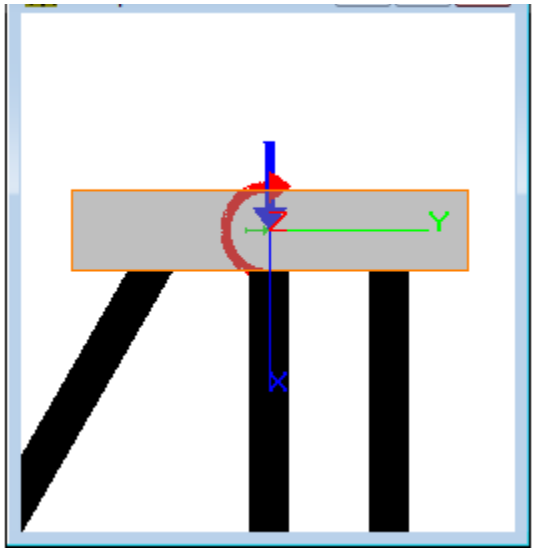


Figure 3.286 Effects on 3D model display using Concent. Loads at Pile Cap option

3.7.28 Show 3D View - Distributed Loads on Piles

This option allows the user to show the distributed loads on piles for the current load case.

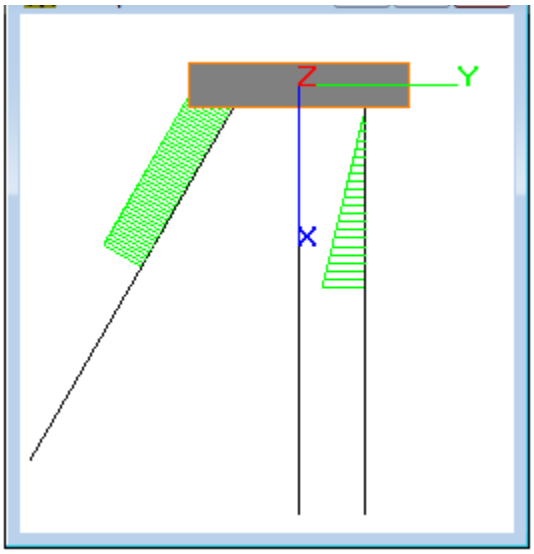


Figure 3.287 Effects on 3D model display using Distributed Loads at Piles option

Note that the distributed loads are drawn at the integration points assuming linear interpolation. If the integration points do not coincide with the depths defined in the distributed load sets, a difference may be shown in the drawing.

3.7.29 Show 3D View – Specified Displacements at Nodes (Flexible Pile Cap)

This option allows the user to show the specified displacements at nodes for the current load case.

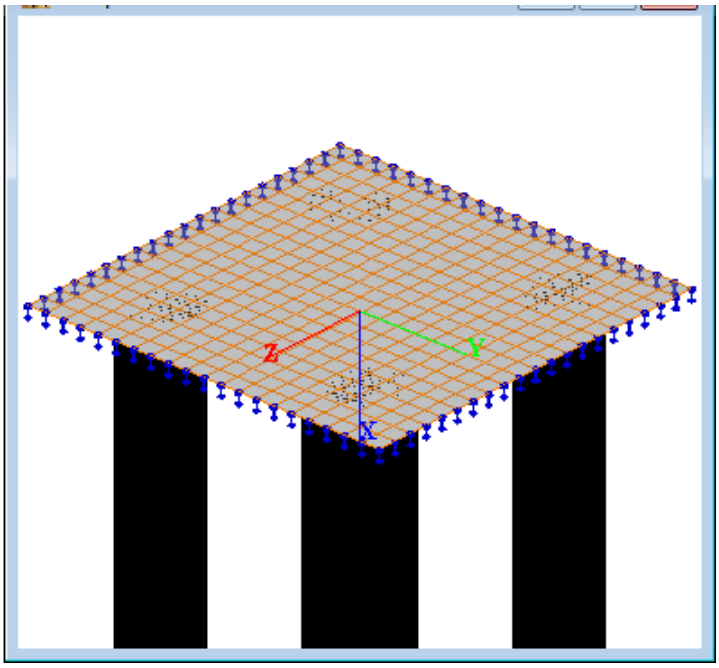


Figure 3.288 Specified Displacements at Nodes option - Flexible Cap Version

3.7.30 Show 3D View – Concentrated Loads at Nodes (Flexible Pile Cap)

This option allows the user to show the concentrated loads at nodes for the current load case.

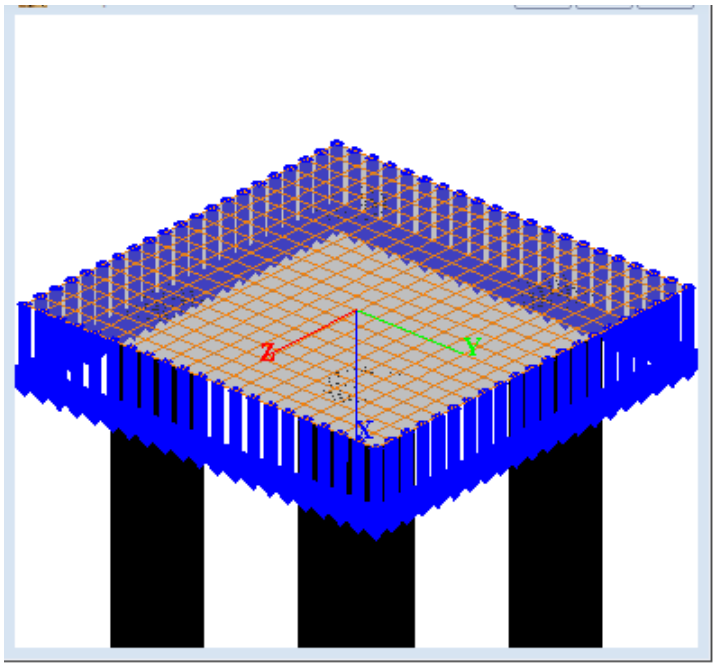


Figure 3.289 Concentrated Loads at Nodes option - Flexible Cap Version

3.7.31 Show 3D View - Distributed Loads on Plates (Flexible Pile Cap)

This option allows the user to show the distributed loads on plates for the current load case.

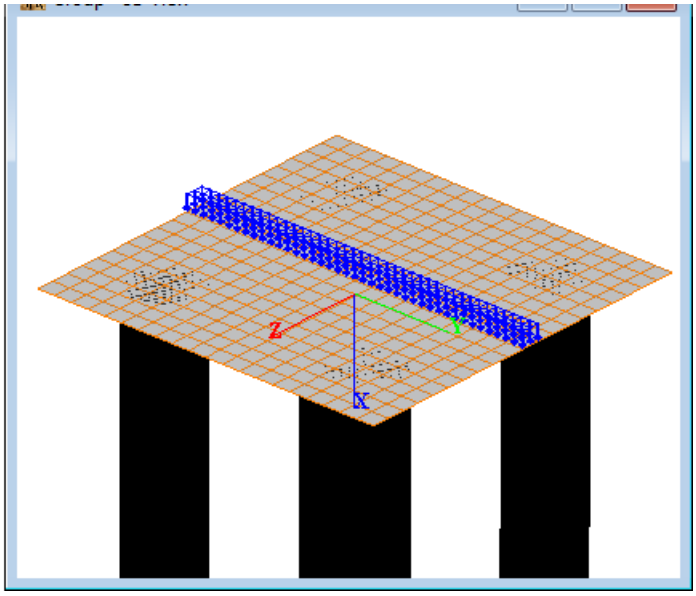


Figure 3.290 Distributed Loads on Plates option - Flexible Cap Version

3.7.32 Show 3D View - Distributed Loads on Beams (Flexible Pile Cap)

This option allows the user to show the distributed loads on beams for the current load case.

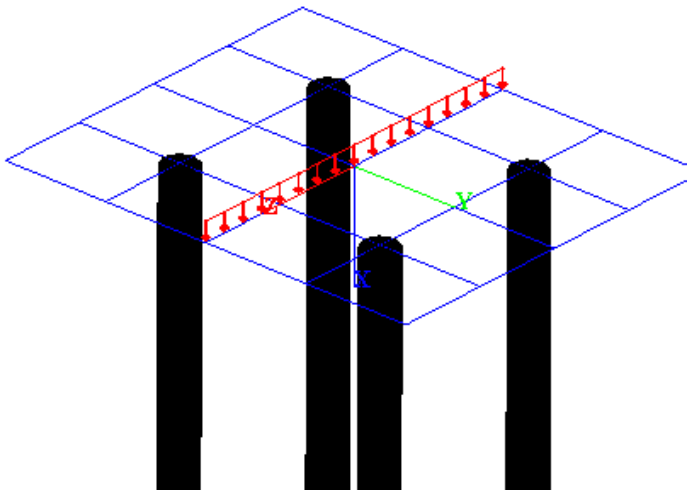


Figure 3.291 Distributed Loads on Beams option - Flexible Cap Version

3.7.33 Show 3D View - Concentrated Loads on Beams (Flexible Pile Cap)

This option allows the user to show the concentrated loads on beams for the current load case.

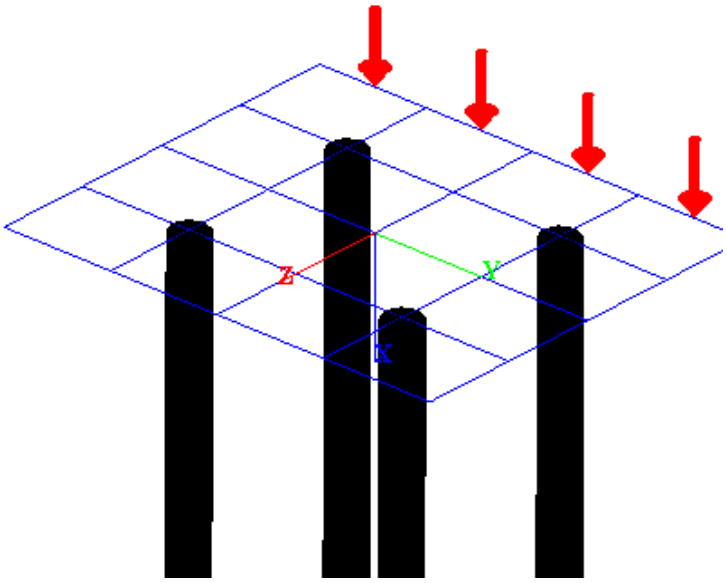


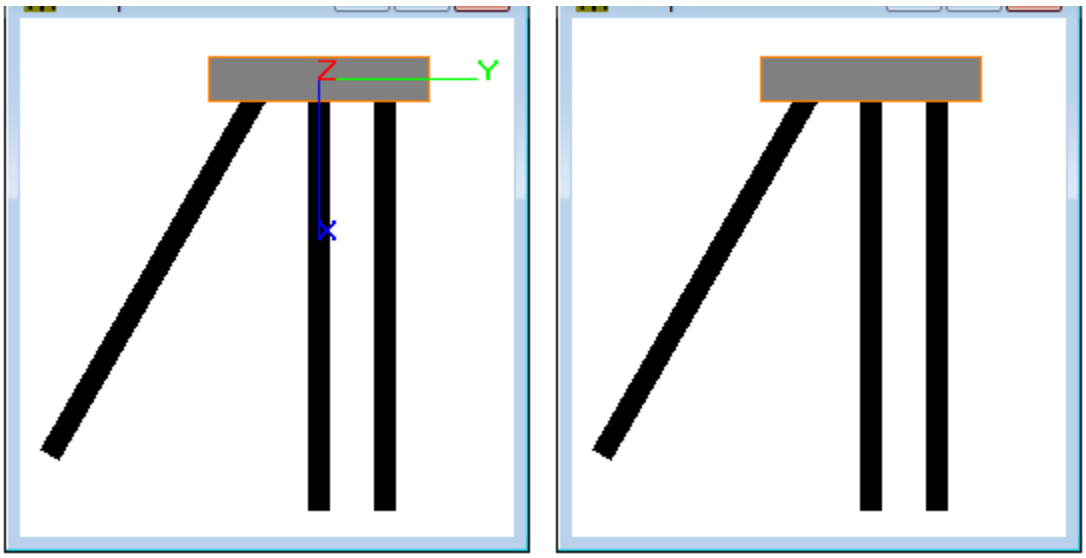
Figure 3.292 Distributed Loads on Beams option - Flexible Cap Version

3.7.34 Show 3D View -Global Axes

This option allows the user to show the global axes at the origin of coordinates.

The following color pattern is used:

- X axis, blue
- Y axis, green
- Z axis, red

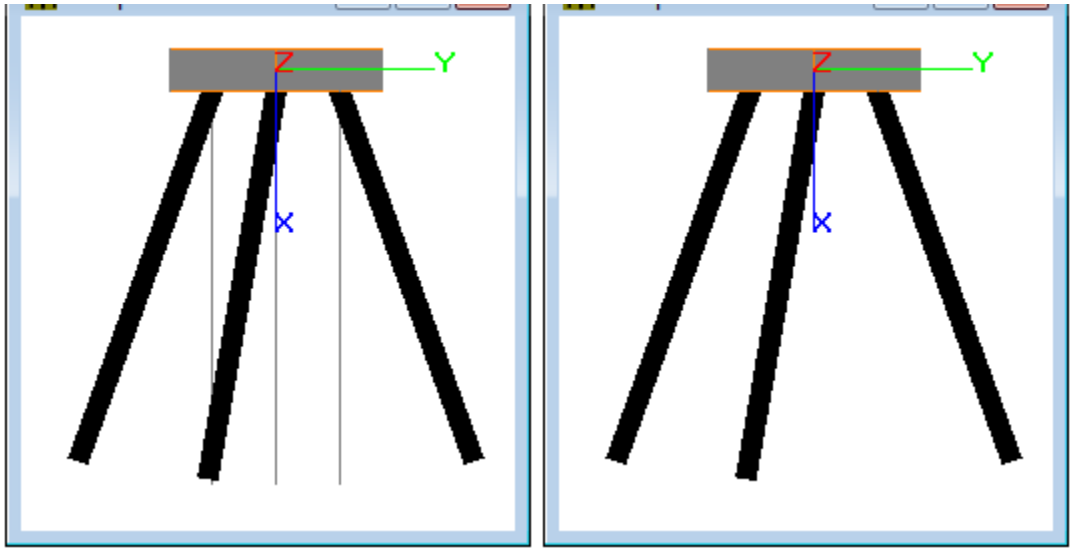


a) Global Axes check-marked b) Global Axes not check-marked

Figure 3.293 Effects on 3D model display using Global Axes option

3.7.35 Show 3D View -Vertical Lines

This option allows the user to show vertical lines that start from the top of each pile head.



a) Vertical Lines check-marked b) Vertical Lines not check-marked

Figure 3.294 Effects on 3D model display using Vertical Lines option

3.7.36 Show 3D View -Show Loading Options

This option opens the Show Loading Options Window that allows the user to change the default drawing for loading and boundary conditions.

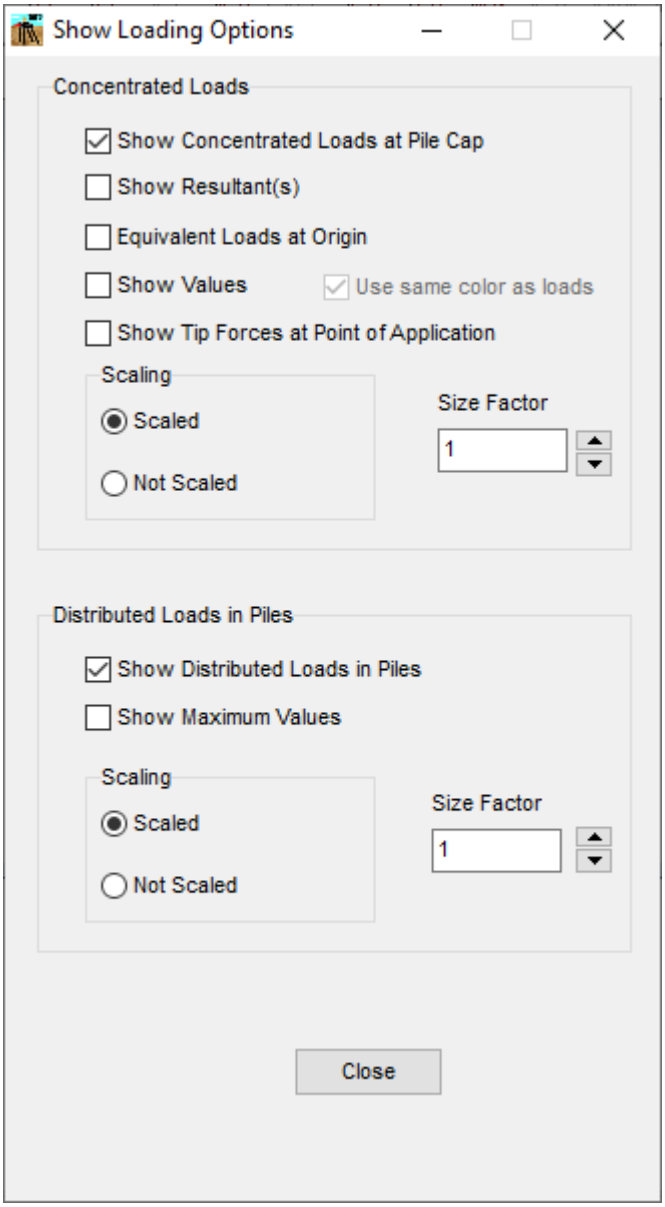


Figure 3.295 Show Loading Options Window

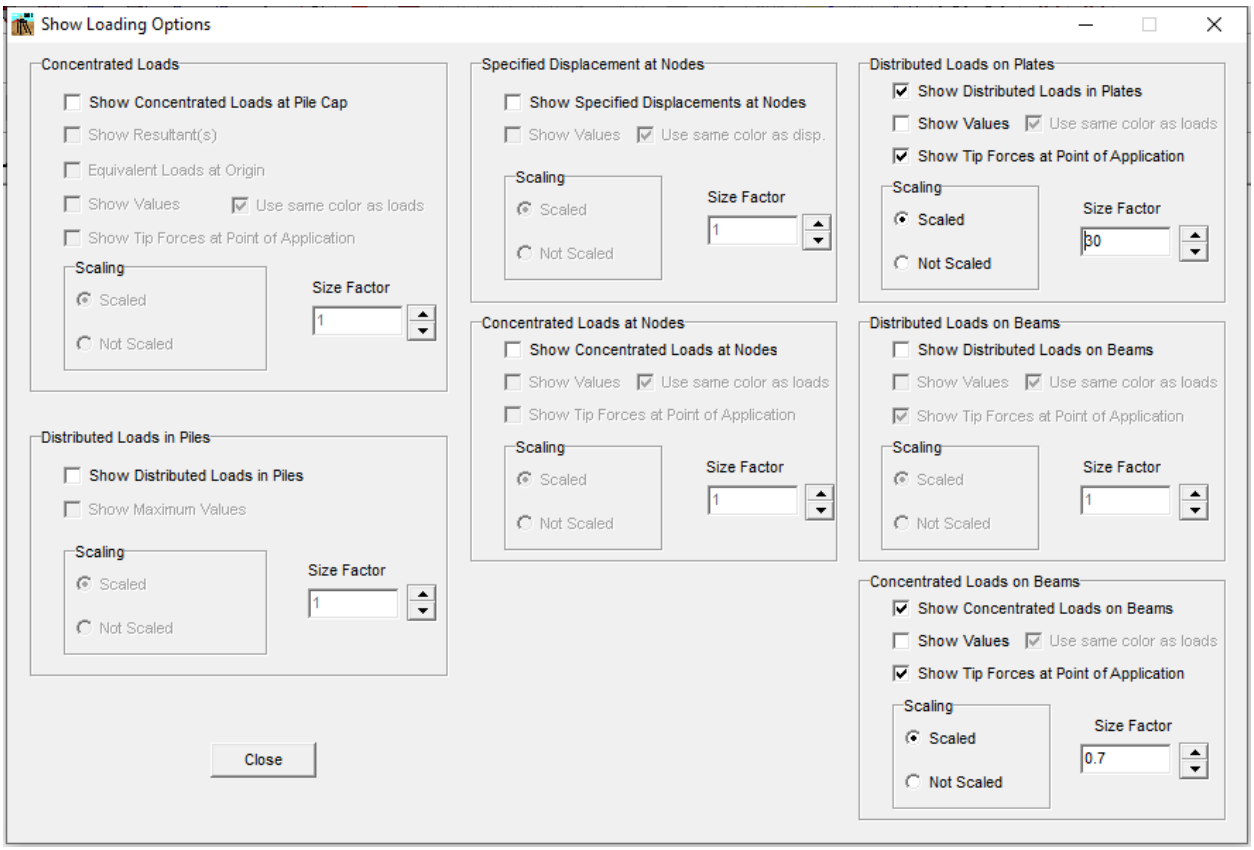


Figure 3.296 Show Loading Options Window - Flexible Cap Version

3.7.37 Show 3D View -Show Springs Options – Flexible Cap Version

This option opens the Show Springs Options Window that allows the user to change the default drawing for springs.

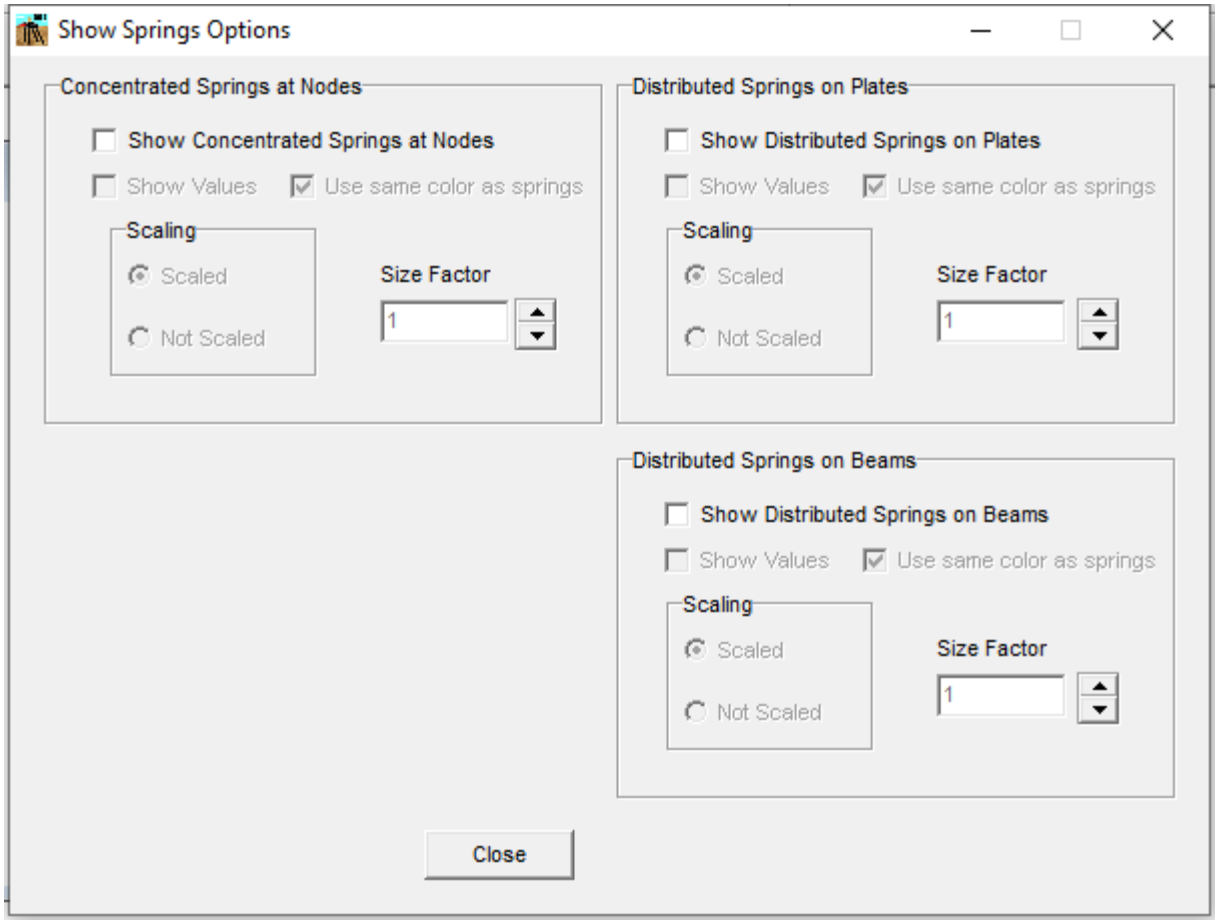


Figure 3.297 Show Sprimgs Options Window – Flexible Cap Version

3.7.38 Show 3D View –Display Values Options

This option opens the Display Values Window that allows the user to change the default format for displaying values in the 3D -View.

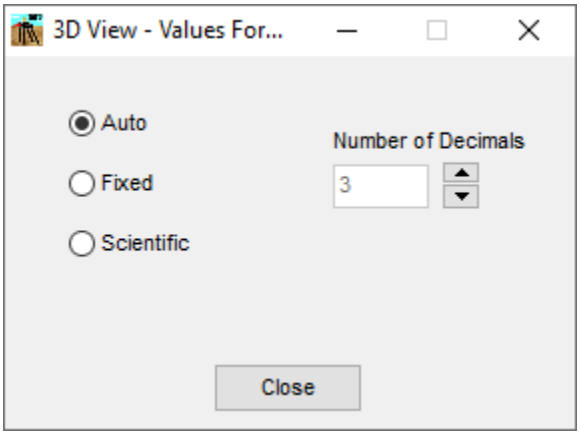
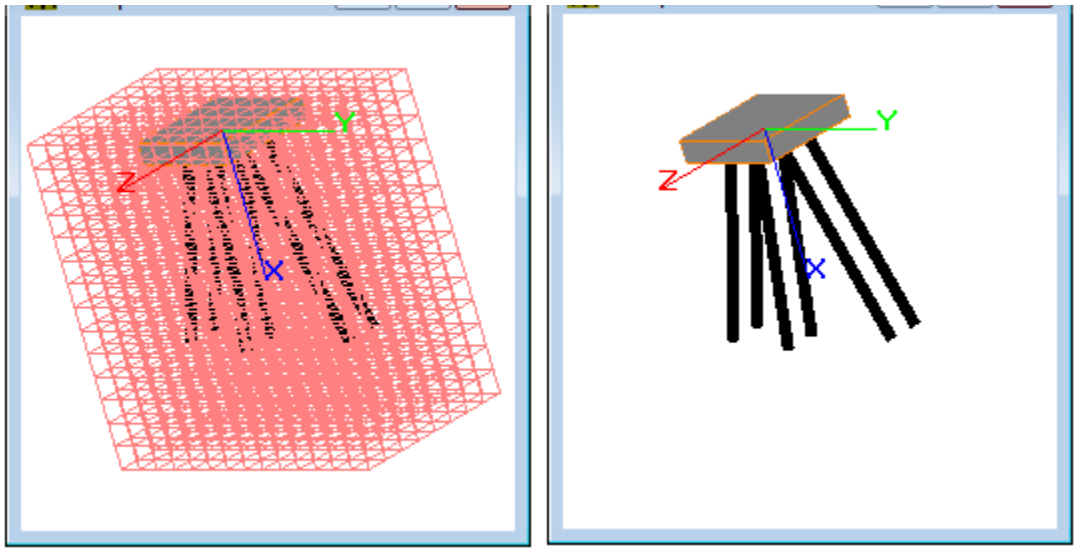


Figure 3.298 Display Values Options Window

3.7.39 Show 3D View -Show Grid

This option allows the user to show a three-dimensional grid.



a) Show Grid check-marked b) Show Grid not check-marked

Figure 3.299 Effects on 3D model display using Show Grid option

3.7.40 Show 3D View - Grid Options

This option allows the user to change the increment values for the 3D grid.

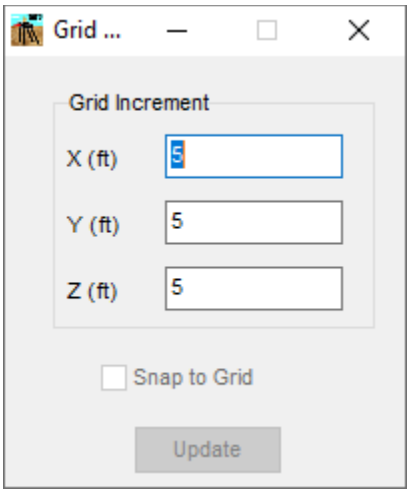
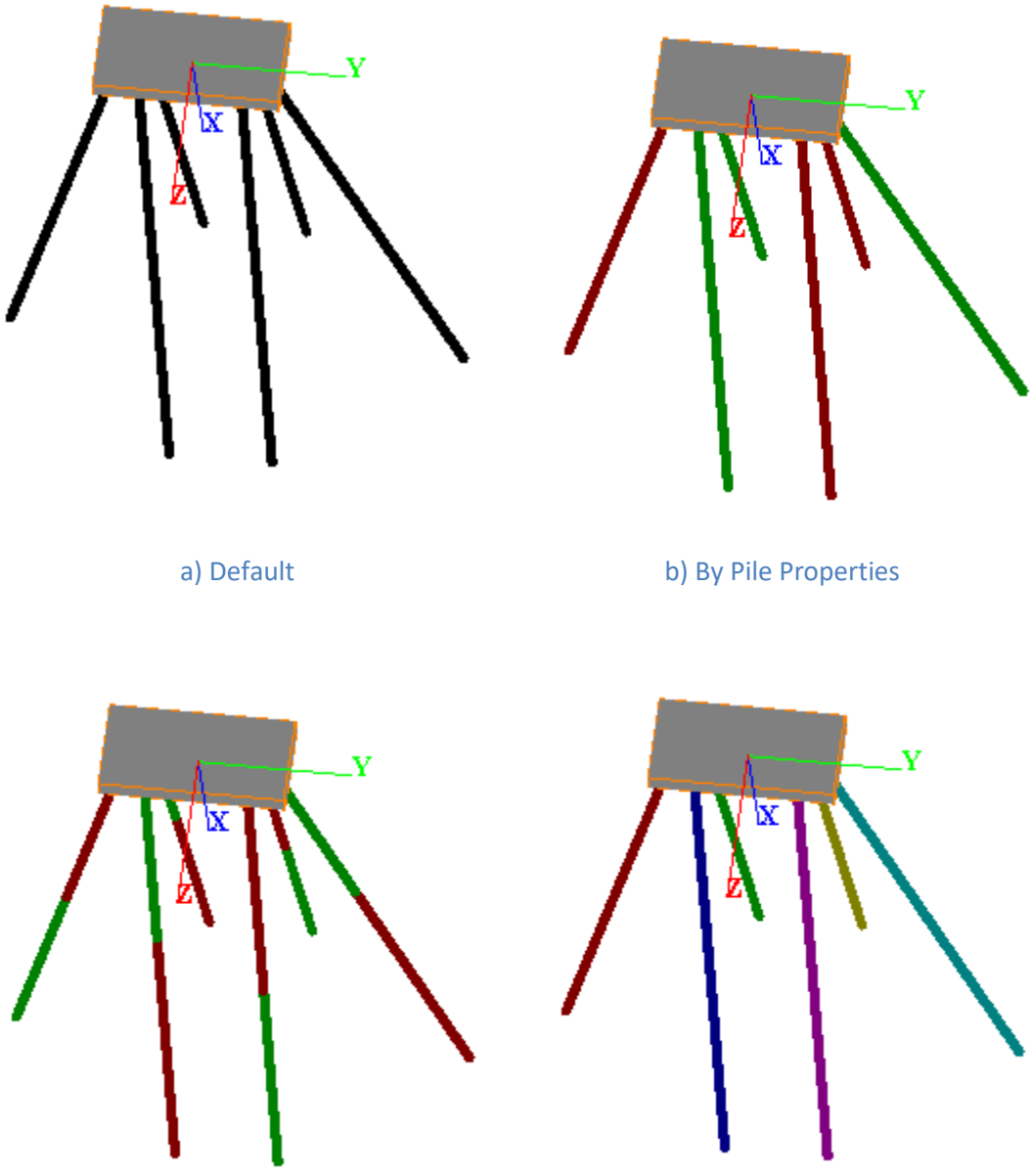


Figure 3.300 Grid Options Window

3.7.41 Show 3D View – Pile Color

This option allows the user to color the piles using six options: Default, by pile properties, by pile sections, by pile numbers, by generated LD curves and by generated TR curves.



a) Default b) By Pile Properties

c) By Pile Sections d) By Pile Numbers

Figure 3.301 Effects on 3D model display using Pile Color option

3.7.42 Show 3D View - Highlight Piles

This option allows the user to highlight piles using various options, showing the highlighted pile(s) with a yellow color.

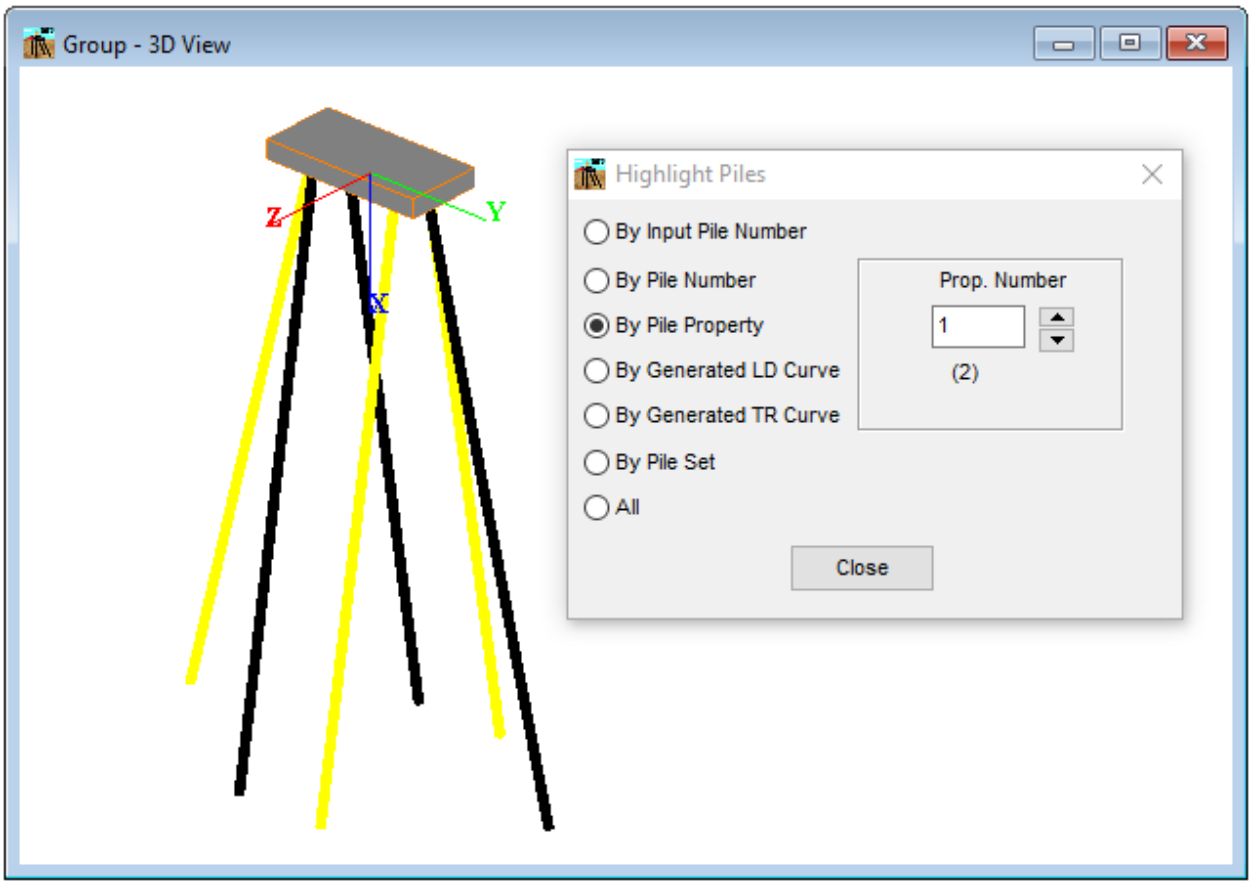


Figure 3.302 Effects on 3D model display using Highlight Piles option

3.7.43 Show 3D View - Highlight Plates - Flexible Cap Version)

This option allows the user to highlight plates using various options, showing the highlighted plate(s) with a light blue color.

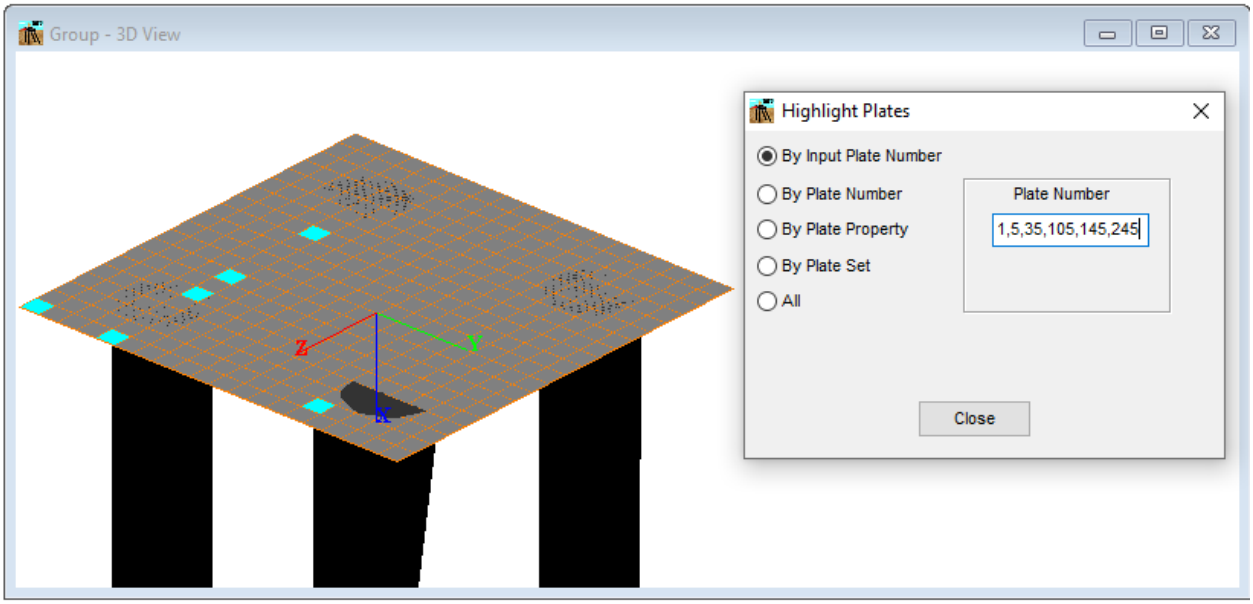


Figure 3.303 Effects on 3D model display using Highlight Plates option – Flexible Cap Version

3.7.44 Show 3D View - Highlight Beams - Flexible Cap Version)

This option allows the user to highlight beams using various options, showing the highlighted beam(s) with a yellow color.

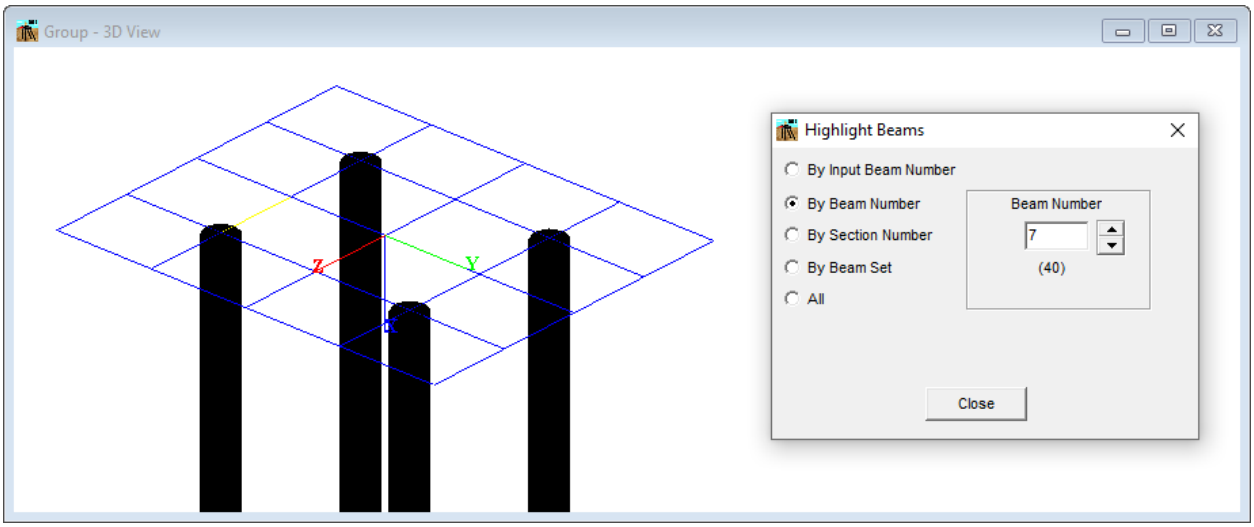


Figure 3.304 Effects on 3D model display using Highlight Beams option – Flexible Cap Version

3.7.45 Show 3D View - Highlight Nodes - Flexible Cap Version

This option allows the user to highlight nodes using various options, showing the highlighted node(s) with a yellow color.

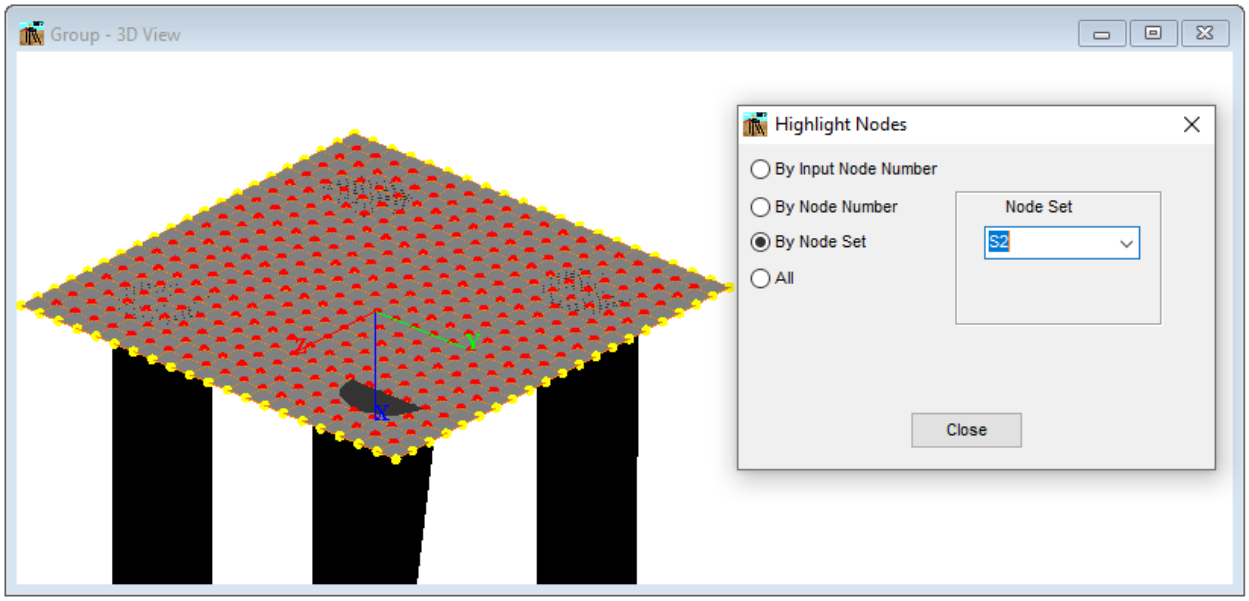


Figure 3.305 Effects on 3D model display using Highlight Nodes option – Flexible Cap Version

3.7.46 Show 3D View - Select Piles

This option allows the user to select piles using various options. The selected pile(s) will be displayed on red color.

This option is enabled if the option Select Piles is checked under the Edit Menu or the Select Piles Tool Button was checked.

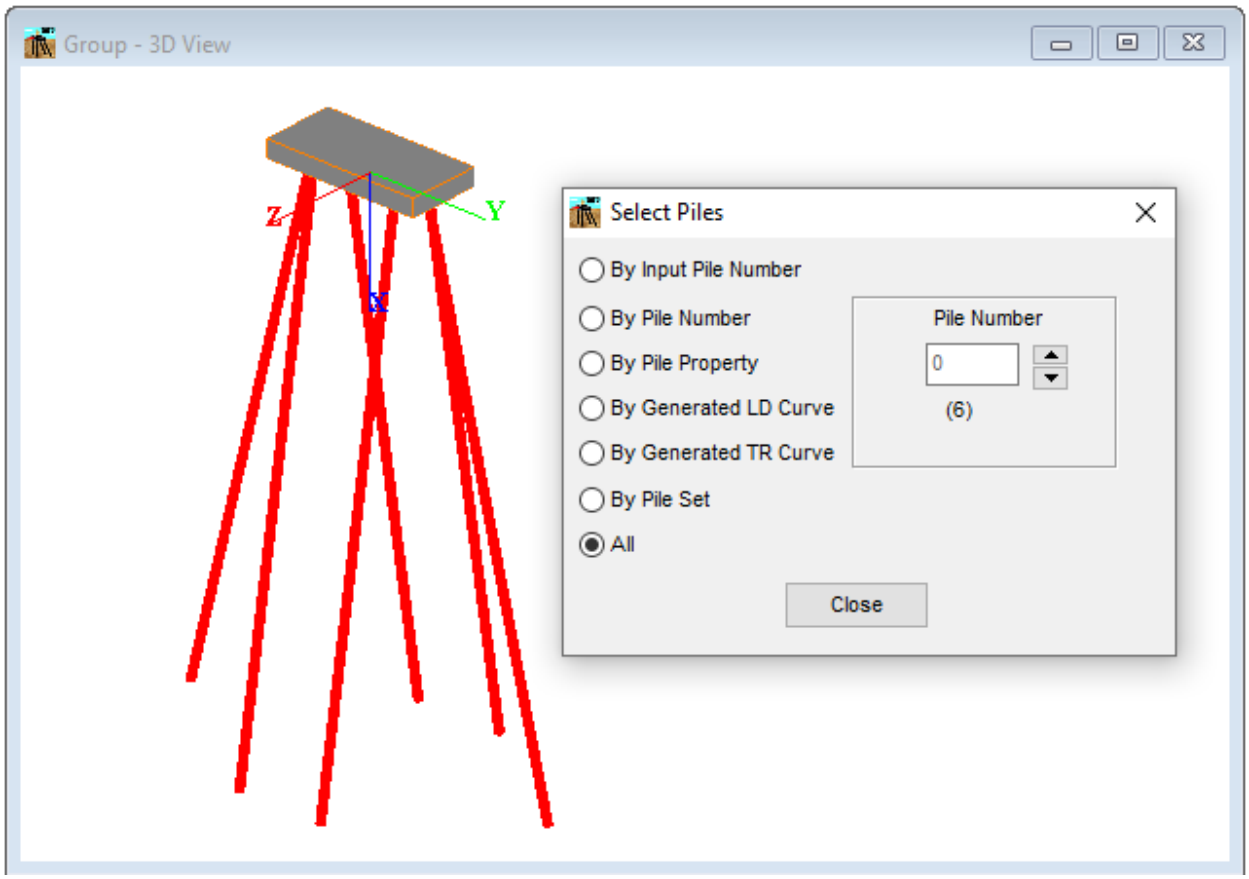


Figure 3.306 Effects on 3D model display using Select Piles option

3.7.47 Show 3D View - Select Plates - Flexible Cap Version

This option allows the user to select plates using various options. The selected plates(s) will be displayed on magenta color.

This option is enabled if the option Select Plates is checked under the Edit Menu or the Select Plates Tool Button was checked.

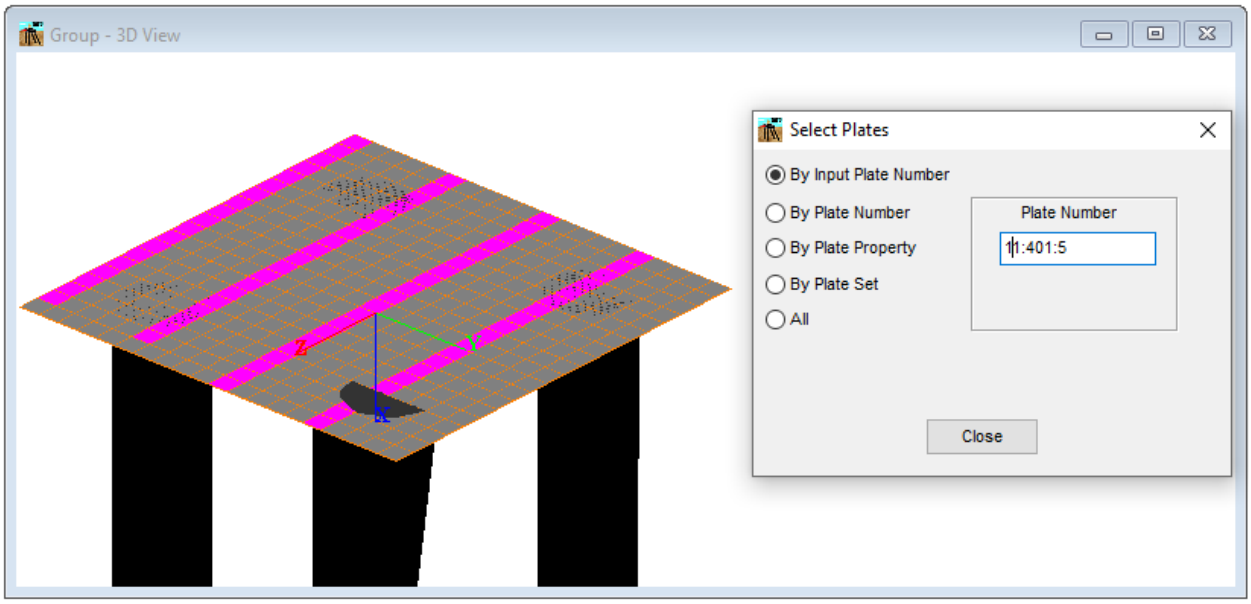


Figure 3.307 Effects on 3D model display using Select Plates option- Flexible Cap Version

3.7.48 Show 3D View - Select Beams - Flexible Cap Version

This option allows the user to select beams using various options. The selected beam(s) will be displayed on red color.

This option is enabled if the option Select Beams is checked under the Edit Menu or the Select Beams Tool Button was checked.

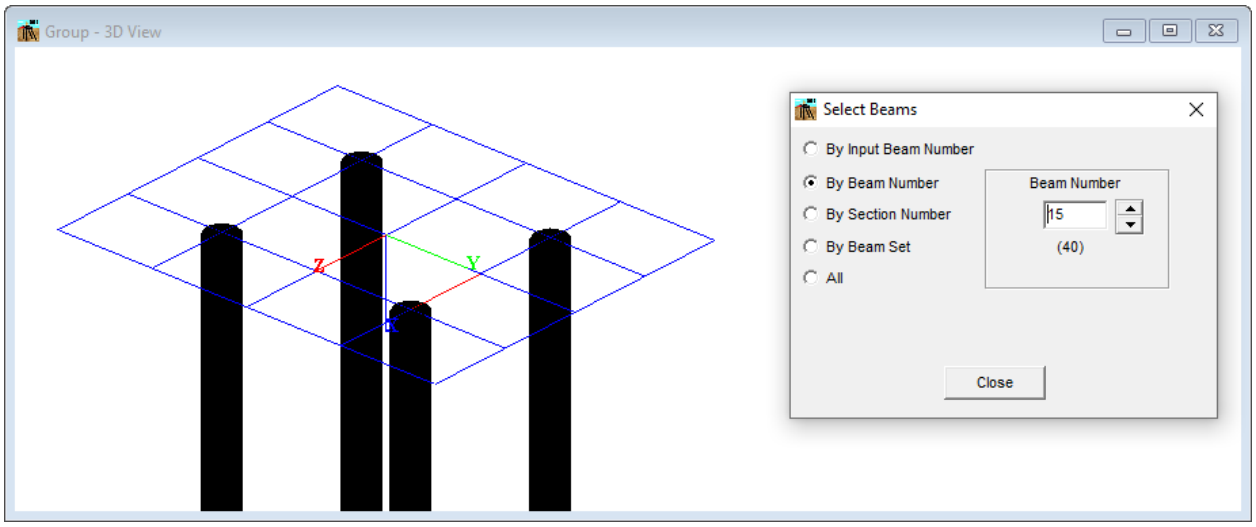


Figure 3.308 Effects on 3D model display using Select Beams option- Flexible Cap Version

3.7.49 Show 3D View - Select Nodes - Flexible Cap Version

This option allows the user to select nodes using various options. The selected node(s) will be displayed on light blue color.

This option is enabled if the option Select Nodes is checked under the Edit Menu or the Select Nodes Tool Button was checked.

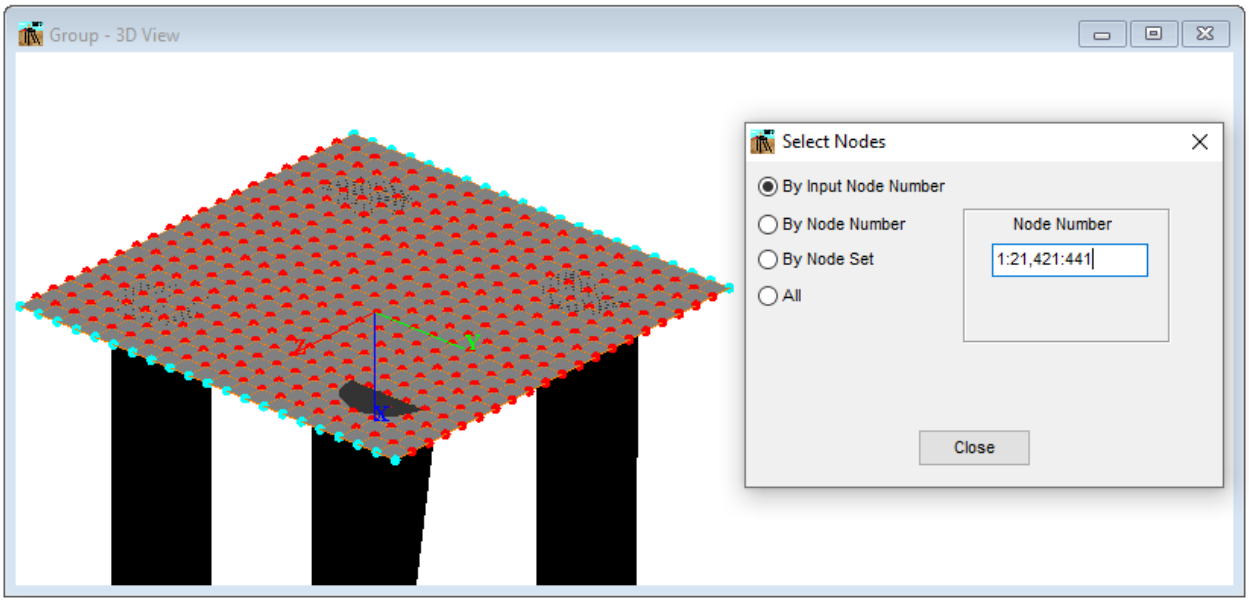
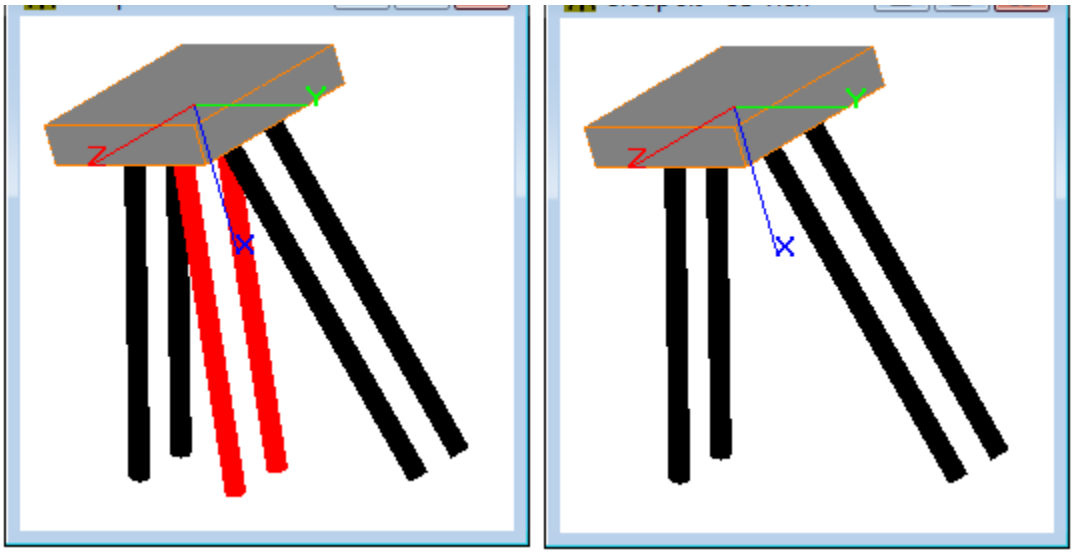


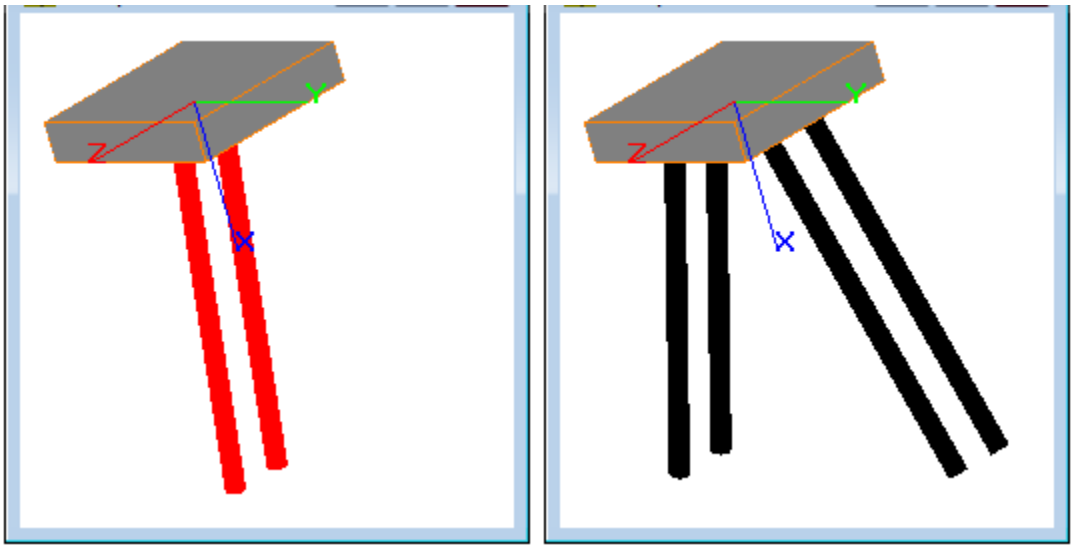
Figure 3.309 Effects on 3D model display using Select Nodes option- Flexible Cap Version

3.7.50 Show 3D View -Hide/Show Piles

This option allows the user to hide the selected piles.



a) Piles Select b) Using Hide Selected Piles option

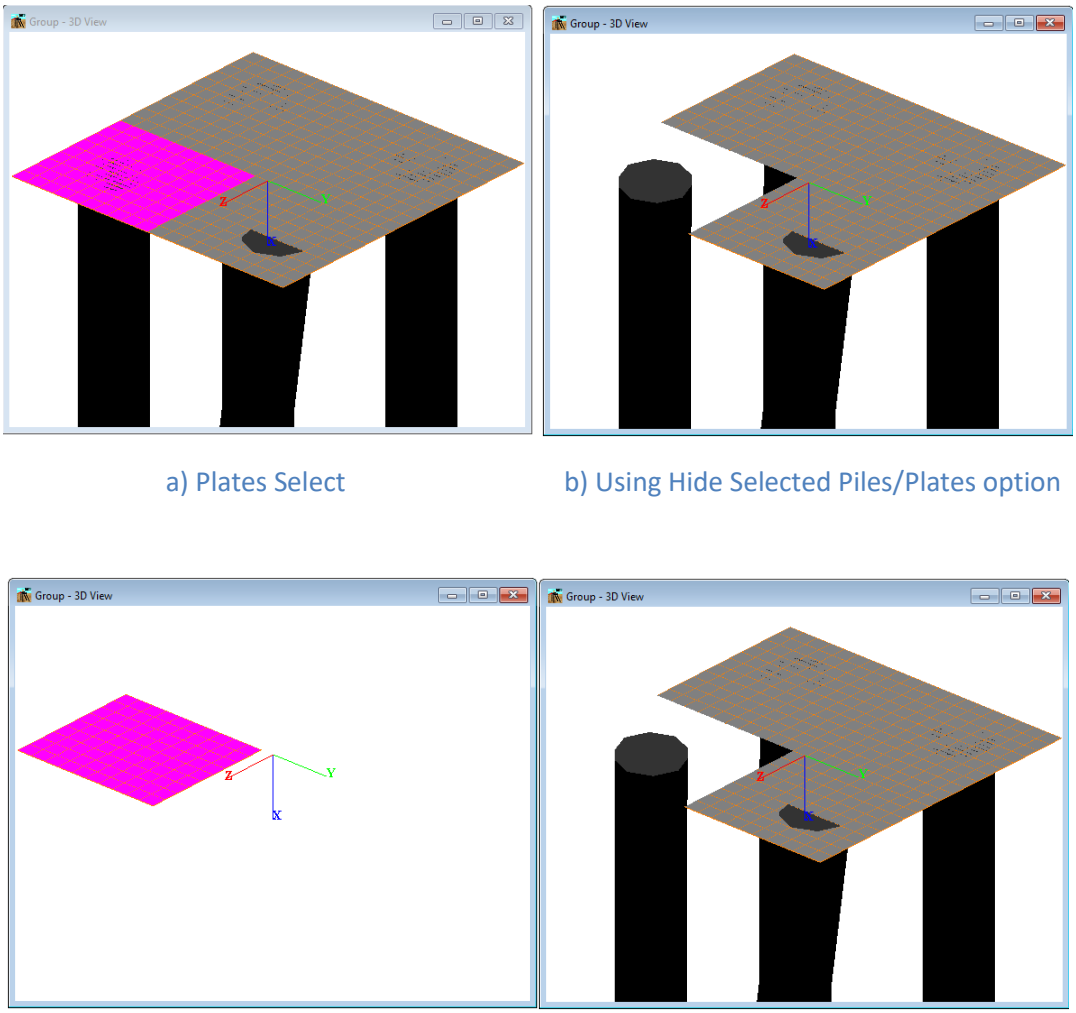


c) Using Hide Except Selected Piles option d) Using Reverse Hiding option

Figure 3.310 Effects on 3D model display using Show/Hide Piles option

3.7.51 Show 3D View -Hide/Show Piles, Plates, Beams - Flexible Cap Version

This option allows the user to hide the selected piles and/or plates and/or beams.



a) Plates Select

b) Using Hide Selected Piles/Plates option

c) Using Hide Except Selected Piles, Plates option

d) Using Reverse Hiding option

Figure 3.311 Effects on 3D model display using Show/Hide Piles, Plates, Beams option – Flexible Cap Version

3.8 Window Menu

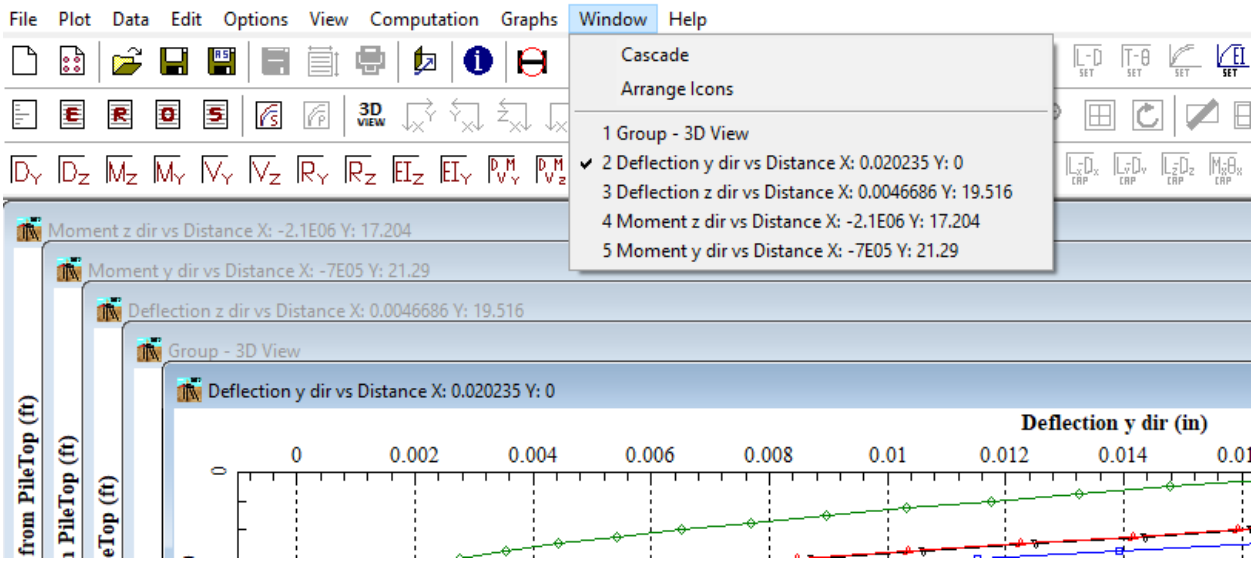


Figure 3.312 Options contained in the Windows Menu

3.8.1 Window - Cascade

This option organizes all open windows so that they are all visible. See Figure 3.312 for an example.

3.8.2 Window - Arrange Icons

Any minimized windows will be aligned at the bottom left of the screen.

3.8.3 Window - Windows

After the options **Cascade** and **Arrange Icons**, the windows opened are listed in order of depth. Clicking on a window name through the menu brings it to the top.

3.9 Help Menu

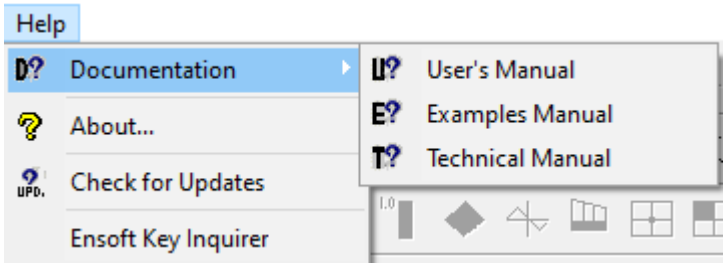


Figure 3.313 Options contained in the Help Menu

3.9.1 Help- Documentation User's Manual

This option opens the User's Manual that is provided as a digital version in pdf format.

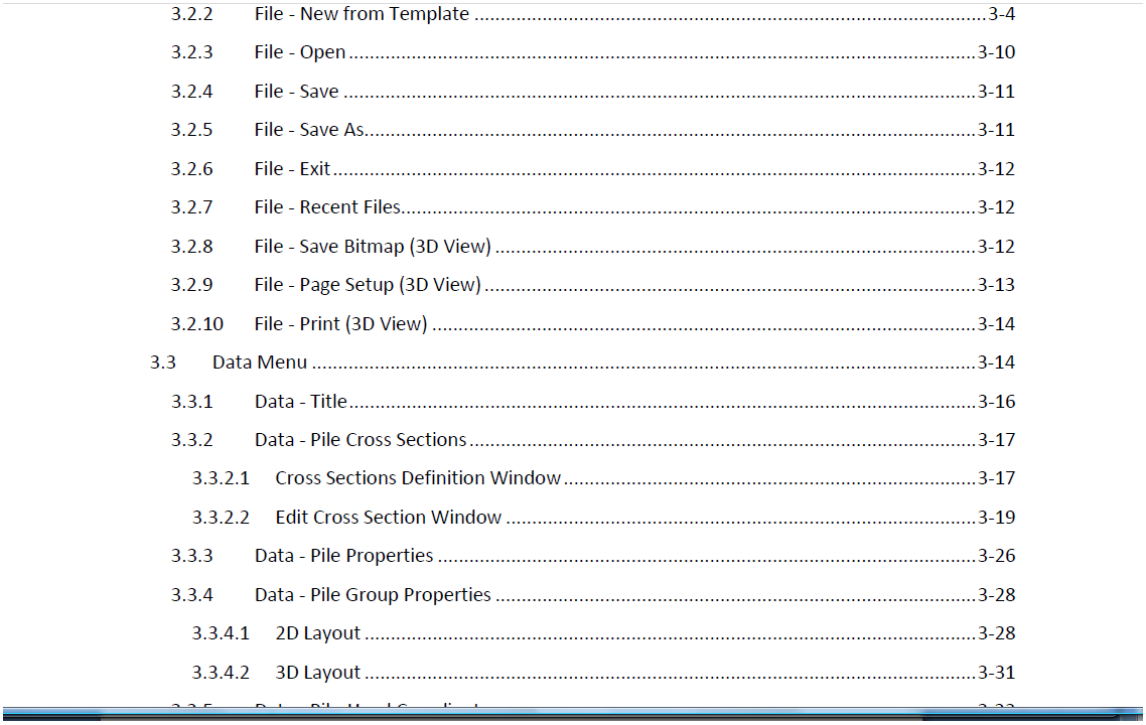
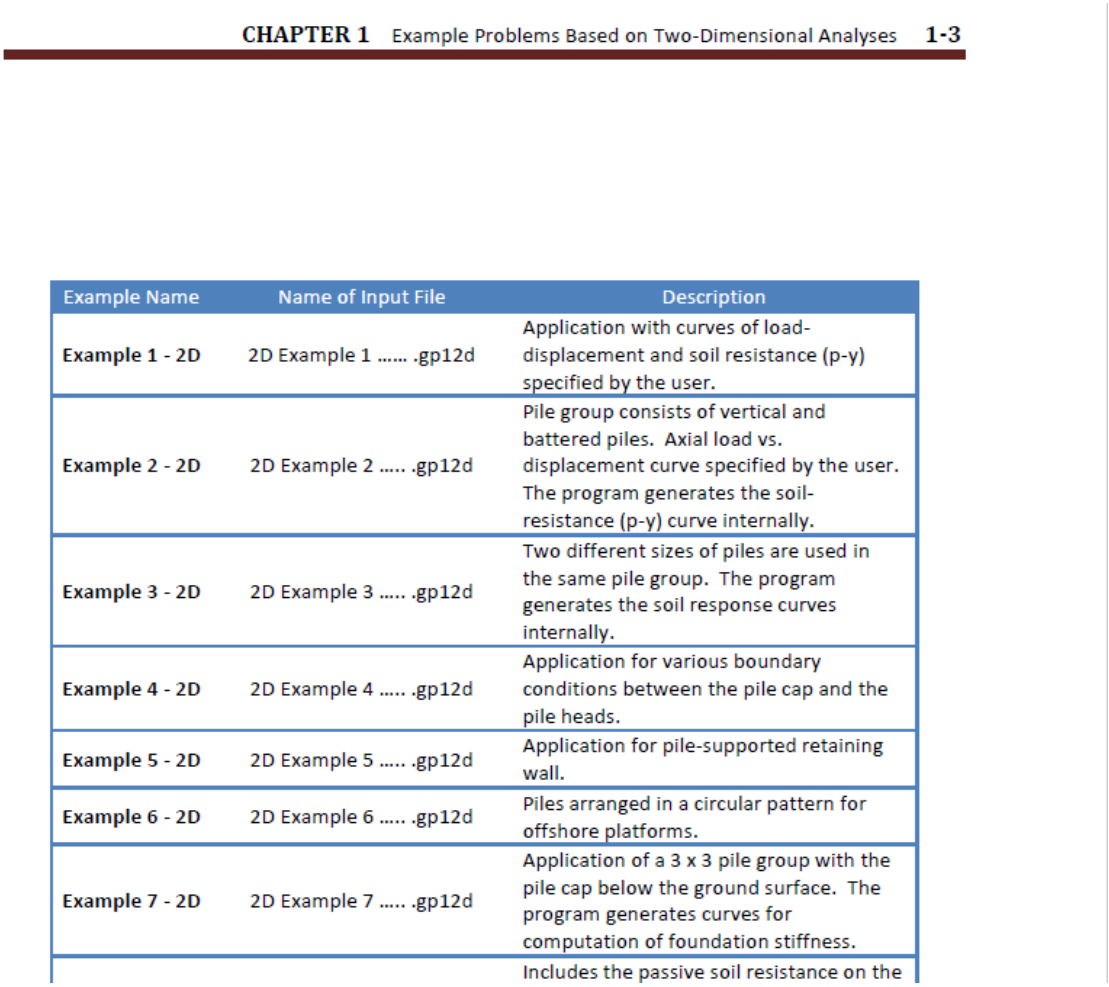

3.2.2 File - New from Template 3-4
3.2.3 File - Open 3-10
3.2.4 File - Save 3-11
3.2.5 File - Save As 3-11
3.2.6 File - Exit 3-12
3.2.7 File - Recent Files 3-12
3.2.8 File - Save Bitmap (3D View) 3-12
3.2.9 File - Page Setup (3D View) 3-13
3.2.10 File - Print (3D View) 3-14
3.3 Data Menu 3-14
3.3.1 Data - Title 3-16
3.3.2 Data - Pile Cross Sections 3-17
3.3.2.1 Cross Sections Definition Window 3-17
3.3.2.2 Edit Cross Section Window 3-19
3.3.3 Data - Pile Properties 3-26
3.3.4 Data - Pile Group Properties 3-28
3.3.4.1 2D Layout 3-28
3.3.4.2 3D Layout 3-31

Figure 3.314 Sample of User's Manual

3.9.2 Help- Documentation Example Manual

This option opens the Example Manual that is provided as a digital version in pdf format.



CHAPTER 1 Example Problems Based on Two-Dimensional Analyses 1-3

Example Name	Name of Input File	Description
Example 1 - 2D	2D Example 1 ..... .gp12d	Application with curves of load-displacement and soil resistance (p-y) specified by the user.
Example 2 - 2D	2D Example 2 ..... .gp12d	Pile group consists of vertical and battered piles. Axial load vs. displacement curve specified by the user. The program generates the soil-resistance (p-y) curve internally.
Example 3 - 2D	2D Example 3 ..... .gp12d	Two different sizes of piles are used in the same pile group. The program generates the soil response curves internally.
Example 4 - 2D	2D Example 4 ..... .gp12d	Application for various boundary conditions between the pile cap and the pile heads.
Example 5 - 2D	2D Example 5 ..... .gp12d	Application for pile-supported retaining wall.
Example 6 - 2D	2D Example 6 ..... .gp12d	Piles arranged in a circular pattern for offshore platforms.
Example 7 - 2D	2D Example 7 ..... .gp12d	Application of a 3 x 3 pile group with the pile cap below the ground surface. The program generates curves for computation of foundation stiffness.
		Includes the passive soil resistance on the

Figure 3.315 Sample of Example Manual

3.9.3 Help- Documentation Technical Manual

This option opens the Technical Manual that is provided as a digital version in pdf format.

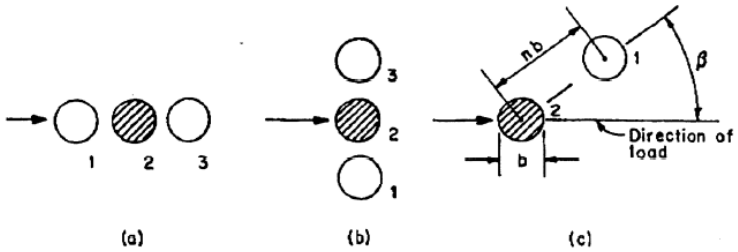


Figure 1.1 Influence of pile spacing on pile-soil-pile interaction

Figure 3.316 Sample of Technical Manual

3.9.4 Help- About

This option provides a screen describing the program version, date, license user and methods for accessing technical support.

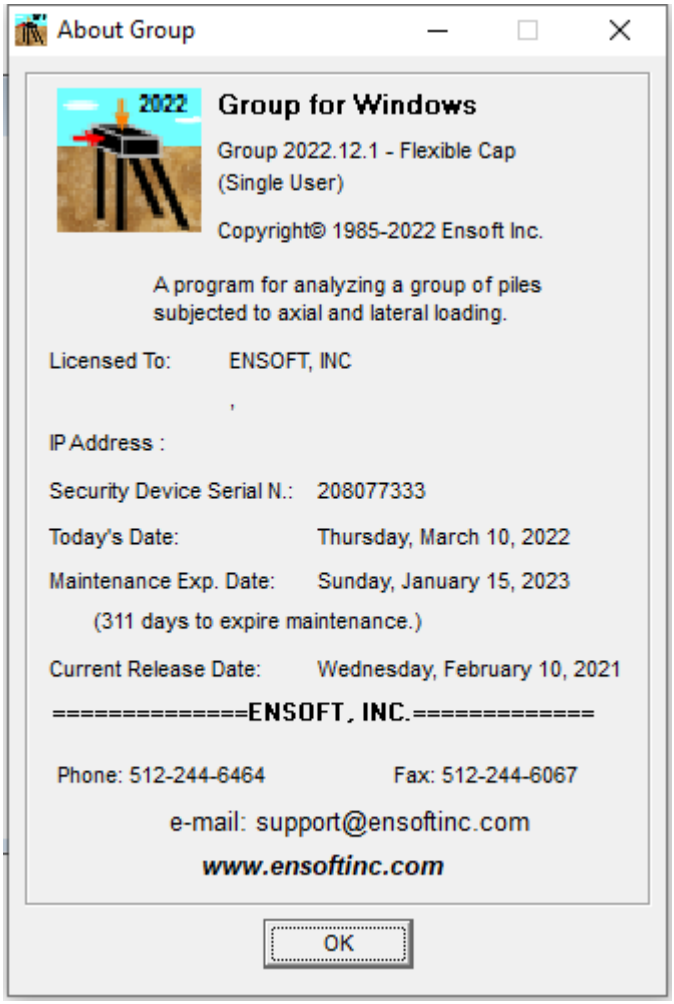


Figure 3.317 About Window

3.9.5 Help- Check for Updates

This opens the user's default internet browser to an ENSOFT webpage with specific update information for GROUP 2022. If a new update is available the page will indicate that. Follow instructions to download and install the update.

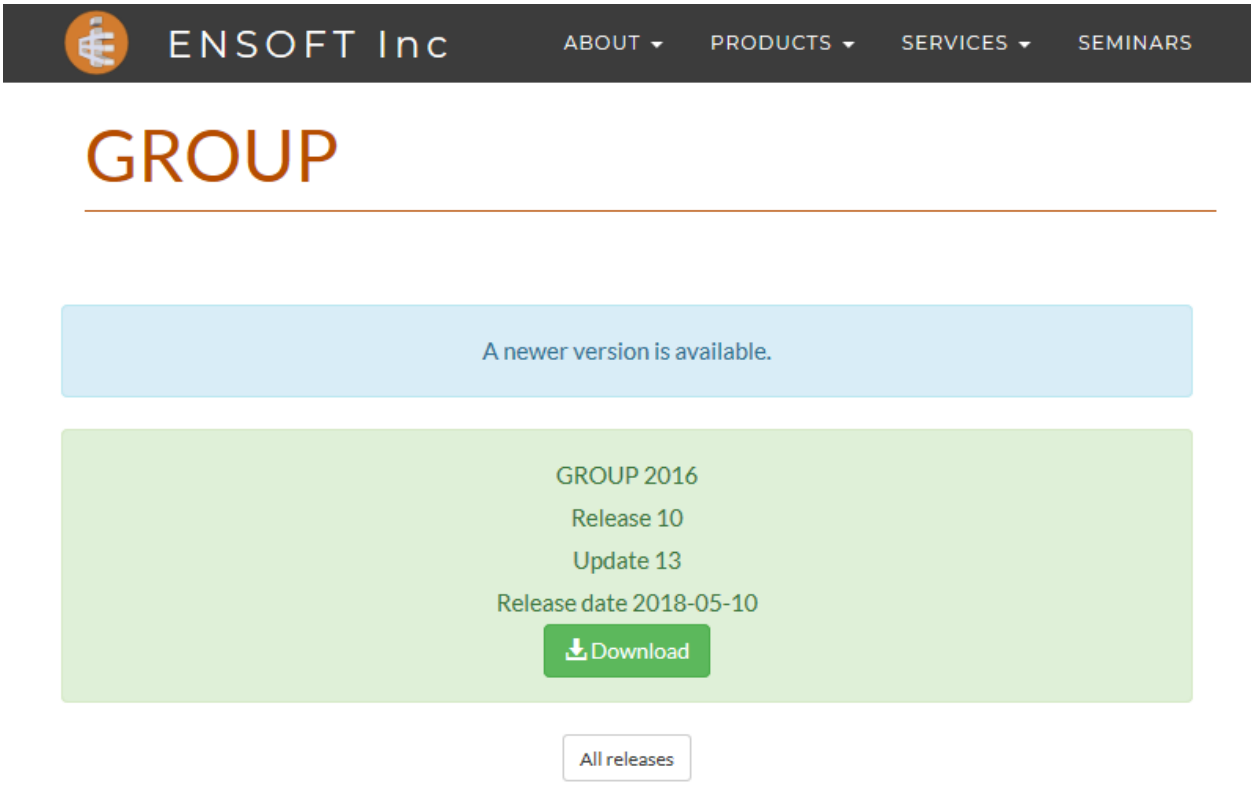


Figure 3.318 Sample of Check for Updates

3.9.6 Help- Ensoft Key Inquirer

This option is available for a single user license. It opens the utiltity Ensoft Key Inquirer that provide information about the ENSOFT licenses available in the USB key.

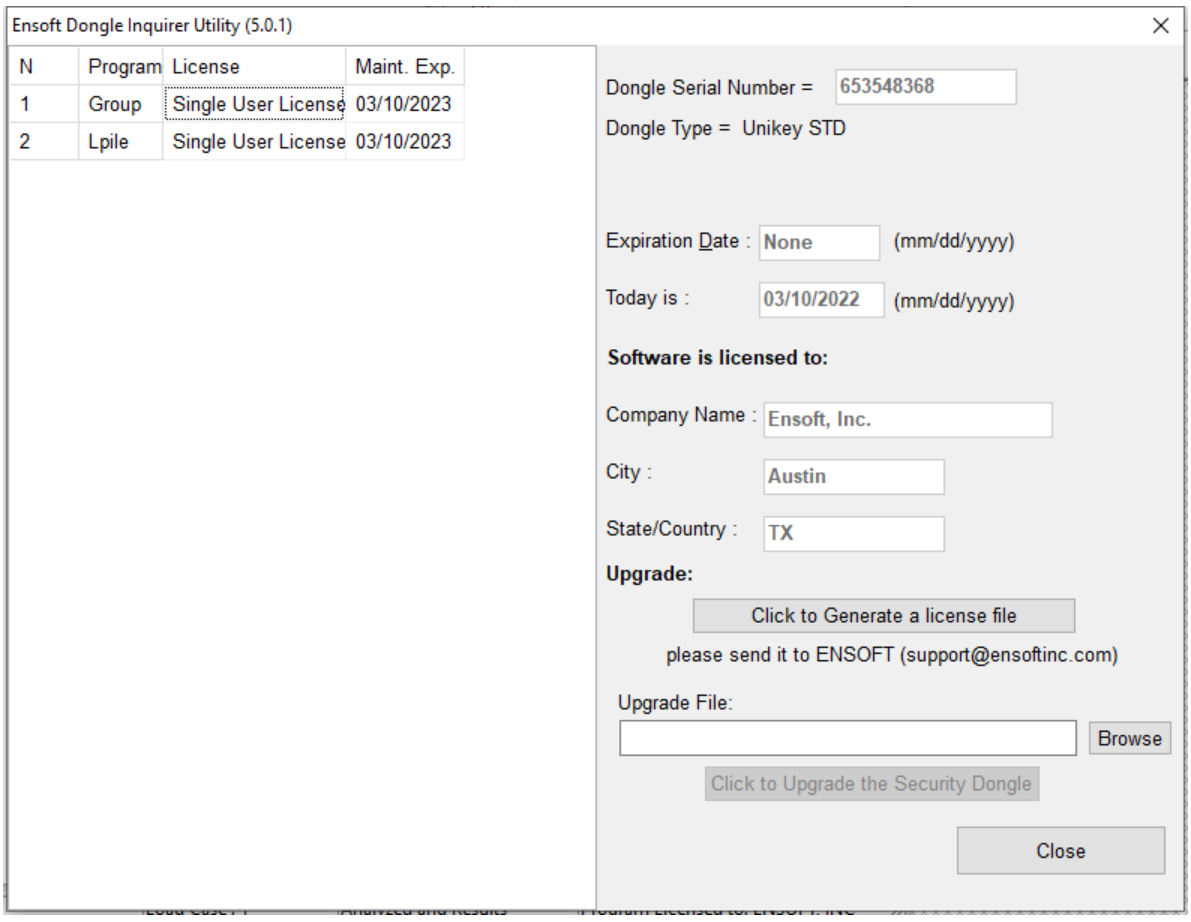


Figure 3.319 Sample of Ensoft Key Inquirer

3.9.7 Help- Check Network Dongle

This option is available for a network license, client user. It opens the utiltity Check Network Dongle that provide information about the ENSOFT network license. This option is useful when the program is not able to get a license from a serve computer. This utility will provide a more detailed information why the program is not able to get a license from the server.

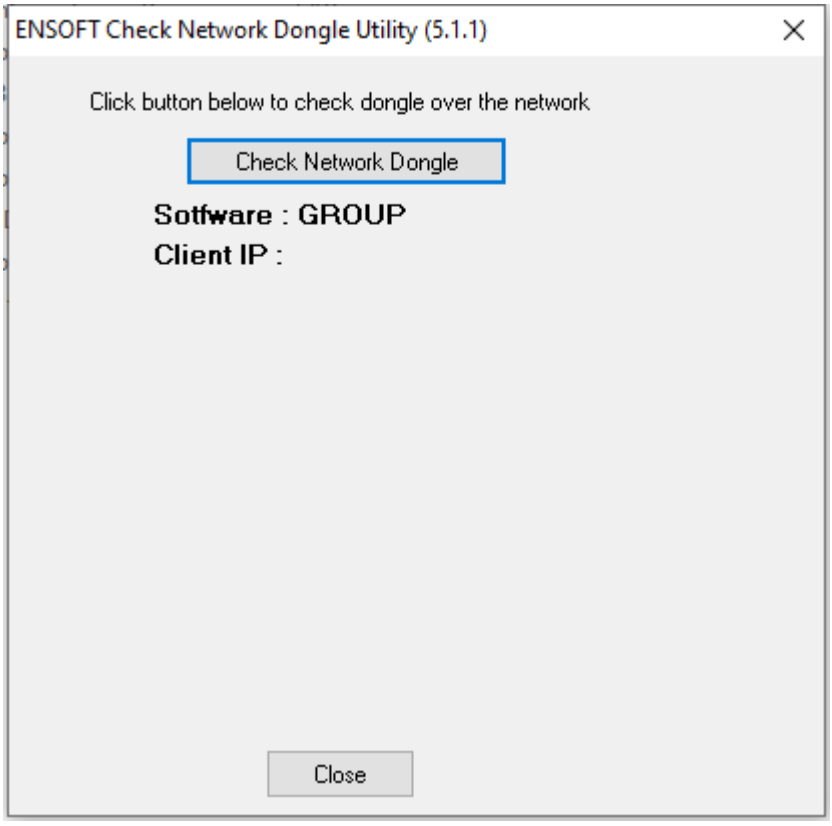


Figure 3.320 Sample of Check Network Dongle

CHAPTER 4. References for Program Execution and Output Reviews

4.1 Introduction

The current Chapter presents options related to execution of the program, and includes methods of addressing run-time errors; suggestions of review of files for input, output, and processor notes; and means for observing results in a graphical form. The commands covered in this chapter are contained under the Computation, Graph, Plot and Show 3D View menu options

4.2 Computation Menu

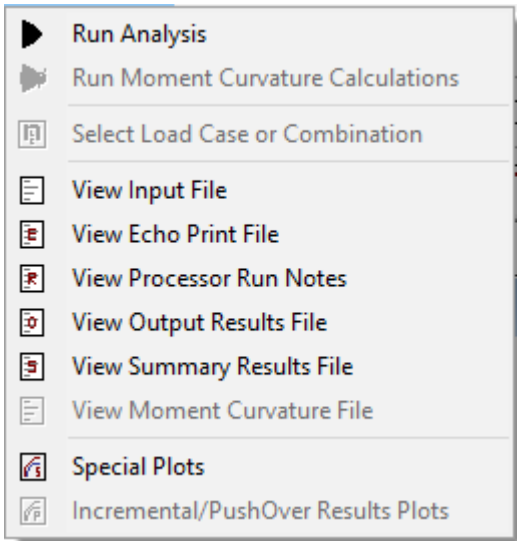


Figure 4.1 Options contained in the Computation Menu

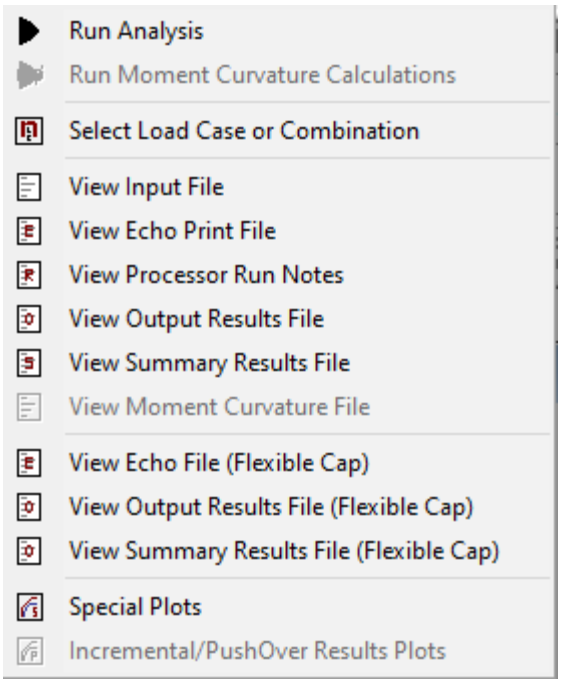


Figure 4.2 Options contained in the Computation Menu – Flexible Cap Version

4.2.1 Computation- Run Analysis

This option runs the analytical computations. Only use this after all data has been entered and saved. The program will do a basic check of the data input before performing the computations; if a data is missing an error message with is description will show up.

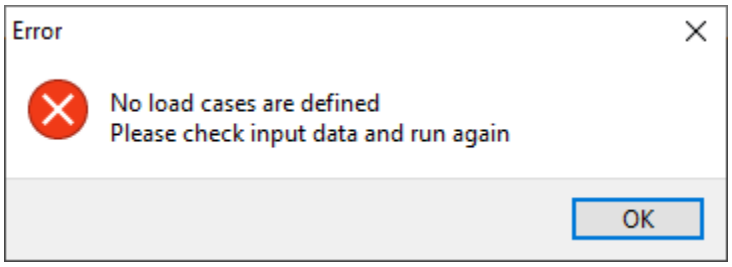


Figure 4.3 Sample message if an input data is missing

During the computation, a window with a progress bar will appear

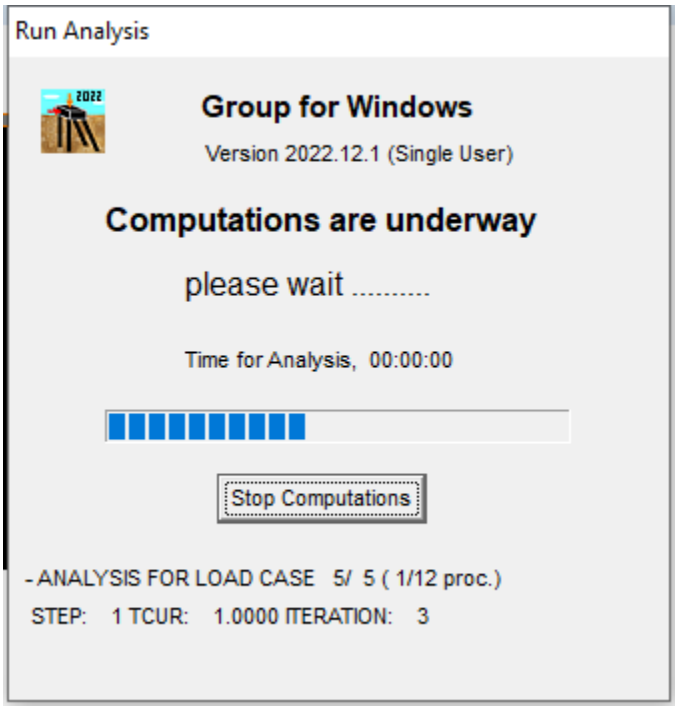


Figure 4.4 Sample message while running the analysis

User may stop the current analysis by clicking on the **Stop Computations** button. After doing so, the program will show a message to confirm it.

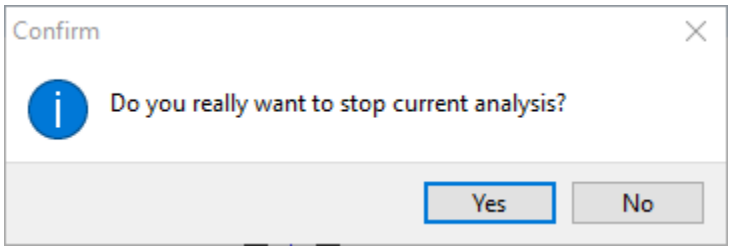


Figure 4.5 Sample message to stop the analysis

By clicking on the **Yes** button, the current analysis will stop, so results may not be available. Clicking on the **No** button will continue the analysis.

If the computation is successful, the program will notify the user.

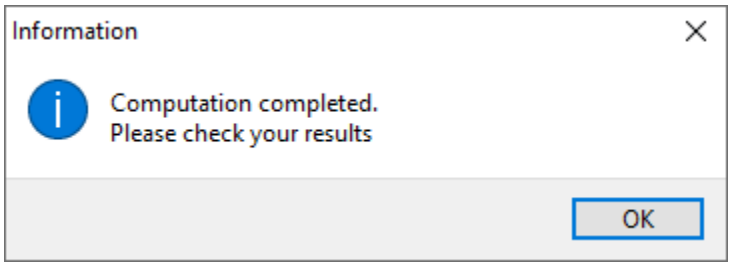


Figure 4.6 Sample message when analysis has finished

The program will also notify the user if the computation is unsuccessful.

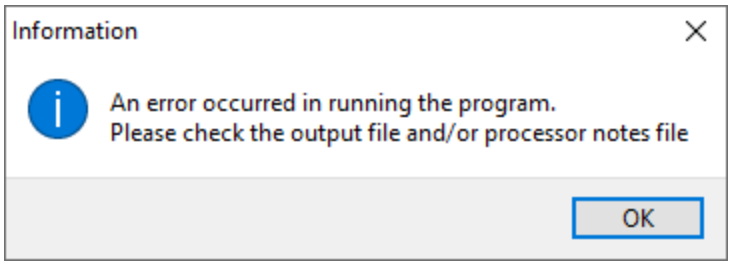


Figure 4.7 Sample message when analysis is unsuccessful

The following text files (can be opened with any text editor) are generated:

File Extension	File Description
.gp12d	Input Data file
.gp12e	Echo Print Data file
.gp12o	Output Results file
.gp12p	Plot Results file
.gp12r	Processor Notes file
.gp12t	Summary Results file

Table 4.1 Files created in GROUP 2022 runs

In addition, the following text files (can be opened with any text editor) may be generated with the Flexible Cap Version:

File Extension	File Description
.sl12e	Echo Print Data file (Flexible Cap)
.sl12o	Output Results file (Flexible Cap)
.sl12t	Summary Results file (Flexible Cap)

Table 4.2 Files created in GROUP 2022 runs – Flexible Cap Version

These files will be created in the same folder as the input file.

It is recommended that users check for the consistency of the results after a successful run.

4.2.2 Computation- Run Moment Curvature Calculations

This option runs the moment curvature calculations. This option is available only if there is at least one nonlinear section. Only use this after all section data has been entered and saved.

After computations are completed, the program will notify the user.

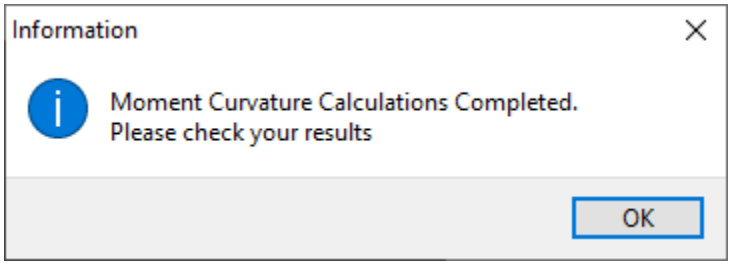


Figure 4.8 Sample message when analysis has finished

The following text file (that can be opened with any text editor) is produced:

File Extension	File Description
.gp12k	Moment Curvature File

Table 4.3 File created using Moment Curvature Calculations

The file will be created in the same folder as the input file.

It is recommended that users check for the consistency of the results after a successful run.

4.2.3 Computation- Select Load Case or Combination

This option opens the **Select Load Case or Combination** window.

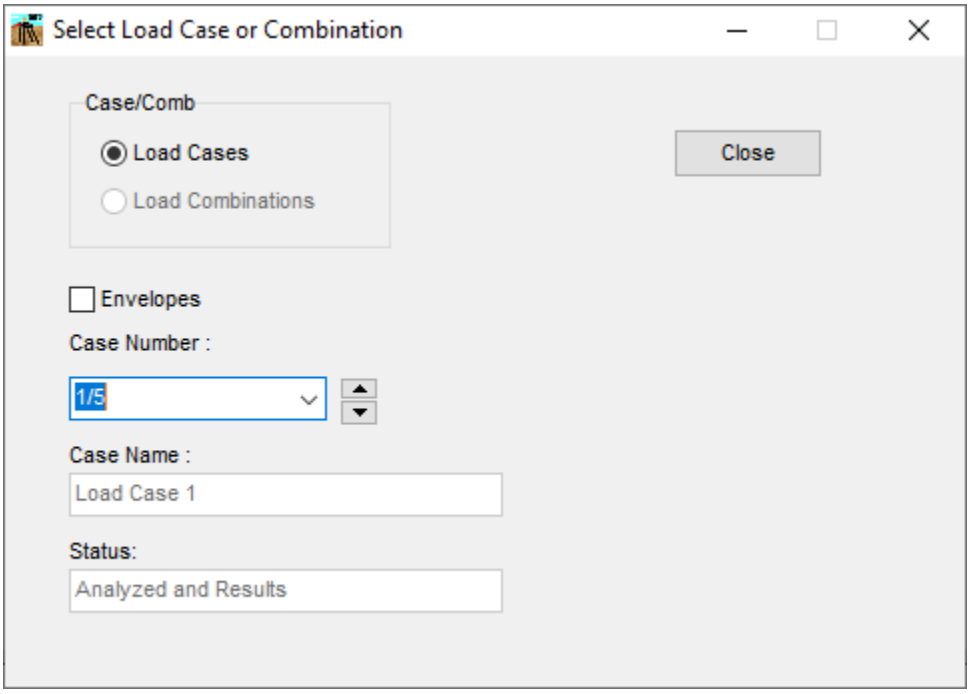


Figure 4.9 Select Load Case or Combination window

In this window, the user can select the load case or load combination that will be active for displaying of the graphical results. This window also shows the status of the different load cases or combinations defined in the model. When a load case or load combination is selected, the current load case will be displayed in the second box of the status line.

4.2.4 Computation- View Input File

This submenu is used to view the input-data file in text mode. This command becomes active after new data files have been saved to disk or when opening existing data files. The command is helpful for experienced users who may just want to change one or two parameters quickly using the text editor, or for those users wishing to observe the prepared-input data in text mode.

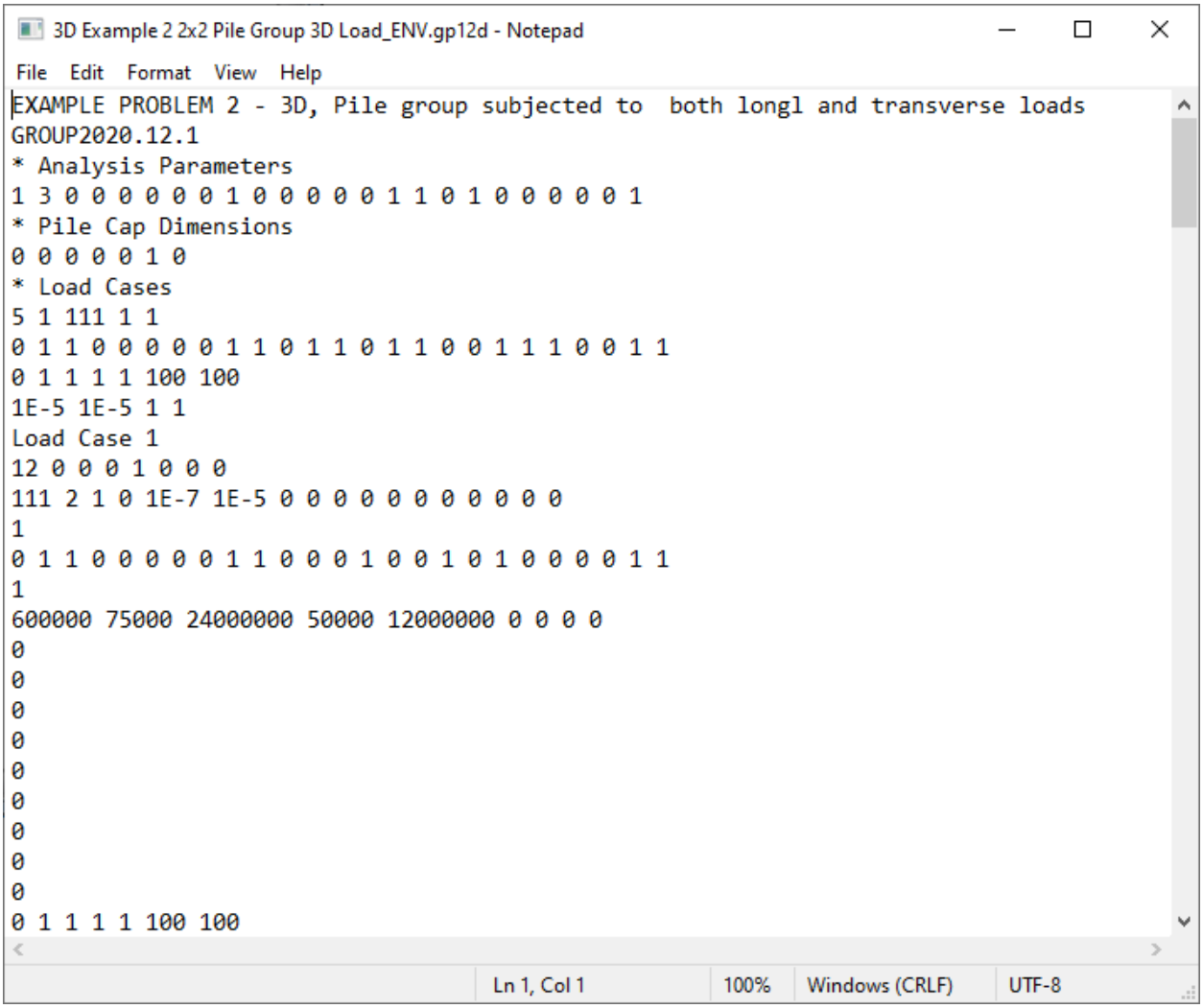


Figure 4.10 Sample of Input Text File

4.2.5 Computation- View Echo Print File

This submenu is used to view the echo print text file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the output files consists of:

- echo-print of input data,
- data for curves of axial load versus displacement
- data for curves of torque vs rotation
- data for internally generated *p-y* curves,
- data for internally generated moment curvature curves,

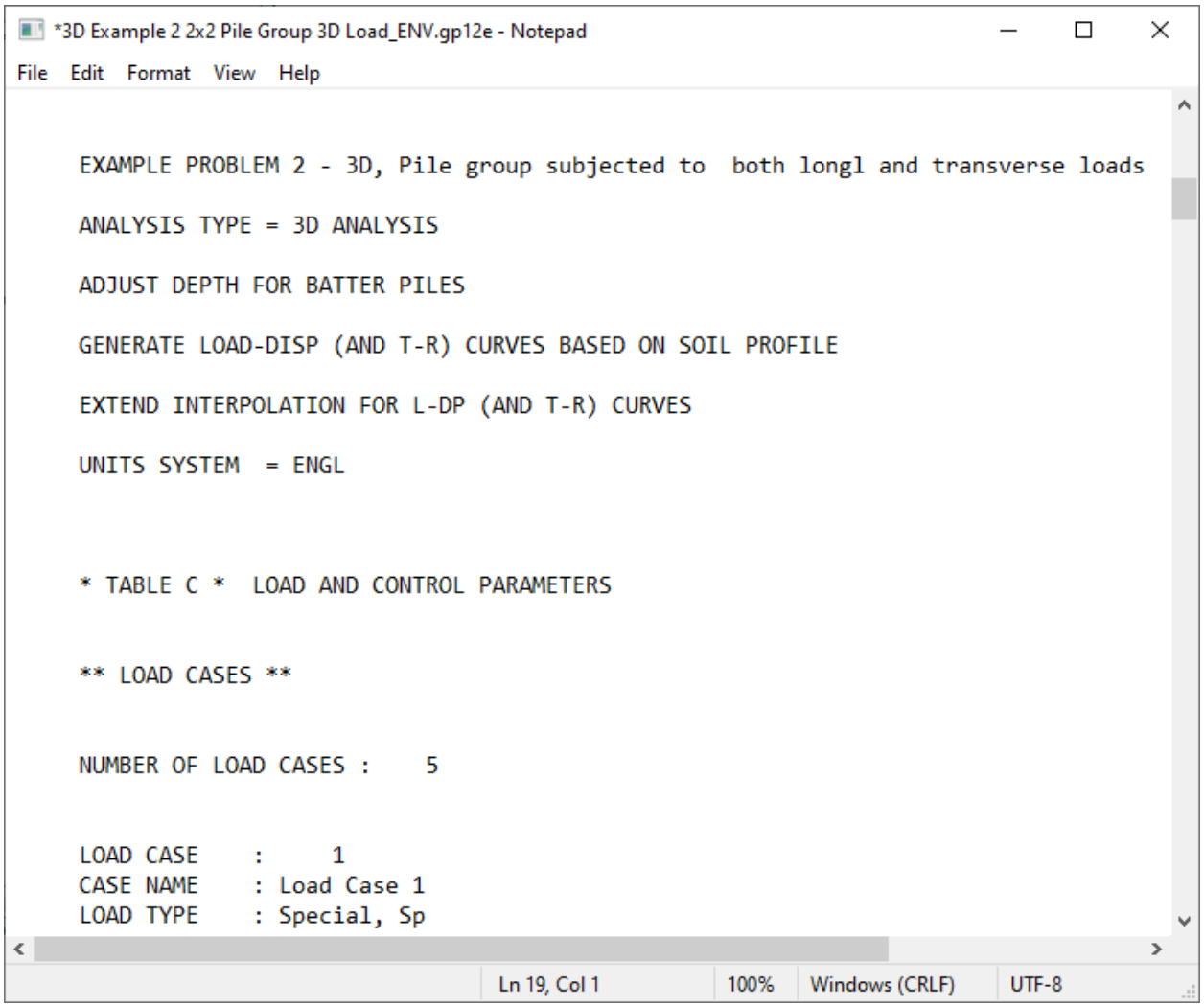


Figure 4.11 Sample of Echo Print File

It is recommended that users check the echo-print of their input data to verify the values that the program is using for the desired analytical case.

4.2.6 Computation- View Processor Run Notes

This submenu is used to view the text file that is automatically produced during each analytical run. The file only includes notes produced during the processing of the input data. The command becomes active after new-data files have been saved to disk and executed, or when opening previously-executed data files. Reviewing the notes is essential for debugging unsuccessful runs.

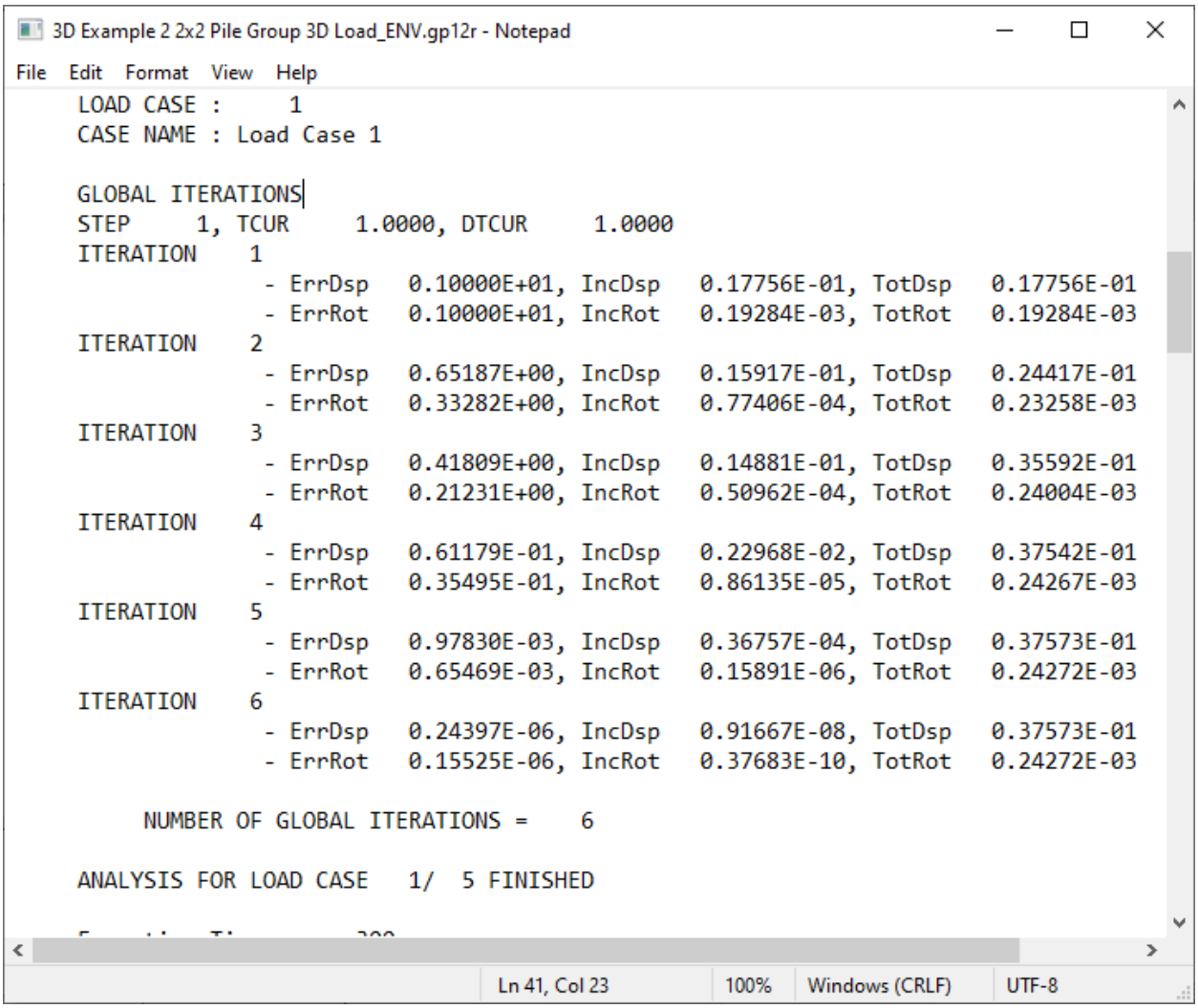


Figure 4.12 Sample of Processor Run Notes Text File

The actual number of lines titled READING depends on the input. If the processor-run notes stop at a certain line without listing the line COMPUTATION IS PROCEEDING..., the user should check the specific input corresponding to the last line read and the line immediately following (that was not read).

After the line COMPUTATION IS PROCEEDING..., the program prints for each load case or load combination the error norm for global iterations (pile cap convergence) and iterations for lateral analysis of each pile if printing is requested for all the piles.

Common types of analytical-run failures are those providing error messages like Pile Failure in Uplift or Pile Failure in Bearing. These two conditions will occur when the axial load on a pile is greater than the ultimate axial capacity of the corresponding pile. To correct these types of problems, the user should check the pile size and soil parameters and make sure that the input data is reasonable. If the pile and soil properties are proper, the user should reduce the axial load on the pile cap for a new analysis. Also, these errors may be triggered if the analysis diverges under the current loading. There may be different causes for divergence, so a complete review of the input data is recommended.

4.2.7 Computation- View Output Results File

This submenu is used to view the output-text file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the output files consists of:

- results for deformation of the pile cap,
- results for loads and deformation at the pile head for each pile (or a standard pile of each subgroup for 2D analysis), and
- results for deflection, bending moment, shear, soil resistance, and related information, as a function of depth for each pile (or a standard pile of each subgroup for 2D analysis).

It is recommended that users check the output files to verify the consistency of the solution estimated by the program.

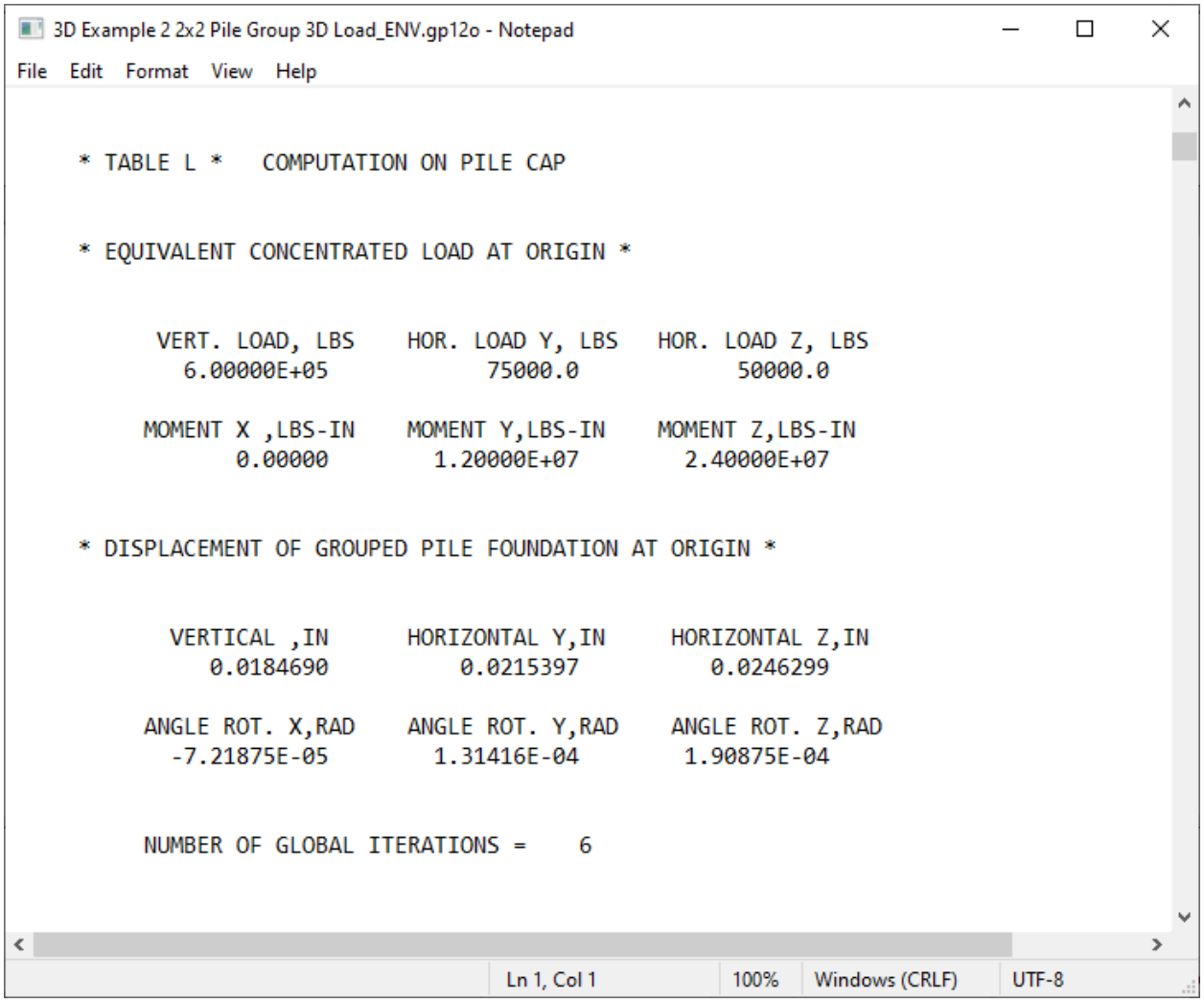
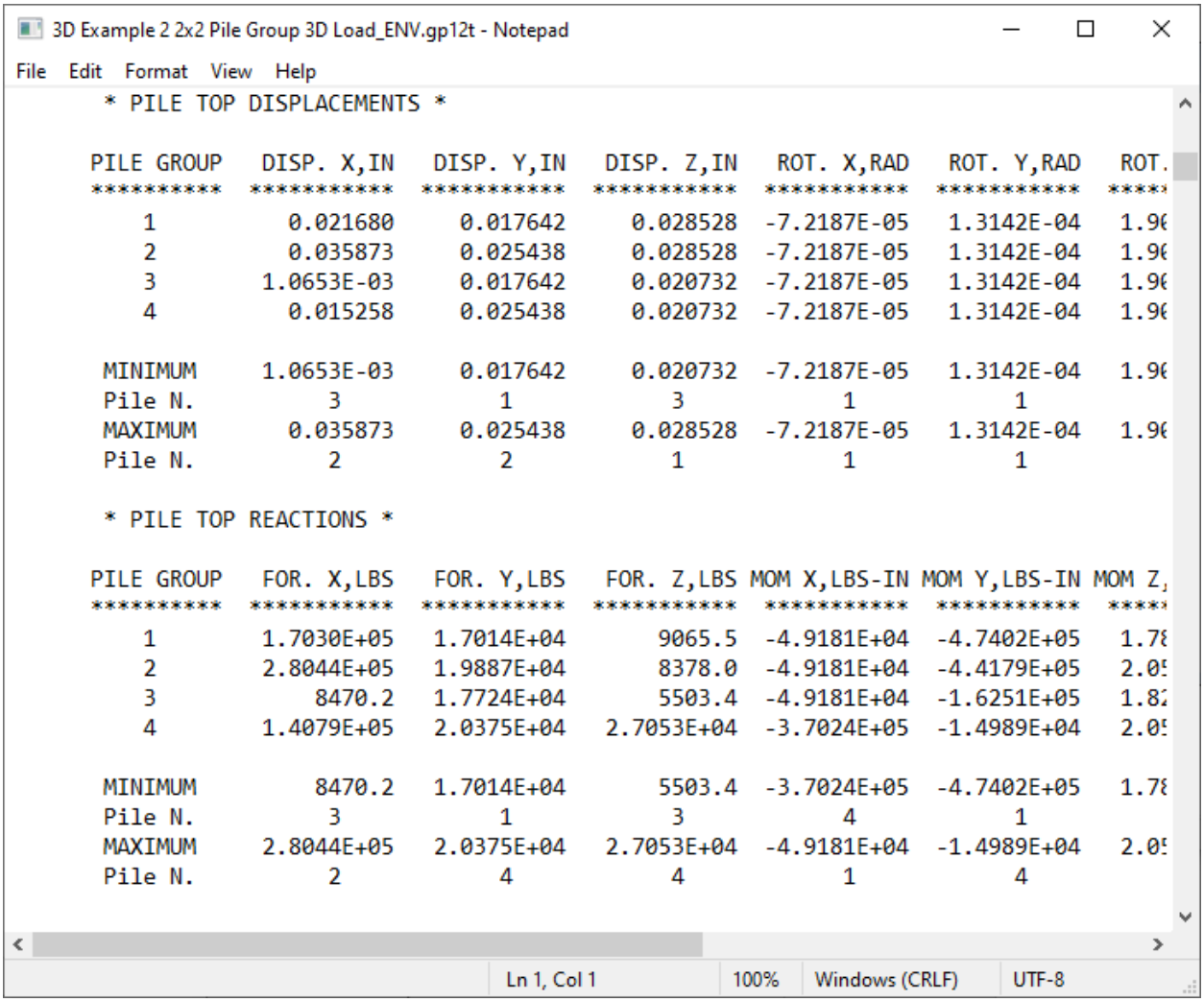


Figure 4.13 Sample of Output Results File

4.2.8 Computation- View Summary Results File

This submenu is used to view the summary file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the summary file consists of:

- results for deformation of the pile cap,
- summary table for results of loads and deformation at the pile head for each pile, including maximum and minimum values and location of those.
- summary table for maximum and minimum results of deflection, bending moment, shear, soil resistance at each pile.


3D Example 2 2x2 Pile Group 3D Load_ENV.gp12t - Notepad

File Edit Format View Help

```
 * PILE TOP DISPLACEMENTS *

PILE GROUP   DISP. X,IN   DISP. Y,IN   DISP. Z,IN   ROT. X,RAD   ROT. Y,RAD   ROT.
**********  ***********  ***********  ***********  ***********  ***********  *****
    1          0.021680     0.017642     0.028528  -7.2187E-05   1.3142E-04   1.96
    2          0.035873     0.025438     0.028528  -7.2187E-05   1.3142E-04   1.96
    3         1.0653E-03    0.017642     0.020732  -7.2187E-05   1.3142E-04   1.96
    4          0.015258     0.025438     0.020732  -7.2187E-05   1.3142E-04   1.96

 MINIMUM      1.0653E-03    0.017642     0.020732  -7.2187E-05   1.3142E-04   1.96
 Pile N.           3            1            3            1            1
 MAXIMUM       0.035873     0.025438     0.028528  -7.2187E-05   1.3142E-04   1.96
 Pile N.           2            2            1            1            1

 * PILE TOP REACTIONS *

PILE GROUP   FOR. X,LBS   FOR. Y,LBS   FOR. Z,LBS MOM X,LBS-IN MOM Y,LBS-IN MOM Z,
**********  ***********  ***********  ***********  ***********  ***********  *****
    1         1.7030E+05   1.7014E+04       9065.5  -4.9181E+04  -4.7402E+05   1.78
    2         2.8044E+05   1.9887E+04       8378.0  -4.9181E+04  -4.4179E+05   2.05
    3             8470.2   1.7724E+04       5503.4  -4.9181E+04  -1.6251E+05   1.82
    4         1.4079E+05   2.0375E+04    2.7053E+04  -3.7024E+05  -1.4989E+04   2.05

 MINIMUM          8470.2   1.7014E+04       5503.4  -3.7024E+05  -4.7402E+05   1.78
 Pile N.           3            1            3            4            1
 MAXIMUM       2.8044E+05   2.0375E+04    2.7053E+04  -4.9181E+04  -1.4989E+04   2.05
 Pile N.           2            4            4            1            4
```

Ln 1, Col 1 100% Windows (CRLF) UTF-8

Figure 4.14 Sample of Summary Results File

4.2.9 Computation- View Moment Curvature File

This submenu is used to view the moment curvature file that is automatically produced after selecting run moment curvature calculations. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the output files consists of:

- Axial thrust forces
- Bending curvature, bending moment, bending stiffness.
-

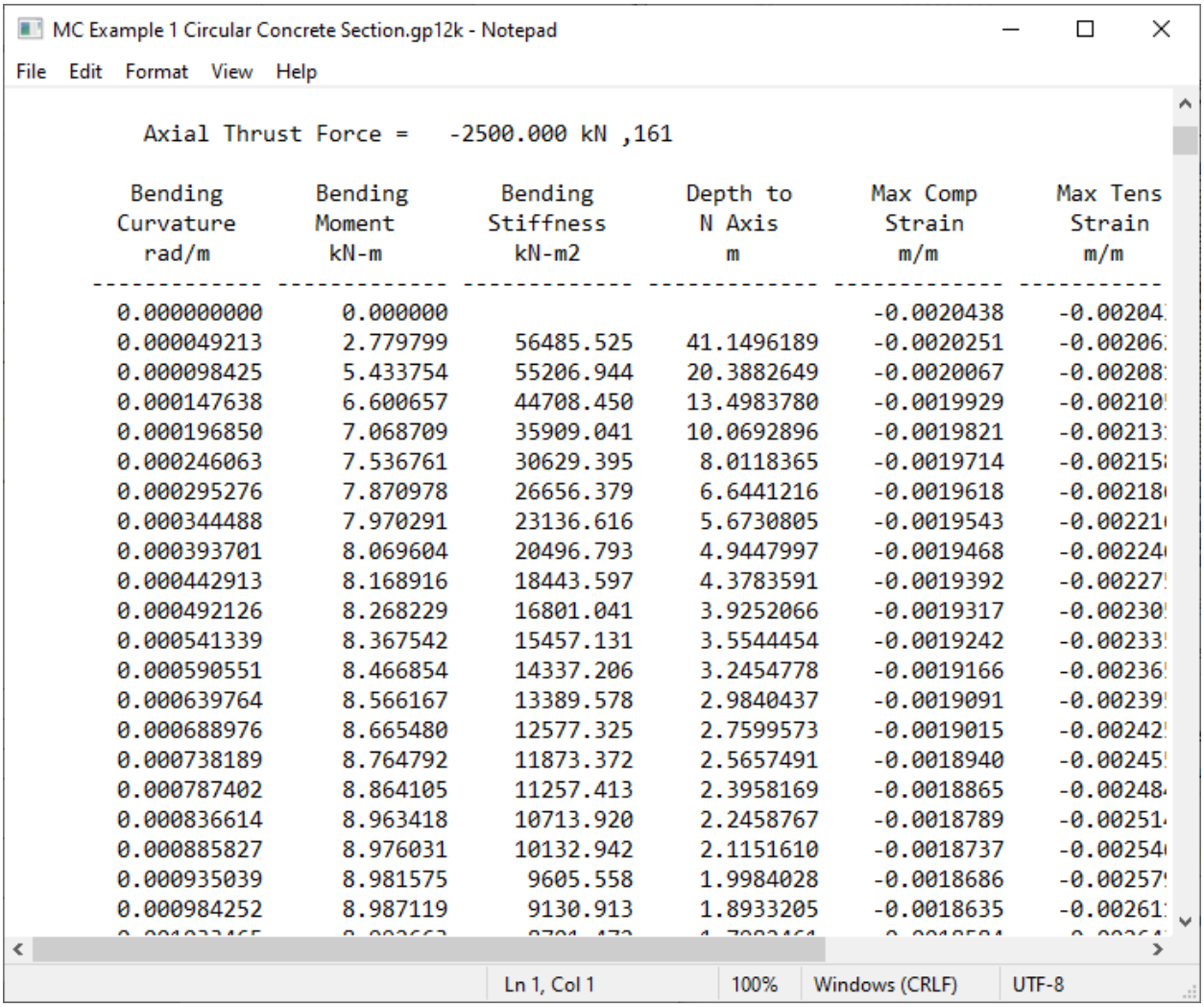


Figure 4.15 Sample of Moment Curvature File

4.2.10 Computation- View Echo File (Flexible Cap) – Flexible Cap Version

This submenu is used to view the echo print file that is automatically produced during each analytical run if the pile cap is modeled as flexible. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the output files consists of:

- echo-print of finite element nodes,
- echo-print of finite element connectivity,
- data for loading on flexible cap

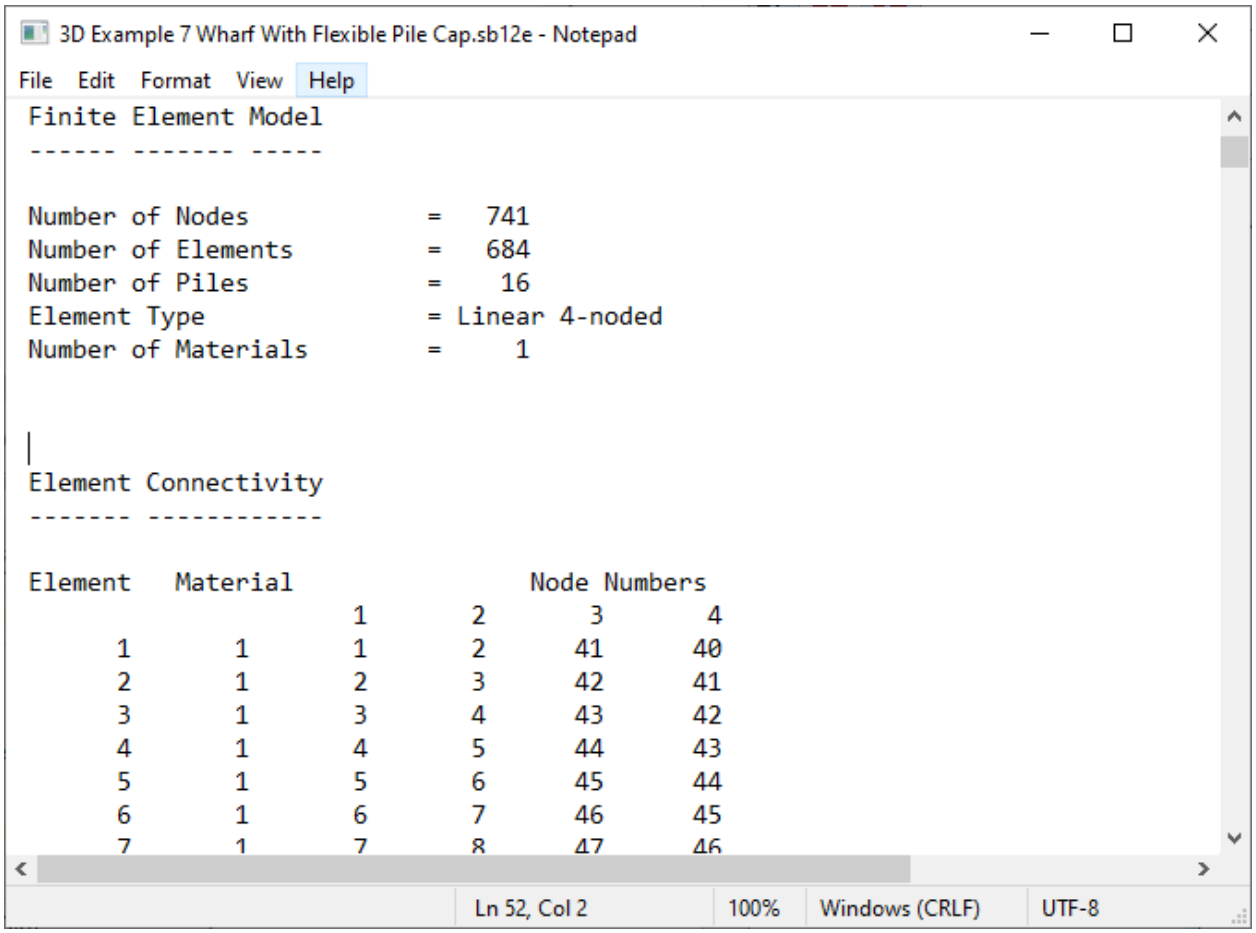


Figure 4.16 Sample of Echo File (Flexible Cap) – Flexible Cap Version

It is recommended that users check the echo-print of their input data to verify the values that the program is using for the desired analytical case.

4.2.11 Computation- View Output Results File (Flexible Cap) – Flexible Cap Version

This submenu is used to view the output-text file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files. Information usually contained in the output files consists of:

- results for displacements on the nodes that belong to the flexible pile cap,
- results for internal forces and stresses on plates that belong to the flexible pile cap
- results for internal forces and deformations of the beams that belong to the flexible pile cap

It is recommended that users check the output files to verify the consistency of the solution estimated by the program.

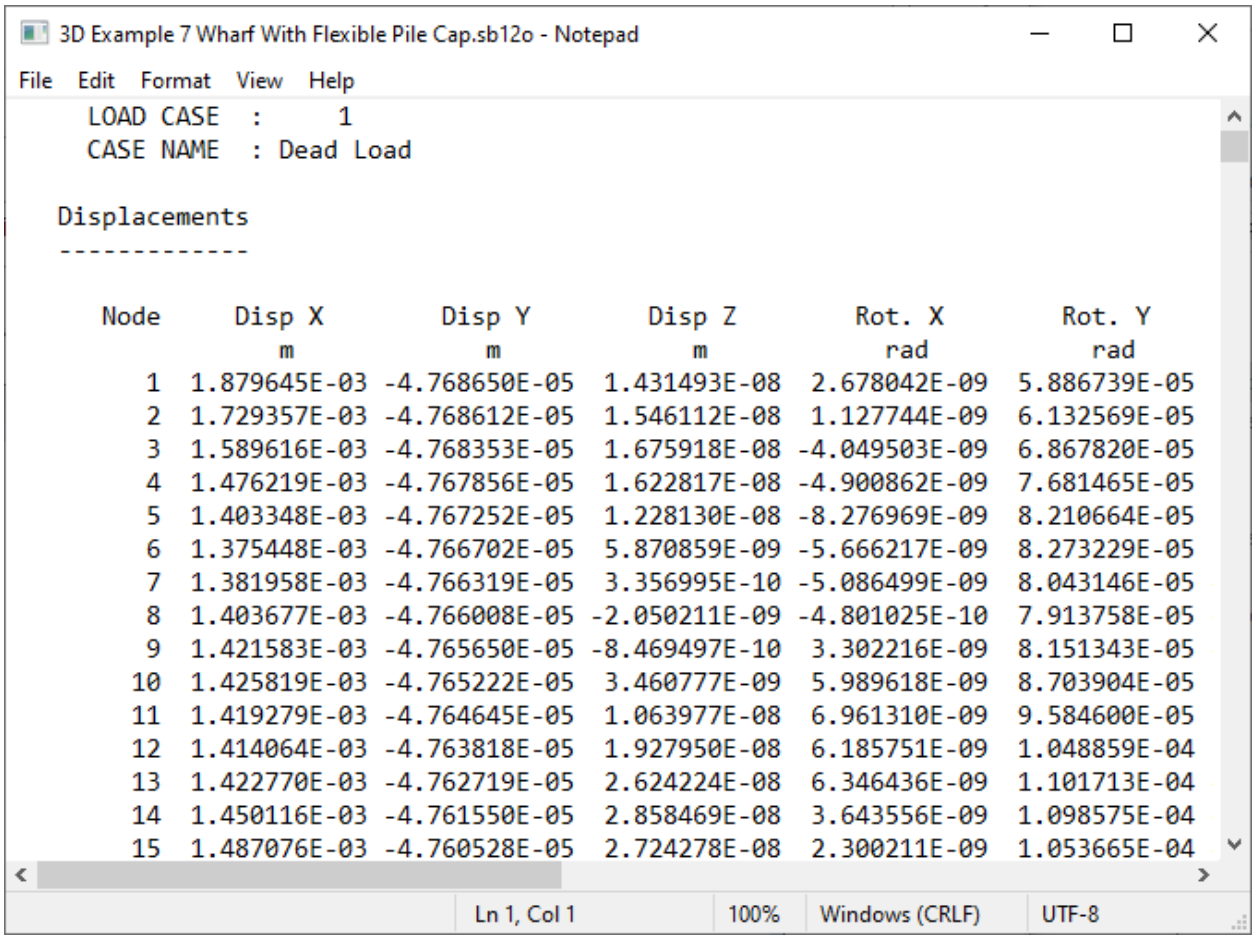


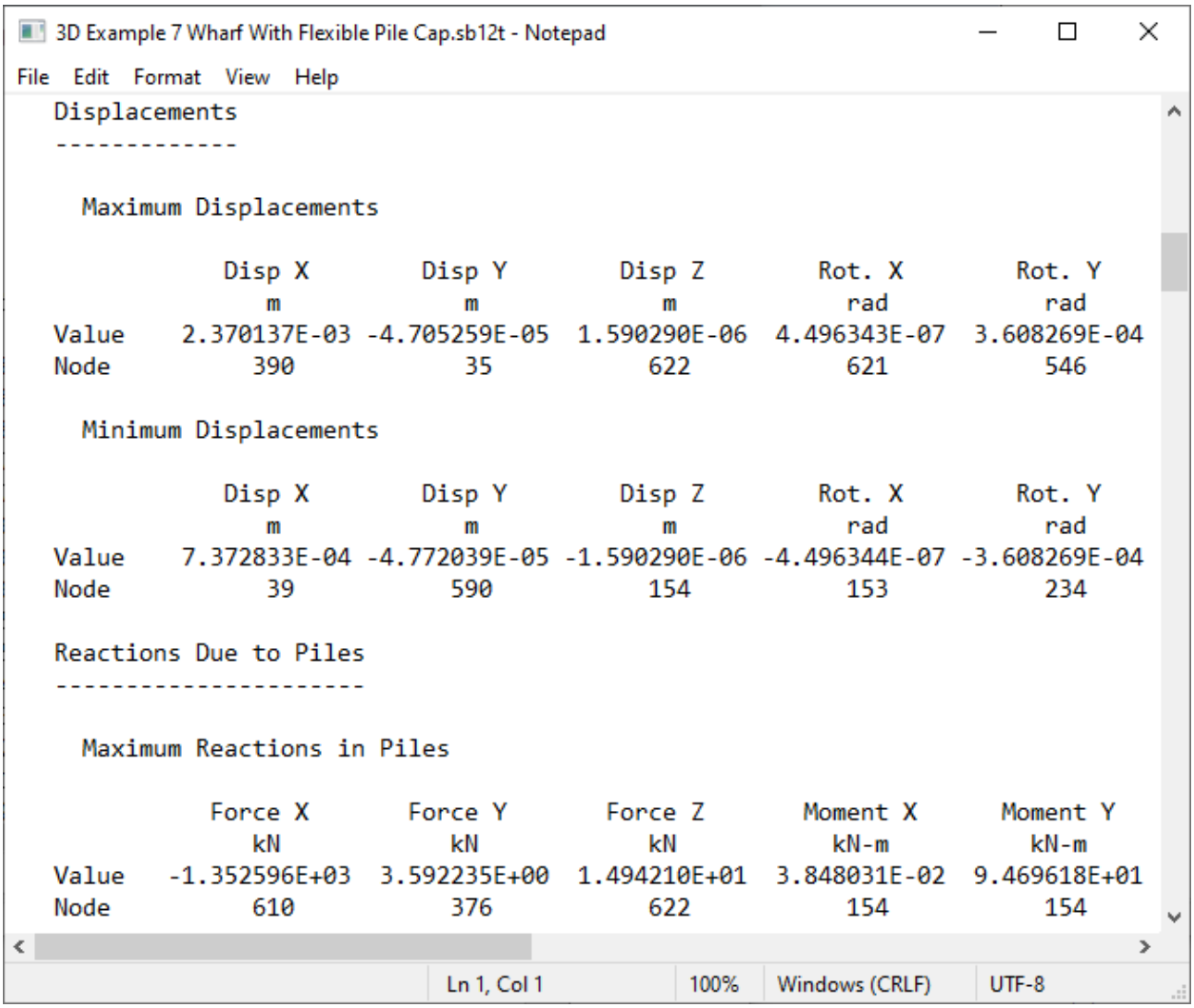
Figure 4.17 Sample of Output Results File (Flexible Cap) – Flexible Cap Version

4.2.12 Computation- View Summary Results File (Flexible Cap) – Flexible Cap Version

This submenu is used to view the summary file that is automatically produced during each analytical run. This command becomes active after new data files have been saved to disk and

executed, or when opening previously-executed data files. Information usually contained in the summary file consists of:

- maximum and minimum values of displacements at nodes, internal forces and stresses on plates and internal forces and deformations on beams on the flexible pile cap.



3D Example 7 Wharf With Flexible Pile Cap.sb12t - Notepad

File Edit Format View Help

Displacements

Maximum Displacements

	Disp X	Disp Y	Disp Z	Rot. X	Rot. Y
	m	m	m	rad	rad
Value	2.370137E-03	-4.705259E-05	1.590290E-06	4.496343E-07	3.608269E-04
Node	390	35	622	621	546

Minimum Displacements

	Disp X	Disp Y	Disp Z	Rot. X	Rot. Y
	m	m	m	rad	rad
Value	7.372833E-04	-4.772039E-05	-1.590290E-06	-4.496344E-07	-3.608269E-04
Node	39	590	154	153	234

Reactions Due to Piles

Maximum Reactions in Piles

	Force X	Force Y	Force Z	Moment X	Moment Y
	kN	kN	kN	kN-m	kN-m
Value	-1.352596E+03	3.592235E+00	1.494210E+01	3.848031E-02	9.469618E+01
Node	610	376	622	154	154

Ln 1, Col 1 100% Windows (CRLF) UTF-8

Figure 4.18 Sample of Summary Results File

4.2.13 Computation- Special Plots

This submenu will open the Special Plots Window, that allows the user to plot the results after a successful run.

The Special Plots Window allows the user to plot results for both a specific Load Case or Combination and also for various Load Cases or Combinations.

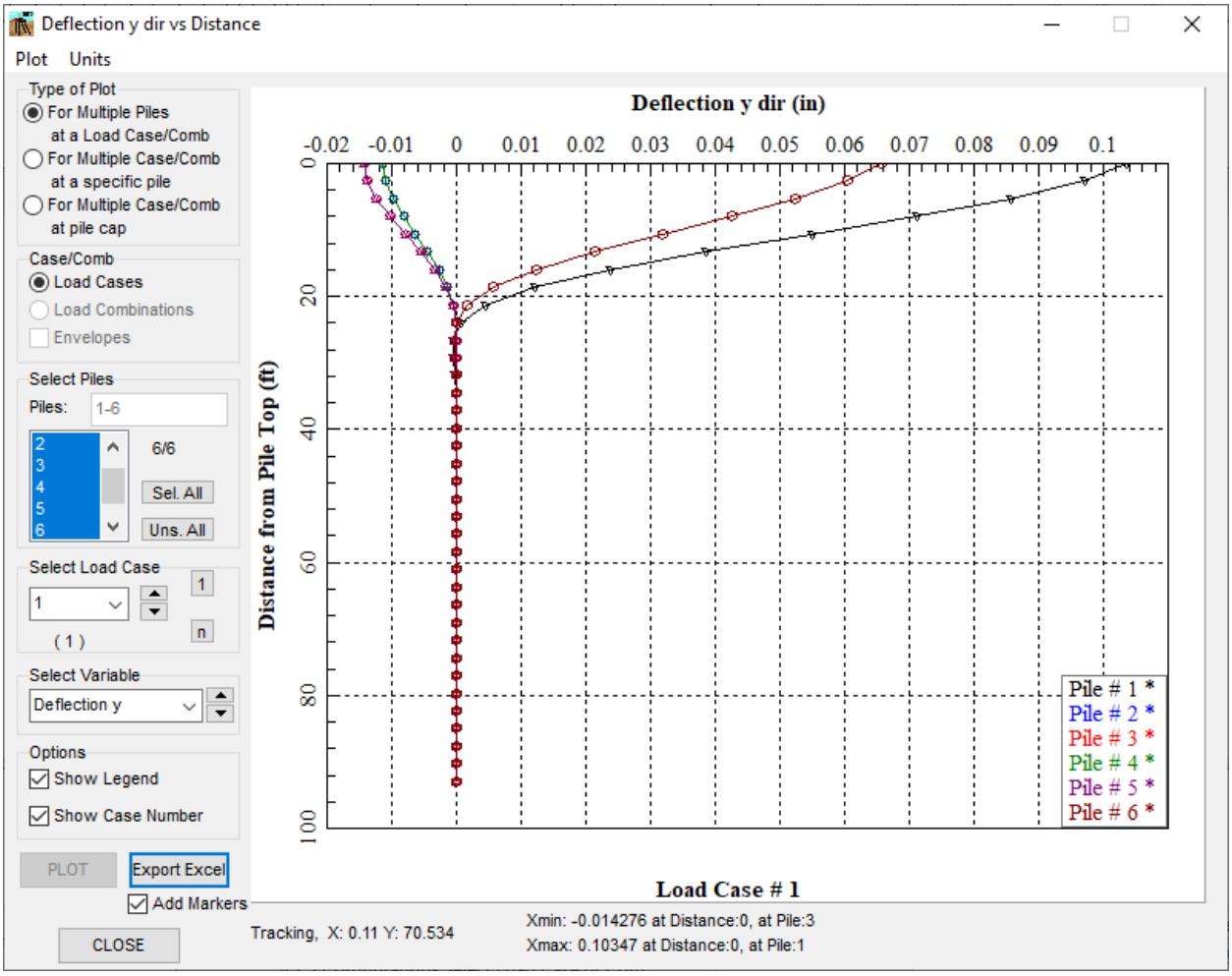


Figure 4.19 Special Plots Window

4.2.14 Computation- Incremental/PushOver Plots

This submenu will open the Incremental/PushOver Plots Window, that allows the user to plot the results after a successful run.

The Incremental/PushOver Plots Window allows the user to plot results available for incremental/pushover analysis.

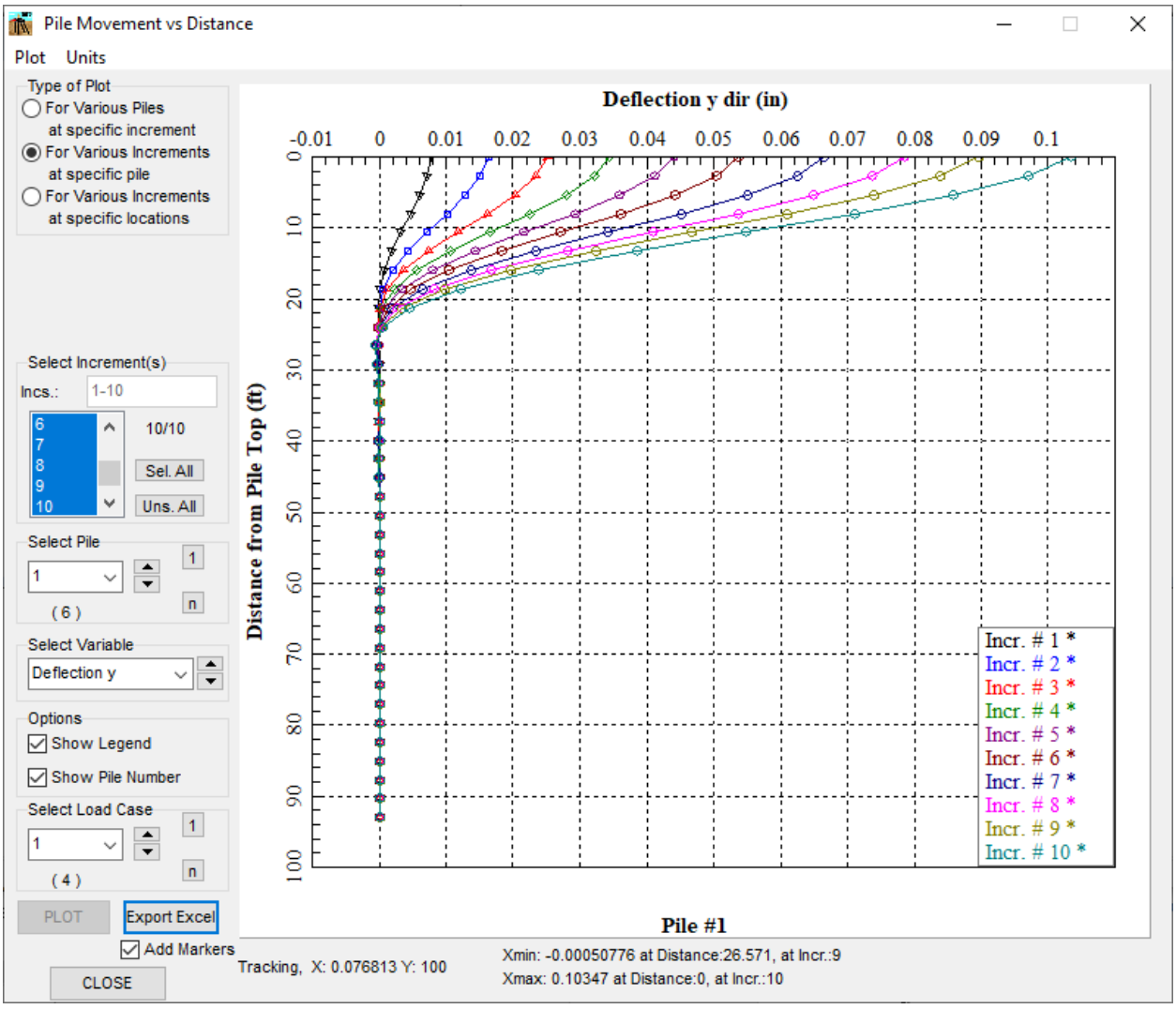


Figure 4.20 Incremental/PushOver Plots Window

4.3 General Comments for Graphics Mode

Graphics mode is accessed when the user selects to observe any one of the output curves. Graphical representation of output curves is produced under the following commands in the Graphs menu:

- Generated p-y Curves
- Generated Axial Load vs. Displacement
- Generated Torque vs. Rotation
- User Defined Axial Load vs. Displacement
- Deflection vs. Distance
- Bending Moment vs. Distance
- Shear vs. Distance
- Soil Reaction vs. Distance
- Def., Mom., Shear vs. Distance
- Bending Stiffness vs. Distance
- Deflection y dir vs. Distance
- Deflection z dir vs. Distance
- Moment z dir vs. Distance
- Moment y dir vs. Distance
- Shear y dir vs. Distance
- Shear z dir vs. Distance
- Soil Reaction y dir vs. Distance
- Soil Reaction z dir vs. Distance
- Bending Stiffness z dir vs. Distance
- Bending Stiffness y dir vs. Distance
- Def. y, Mom. z, Shear y vs. Distance
- Def. z, Mom. y, Shear z vs. Distance
- Deflection x dir vs. Distance
- Axial Force vs. Distance
- Resultant Deflection vs. Distance
- Resultant Moment vs. Distance
- Resultant Shear vs. Distance
- Resultant Soil Reaction vs. Distance
- Res. Def., Mom., Shear vs. Distance
- Pile Cap Vertical Load vs. Deflection
- Pile Cap Load vs. Lateral Deflection
- Pile Cap Load vs. Rotation
- Pile Cap Moment vs. Deflection
- Pile Cap Moment vs. Rotation

- Pile Cap Vertical Load vs. Deflection X
- Pile Cap Lateral Load vs. Deflection Y
- Pile Cap Lateral Load vs. Deflection Z
- Pile Cap Moment vs. Rotation X
- Pile Cap Moment vs. Rotation Y
- Pile Cap Moment vs. Rotation Z
- Pile Cap K11 vs. Deflection X
- Pile Cap K22 vs. Deflection Y
- Pile Cap K33 vs. Rotation Z
- Pile Cap K33 vs. Deflection Z
- Pile Cap K44 vs. Rotation X
- Pile Cap K55 vs. Rotation Y
- Pile Cap K66 vs. Rotation Z

Additionally, the program produces the nonlinear EI plots, like the moment curvature diagram; however, the description below does not apply to those curves.

The activation of any of the above listed curves will activate the graphics mode of GROUP 2022. Several changes occur during use of the graphics mode: new mouse commands are enabled and a new top-menu option becomes available.

The following table describes mouse commands that become enabled automatically during all graphical observation of the output curves.

Event	Description
Mouse Left Click	Zooms in the graphics at the clicked position (up to five zoom levels)
Mouse Right Click	Zoom out the graphics
Mouse Double Click on Legend	Turns the selected curve on/off at every click

Table 4.4 Mouse commands in the graphics mode

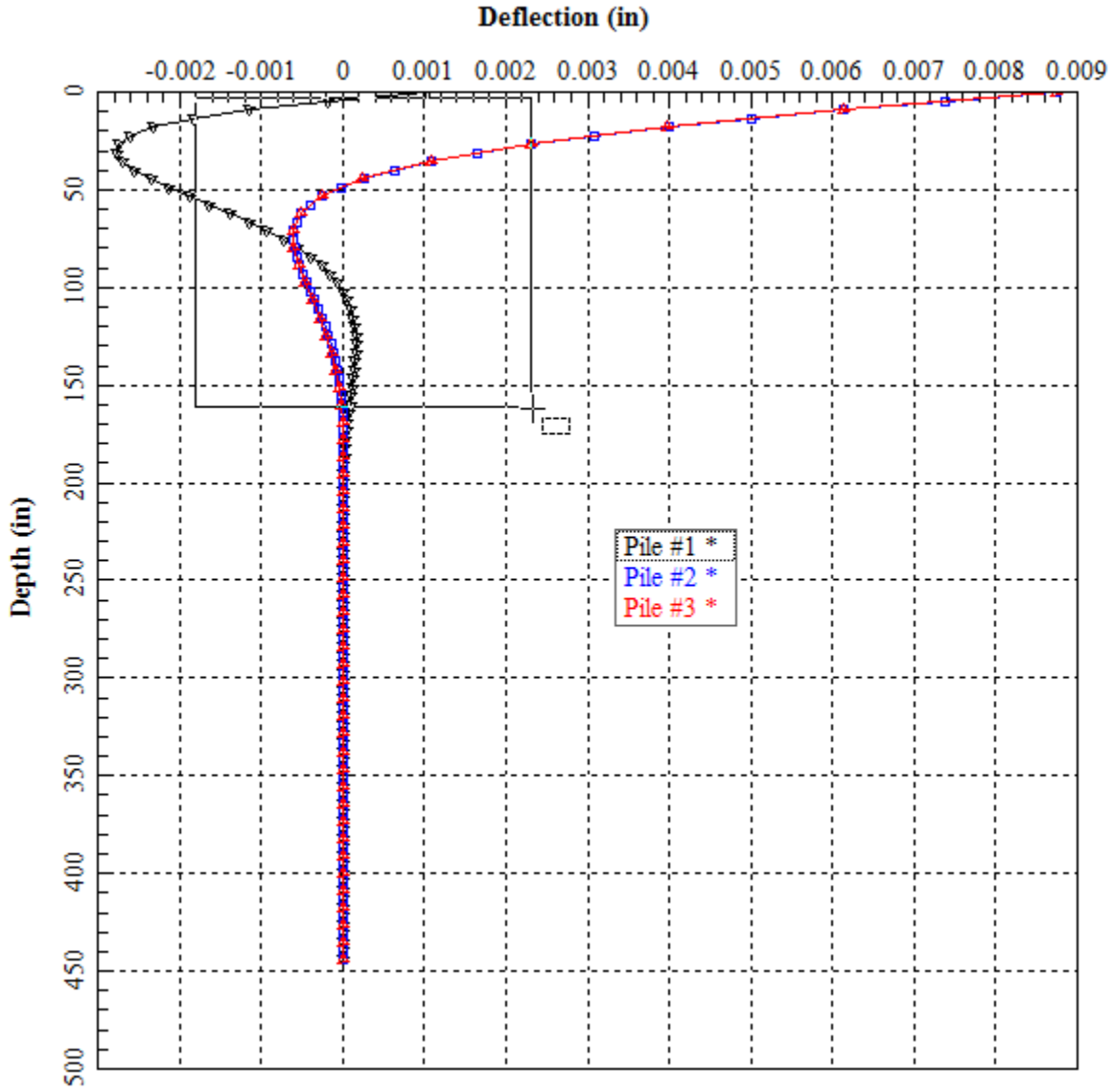


Figure 4.21 Effects on graph display using mouse commands. Defining box for zooming in (Left click)

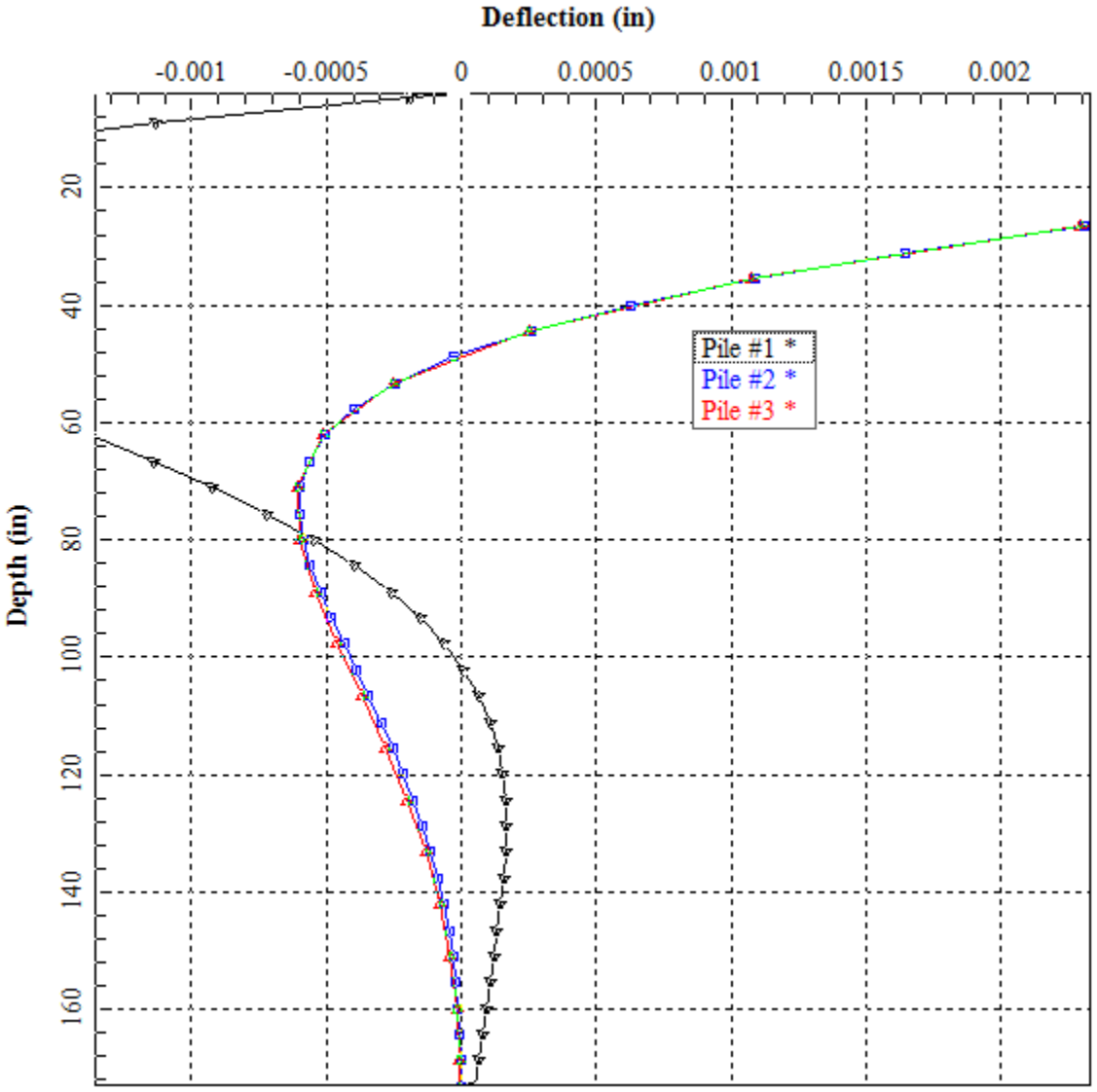


Figure 4.22 Effects on graph display using mouse commands. After zooming in

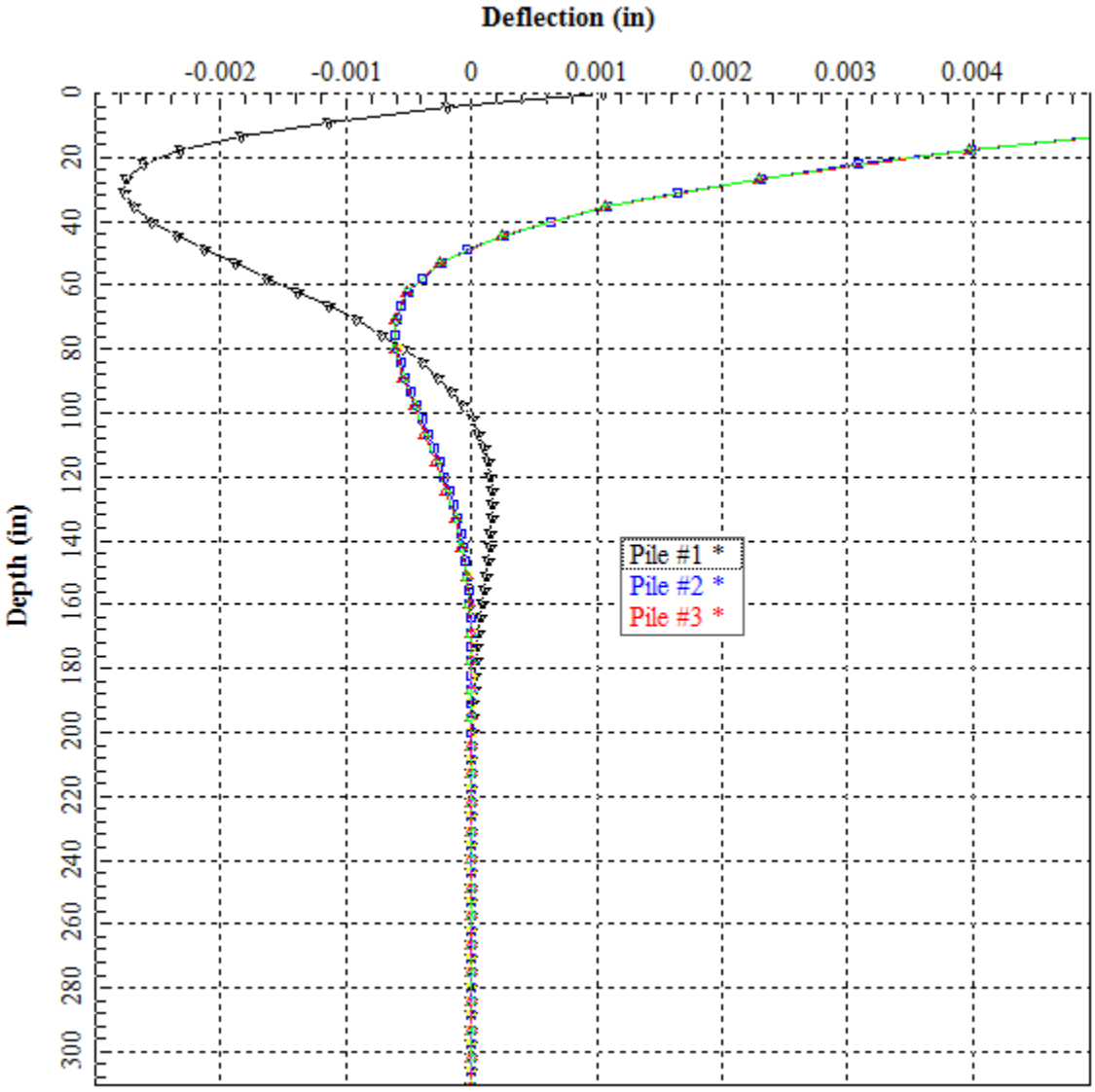


Figure 4.23 Effects on graph display using mouse commands. Zooming out (Right Click)

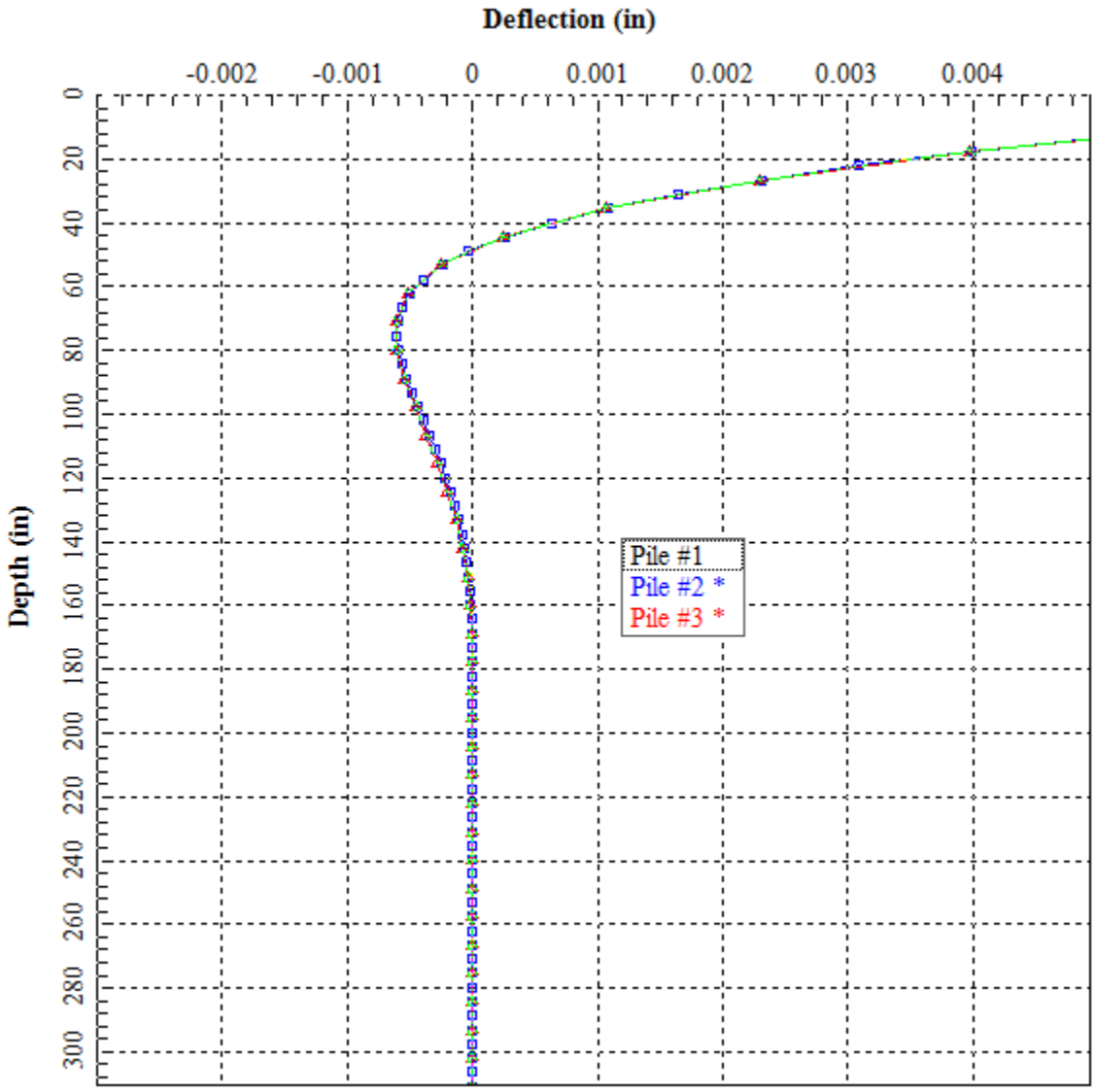


Figure 4.24 Effects on graph display using mouse commands. Hiding a curve (Double Click on Legend)

4.4 Plot Menu

The **Plot** menu command appears in the top-menu bar while in graphics mode. The changes to the graphs are also valid to the current graph and while the graph is active. Closing the graph will discard automatically all the changes. Options contained in this menu are explained below:

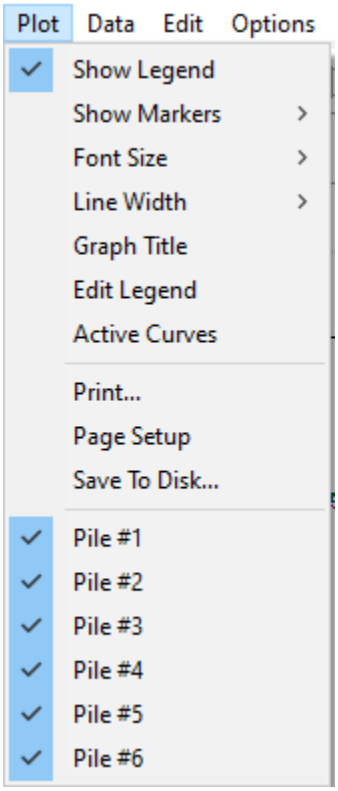


Figure 4.25 Options contained in the Plot Menu

4.4.1 Plot - Show Legend

Activates or deactivates the legend in the graphics window. The legend box can be moved using click-and-drag.

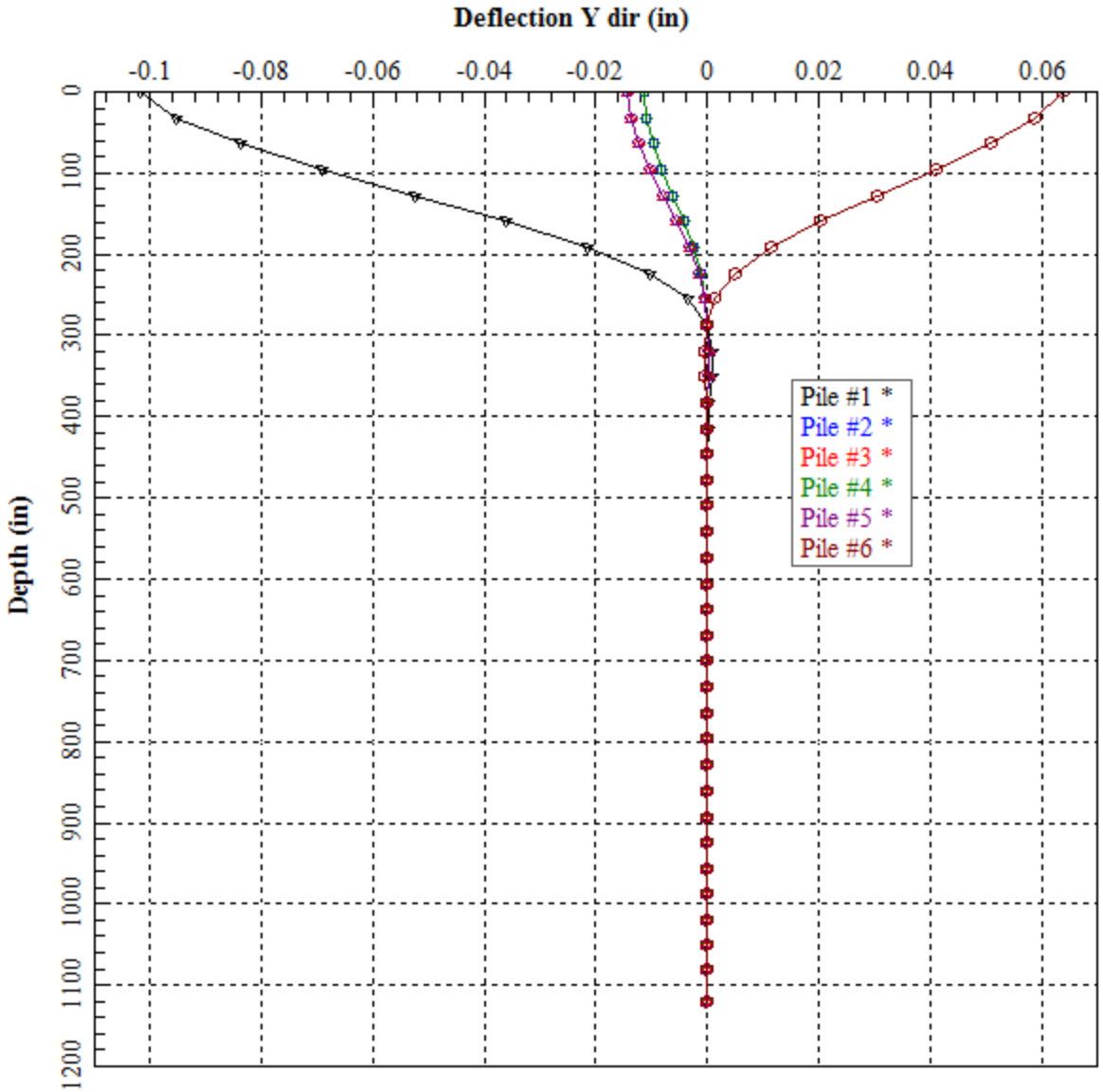


Figure 4.26 Effects on graph display using Plot Legend option. Plot Legend check-marked

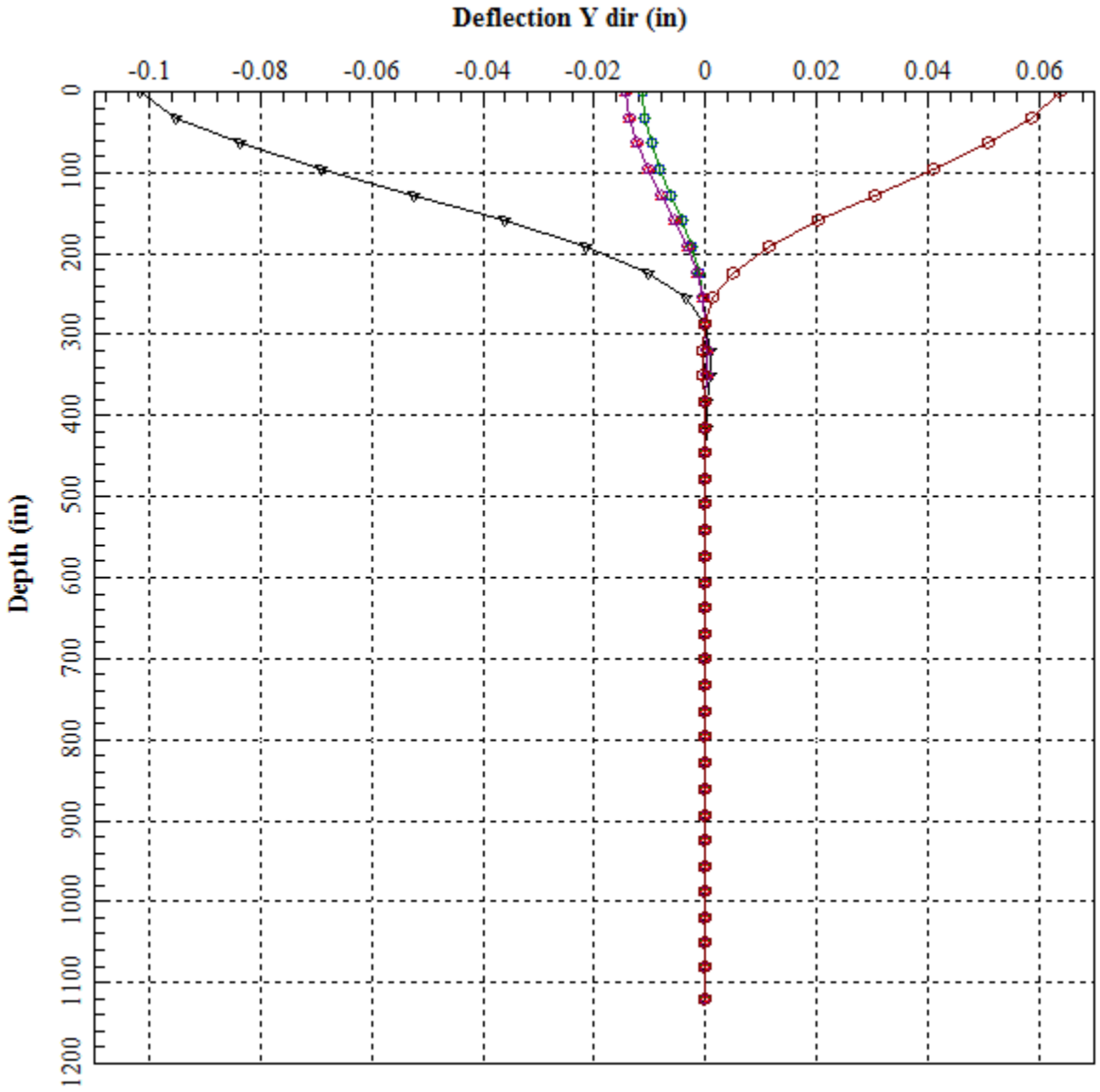


Figure 4.27 Effects on graph display using Plot Legend option. Plot Legend not check-marked

4.4.2 Plot - Show Markers

This activates or deactivates the markers for each point of all the curves displayed in the active screen of the graphics mode. The type of marker used in each curve is automatically selected by the program. Once activated, the markers are enabled for all visible curves of the active graphics screen.

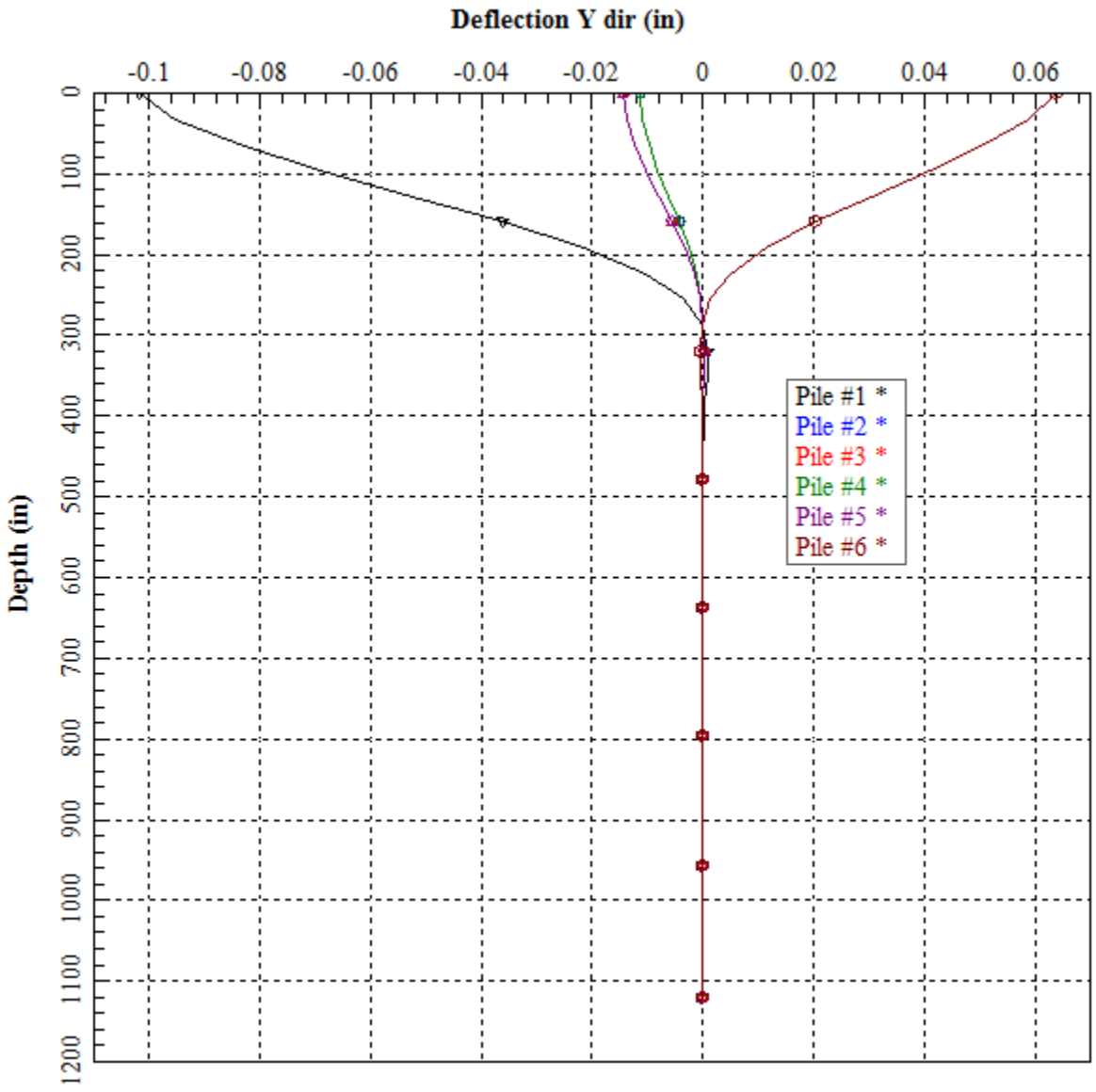


Figure 4.28 Effects on graph display using Plot Markers option. Using Markers every 5 increments

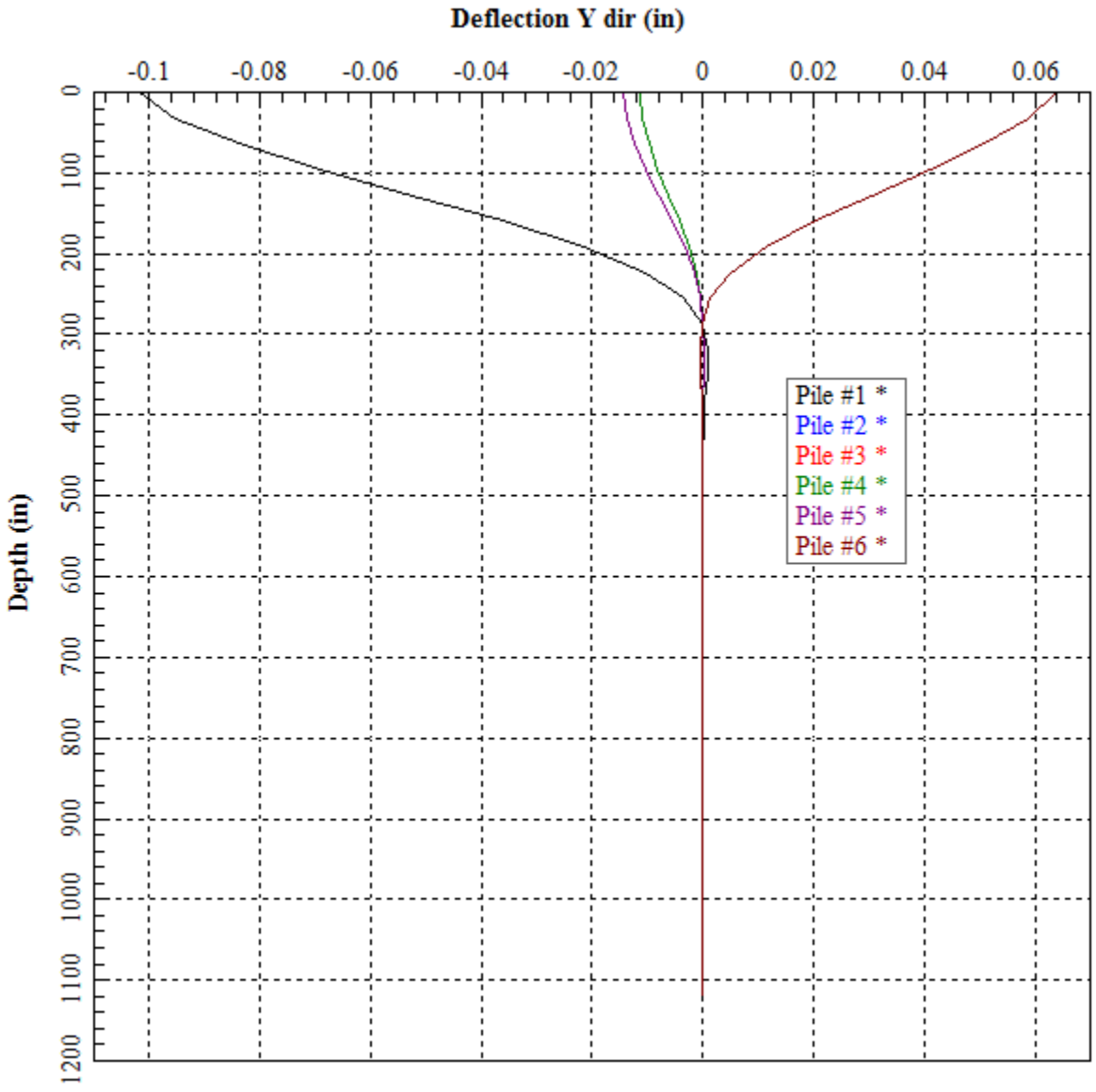


Figure 4.29 Effects on graph display using Plot Markers option. Using no markers (Never)

4.4.3 Plot - Font Size

This option set the font size for the current graph.

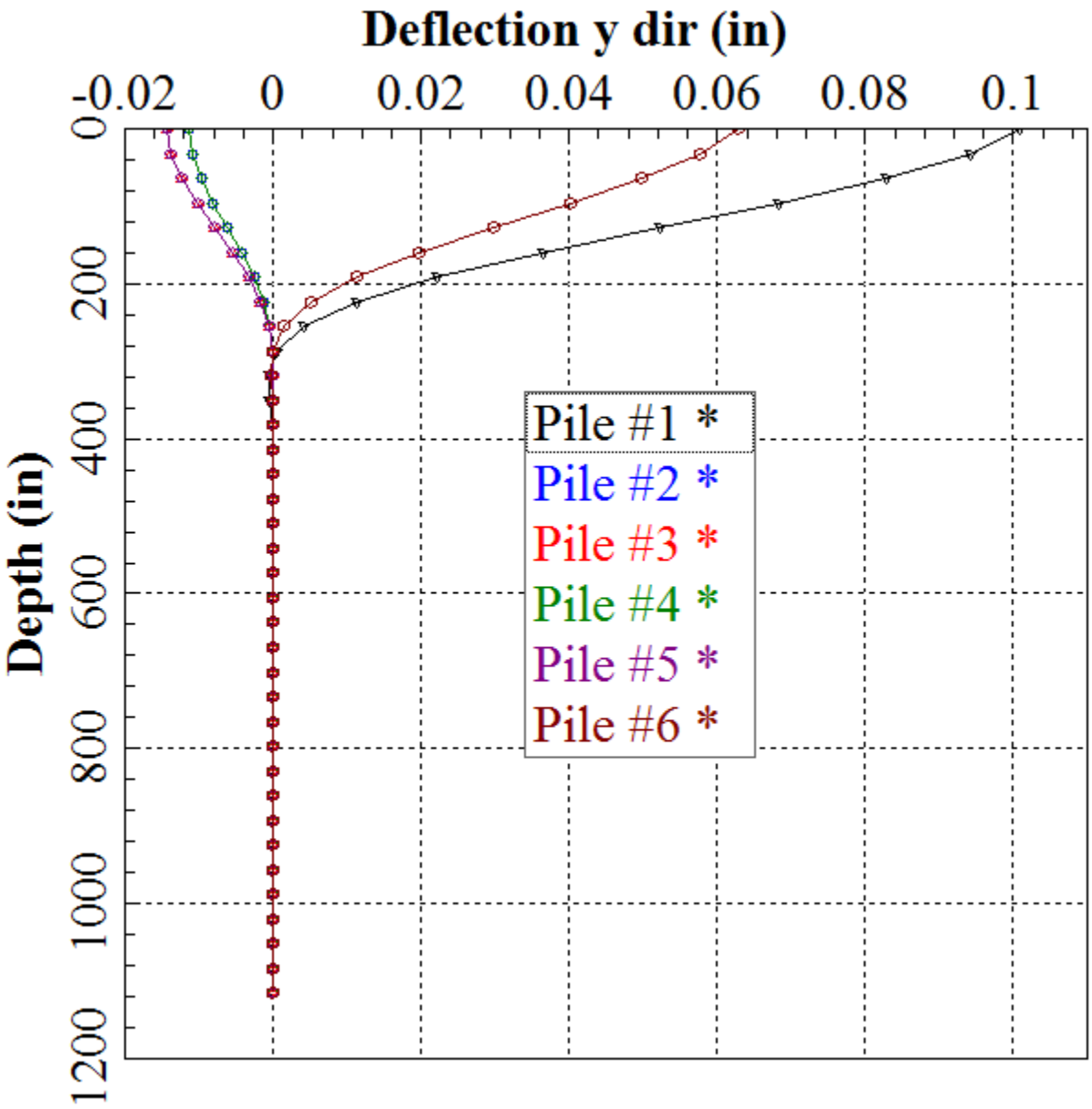


Figure 4.30 Effects on graph display using Font Size option. Using twice the Default Size

4.4.4 Plot - Line Width

This option set the line width for the current graph.

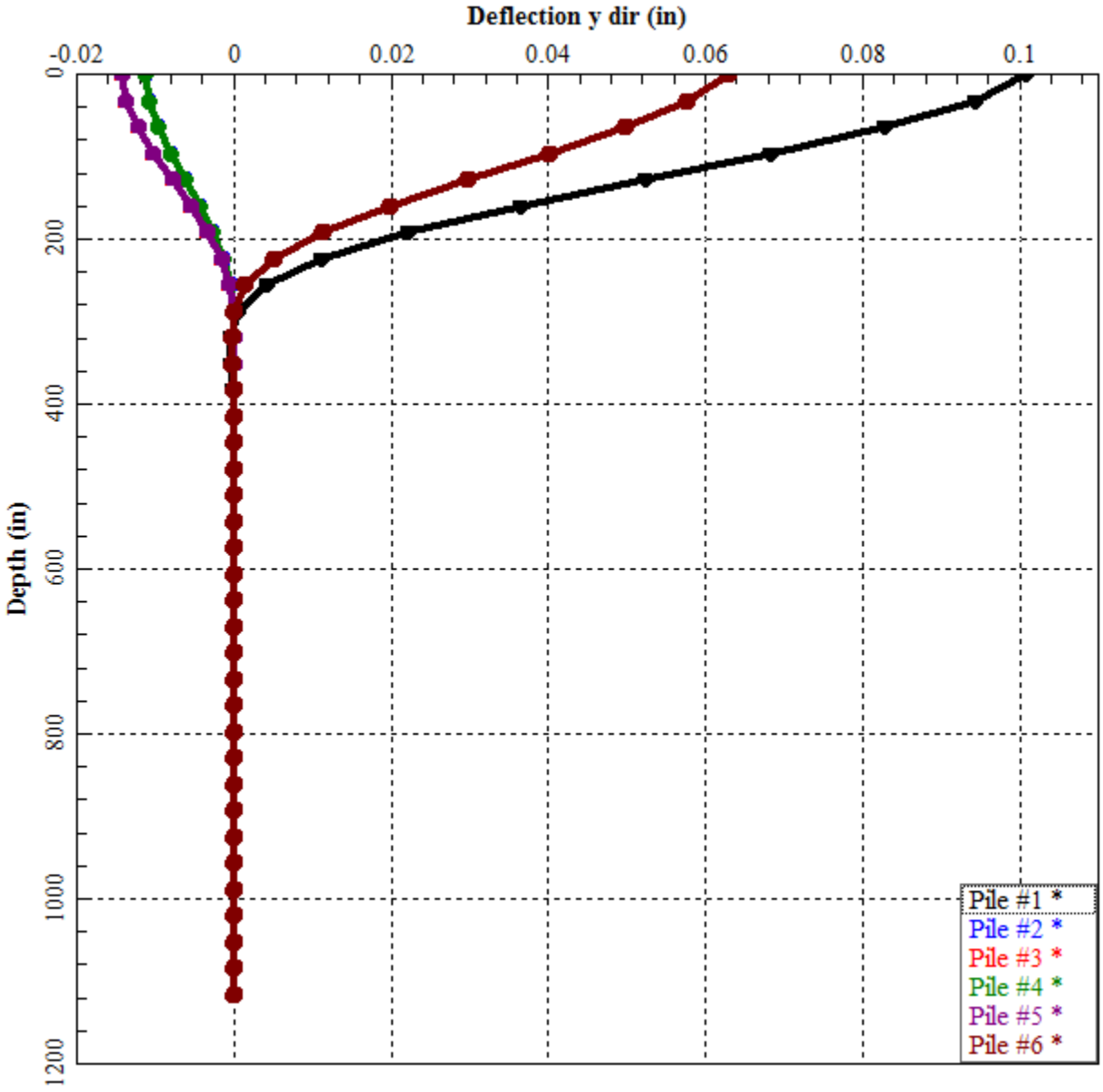


Figure 4.31 Effects on graph display using Line Width option. Using four times the Default Size

4.4.5 Plot - Graph Title

This command is used to define the title for the current graph.

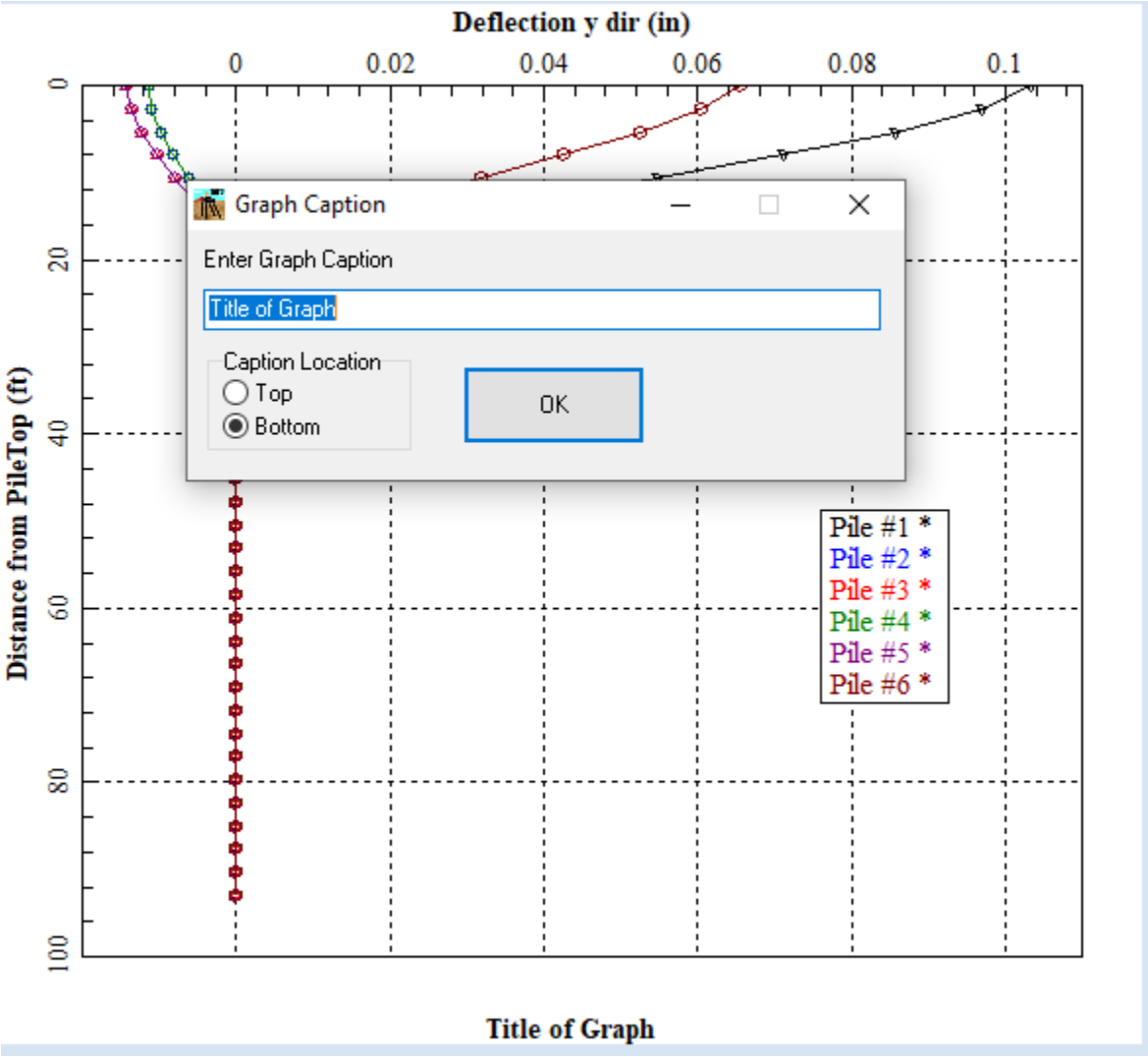


Figure 4.32 Using Graph Title option

4.4.6 Plot - Edit Legend

This command is used to edit the labels for the legend.

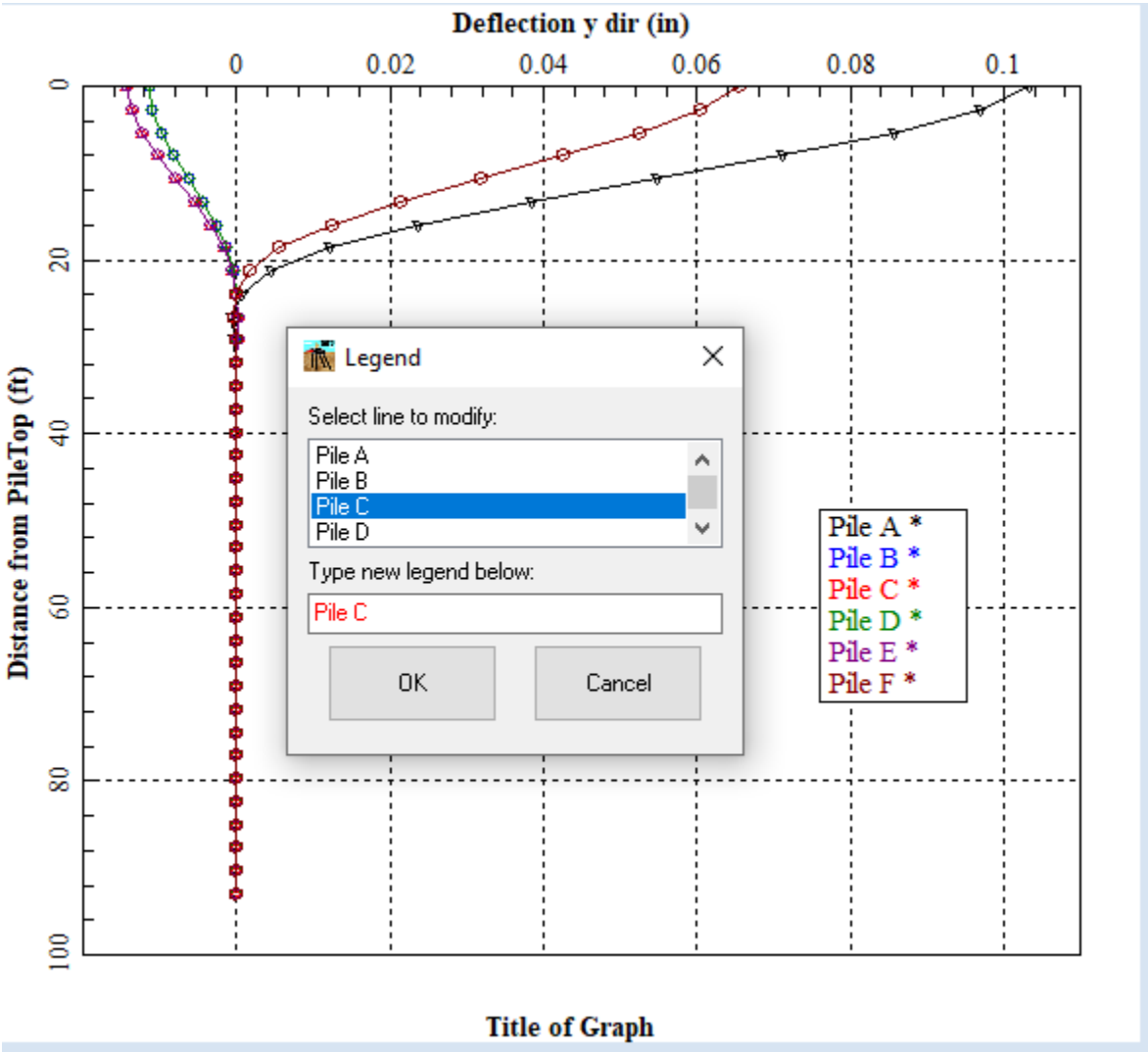


Figure 4.33 Using Edit Legend option

4.4.7 Plot - Active Curves

This option opens a new window that gives the user the ability to easily activate curves on the current graph. Additionally, below the **Active Curves** option the names of the curves as they are referred to in the legend appear—clicking on those names also deactivates or activates them.

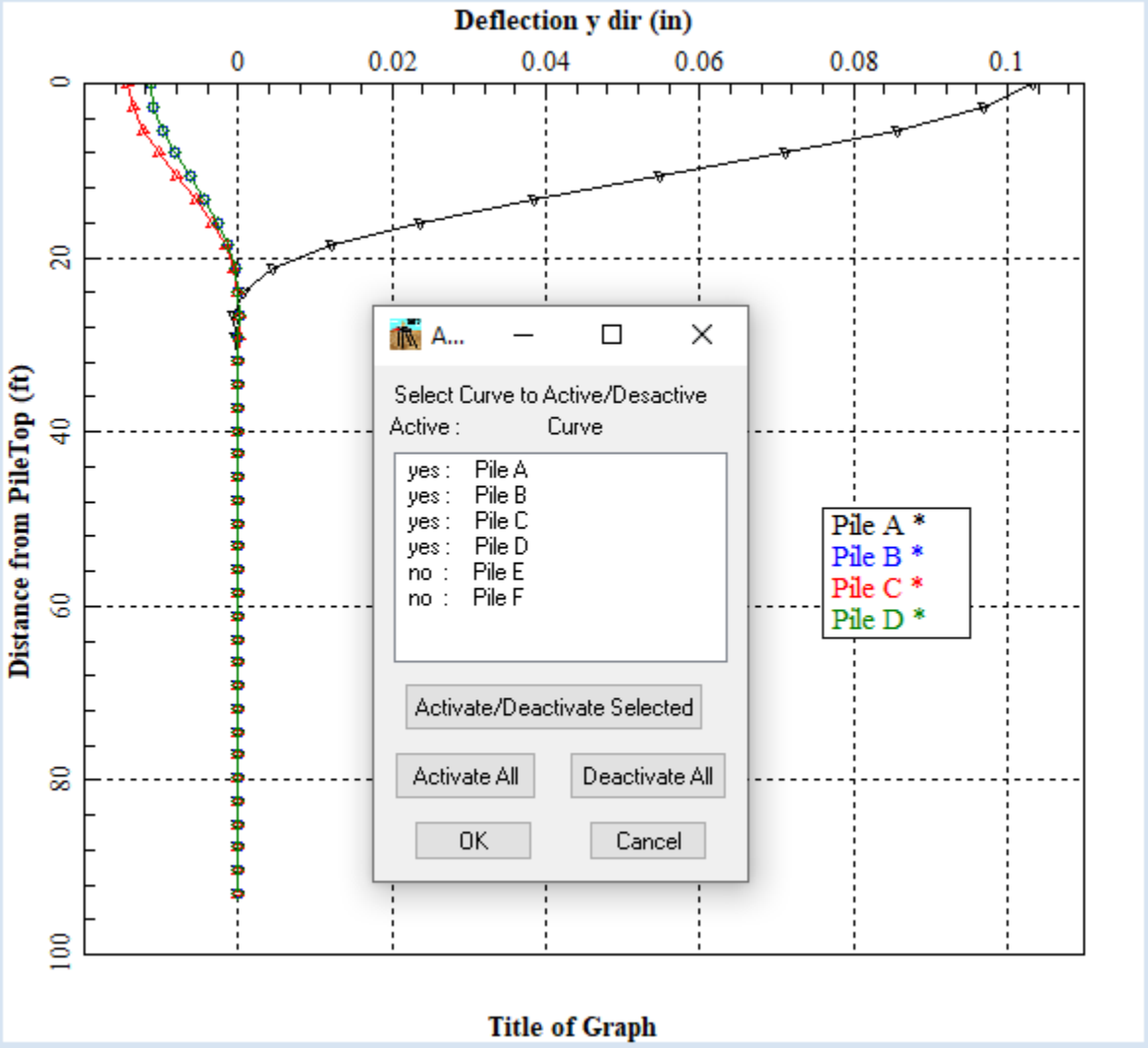


Figure 4.34 Using Active Graphs option

4.4.8 Plot - Print

This command calls up the standard Windows print dialog. The printed graphics are automatically sized to occupy the maximum width of the page.

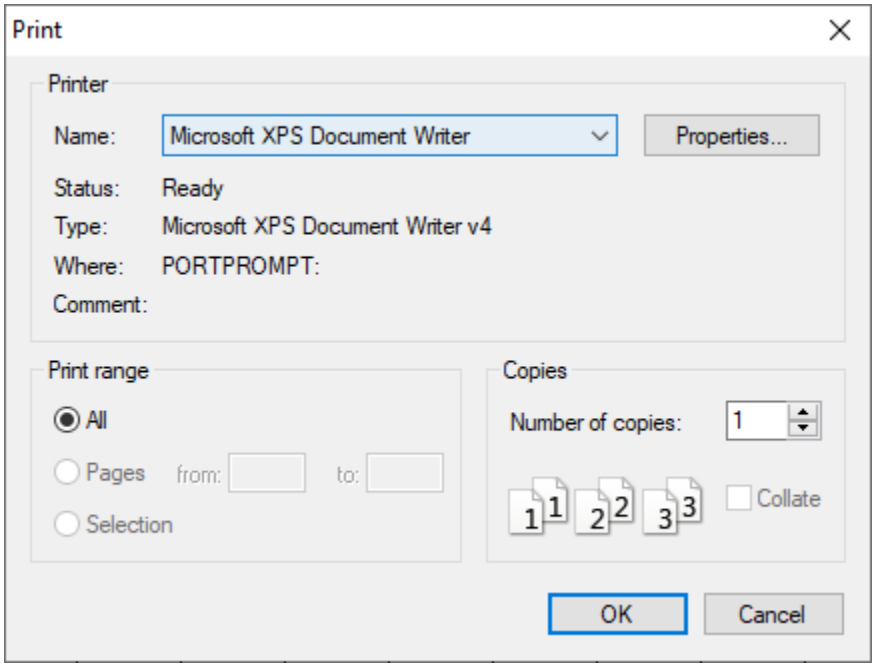


Figure 4.35 Using Print option

4.4.9 Plot - Page Setup

This command allows the user to customize the arrangement of the graph on the page for a hard copy print out.

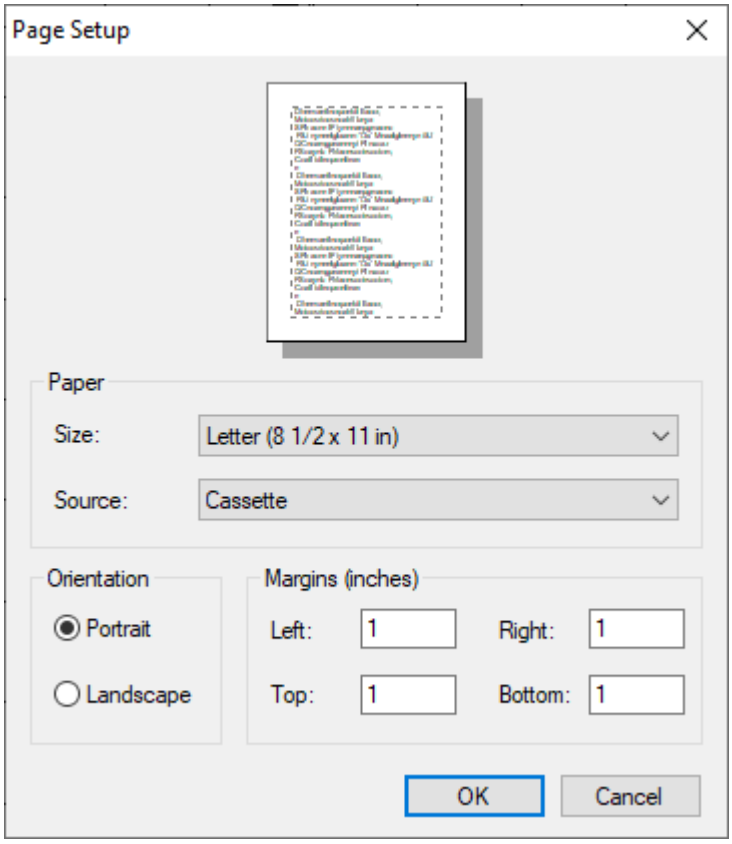


Figure 4.36 Using Page Setup option

4.4.10 Plot - Save To Disk

This command is used to save the active graphics screen to disk. The format of the saved file is **.bmp*.

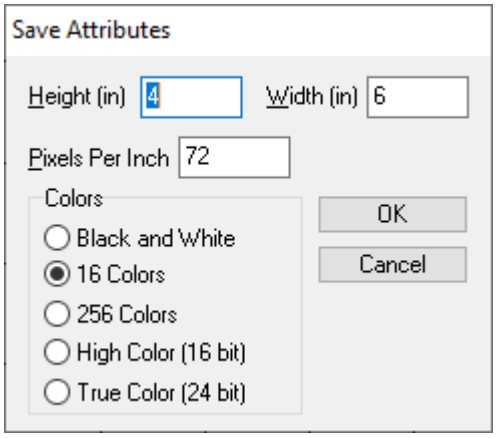


Figure 4.37 Using Save To Disk option

4.5 Graphs Menu

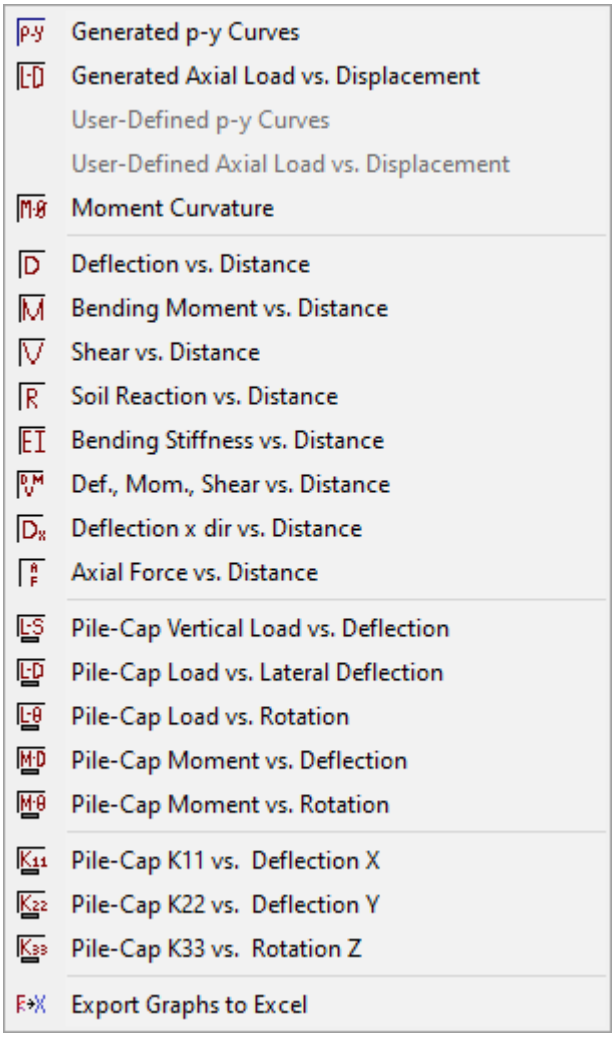


Figure 4.38 Options contained in the Graphs Menu. 2D Model

Figure 4.39 Options contained in the Graphs Menu. 3D Model

4.5.1 Sign Convention for results along the piles

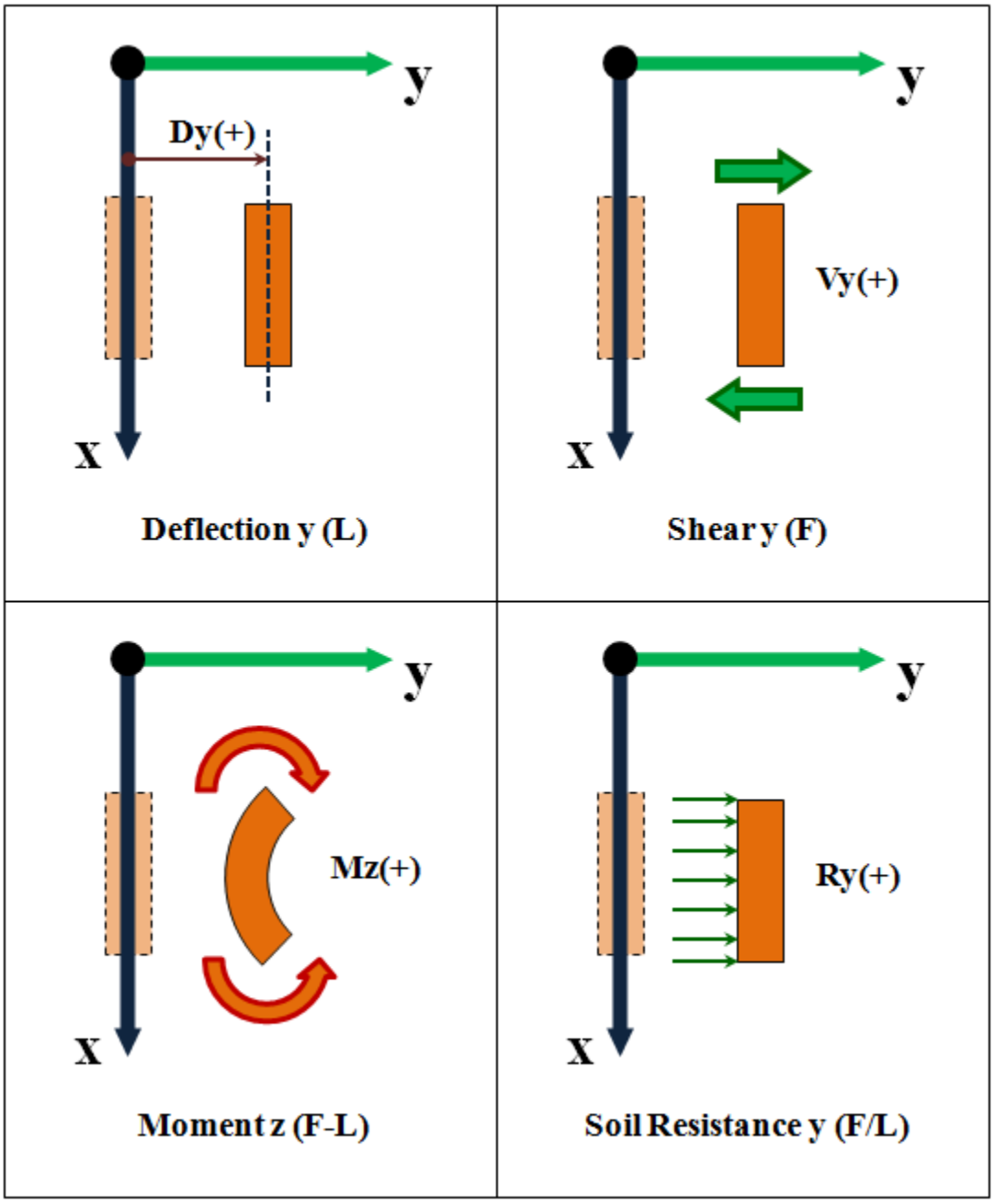


Figure 4.40 Sign Convention for pile under lateral loading. (xy-plane loading)

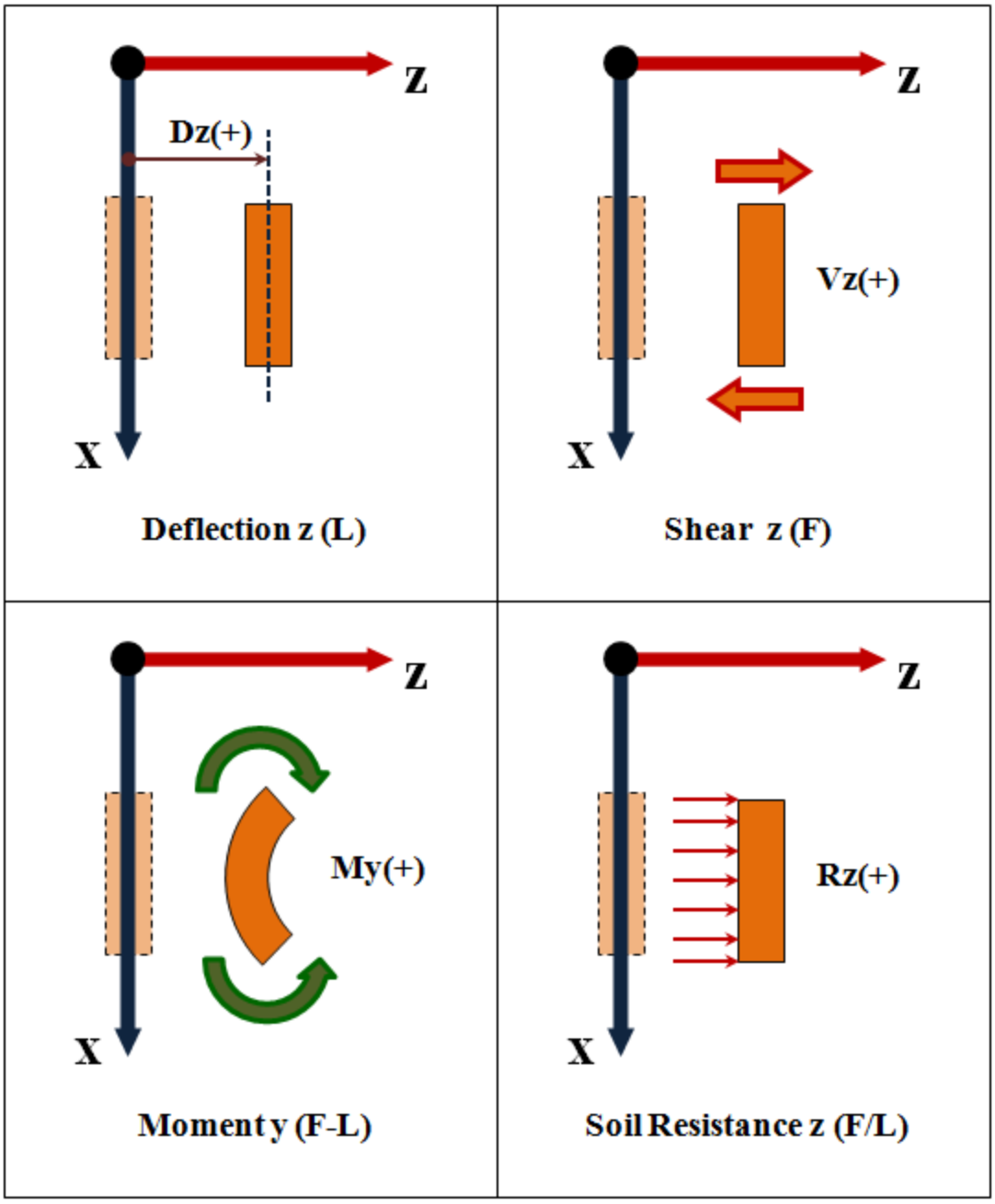


Figure 4.41 Sign Convention for pile under lateral loading. (xz-plane loading)

4.5.2 Graphs- Generated p-y Curves

GROUP 2022 will generate p-y curves at user-specified depths for any particular pile or subgroup selected by the user. This graphics option becomes available only if the user asked the program to print p-y curves for verification purposes using the command for Print p-y curves under the **Options** menu and the option Set of Data to Print p-y curves under the **Data** menu.

When available, the graphics screen will show the p-y curves for all selected depths.

In the output data file and graphics, the labels of depths of *p-y* curves are computed as depths below the ground line. These labels will thus be different than requested depths of printout when the ground line is not at the same level as the origin of the global coordinate system (usually at the level of the pile cap).

If the user selects Pile-Cap Embedded under the **Pile Cap Options** menu item and also requests to print an internally generated *p-y* curve within the height of the pile cap, then two different *p-y* curves will be plotted. One will correspond to the *p-y* curve for the pile cap and the other will correspond to the *p-y* curve for the specified pile subgroup. The difference can easily be identified in the pile diameters. In addition, the *p-y* curve corresponding to the pile cap will usually be a stiffer curve.

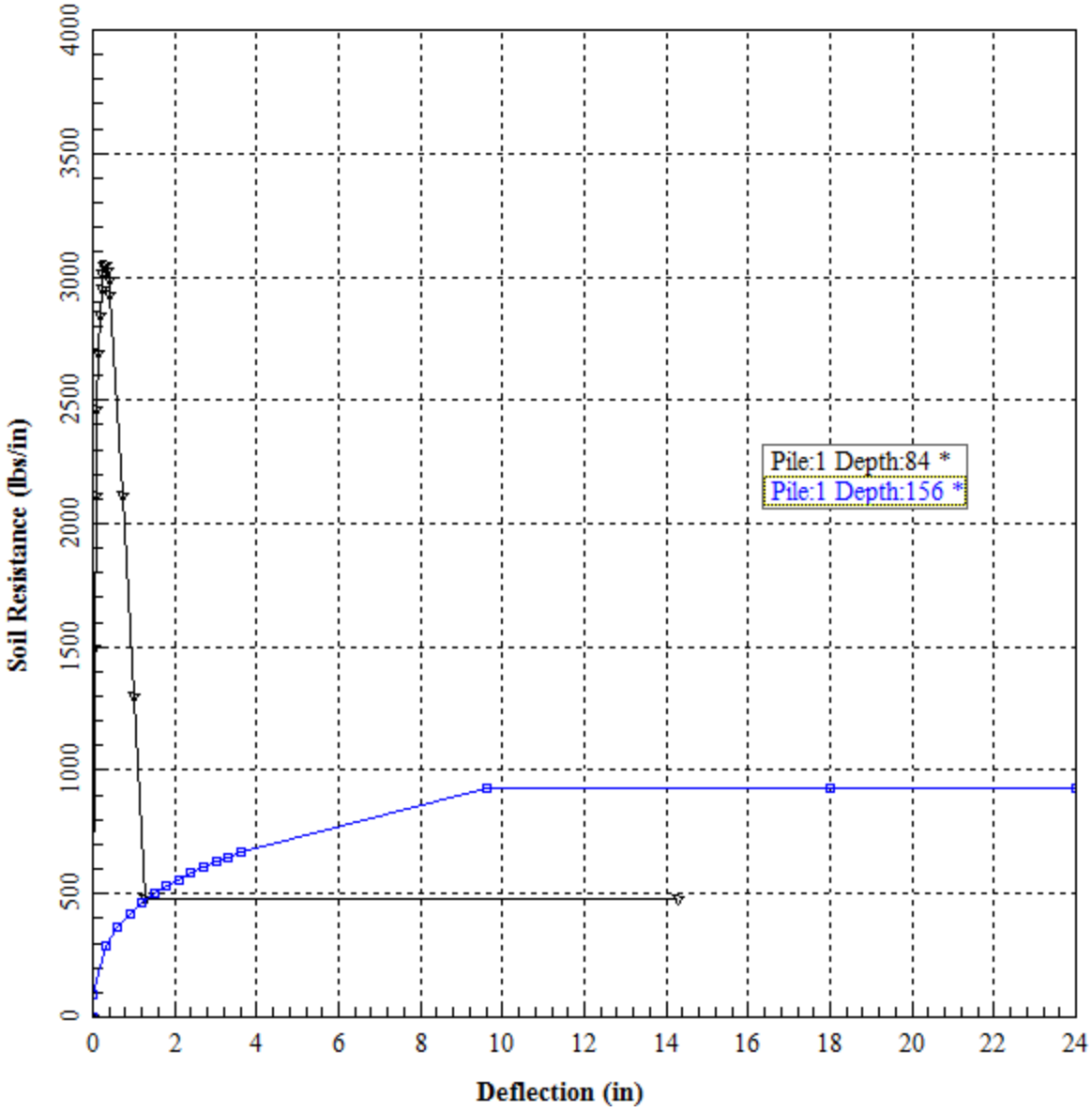


Figure 4.42 Sample graphics screen of p-y Curves command under the Graphs menu

4.5.3 Graphs- Generated Axial Load vs. Displacement

If the program was asked to generate internally the curve of axial load versus displacement of a single pile, the user may select this item to observe a graphical representation of the curve.

Two curves are always generated for each pile type. One curve corresponds to movements due to tensile loads while the other corresponds to movements due to compressive loads. Curves corresponding to tensile loads are always the ones ending on a vertical slope. Theoretically, curves corresponding to tensile and compressive loads may be equal if soils offer no resistance in end bearing.

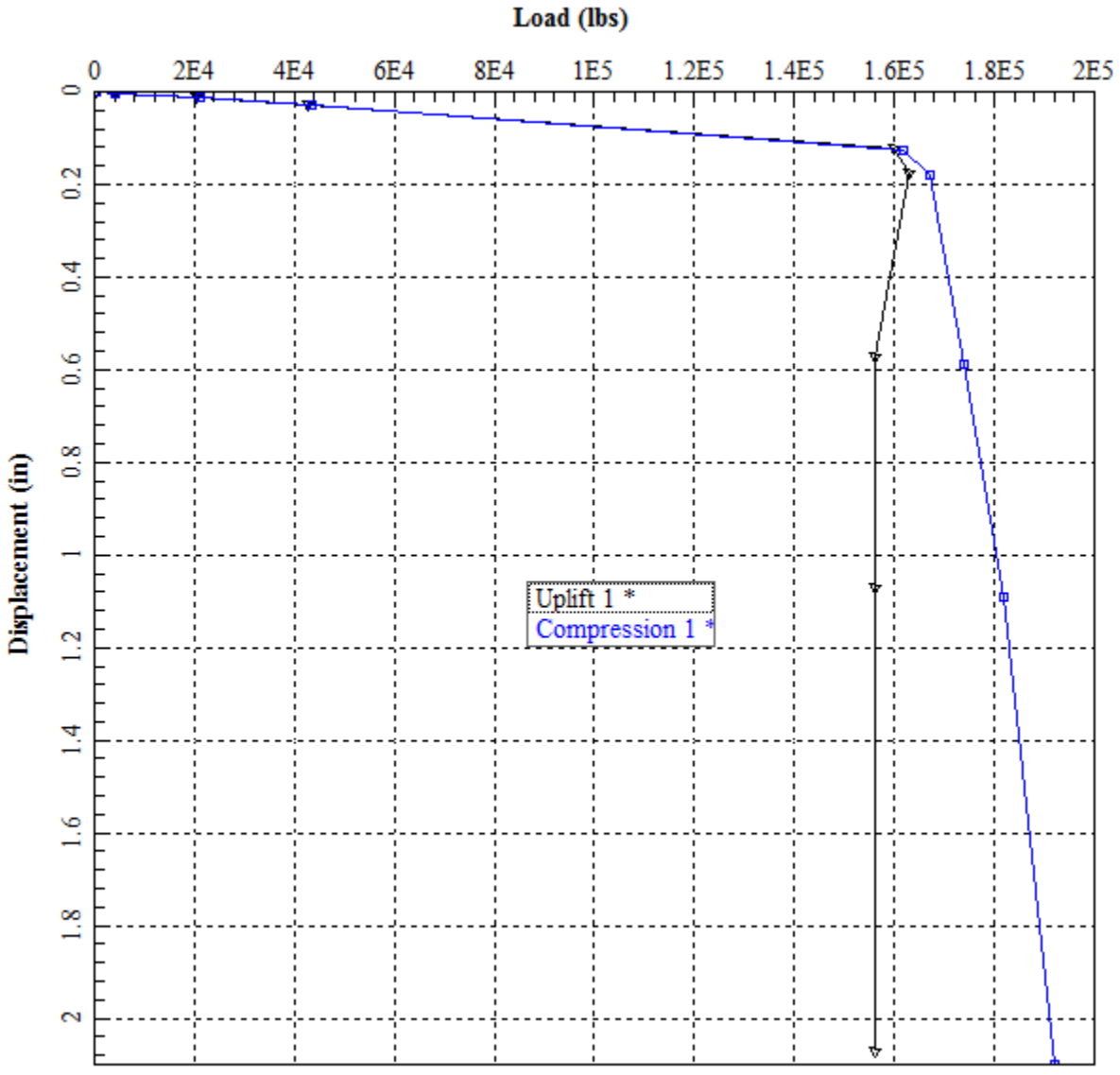


Figure 4.43 Sample graphics screen of Axial Load vs. Displacement command under the Graphs menu

4.5.4 Graphs- Generated Torque vs. Rotation

If the program was asked to generate internally the curve of torque versus rotation of a single pile, the user may select this item to observe a graphical representation of the curve.

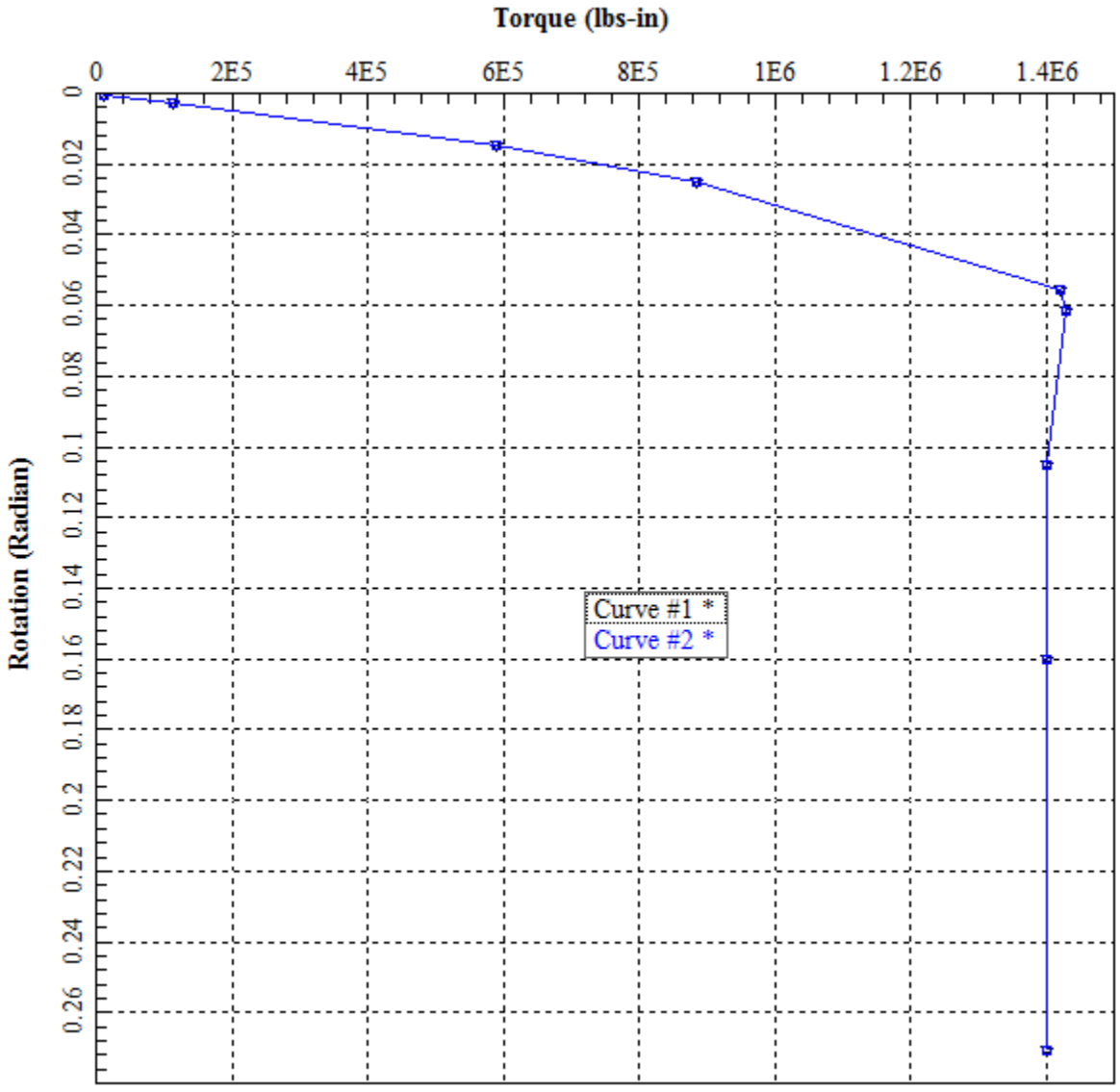


Figure 4.44 Sample graphics screen of Torque vs. Rotation command under the Graphs menu

4.5.5 Graphs- User Defined p-y Curves

This graphics option become available only if the user enters a User-Defined p-y curve

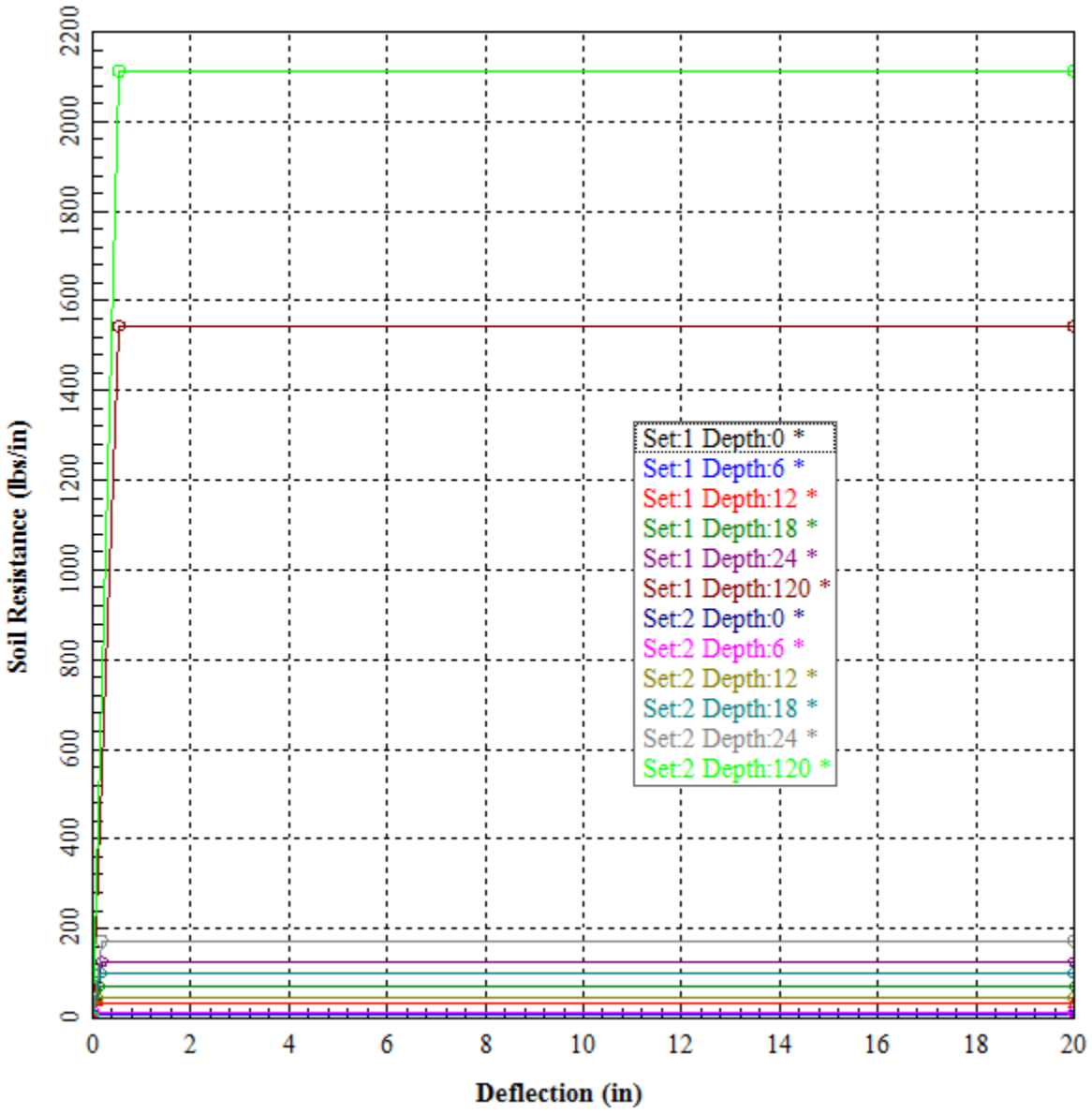


Figure 4.45 Sample graphics screen of p-y Curves command under the Graphs menu

4.5.6 Graphs- User Defined Axial Load vs. Displacement

This graphics option becomes available only if the user enters a User-Defined Axial Load-Displacement curve.

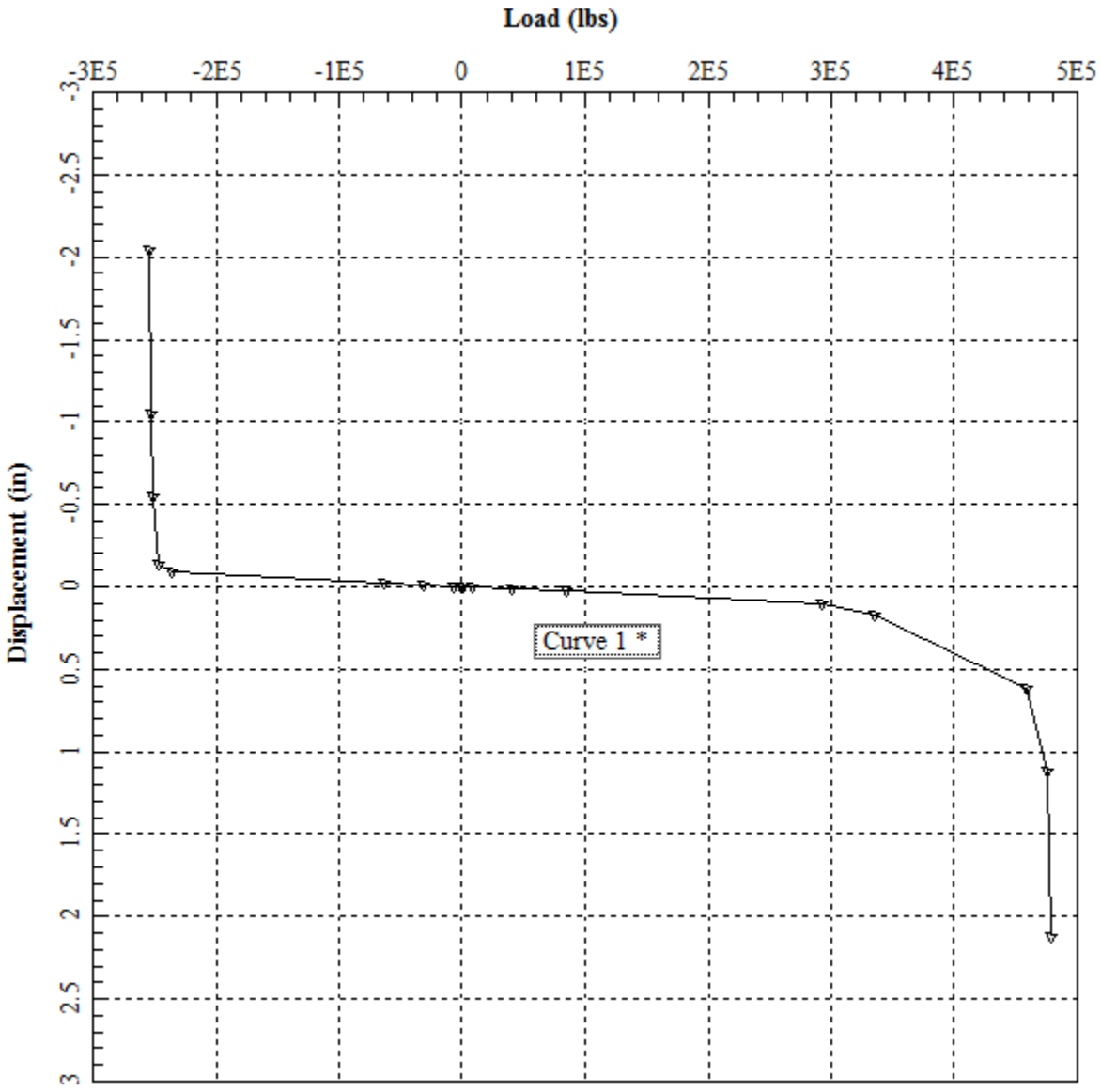


Figure 4.46 Sample graphics screen of Axial Load vs. Displacement command under the Graphs menu

4.5.7 Graphs- User Defined Torque vs. Rotation

This graphics option becomes available only if the user enters a User-Defined Torque-Rotation curve.

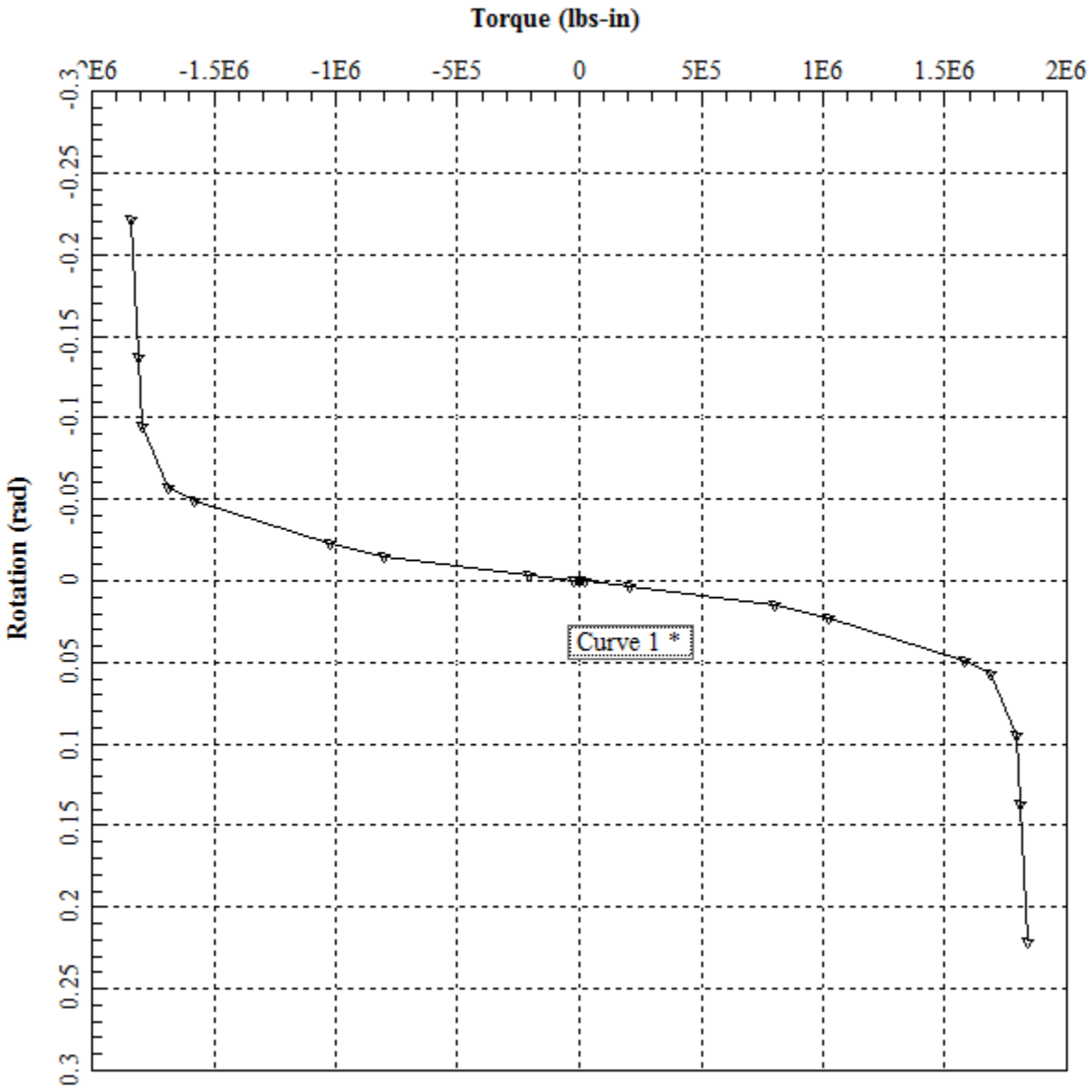


Figure 4.47 Sample graphics screen of Torque vs. Rotation command under the Graphs menu

4.5.8 Graphs - Deflection vs. Distance

This option provides a plot of lateral pile deflection versus distance from pile top of each individual pile in a 2-dimensional pile group. Notice that the lateral deflections are provided in values for the local axis of each pile (which is different for piles that have different batter).

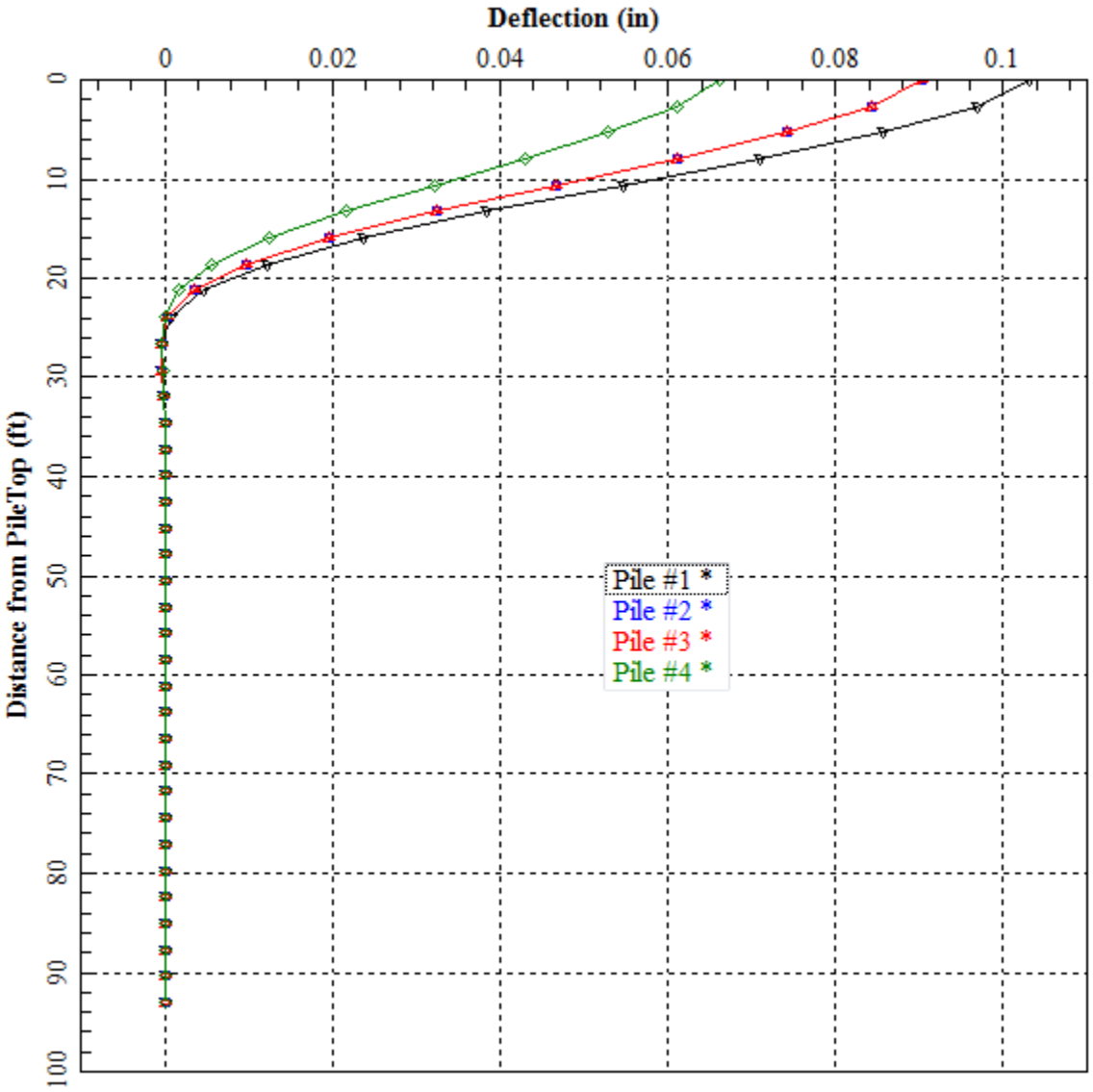


Figure 4.48 Sample graphics screen of Deflection vs. Distance command under the Graphs menu

4.5.9 Graphs - Moment vs. Distance

This option provides a plot of bending moment versus distance from pile top of each individual pile in a 2-dimensional pile group. Notice that the moments are provided in values for the local axis of each pile (which is different for piles that have different batter).

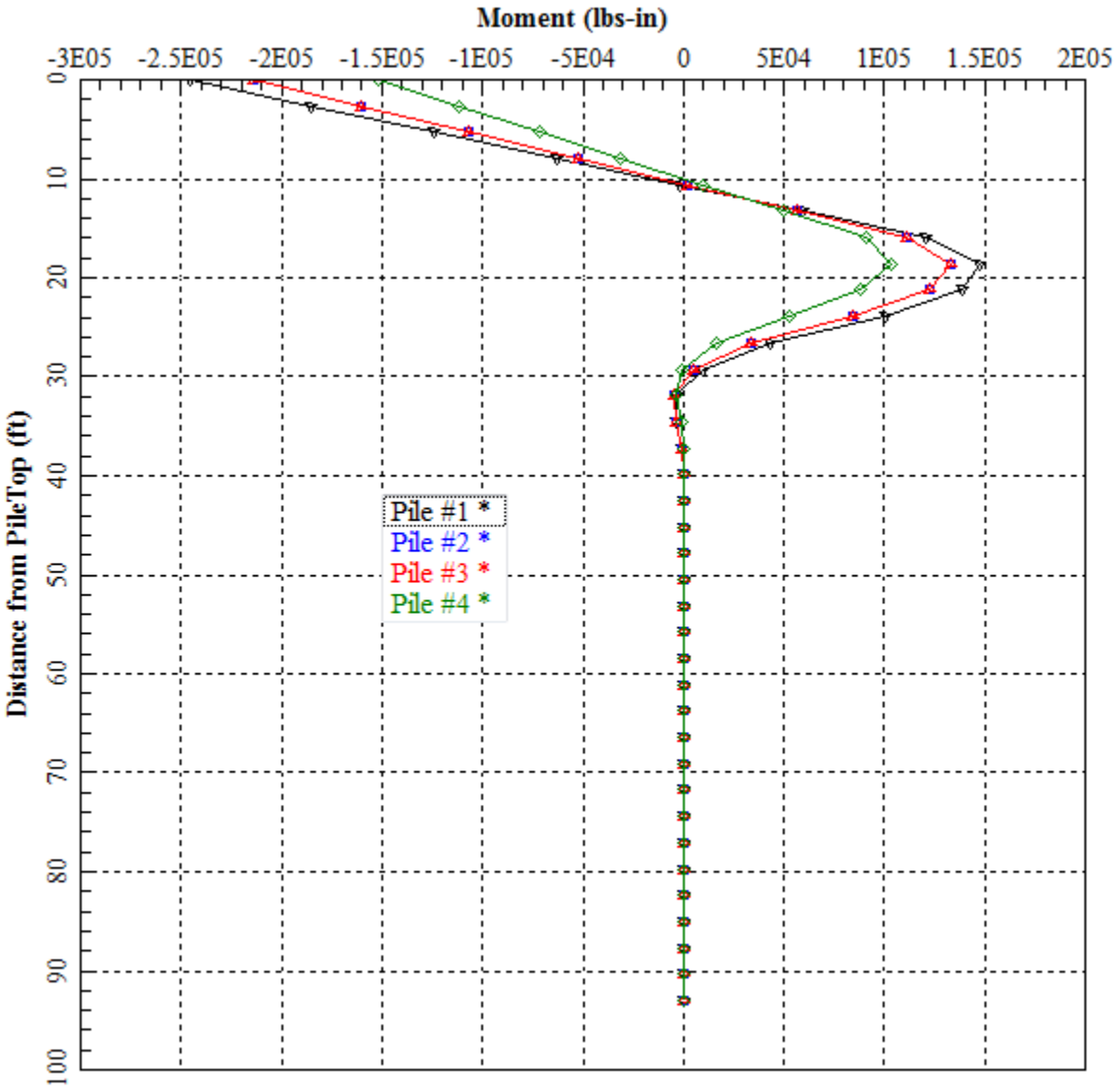


Figure 4.49 Sample graphics screen of Moment vs. Distance command under the Graphs menu

4.5.10 Graphs - Shear vs. Distance

This option provides a plot of shear versus distance from pile top of each individual pile in a 3-dimensional pile group. Notice that the shear forces are provided in values for the local axis of each pile (which is different for piles that have different batter).

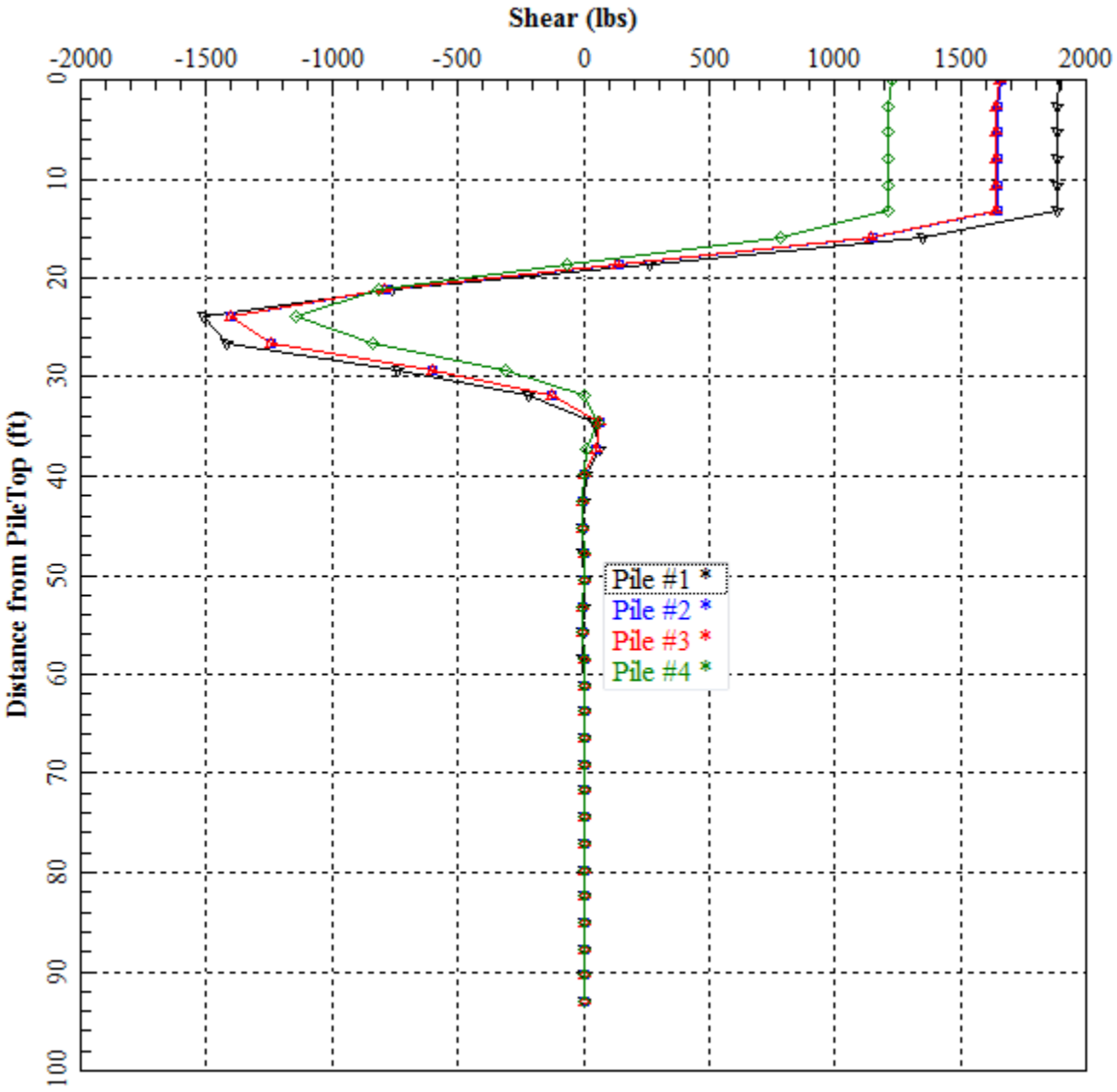


Figure 4.50 Sample graphics screen of Shear vs. Distance command under the Graphs menu

4.5.11 Graphs - Soil Reaction vs. Distance

This option provides a plot of lateral soil resistance versus distance from pile top of each individual pile in a 2-dimensional pile group. Notice that the soil reactions are provided in values for the local axis of each pile (which is different for piles that have different batter).

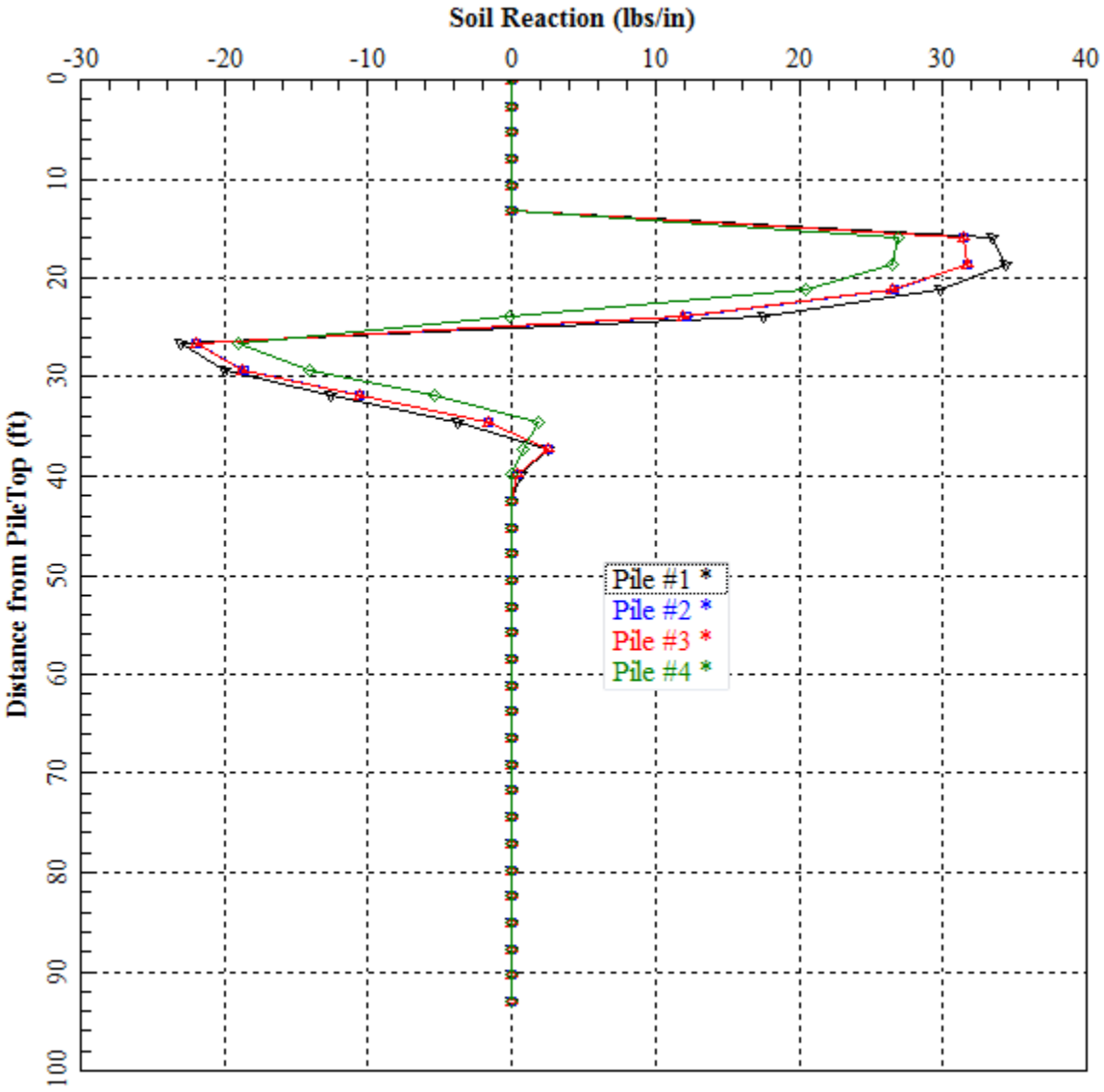


Figure 4.51 Sample graphics screen of Soil Reaction vs. Distance command under the Graphs menu

4.5.12 Graphs – Bending Stiffness vs. Distance

This option provides a plot of the pile bending stiffness versus distance from pile top of each individual pile in a 2-dimensional pile group. Notice that the bending stiffness are provided in values along the local axis of each pile

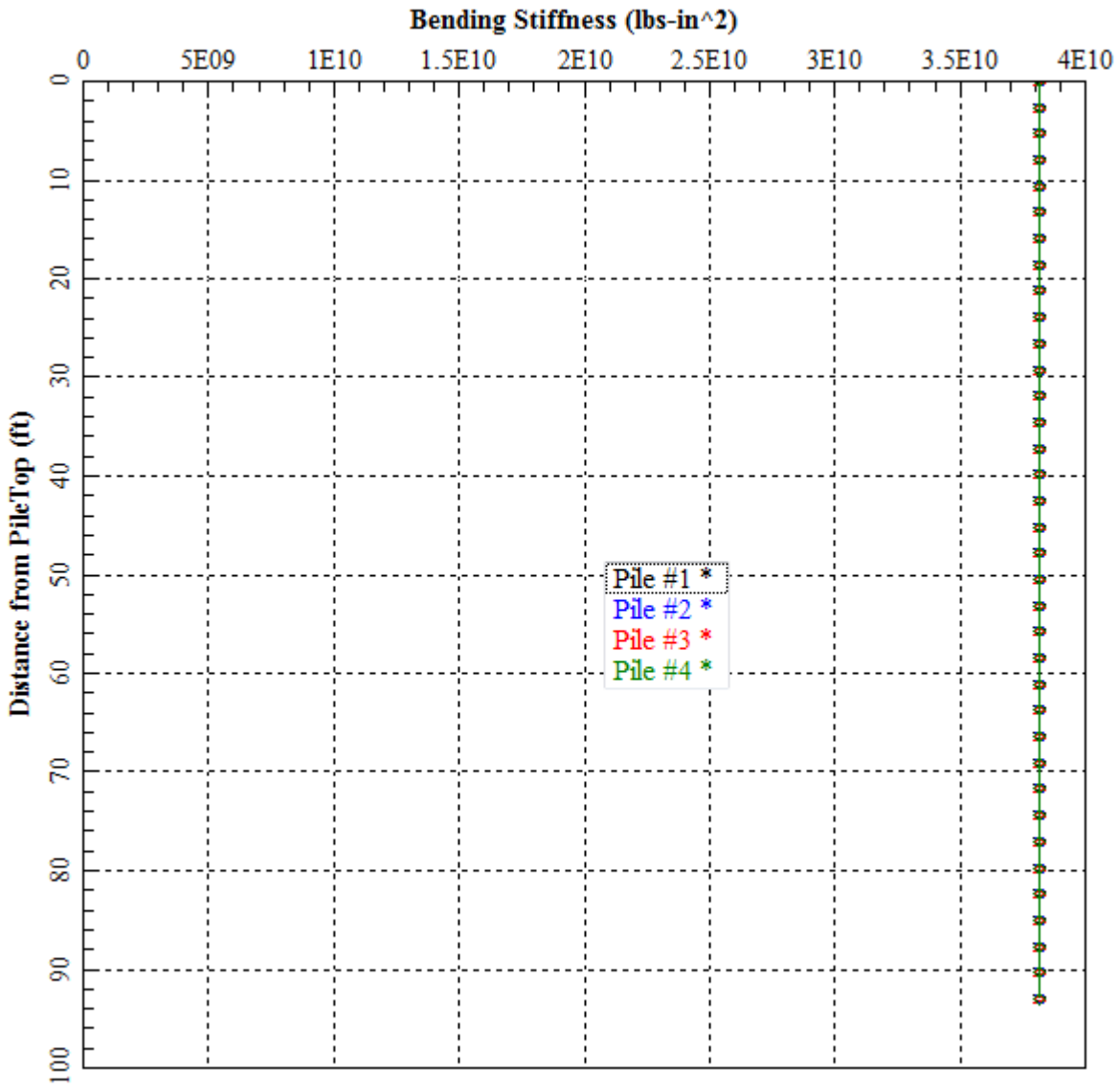


Figure 4.52 Sample graphics screen of Bending Stiffness vs. Distance command under the Graphs menu

4.5.13 Graphs – Def., Mom., Shear vs. Distance

This option provides a plot (in the same graph) of the lateral deflection, bending moment and shear versus distance from pile top of each individual pile in a 2-dimensional pile group. Notice that the deflection, moment and shear are provided in values for the local axis of each pile (which is different for piles that have different batter).

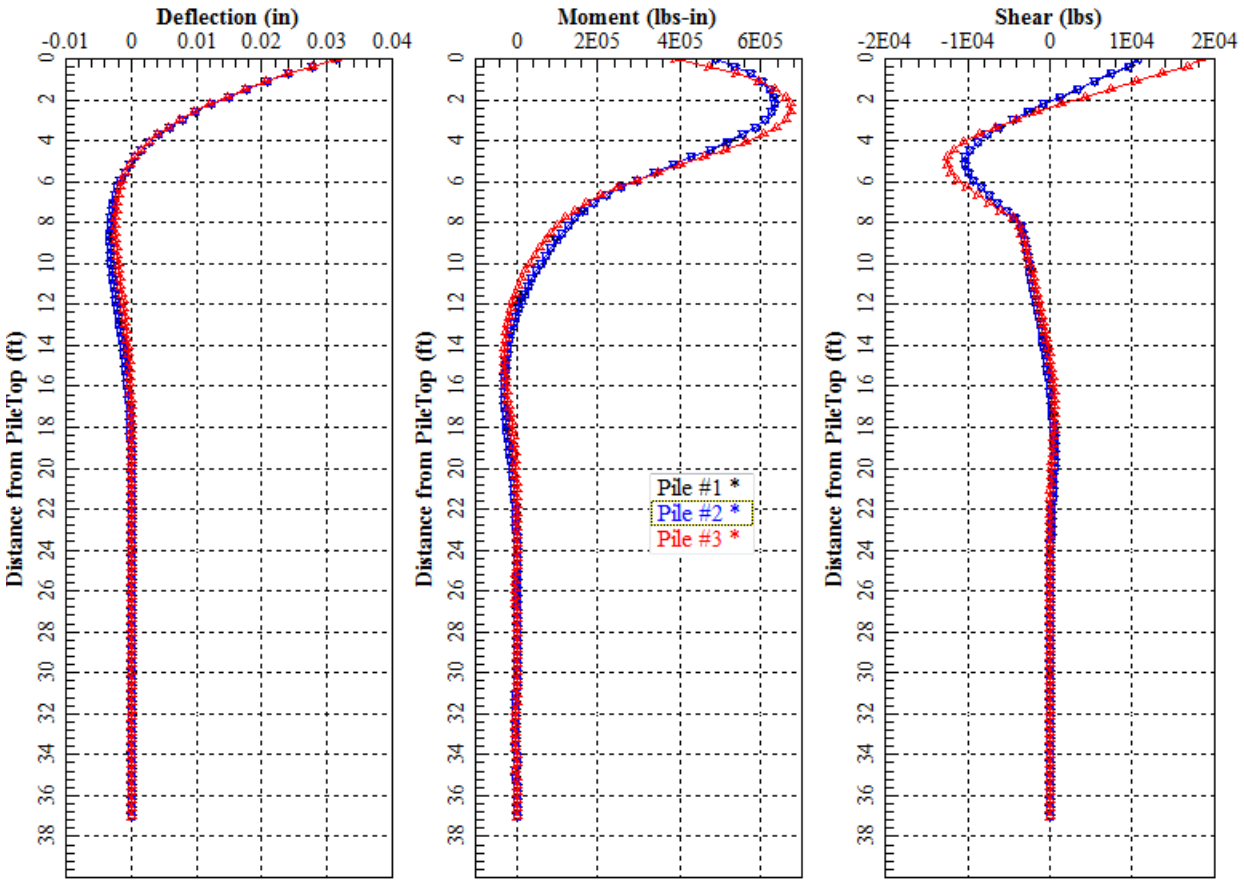


Figure 4.53 Sample graphics screen of Def., Mom., Shear vs. Distance command under the Graphs menu

4.5.14 Graphs - Deflection y dir vs. Distance

This option provides a plot of lateral pile deflection versus distance from pile top on the local y-axis of each individual pile in a 3-dimensional pile group. Notice that the lateral deflections are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

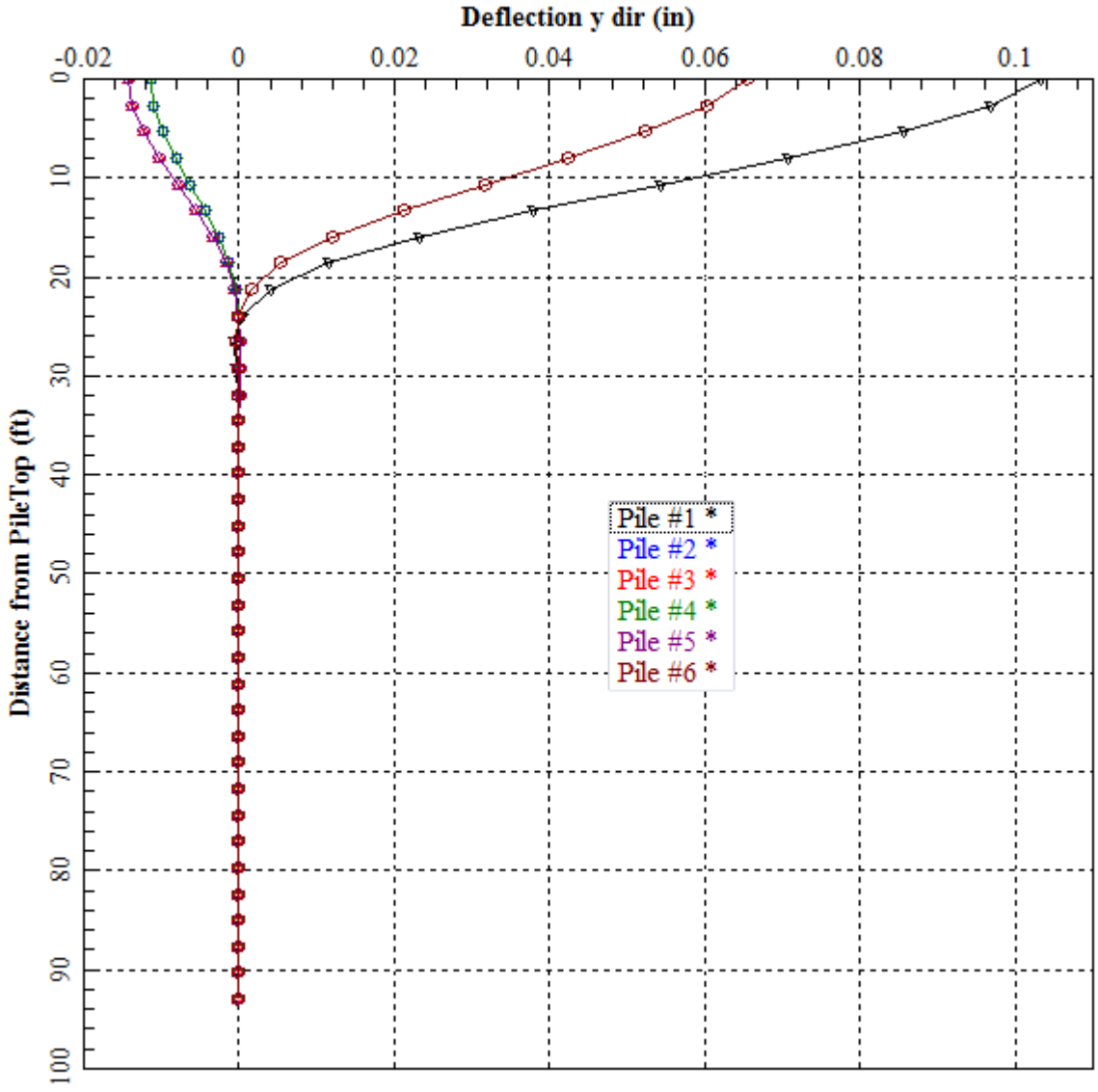


Figure 4.54 Sample graphics screen of Deflection y dir vs. Distance command under the Graphs menu

4.5.15 Graphs - Deflection z dir vs. Distance

This option provides a plot of lateral pile deflection versus distance from pile top on the local z-axis of each individual pile in a 3-dimensional pile group. Notice that the lateral deflections are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

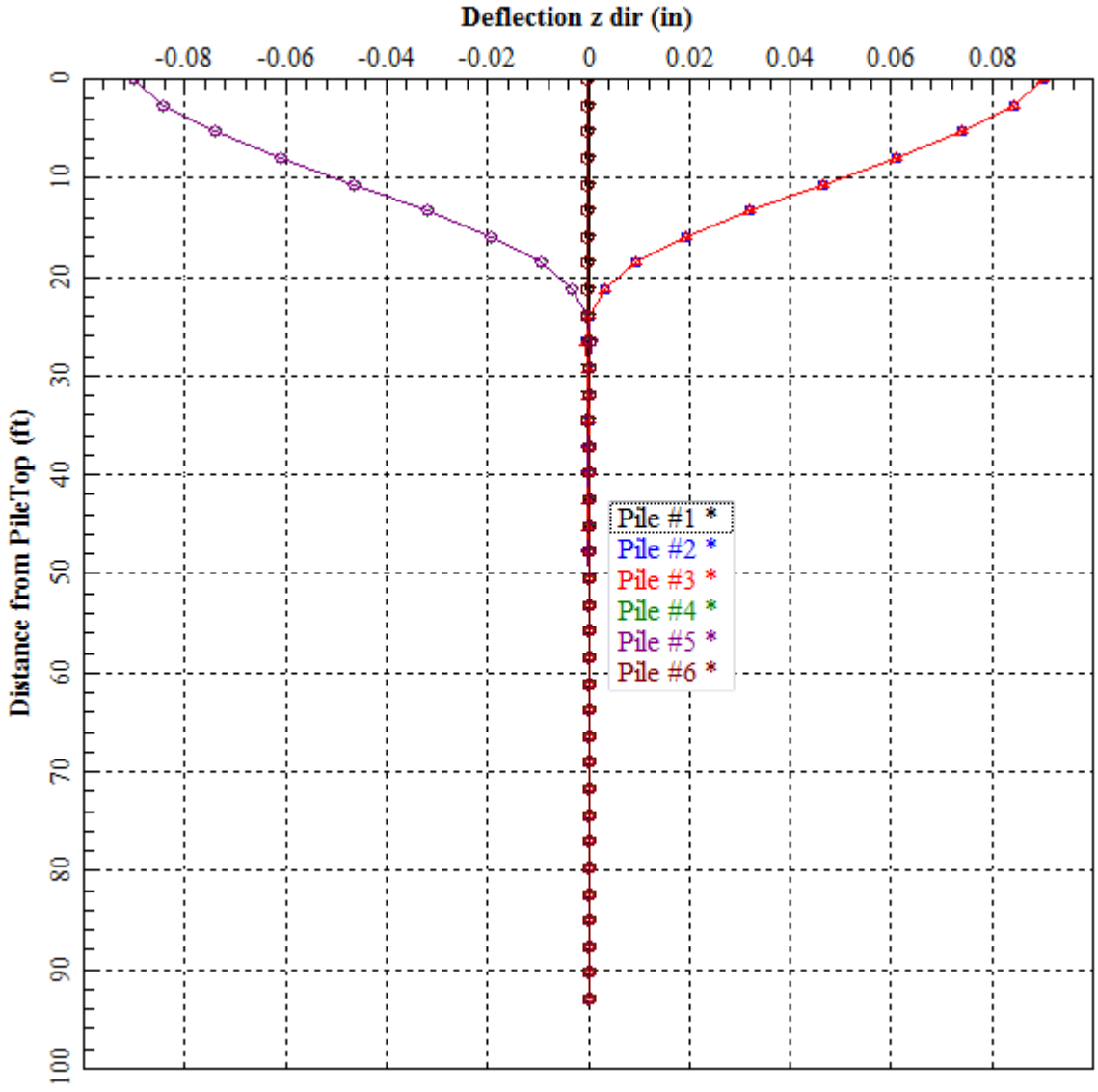


Figure 4.55 Sample graphics screen of Deflection z dir vs. Distance command under the Graphs menu

4.5.16 Graphs - Moment z dir vs. Distance

This option provides a plot of bending moment about the local z-axis versus distance from pile top on the xy-plane of each individual pile in a 3-dimensional pile group. Notice that the moments are provided in values along the local axis of each pile (which may be different for piles that have different batter and rotation angles).

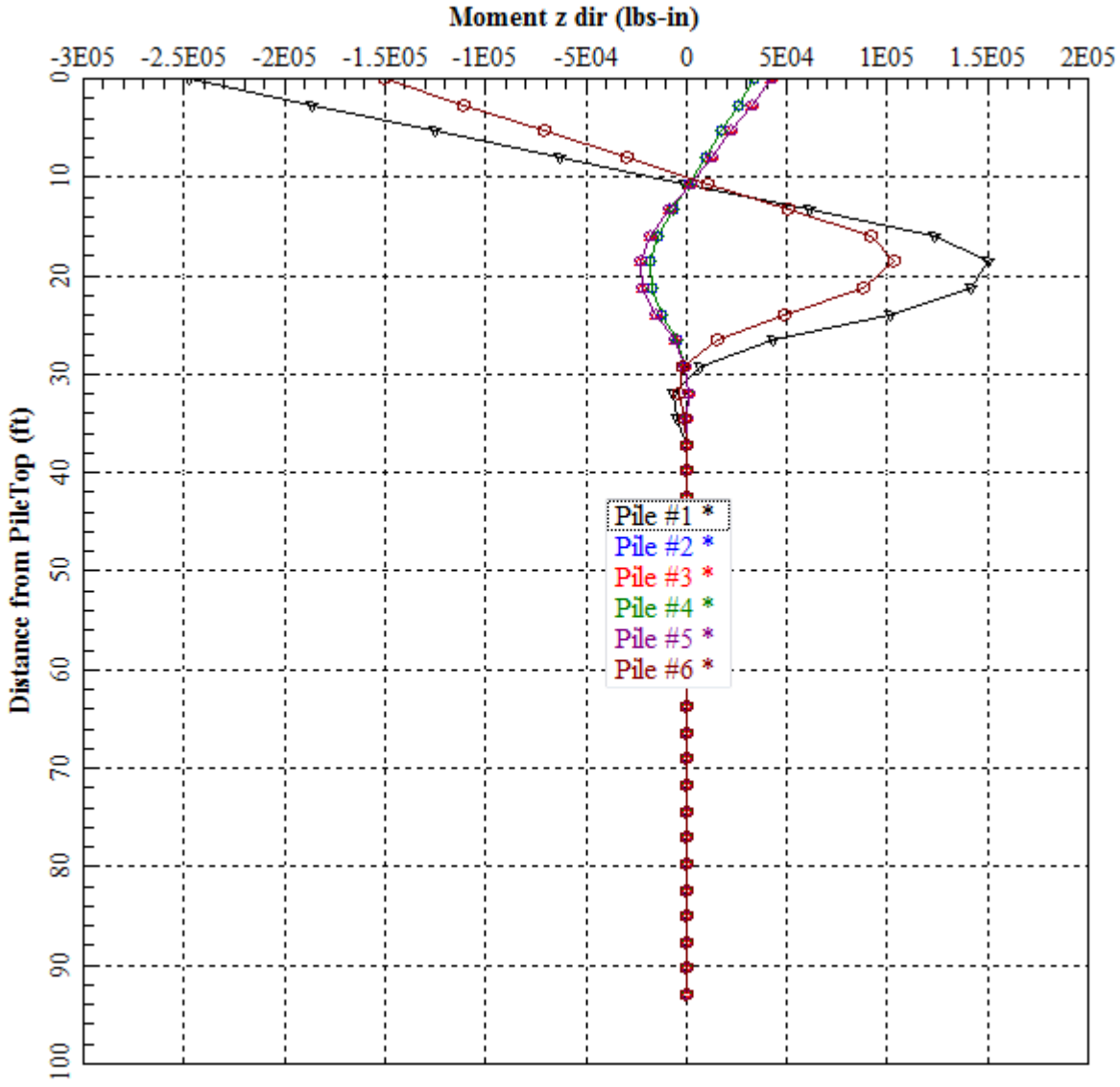


Figure 4.56 Sample graphics screen of Moment z dir vs. Distance command under the Graphs menu

4.5.17 Graphs - Moment y dir vs. Distance

This option provides a plot of bending moment about the local y-axis versus distance from pile top on the xz-plane of each individual pile in a 3-dimensional pile group. Notice that the moments are provided in values along the local axis of each pile (which may be different for piles that have different batter and rotation angles).

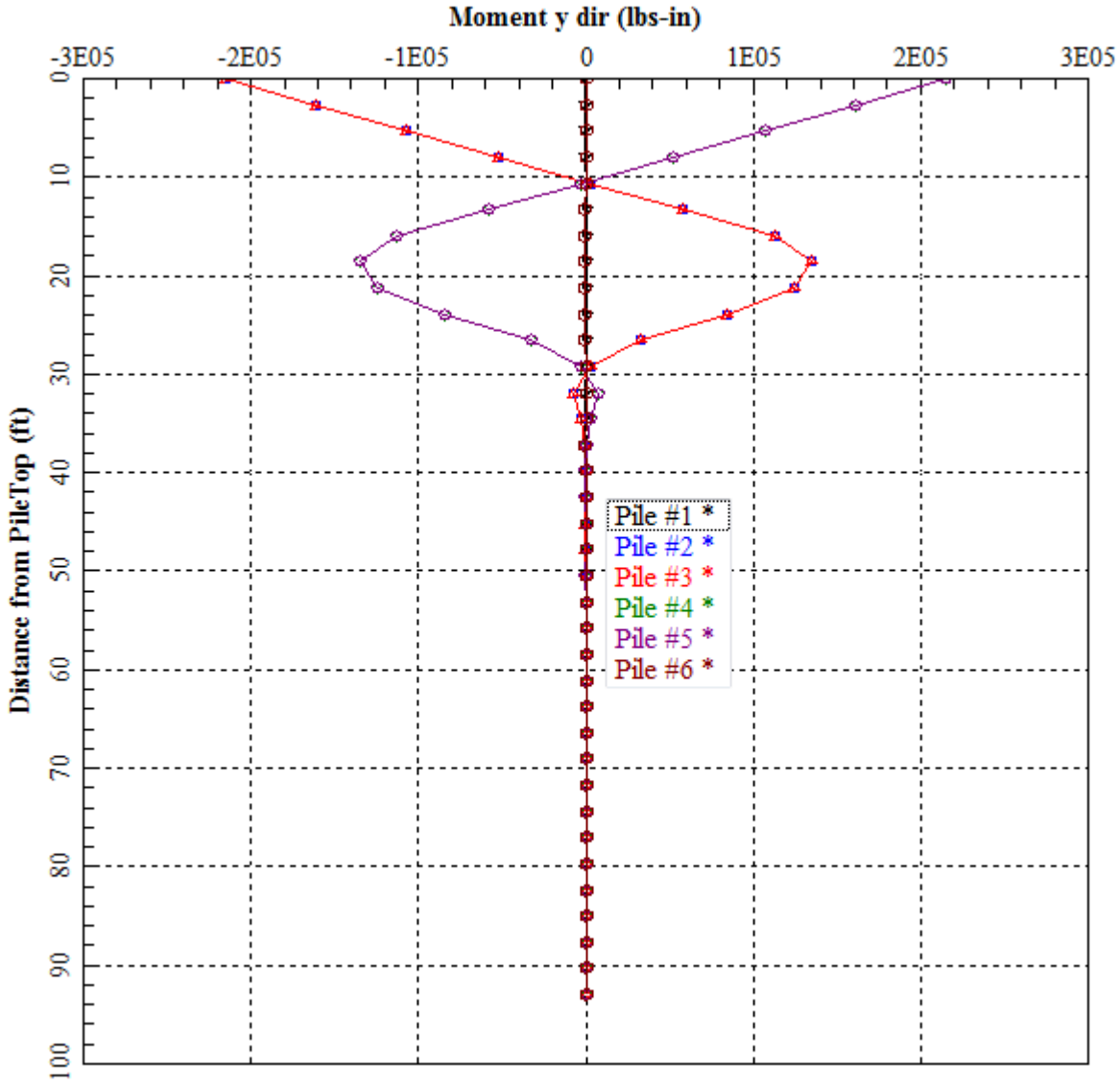


Figure 4.57 Sample graphics screen of Moment y dir vs. Distance command under the Graphs menu

4.5.18 Graphs - Shear y dir vs. Distance

This option provides a plot of shear versus distance from pile top on the local y-axis of each individual pile in a 3-dimensional pile group. Notice that the shear forces are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

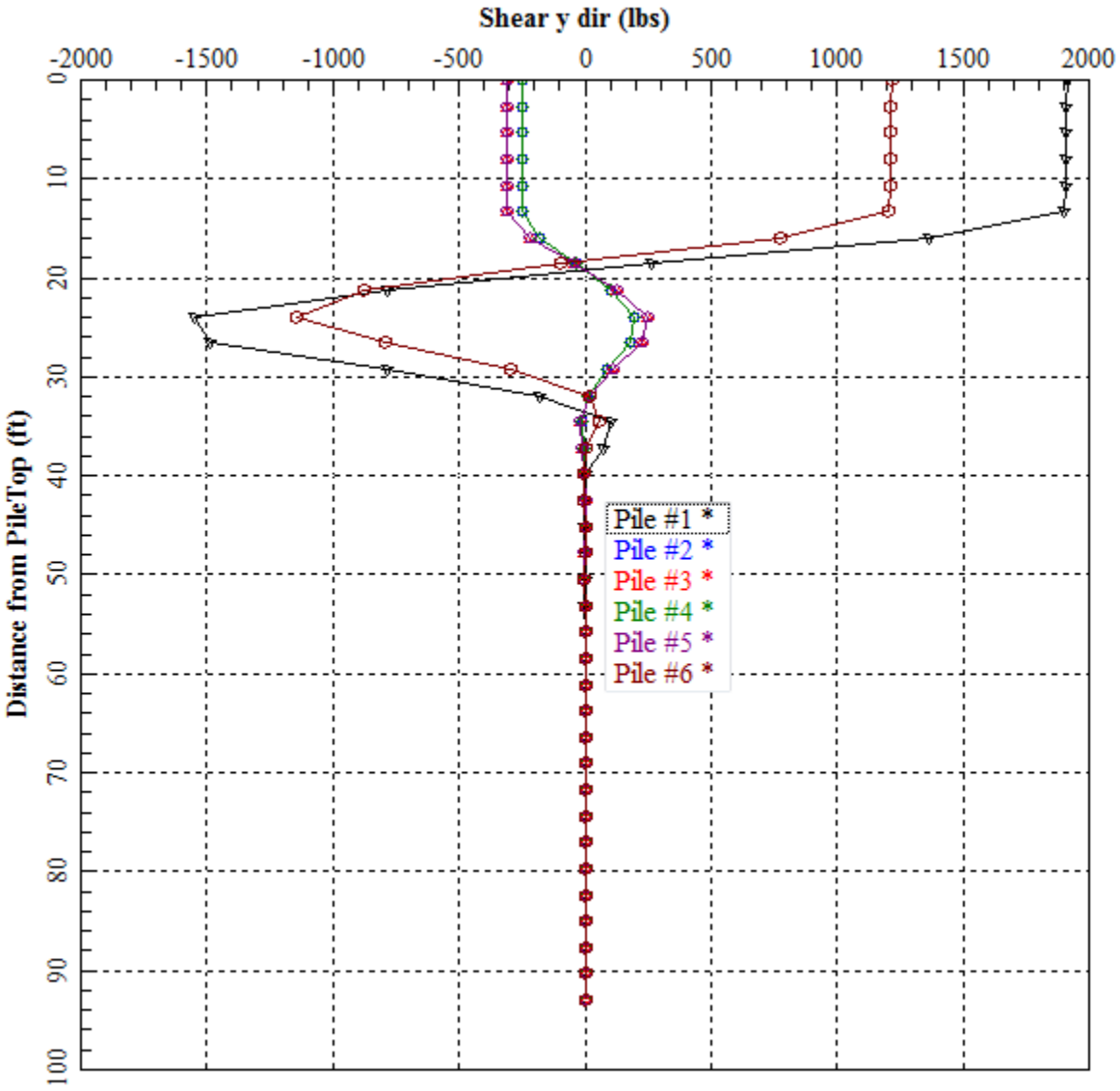


Figure 4.58 Sample graphics screen of Shear y dir vs. Distance command under the Graphs menu

4.5.19 Graphs - Shear z dir vs. Distance

This option provides a plot of shear versus distance from pile top on the local z-axis of each individual pile in a 3-dimensional pile group. Notice that the shear forces are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

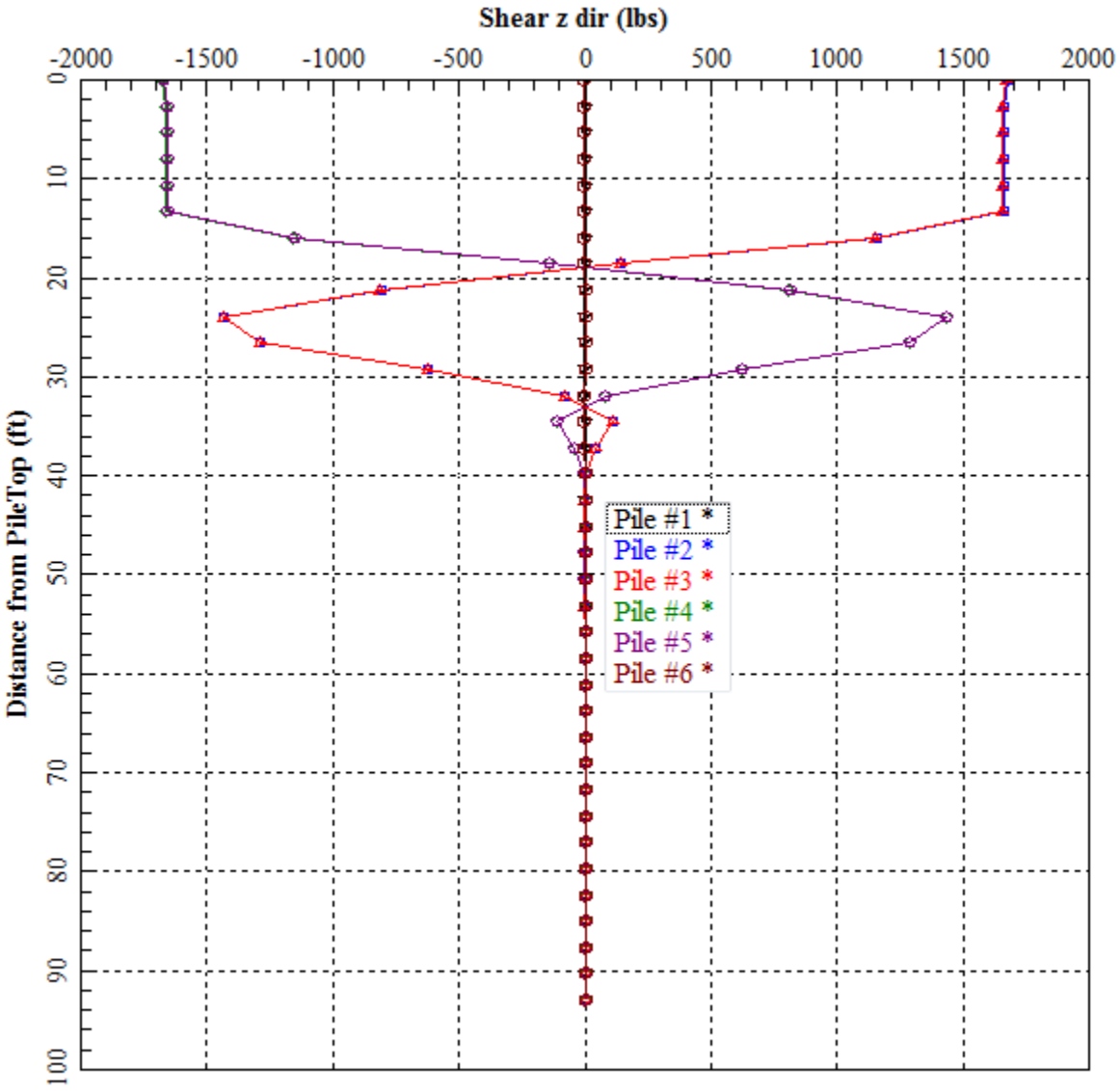


Figure 4.59 Sample graphics screen of Shear z dir vs. Distance command under the Graphs menu

4.5.20 Graphs - Soil Reaction y dir vs. Distance

This option provides a plot of lateral soil resistance versus distance from pile top on the local y-axis of each individual pile in a 3-dimensional pile group. Notice that the soil reactions are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

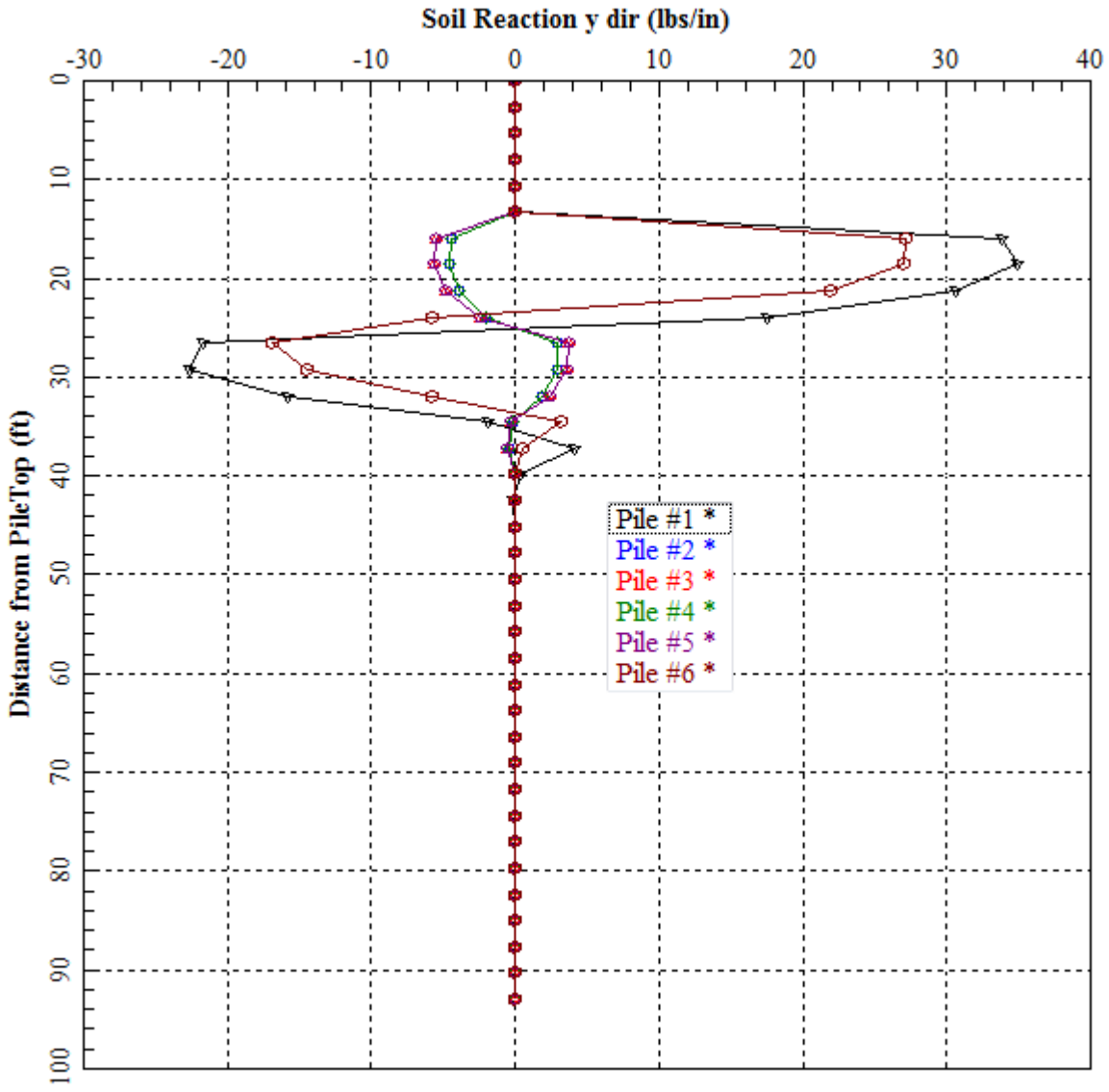


Figure 4.60 Sample graphics screen of Soil Reaction y dir vs. Distance command under the Graphs menu

4.5.21 Graphs - Soil Reaction z dir vs. Distance

This option provides a plot of lateral soil resistance versus distance from pile top on the local z-axis of each individual pile in a 3-dimensional pile group. Notice that the soil reactions are provided in values for the local axis of each pile (which may be different for piles that have different batter and rotation angles).

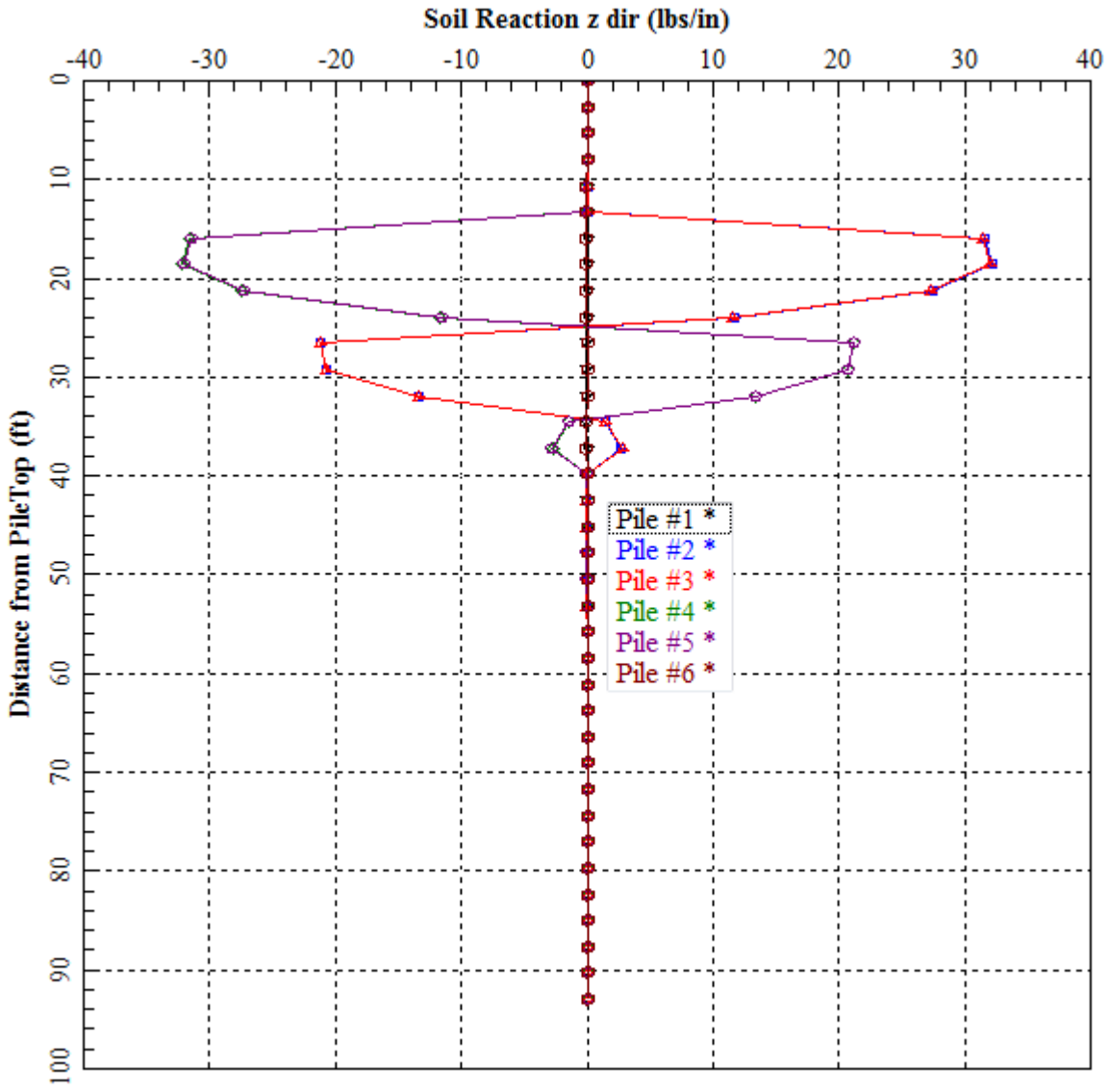


Figure 4.61 Sample graphics screen of Soil Reaction z dir vs. Distance command under the Graphs menu

4.5.22 Graphs – Bending Stiffness z dir vs. Distance

This option provides a plot of pile bending stiffness about the local z-axis versus distance from pile top of each individual pile in a 3-dimensional pile group. Notice that the bending stiffness are provided in values along the local axis of each pile.

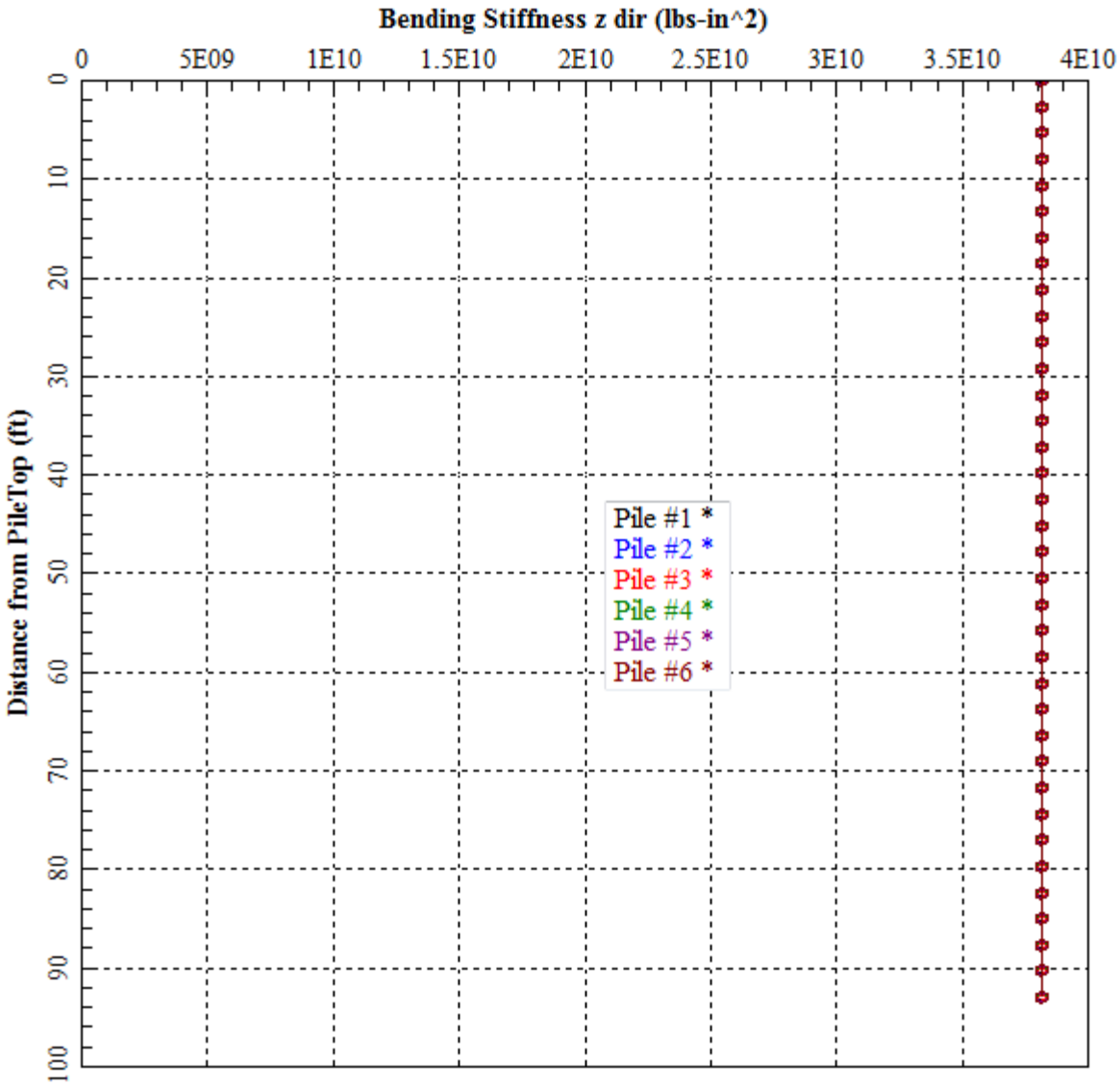


Figure 4.62 Sample graphics screen of Bending Stiffness z dir vs. Distance command under the Graphs menu

4.5.23 Graphs – Bending Stiffness y dir vs. Distance

This option provides a plot of bending stiffness about the local y-axis versus distance from pile top of each individual pile in a 3-dimensional pile group. Notice that the bending stiffness are provided in values along the local axis of each pile.

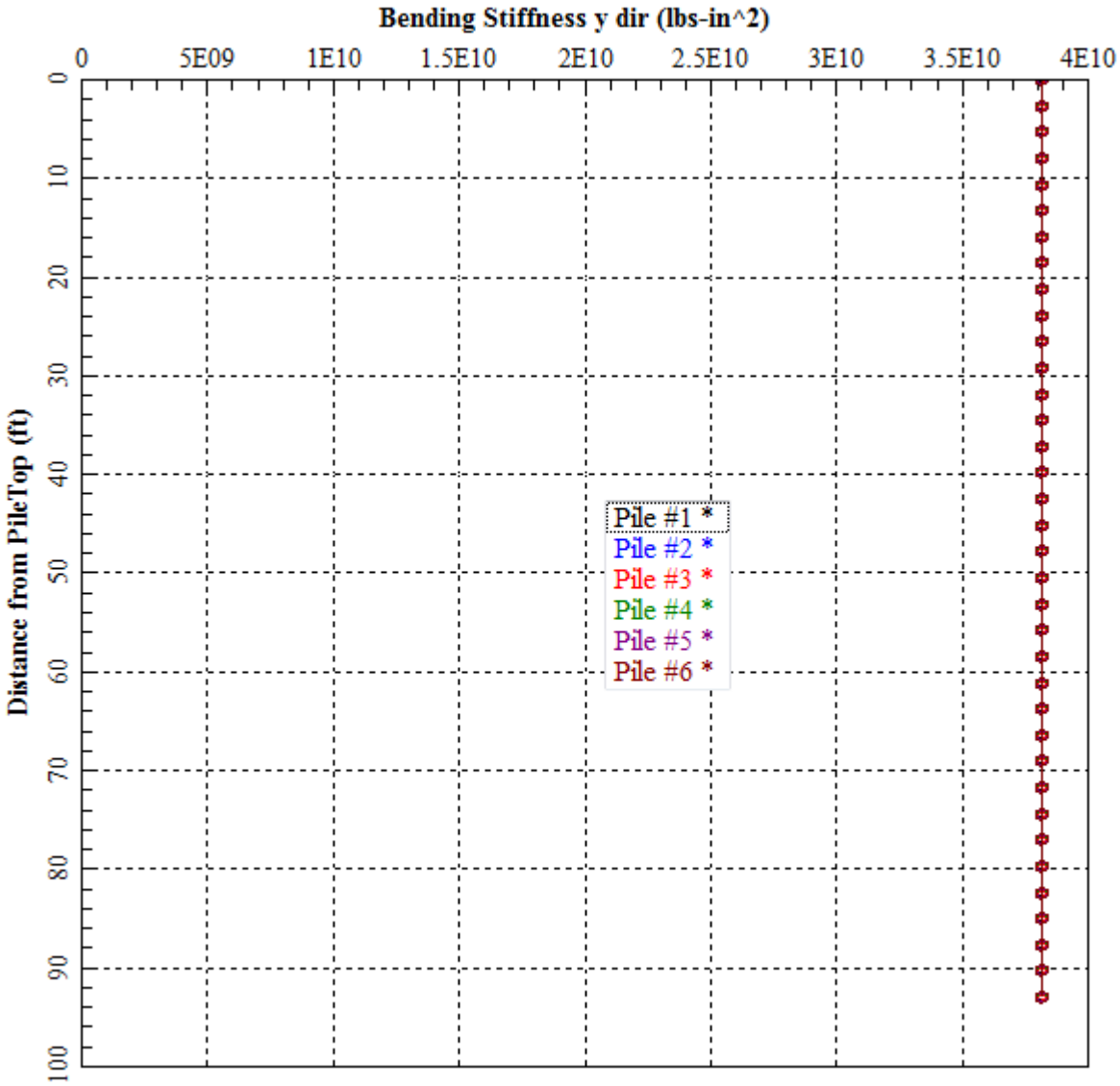


Figure 4.63 Sample graphics screen of Bending Stiffness y dir vs. Distance command under the Graphs menu

4.5.24 Graphs – Def. y, Mom. y, Shear z vs. Distance

This option provides a plot (in the same graph) of the lateral deflection y, bending moment z and shear y versus distance from pile top of each individual pile in a 3-dimensional pile group. Notice that the deflection, moment and shear are provided in values for the local axis of each pile (which is different for piles that have different batter).

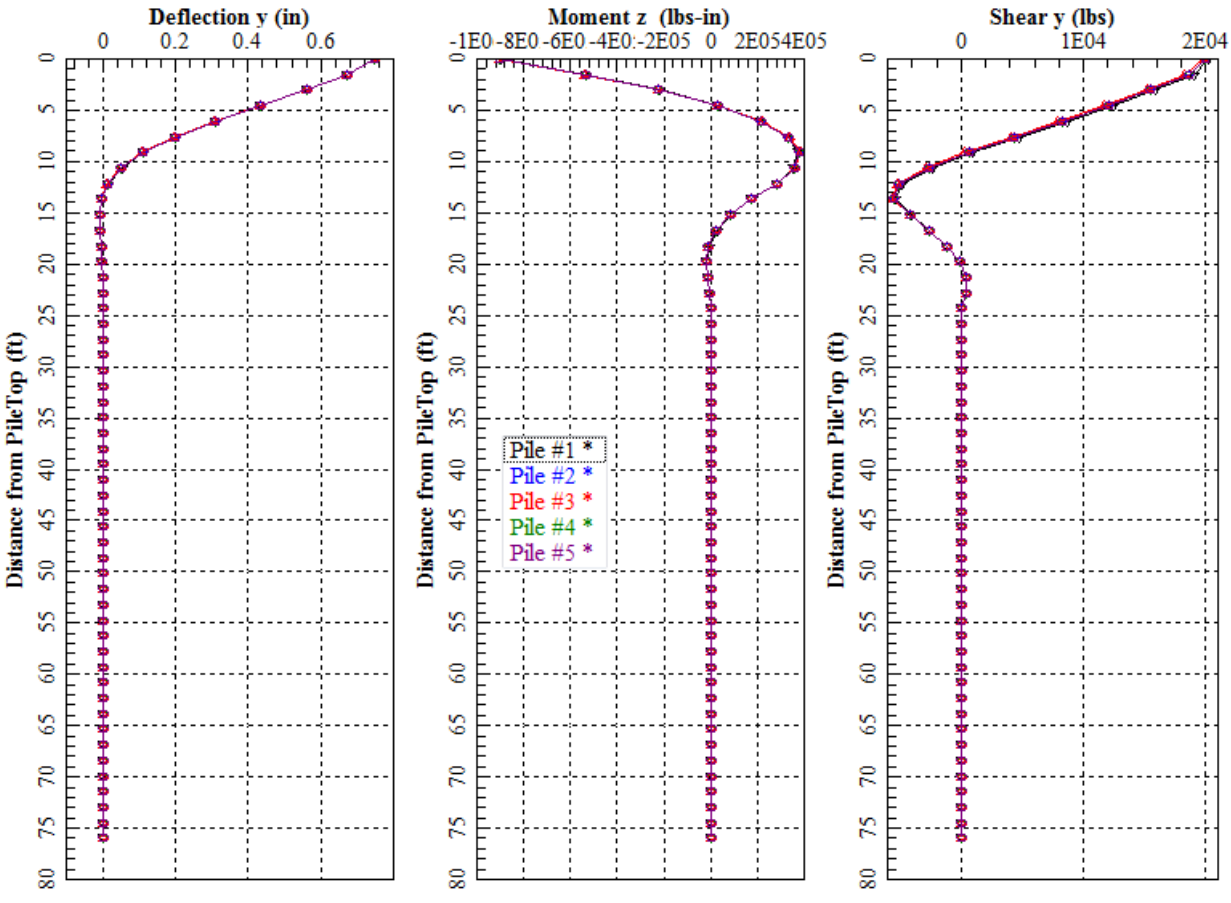


Figure 4.64 Sample graphics screen of Def. y, Mom. z, Shear y vs. Distance command under the Graphs menu

4.5.25 Graphs – Def. z, Mom. y, Shear z vs. Distance

This option provides a plot (in the same graph) of the lateral deflection z, bending moment y and shear z versus distance from pile top of each individual pile in a 3-dimensional pile group. Notice that the deflection, moment and shear are provided in values for the local axis of each pile (which is different for piles that have different batter).

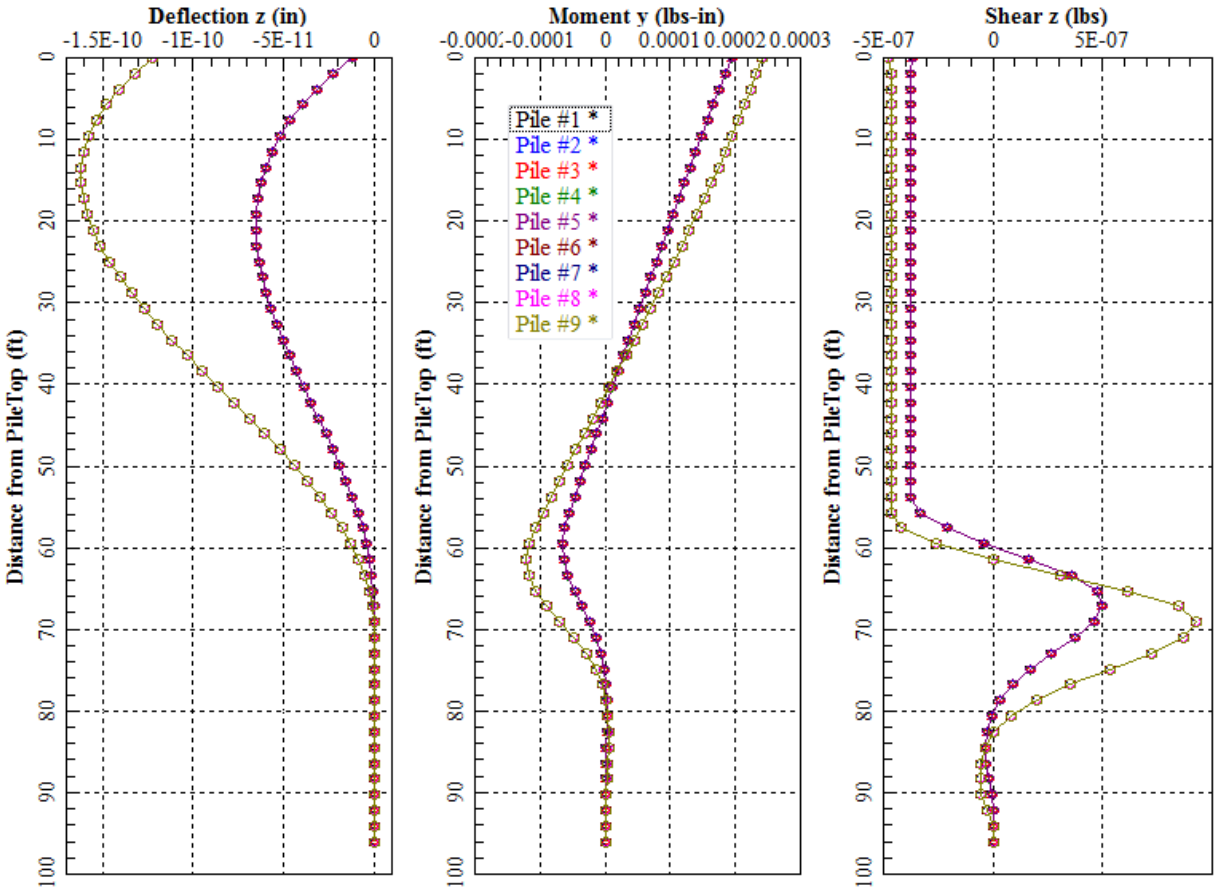


Figure 4.65 Sample graphics screen of Def. z, Mom. y, Shear z vs. Distance command under the Graphs menu

4.5.26 Graphs – Deflection x Dir vs. Distance

This option provides a plot of deflection along the pile versus distance from pile top of each individual pile in a 3-dimensional pile group. If the user select Include Axial Variation along Piles, the variation may be displayed along depth of the pile; otherwise the plot may be constant along the pile and equal to the axial displacement at each pile head.

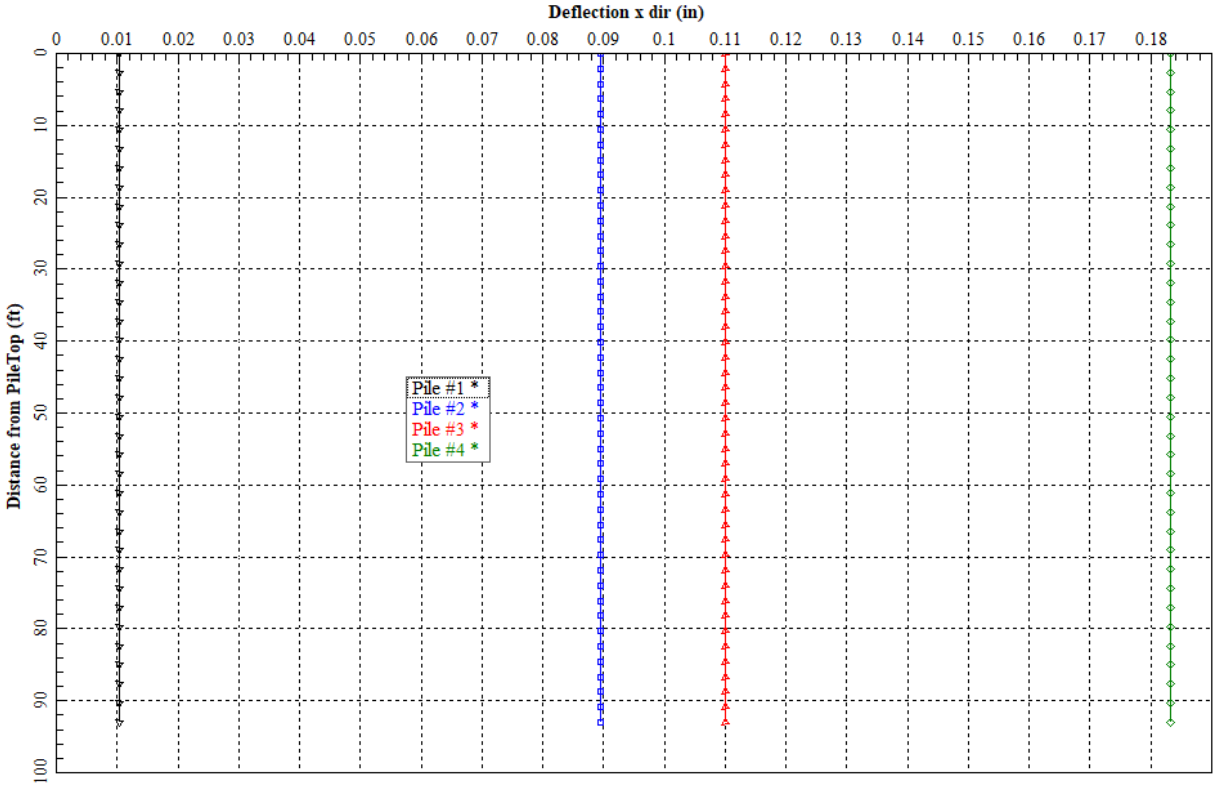


Figure 4.66 Sample graphics screen of Deflection x Dir vs. Distance command under the Graphs menu

4.5.27 Graphs – Axial Force vs. Distance

This option provides a plot of axial force along the pile versus distance from pile top of each individual pile in a 3-dimensional pile group. If the user select Include Axial Variation along Piles, the variation may be displayed along depth of the pile; otherwise the plot may be constant along the pile and equal to the axial force at each pile head.

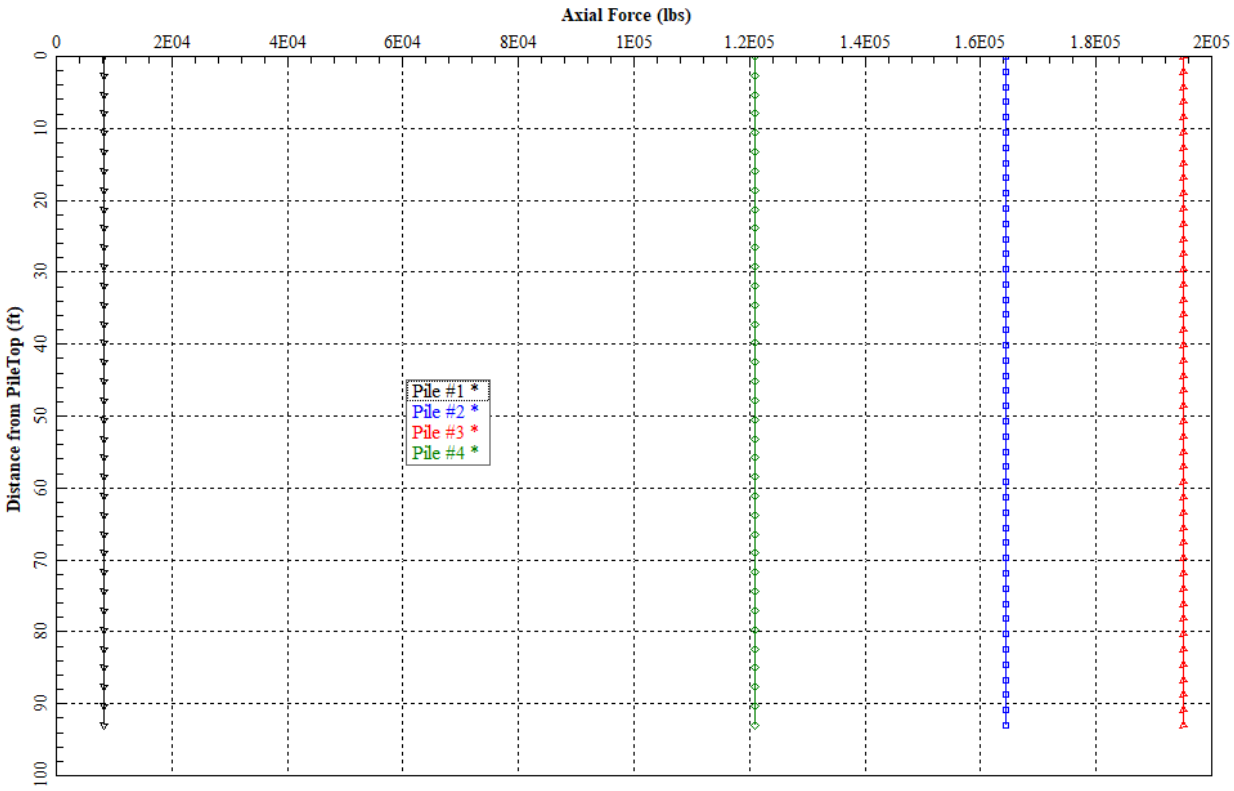


Figure 4.67 Sample graphics screen of Def. z, Mom. y, Shear z vs. Distance command under the Graphs menu

4.5.28 Graphs - Resultant Deflection vs. Distance

This option provides a plot of lateral pile resultant deflection versus distance from pile top of each individual pile in a 3-dimensional pile group. This is calculated for every depth as the square root of the summation of squares from the deflections in the y and z directions.

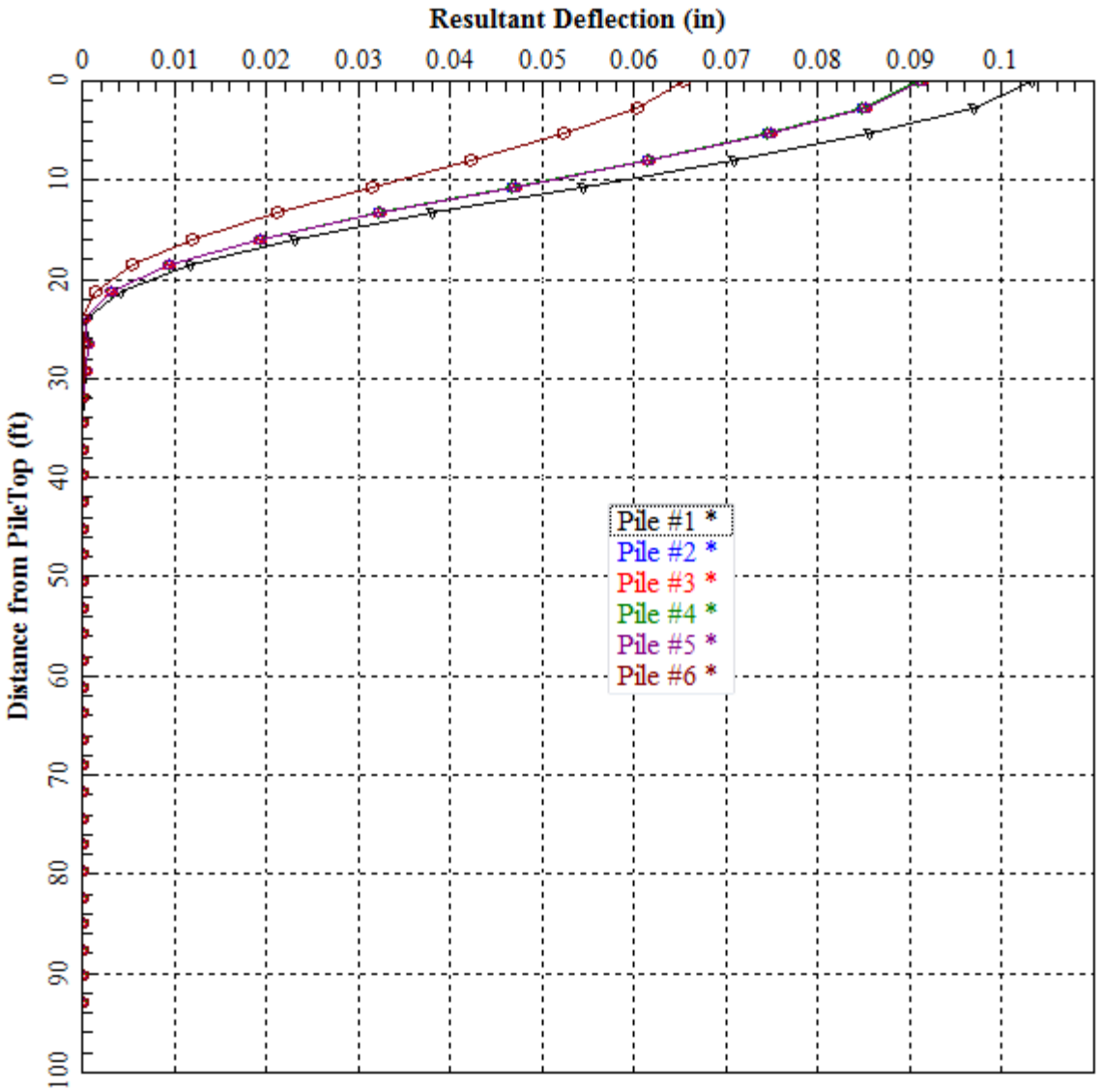


Figure 4.68 Sample graphics screen of Resultant Deflection vs. Distance command under the Graphs menu

4.5.29 Graphs - Resultant Moment vs. Distance

This option provides a plot of resultant bending moment versus distance from pile top of each individual pile in a 3-dimensional pile group. This is calculated for every depth as the square root of the summation of squares from the moments along the y and z directions.

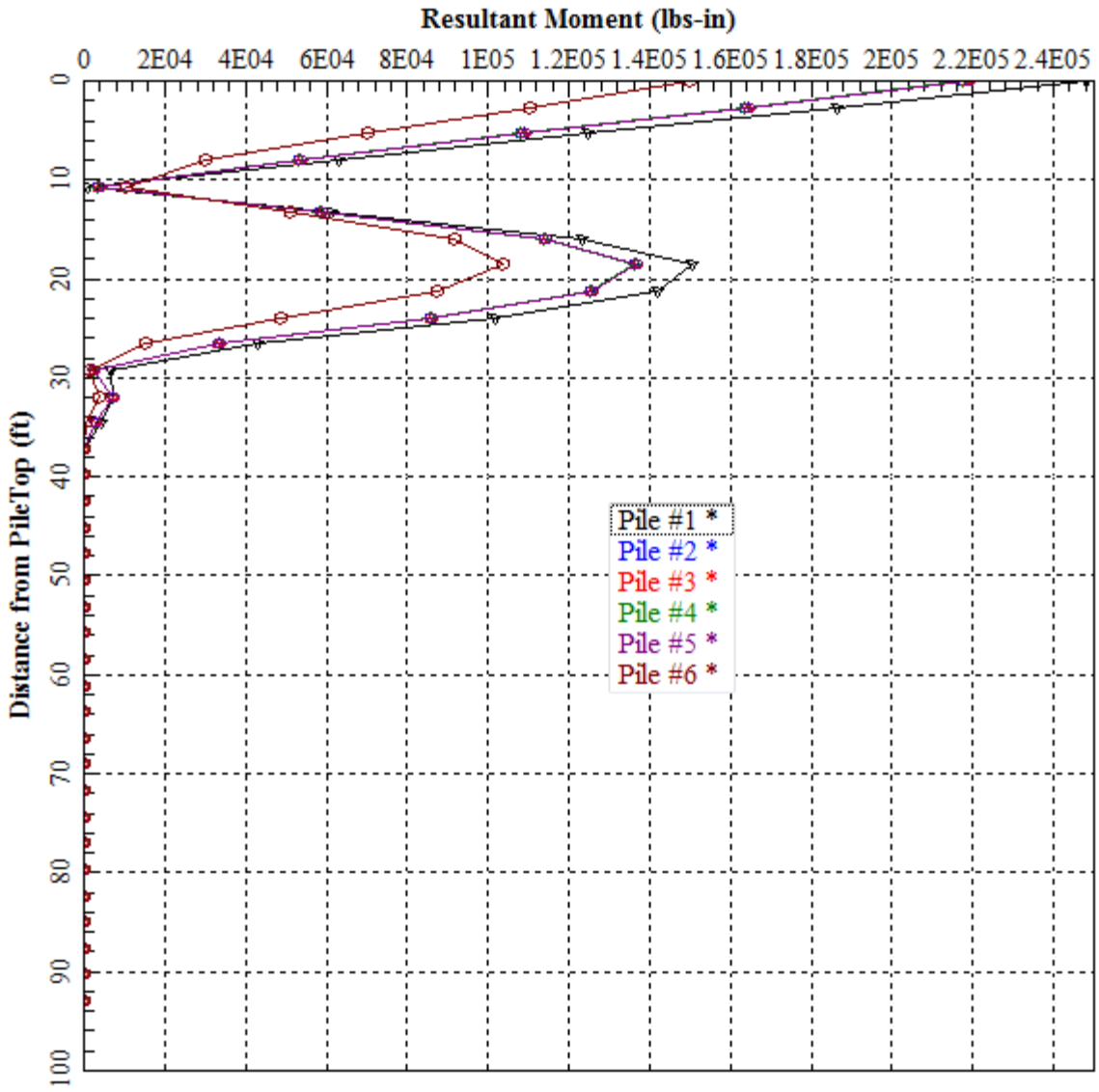


Figure 4.69 Sample graphics screen of Resultant Moment vs. Distance command under the Graphs menu

4.5.30 Graphs - Resultant Shear vs. Distance

This option provides a plot of resultant shear versus distance from pile top of each individual pile in a 2-dimensional pile group. This is calculated for every depth as the square root of the summation of squares from the shear forces in the y and z directions.

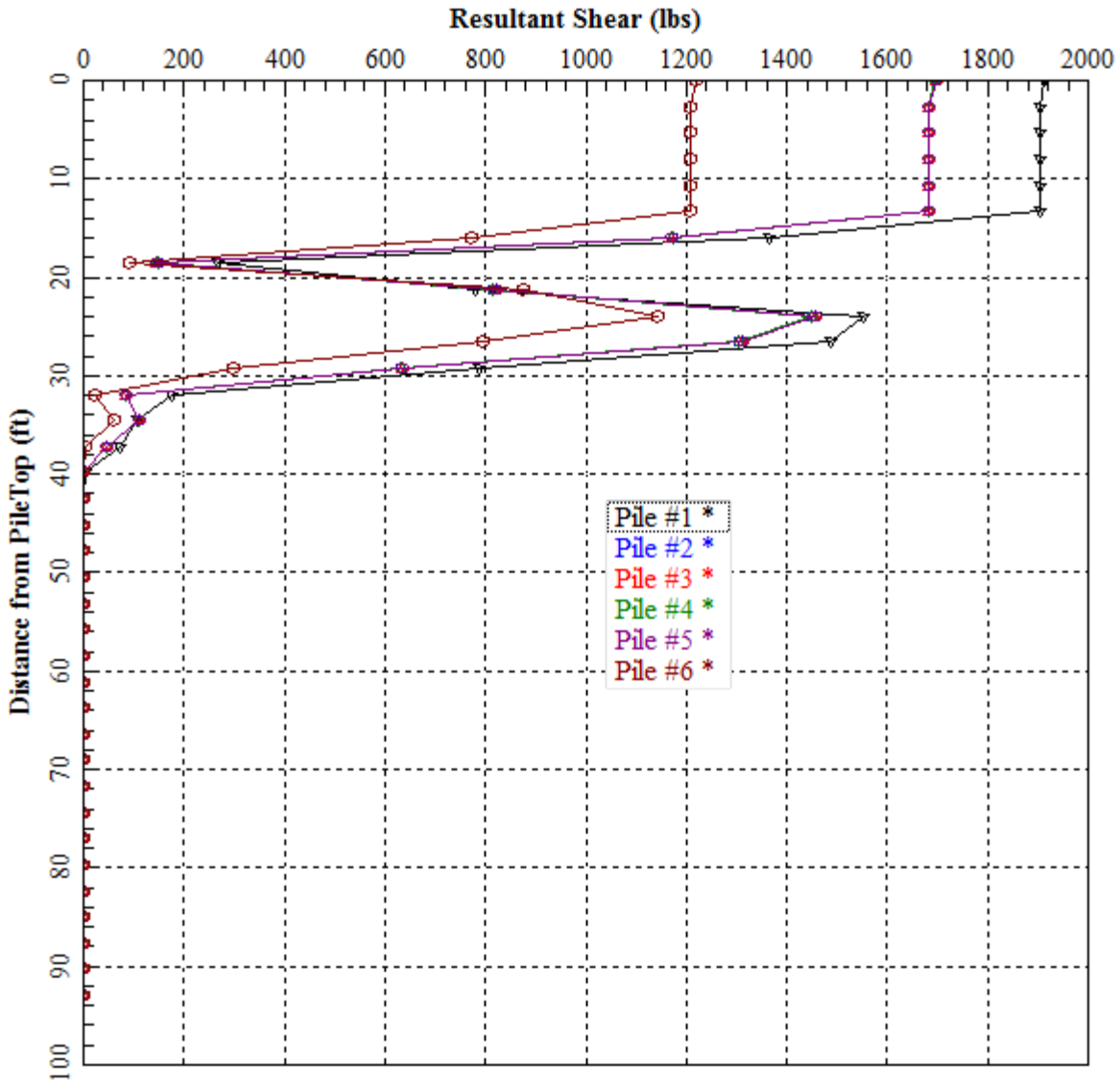


Figure 4.70 Sample graphics screen of Resultant Shear vs. Distance command under the Graphs menu

4.5.31 Graphs - Resultant Soil Reaction vs. Distance

This option provides a plot of resultant lateral soil resistance versus distance from pile top of each individual pile in a 3-dimensional pile group. This is calculated for every depth as the square root of the summation of squares from the soil reactions in the y and z directions.

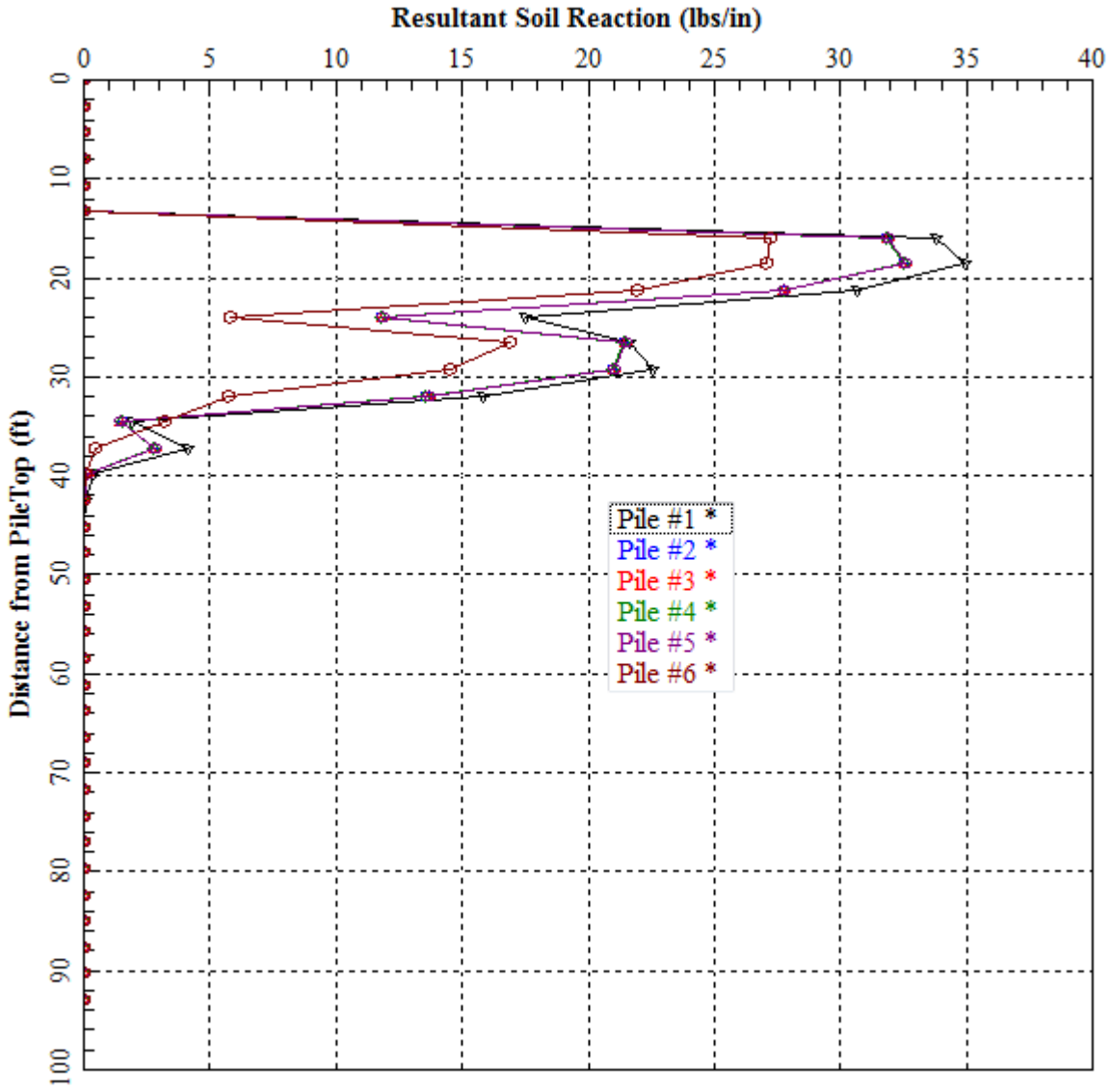


Figure 4.71 Sample graphics screen of Resultant Soil Reaction vs. Distance command under the Graphs menu

4.5.32 Graphs – Res. Def., Mom., Shear vs. Distance

This option provides a plot (in the same graph) of the resultant lateral deflection, resultant bending moment and resultant shear versus distance from pile top of each individual pile in a 3-dimensional pile group.

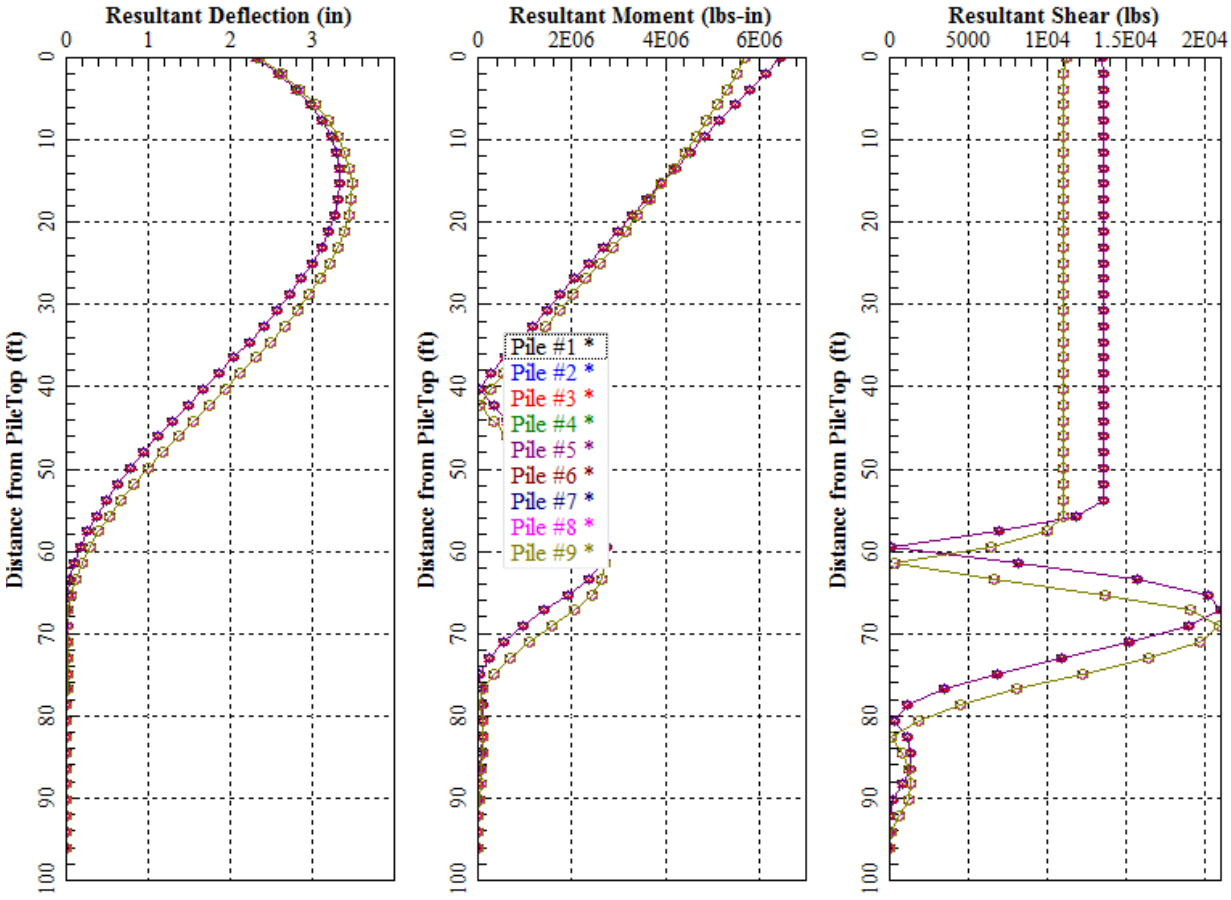


Figure 4.72 Sample graphics screen of Def. y, Mom. z, Shear y vs. Distance command under the Graphs menu

4.5.33 Graphs - Pile Cap Vertical Load vs. Deflection

This option provides a plot of incremental vertical load versus vertical deflection at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 2D models.

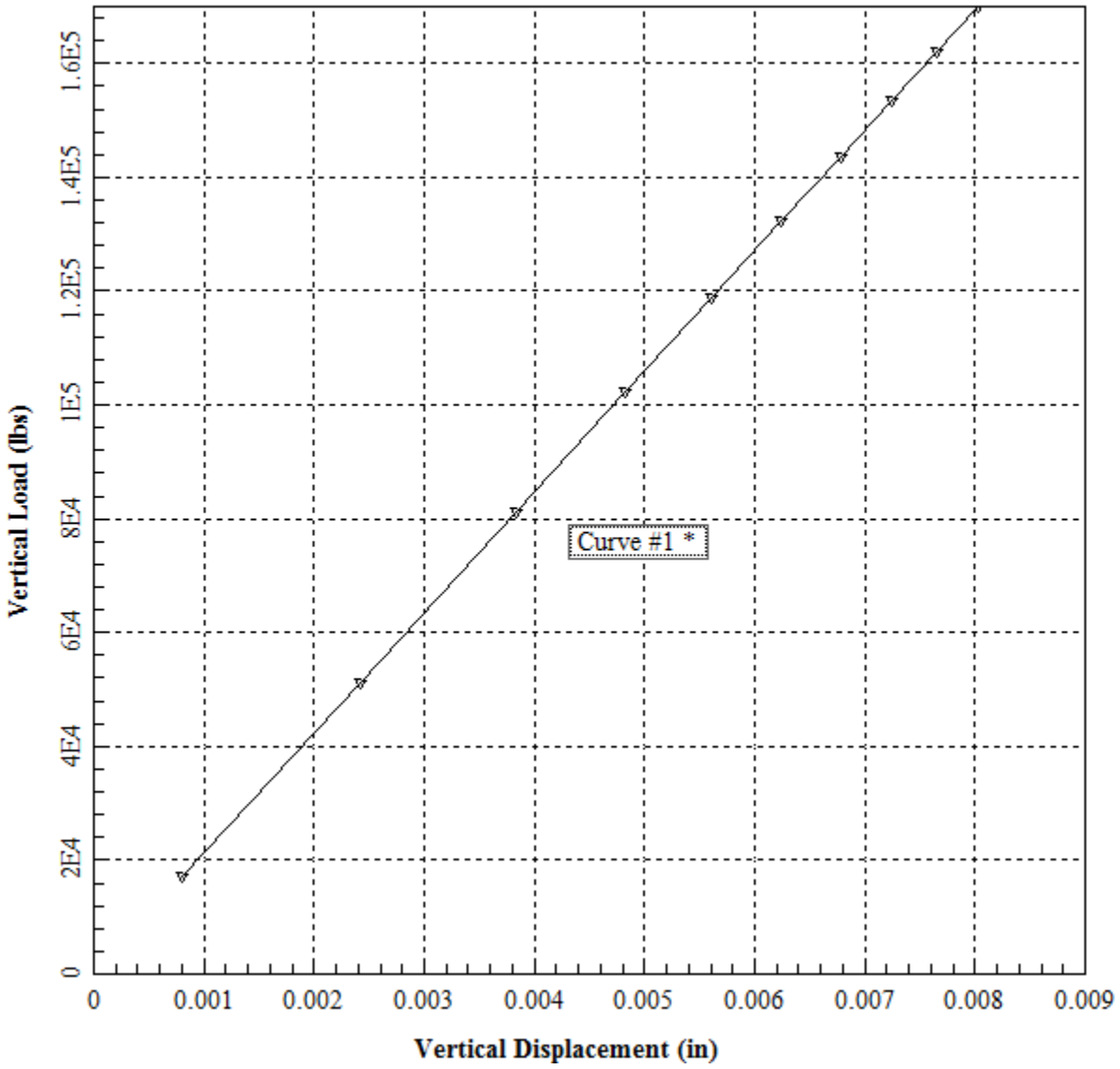


Figure 4.73 Sample graphics screen of Pile Cap Vertical Load vs. Deflection command under the Graphs menu

4.5.34 Graphs - Pile Cap Load vs. Lateral Deflection

This option provides a plot of incremental lateral load versus lateral deflection at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 2D models.

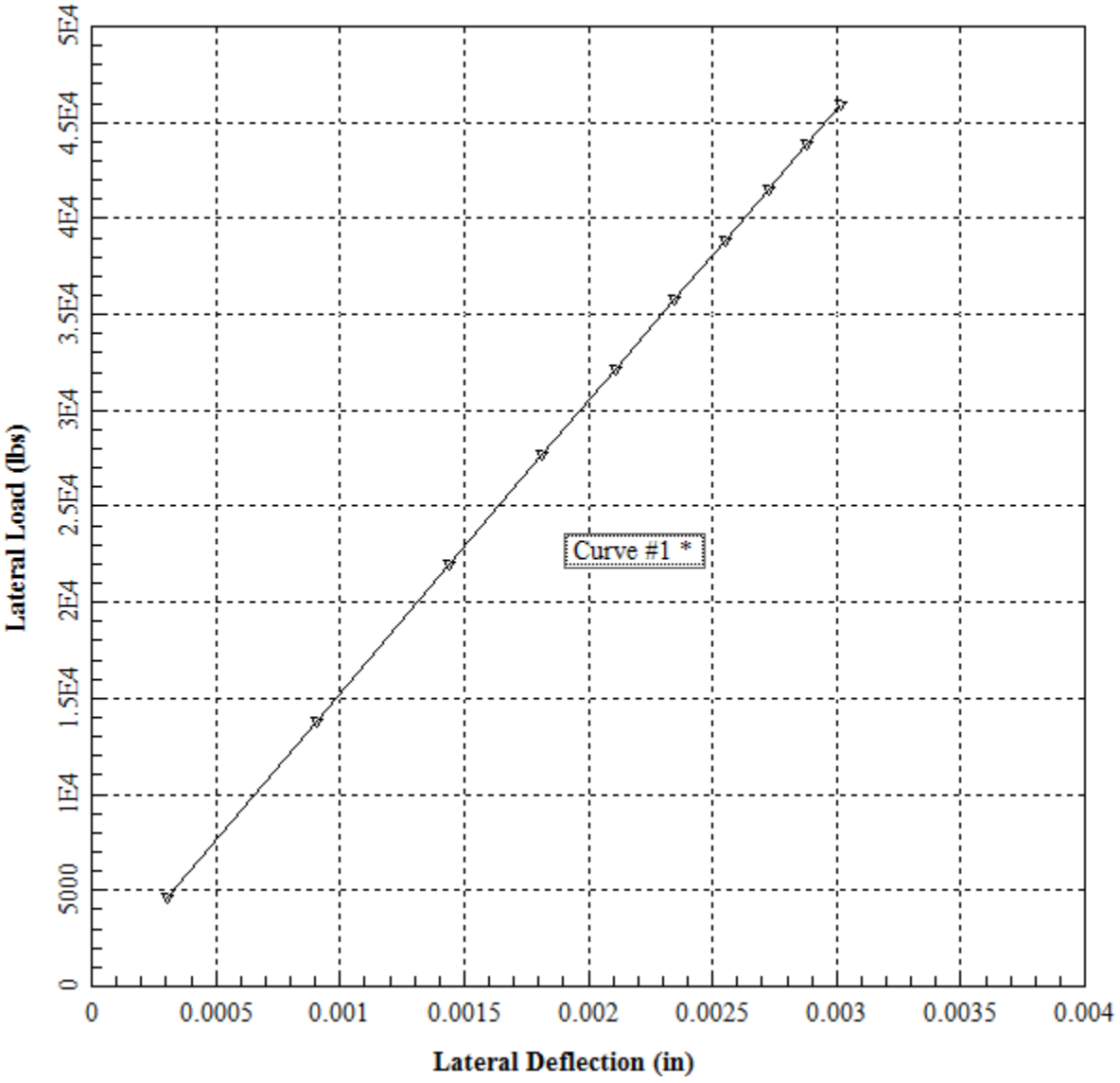


Figure 4.74 Sample graphics screen of Pile Cap Load vs. Lateral Deflection command under the Graphs menu

4.5.35 Graphs - Pile Cap Load vs. Rotation

This option provides a plot of incremental lateral load versus rotation of the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 2D models.

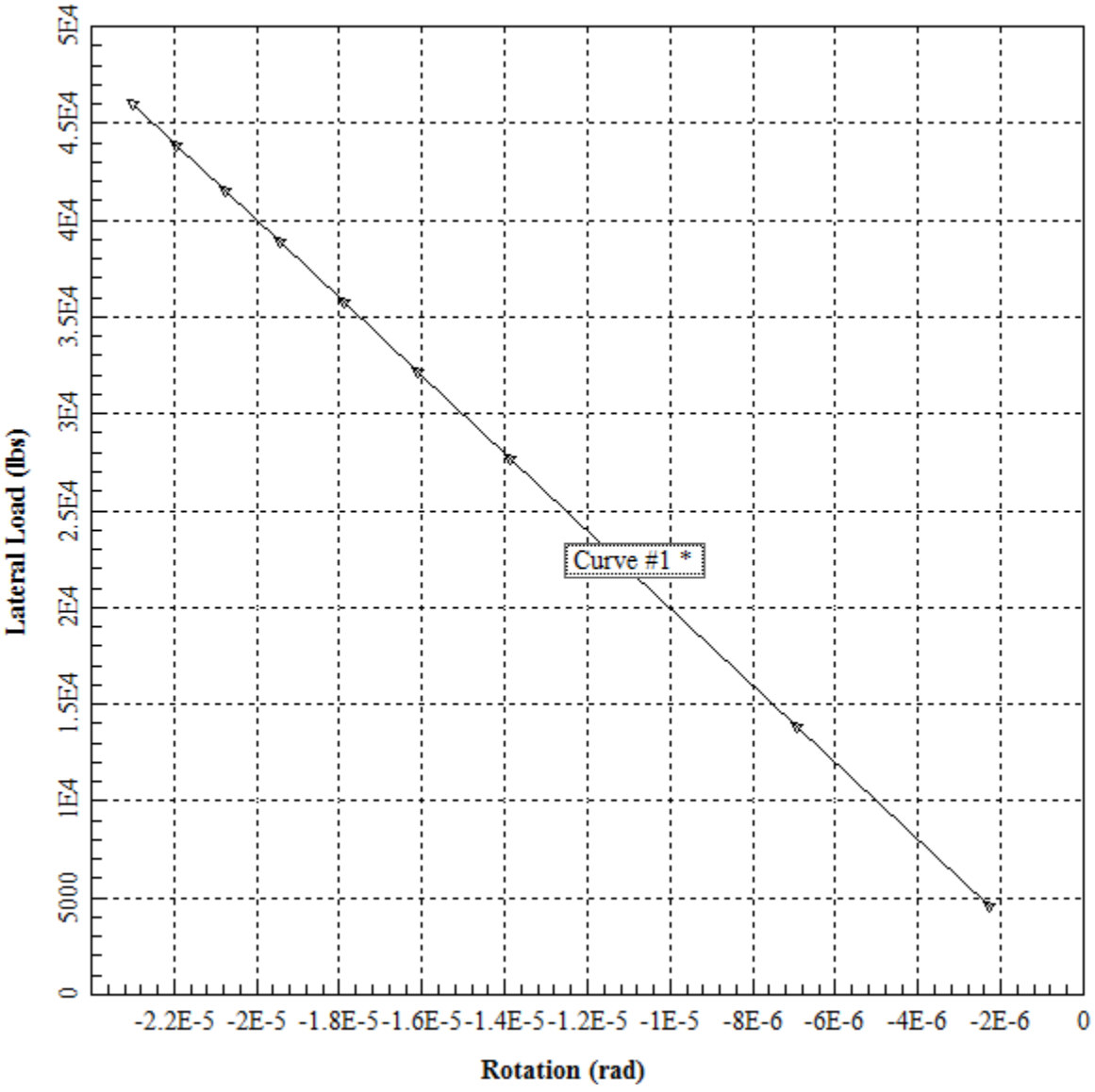


Figure 4.75 Sample graphics screen of Pile Cap Load vs. Rotation command under the Graphs menu

4.5.36 Graphs - Pile Cap Moment vs. Deflection

This option provides a plot of incremental moment versus lateral deflection at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 2D models.

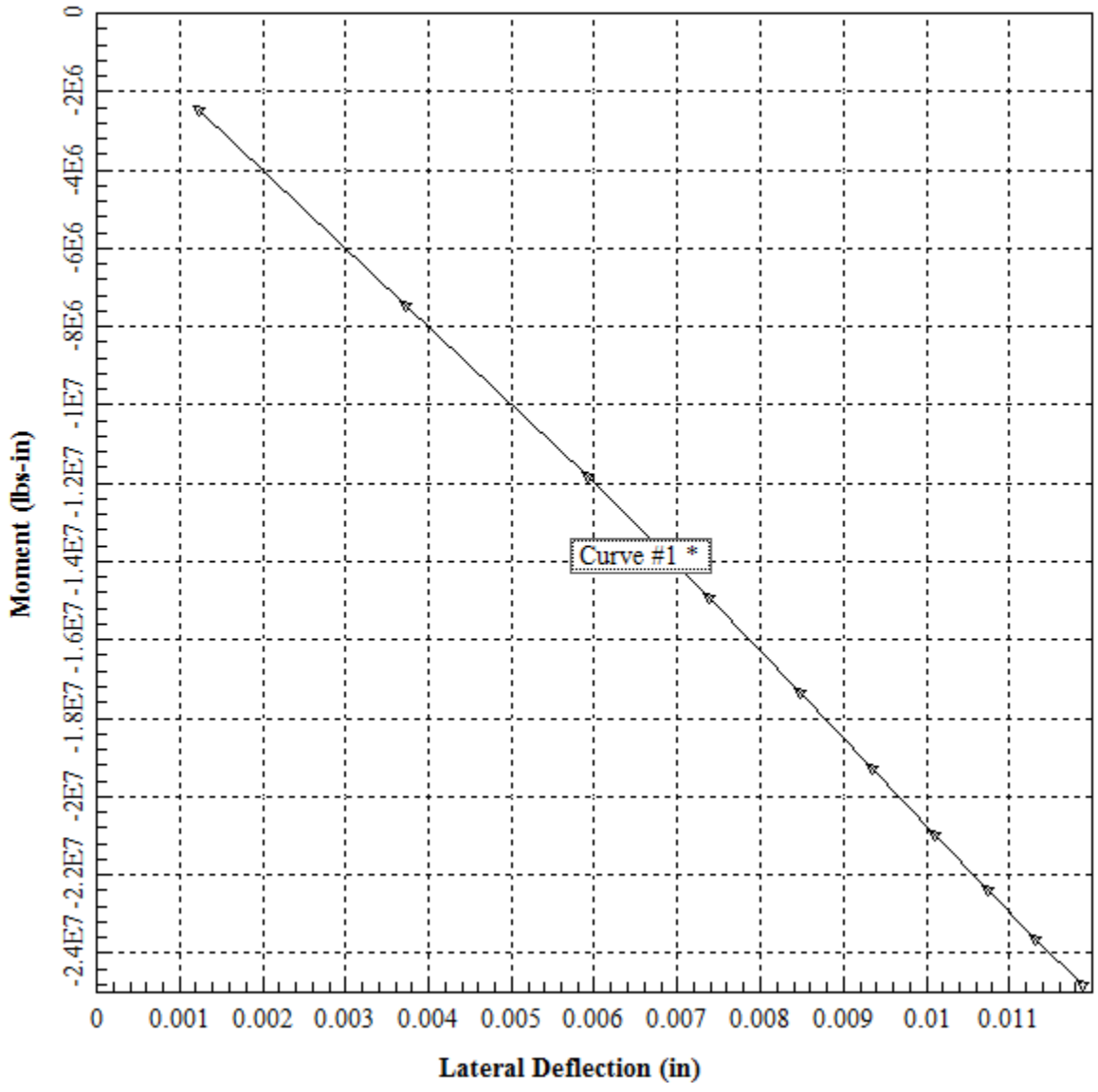


Figure 4.76 Sample graphics screen of Pile Cap Moment vs. Deflection command under the Graphs menu

4.5.37 Graphs - Pile Cap Moment vs. Rotation

This option provides a plot of incremental moment load versus rotation of the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 2D models.

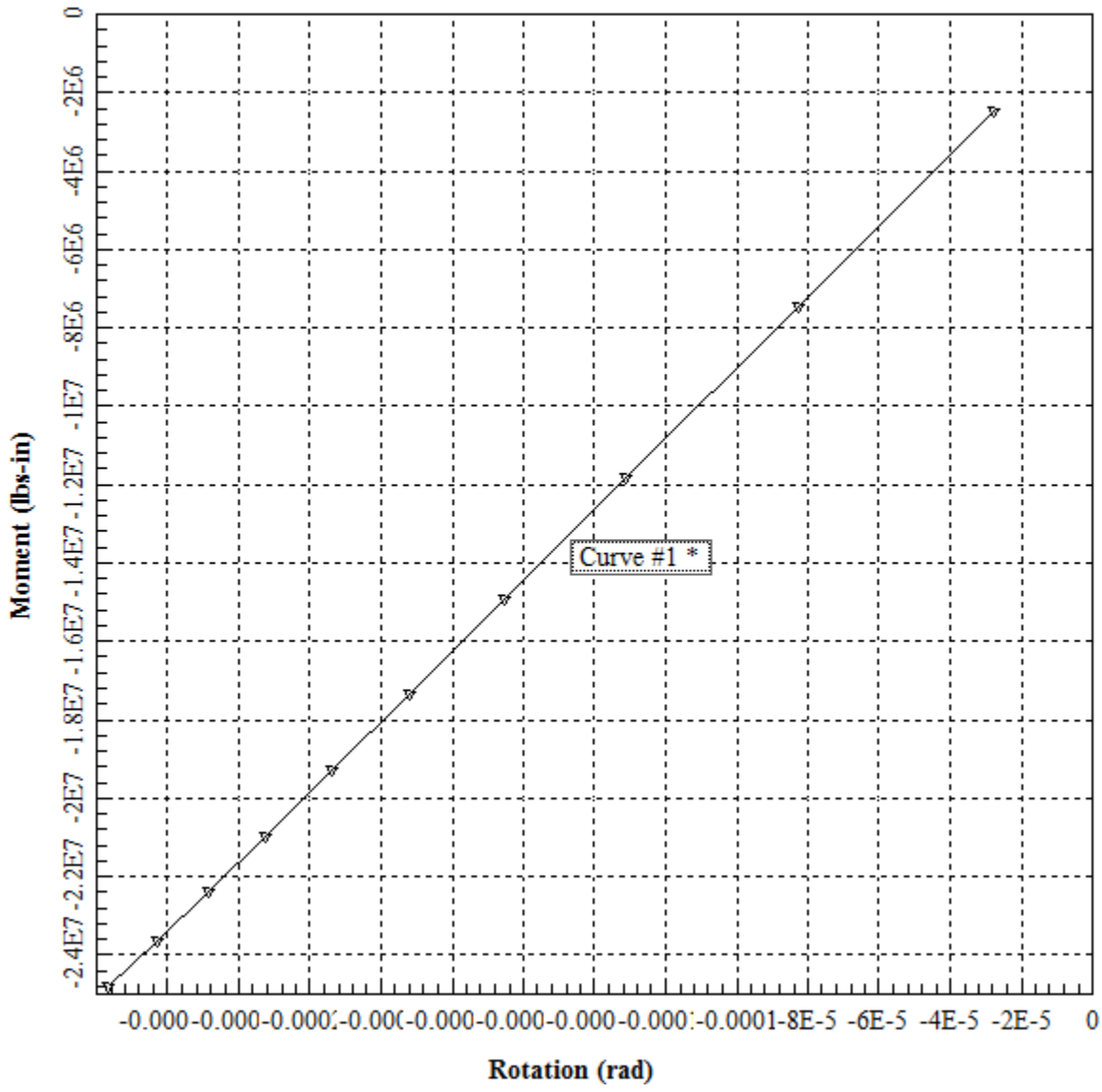


Figure 4.77 Sample graphics screen of Pile Cap Moment vs. Rotation command under the Graphs menu

4.5.38 Graphs - Pile Cap Vertical Load vs. Deflection X

This option provides a plot of incremental vertical load versus vertical deflection at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models.

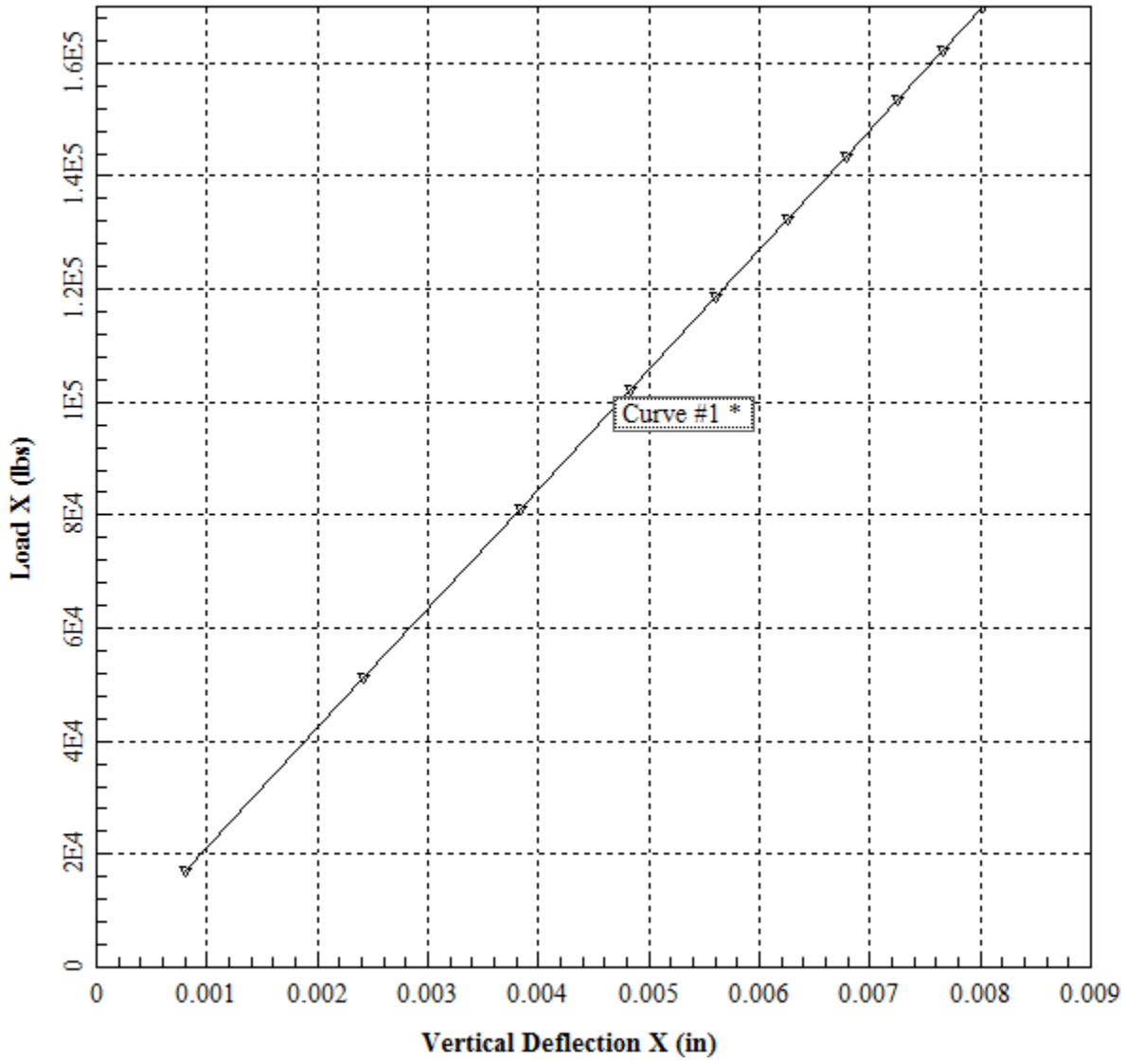


Figure 4.78 Sample graphics screen of Pile Cap Vertical Load vs. Deflection X command under the Graphs menu

4.5.39 Graphs - Pile Cap Lateral Load vs. Deflection Y

This option provides a plot of incremental lateral load versus lateral deflection in the Y-direction at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models.

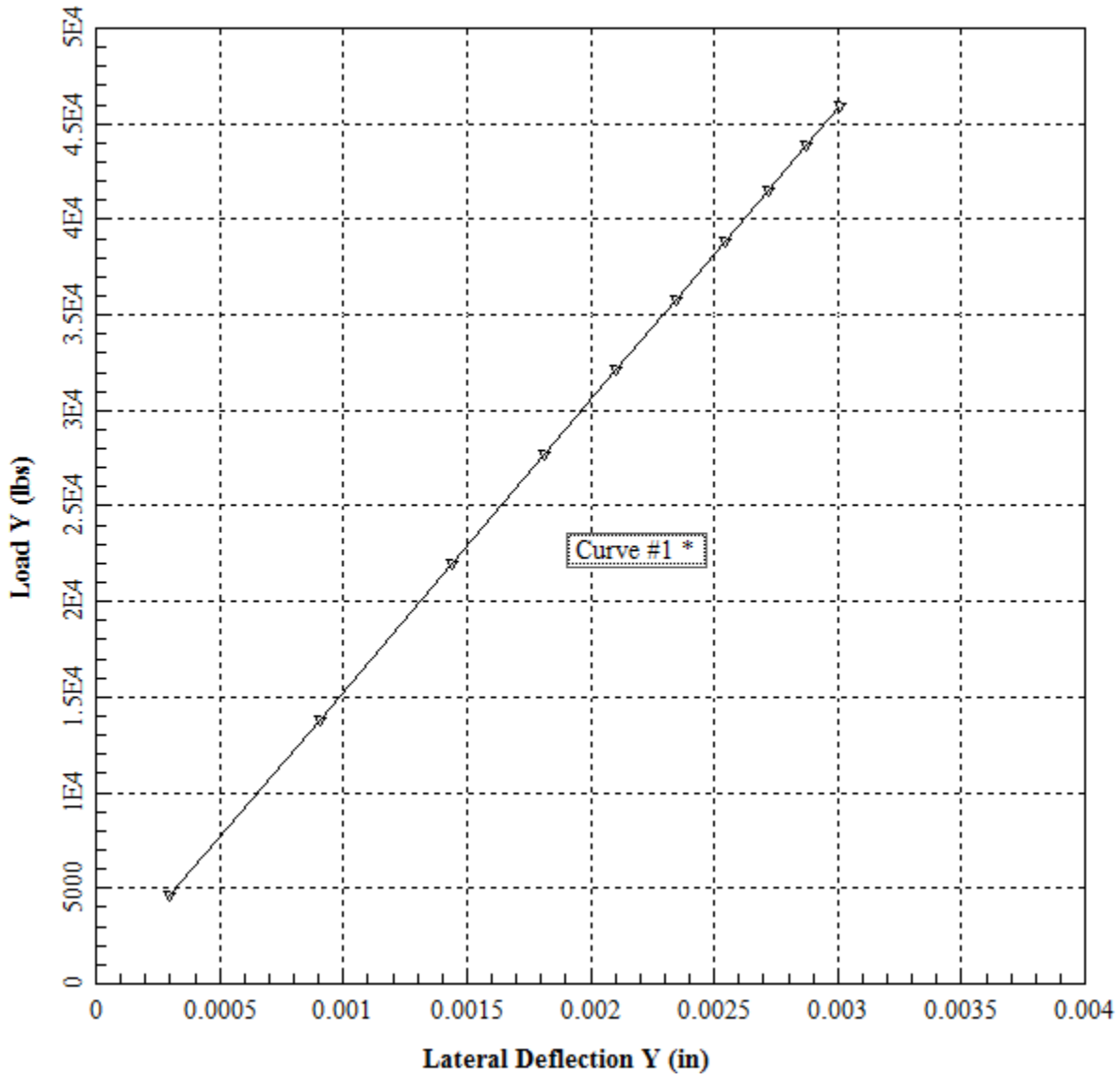


Figure 4.79 Sample graphics screen of Pile Cap Lateral Load vs. Deflection Y command under the Graphs menu

4.5.40 Graphs - Pile Cap Lateral Load vs. Deflection Z

This option provides a plot of incremental lateral load versus lateral deflection in the Z-direction at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models.

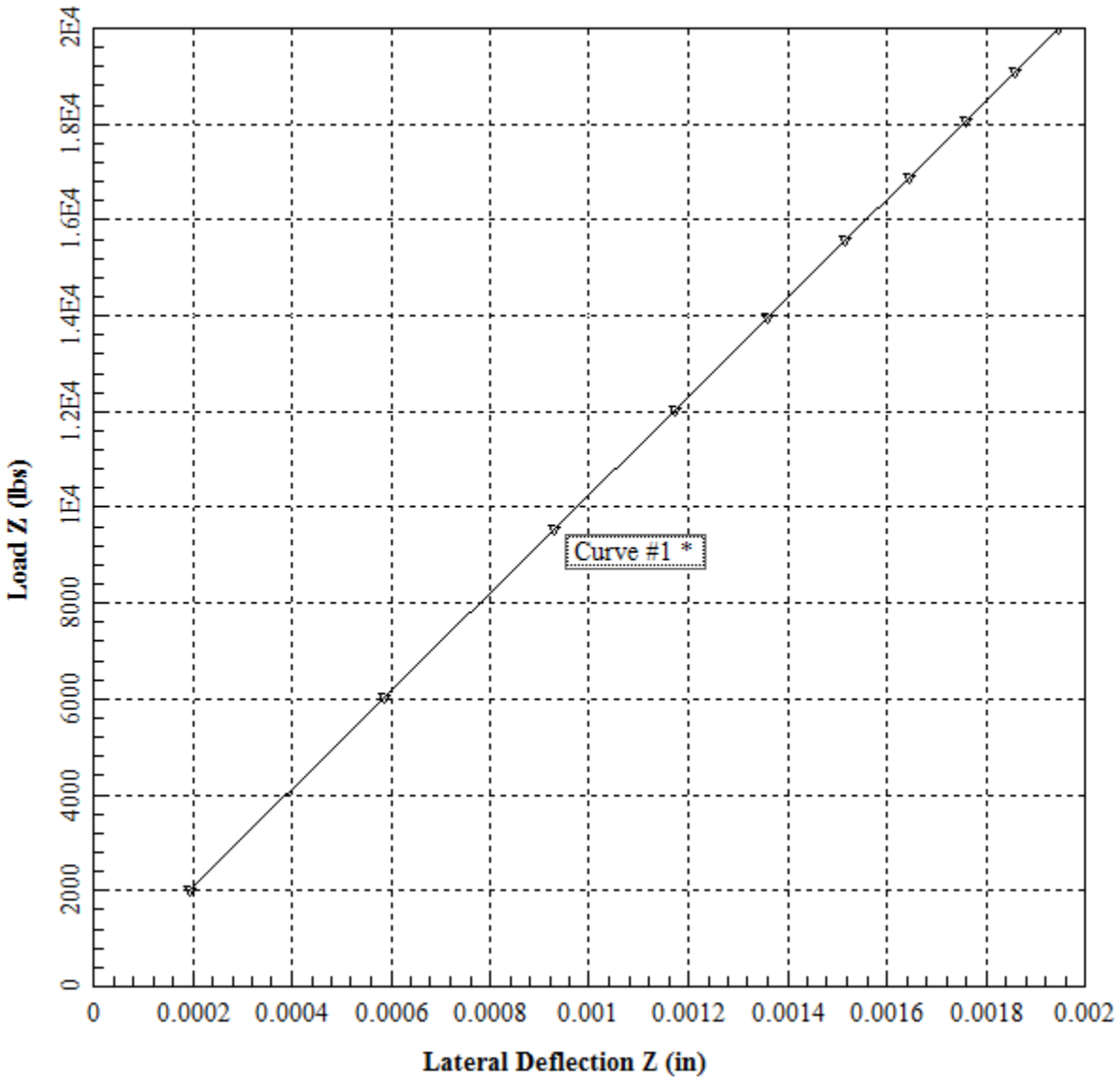


Figure 4.80 Sample graphics screen of Pile Cap Lateral Load vs. Deflection Z command under the Graphs menu

4.5.41 Graphs - Pile Cap Moment vs. Rotation X

This option provides a plot of incremental moment versus rotation about the X-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models

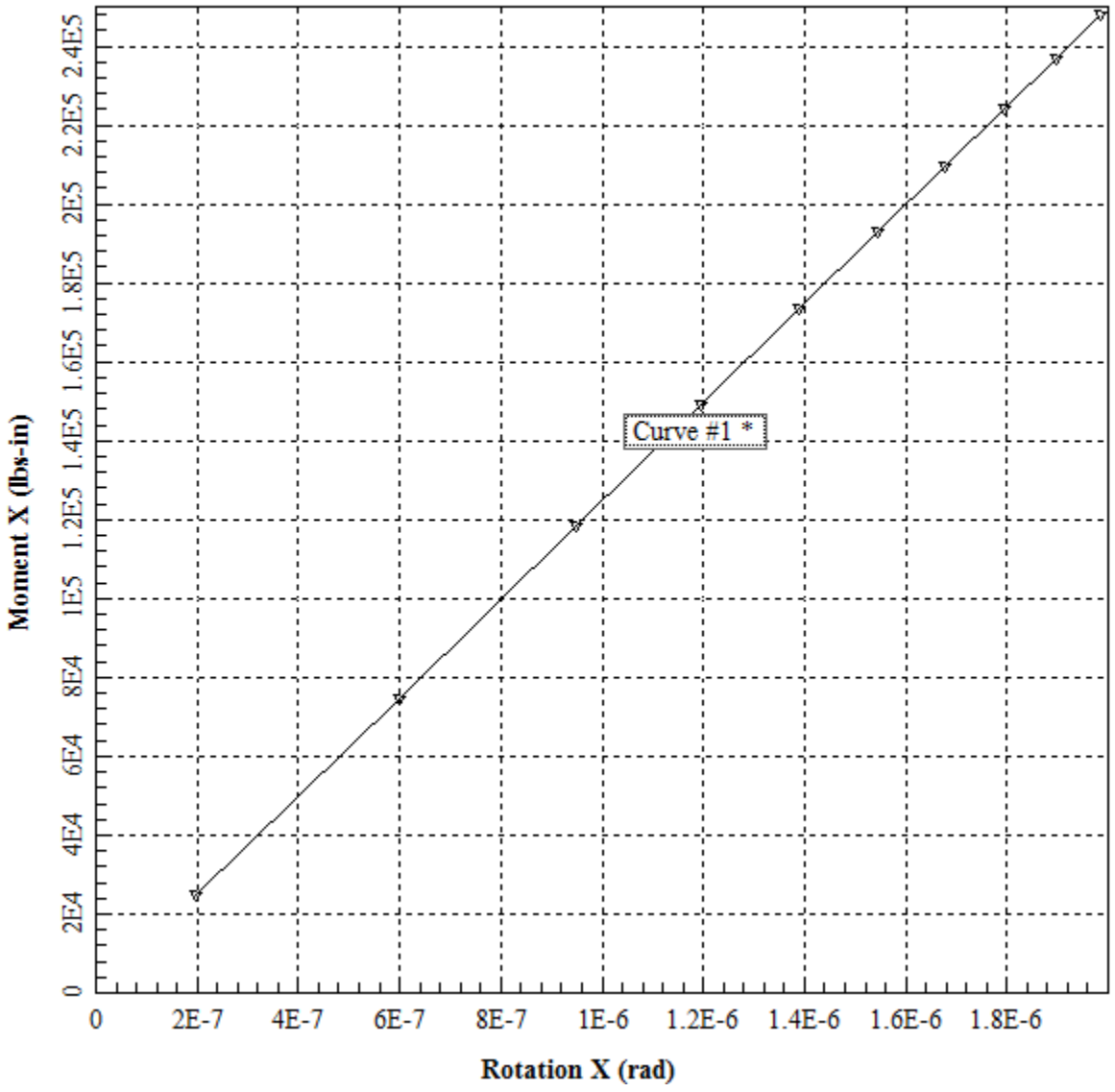


Figure 4.81 Sample graphics screen of Pile Cap Moment vs. Rotation X command under the Graphs menu

4.5.42 Graphs - Pile Cap Moment vs. Rotation Y

This option provides a plot of incremental moment versus rotation about the Y-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models

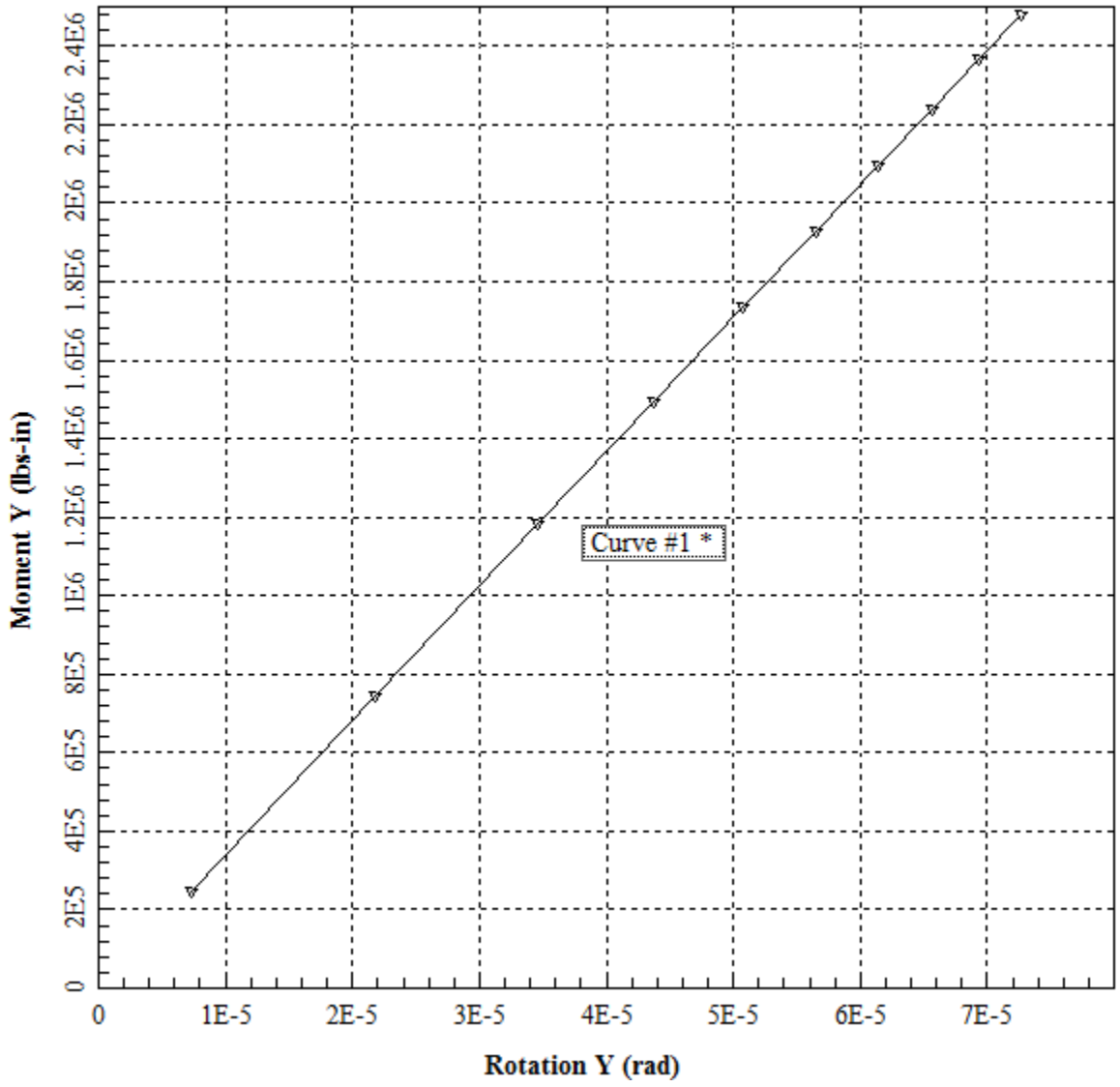


Figure 4.82 Sample graphics screen of Pile Cap Moment vs. Rotation Y command under the Graphs menu

4.5.43 Graphs - Pile Cap Moment vs. Rotation Z

This option provides a plot of incremental moment versus rotation about the Z-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** is specified in the input for the current load case. This option is available for 3D models

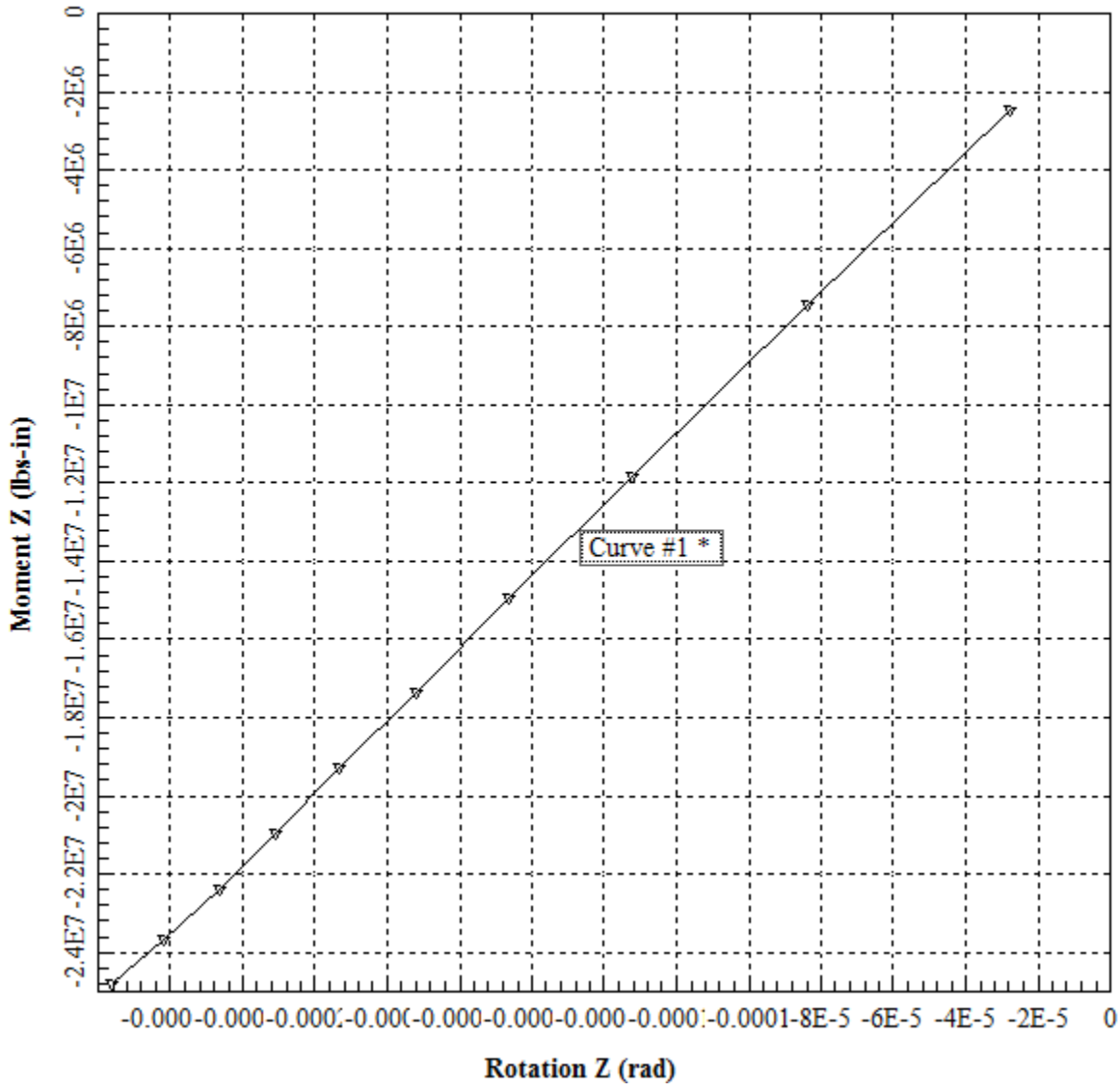


Figure 4.83 Sample graphics screen of Pile Cap Moment vs. Rotation Z command under the Graphs menu

4.5.44 Graphs - Pile Cap K11 vs. Deflection X

This option provides a plot of stiffness coefficient K11(Kxx) versus vertical deflection (X-direction) at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 2D and 3D models

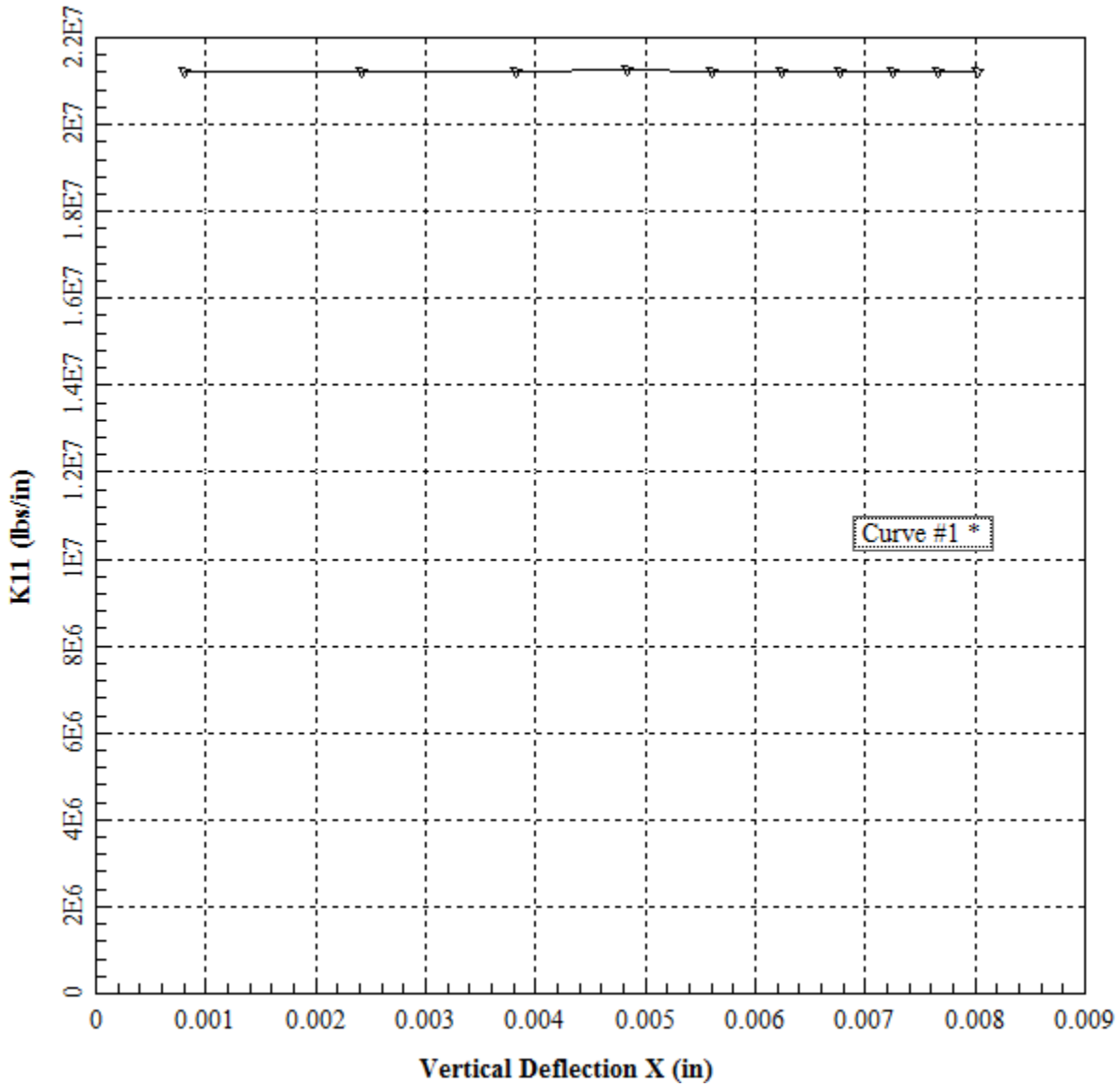


Figure 4.84 Sample graphics screen of Pile Cap K11 vs. Deflection X command under the Graphs menu

4.5.45 Graphs - Pile Cap K22 vs. Deflection Y

This option provides a plot of stiffness coefficient K22(Kyy) versus lateral deflection in the Y-direction at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 2D and 3D models

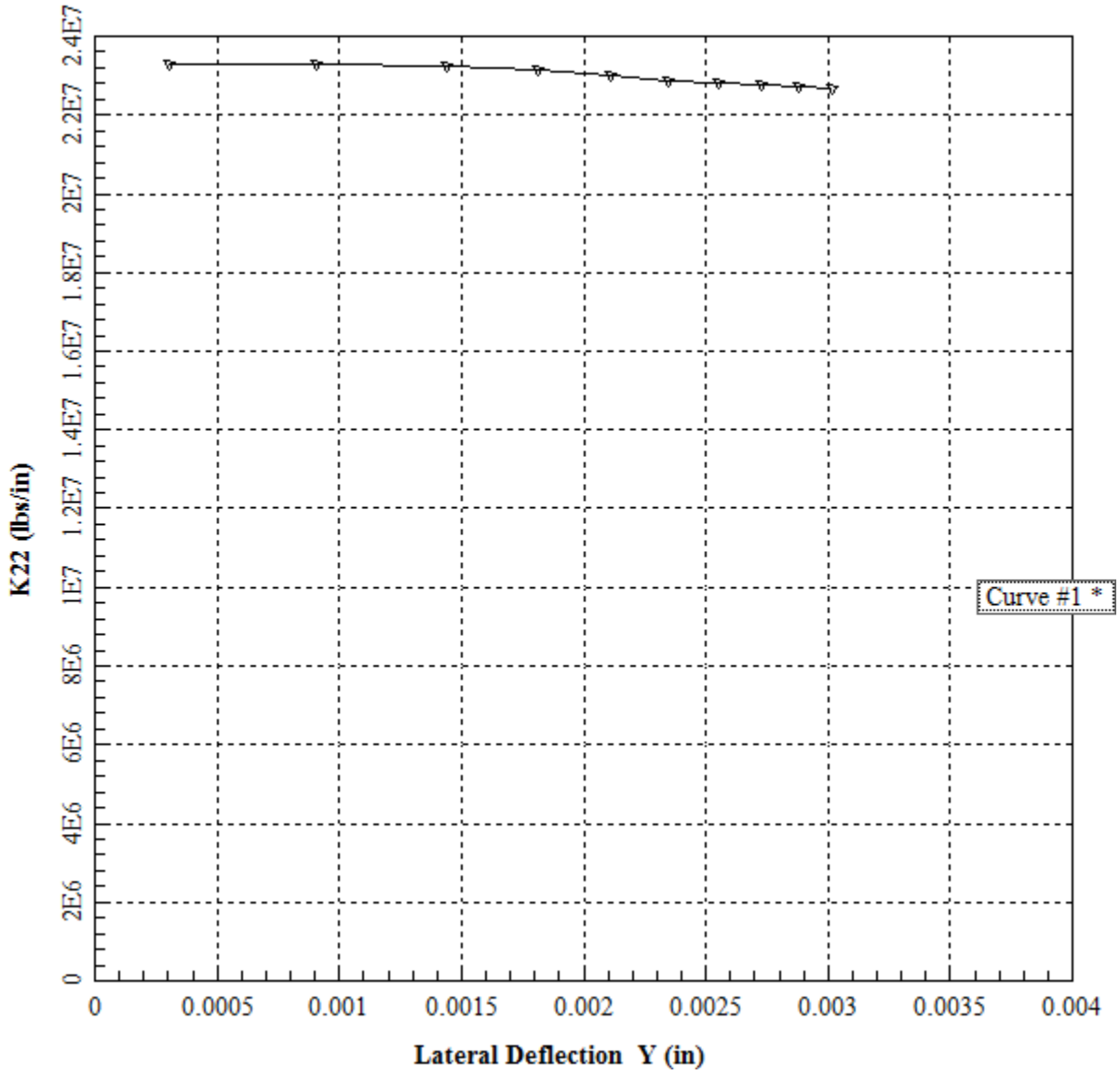


Figure 4.85 Sample graphics screen of Pile Cap K22 vs. Deflection Y command under the Graphs menu

4.5.46 Graphs - Pile Cap K33 vs. Rotation Z

This option provides a plot of stiffness coefficient K33(Krzrz) versus rotation along the Z-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 2D models.

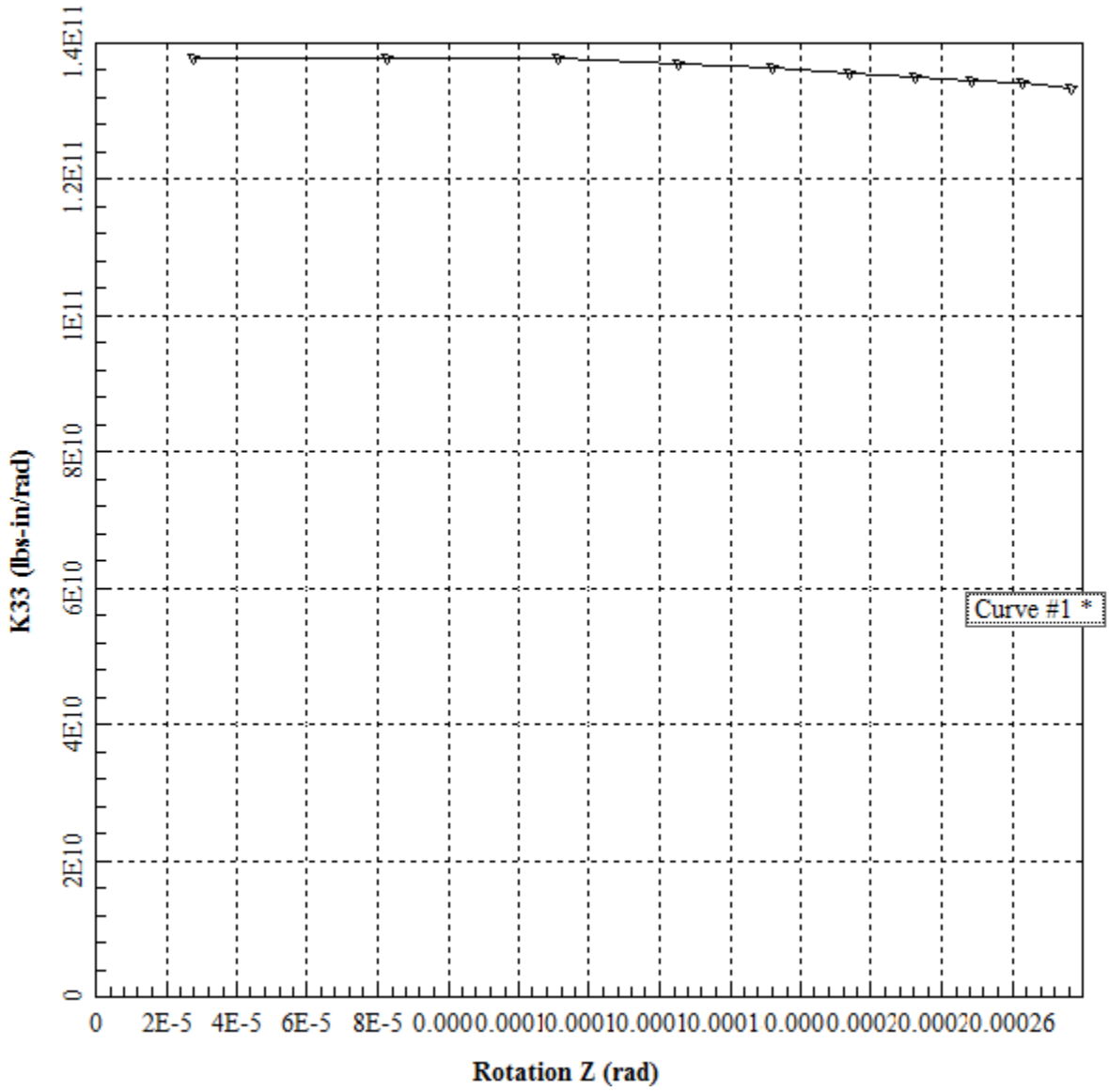


Figure 4.86 Sample graphics screen of Pile Cap K33 vs. Rotation Z command under the Graphs menu

4.5.47 Graphs - Pile Cap K33 vs. Deflection Z

This option provides a plot of stiffness coefficient K33(Kzz) versus lateral deflection in the Z-direction at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 3D models.

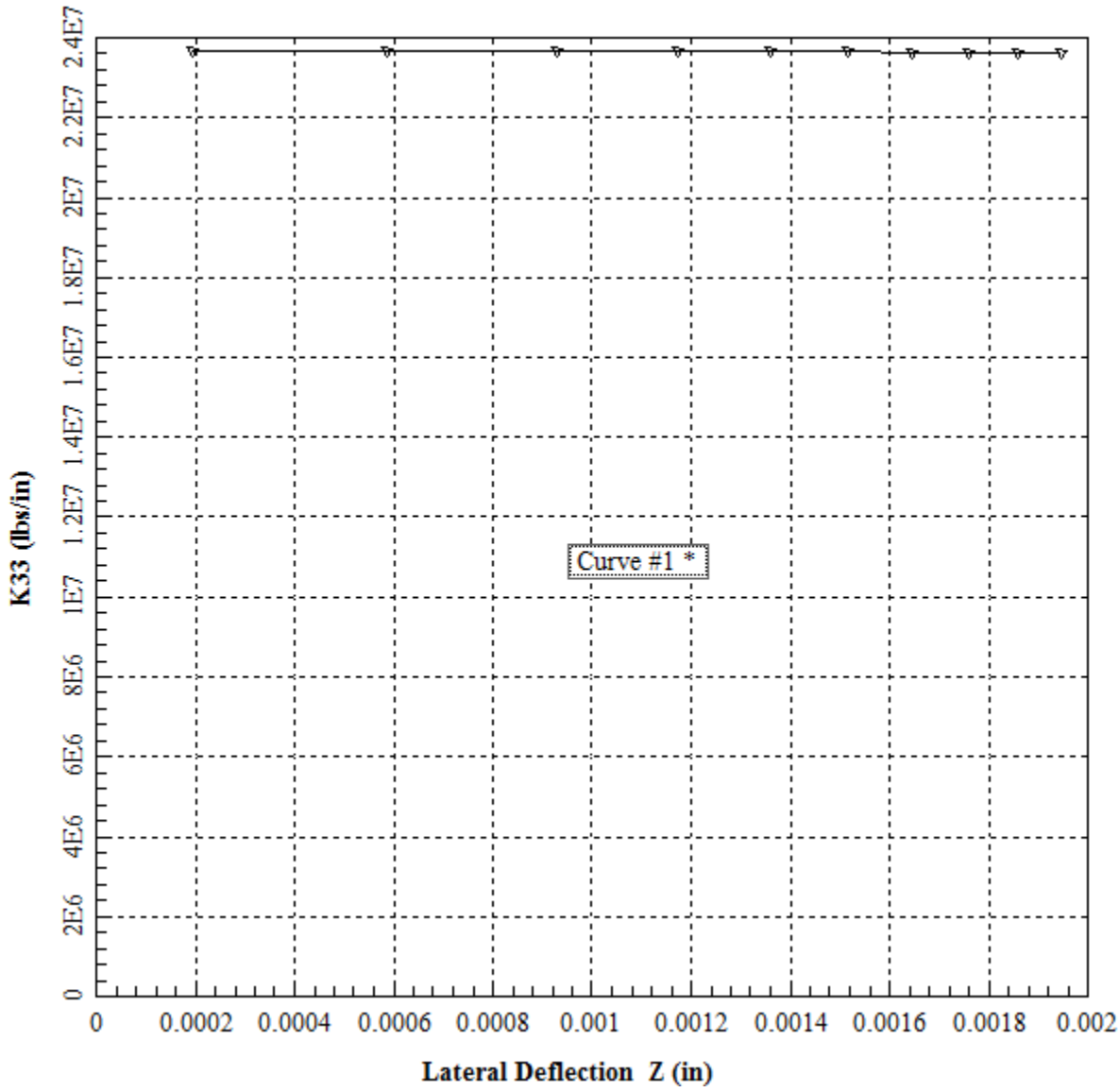


Figure 4.87 Sample graphics screen of Pile Cap K33 vs. Deflection Z command under the Graphs menu

4.5.48 Graphs - Pile Cap K44 vs. Rotation X

This option provides a plot of stiffness coefficient K44(Krxrx) versus rotation along the X-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 3D models.

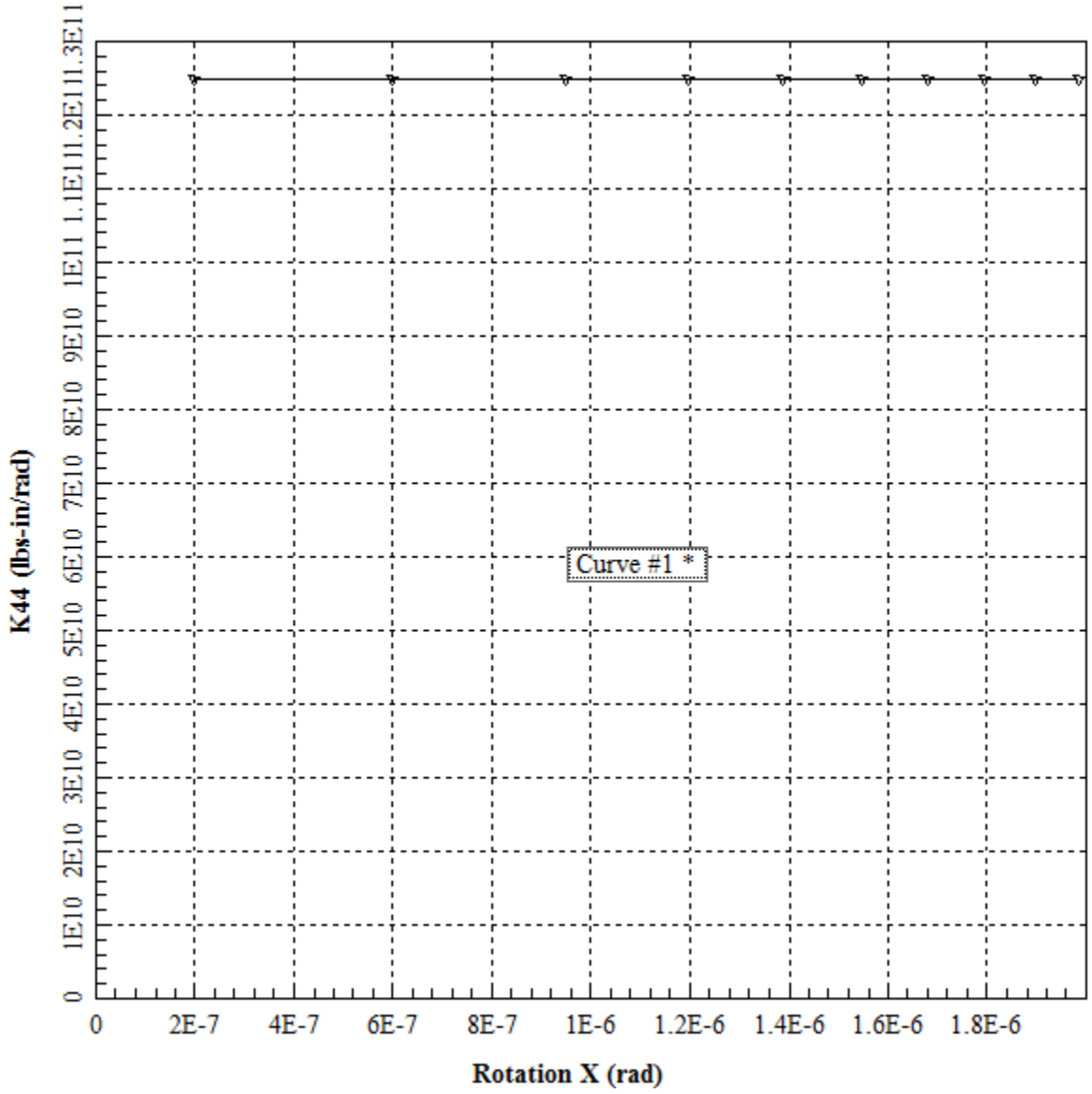


Figure 4.88 Sample graphics screen of Pile Cap K44 vs. Rotation X command under the Graphs menu

4.5.49 Graphs - Pile Cap K55 vs. Rotation Y

This option provides a plot of stiffness coefficient K55(Kryry) versus rotation along the Y-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 3D models.

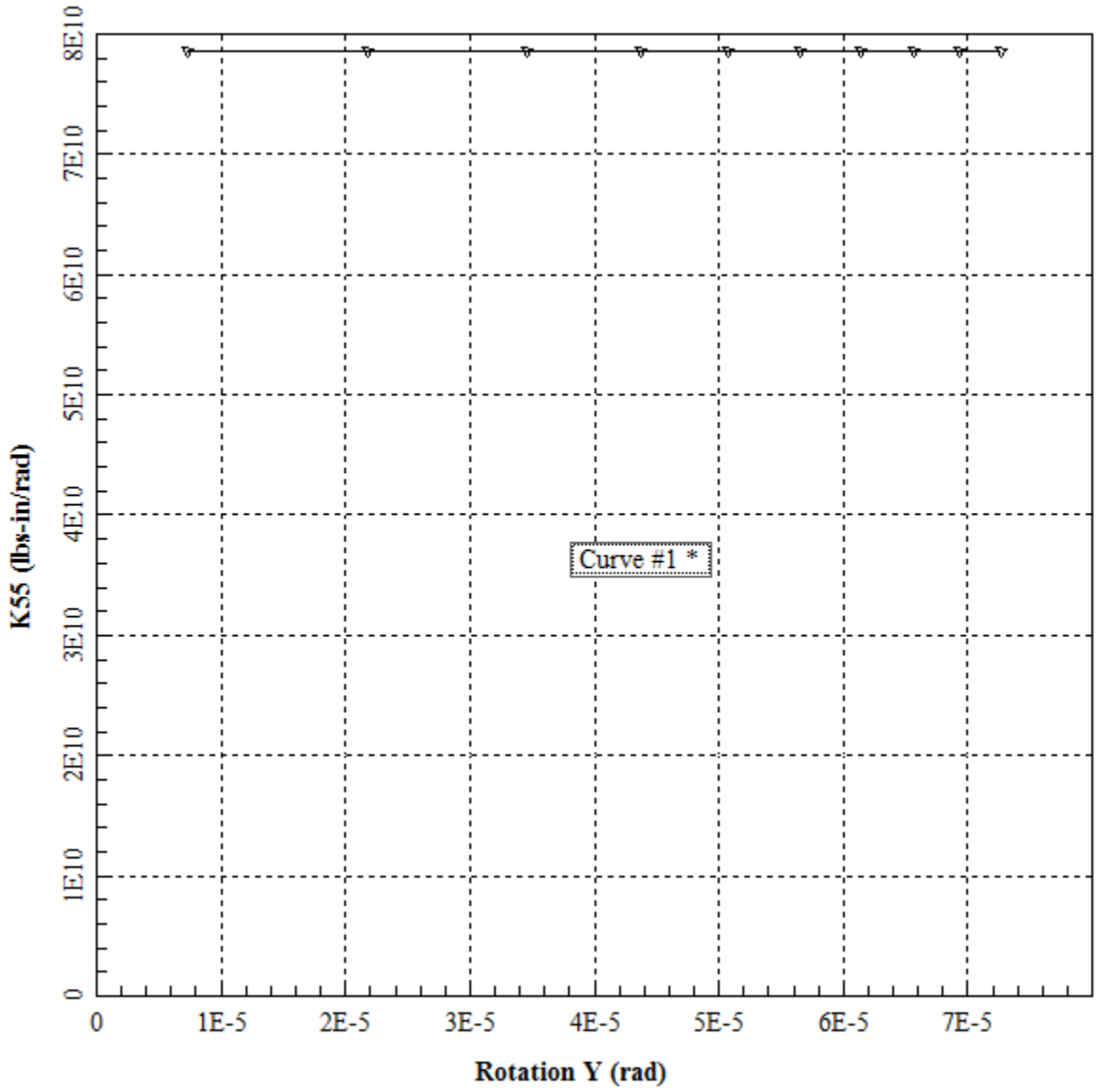


Figure 4.89 Sample graphics screen of Pile Cap K55 vs. Rotation Y command under the Graphs menu

4.5.50 Graphs - Pile Cap K66 vs. Rotation Z

This option provides a plot of stiffness coefficient K66(Krzrz) versus rotation along the X-axis at the pile cap. This option is available if **Generate Data for Foundation Stiffness** and **Compute Foundation Stiffness** are specified in the input for the current load case. This option is available for 3D models.

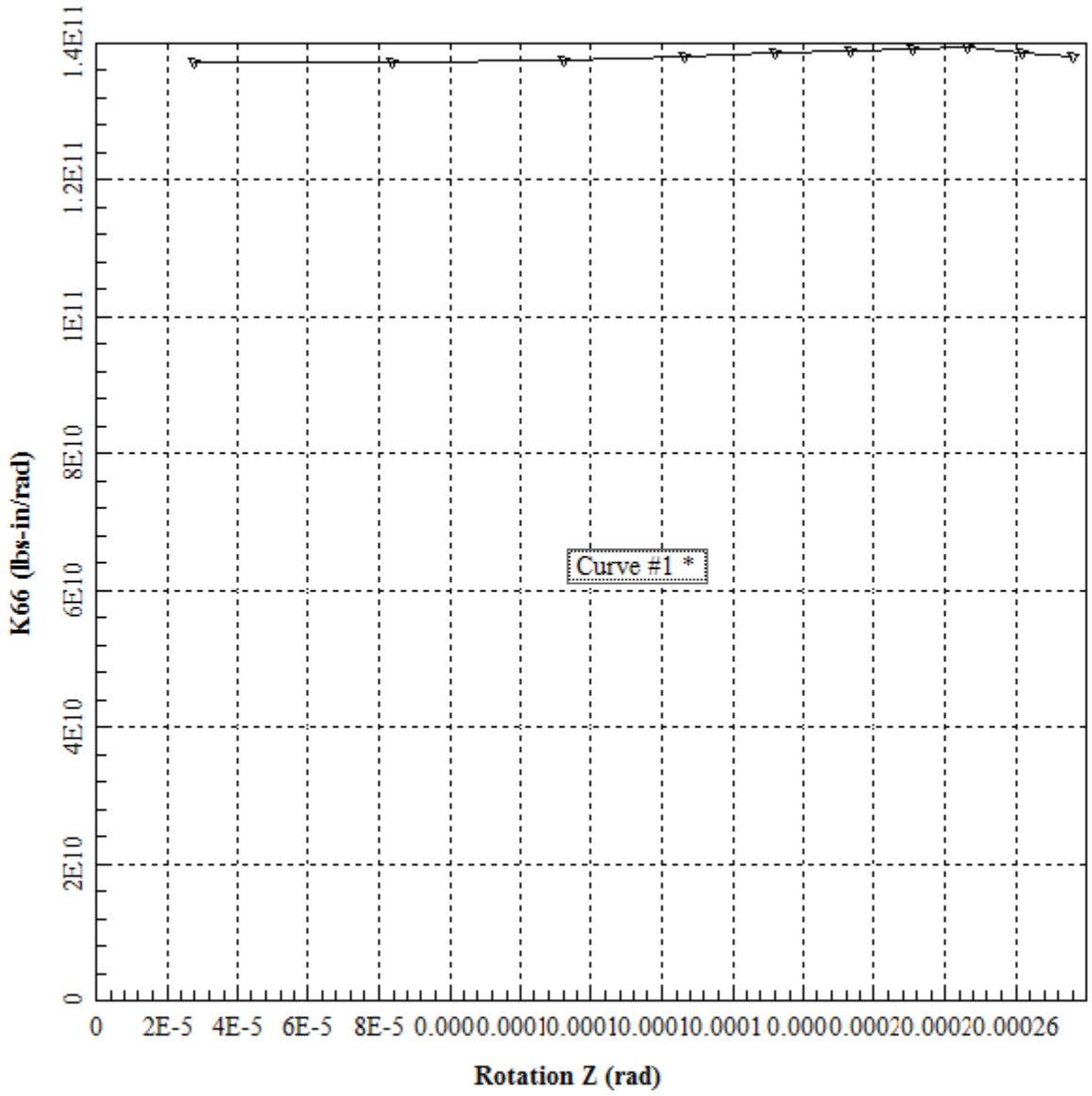


Figure 4.90 Sample graphics screen of Pile Cap K66 vs. Rotation Z command under the Graphs menu

4.5.51 Graphs – Export Graphs to Excel

This option allows the user to export the graphs to an excel file. After selecting this option. The Export To Excel Window will appear. In that window, select the graphs desired to export and then click on the Generate Excel File button. Please note that the user needs to save the Excel file afterwards.

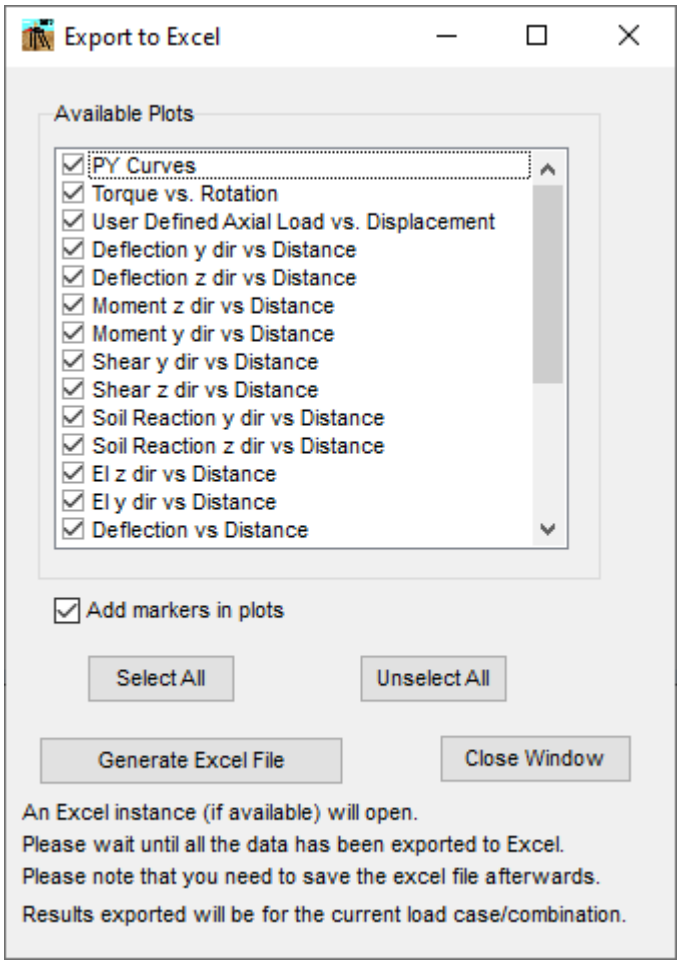


Figure 4.91 Plot to Excel Window

4.5.52 Graphs – Moment Curvature

This option allows the user to display the moment curvature diagrams as well as other section EI diagrams. After selecting this option. The Plot EI Diagrams Window will appear. In that window, select the diagrams desired to displayed.

For non-symmetric sections the program will be able to show only the positive part of the curve, only the negative part or the whole range.

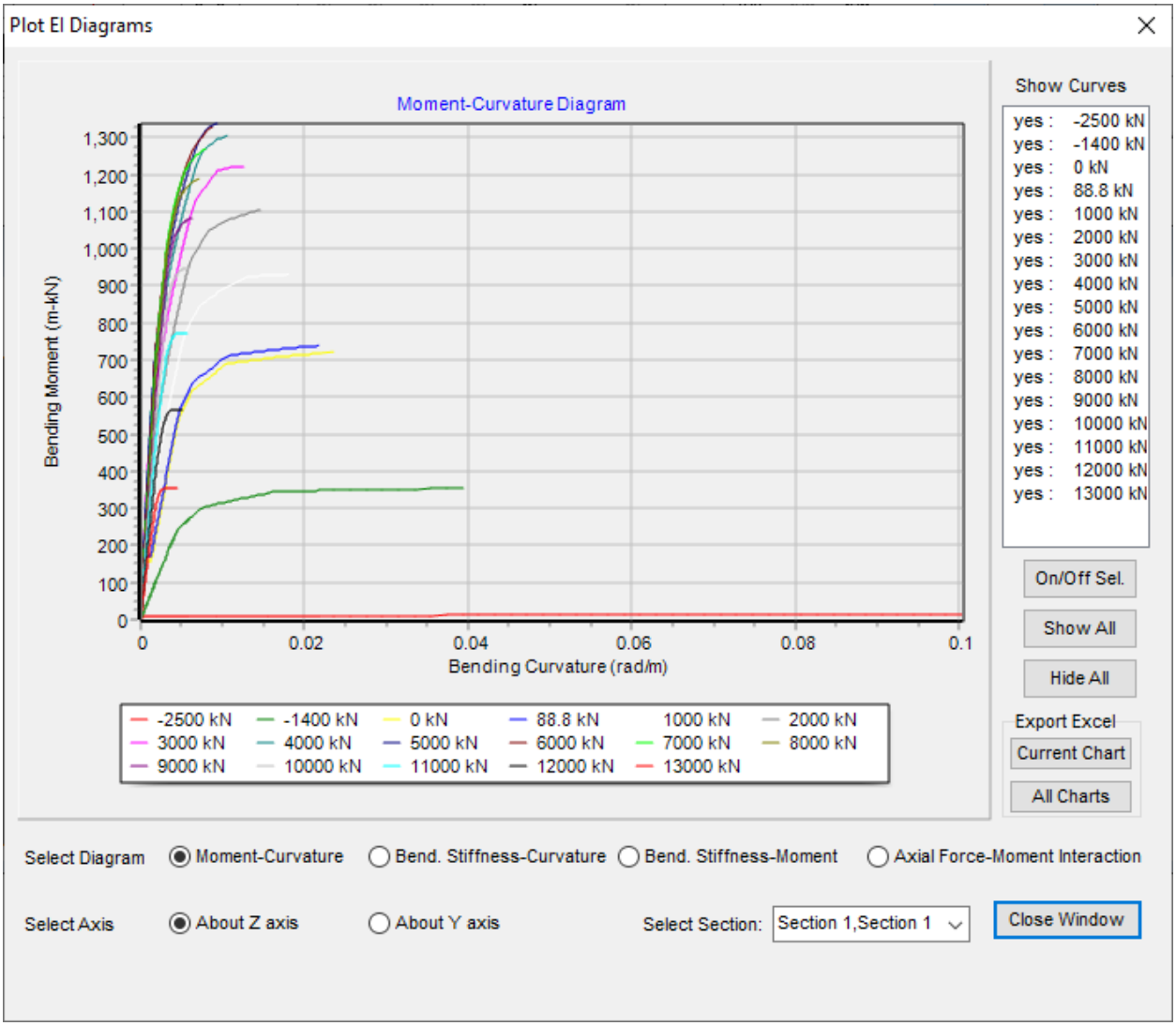


Figure 4.92 Plot EI Diagrams Window

4.6 3D View Results Menu

The **Show 3D View** menu is available when the **3D View** window is displayed. It provides to the user different options for display of the model.

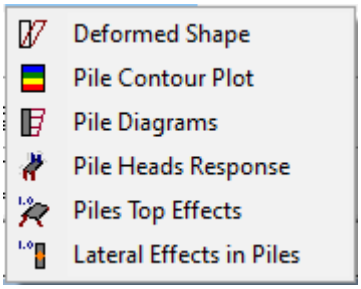


Figure 4.93 Options contained in the 3D View Results Menu

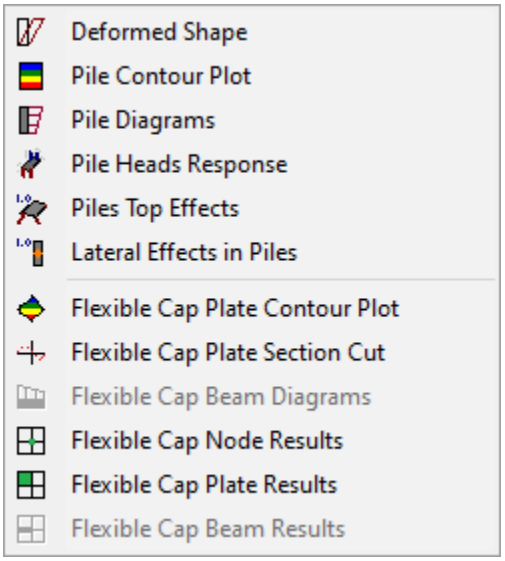


Figure 4.94 Options contained in the 3D View Results Menu – Flexible Cap Version

Below there is a description of the menu items related to display of results. All other items are described in the previous chapter.

4.6.1 Show 3D View -Deformed Shape

This option allows the user to display the deflected shape for the current load case or combination.

Check on **Show Deformed Shape** to display the deformed shape of the model. Check on **Show Undeformed Model** to show the non-deformed model along with the deformed one. Enter the **Deformed Shape Factor** to be used for drawing the deformed model or use the Up or Down buttons to increase or decrease the current one.

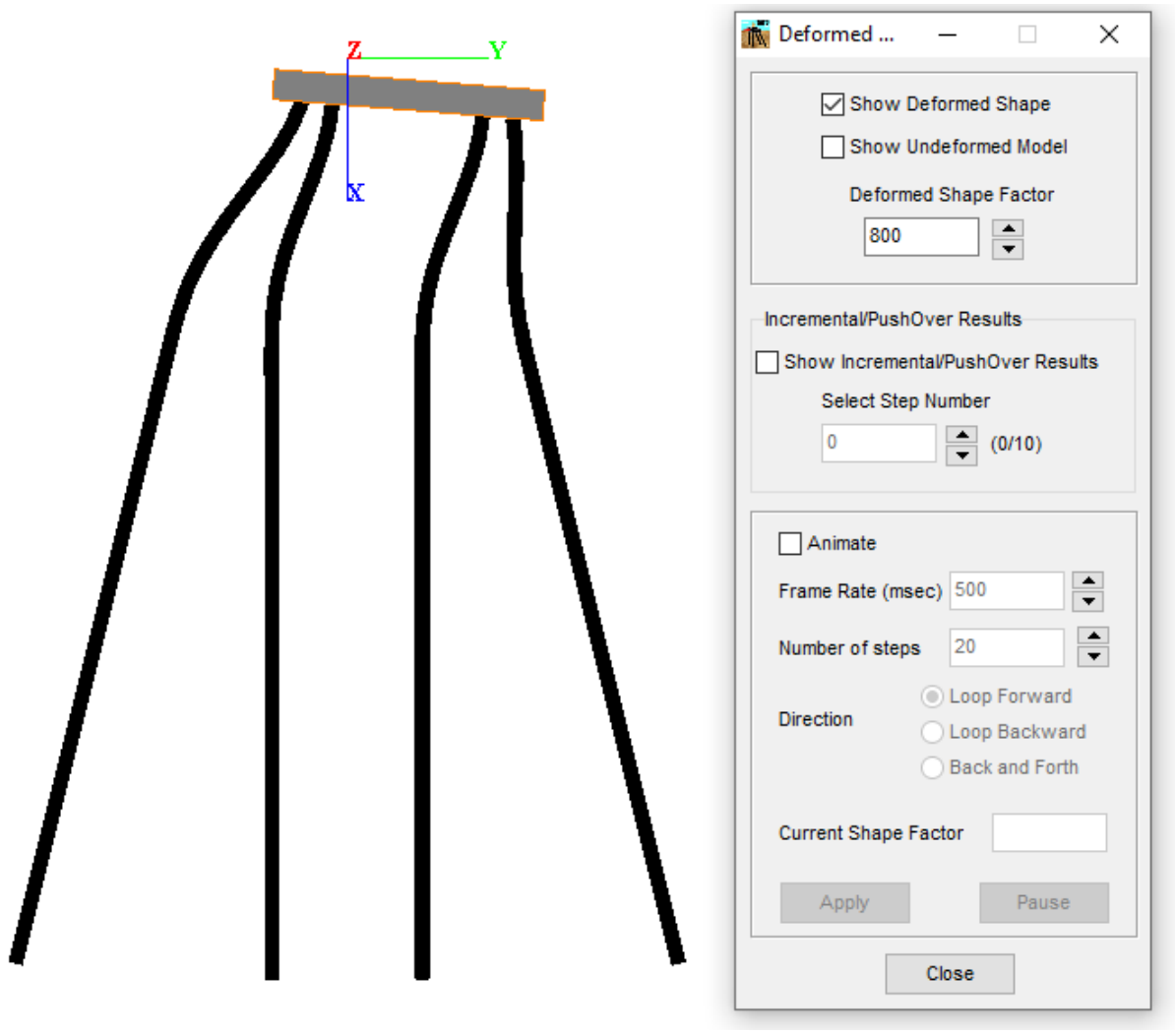


Figure 4.95 Sample graphics screen for Deformed Shape command

4.6.2 Show 3D View -Contour Plot

This option allows the user to display a contour plot for the current load case or combination.

Check on **Show Contour Plot** to display the contour plot of the current type. Select the desired type from the **Contour Plot Type** drop-down menu. The program will assign maximum, minimum, division and increments by default. Click on the Up and Down buttons of each value to increase or decrease it. Check on **Show Contour Legend** to display the legend of the contour.

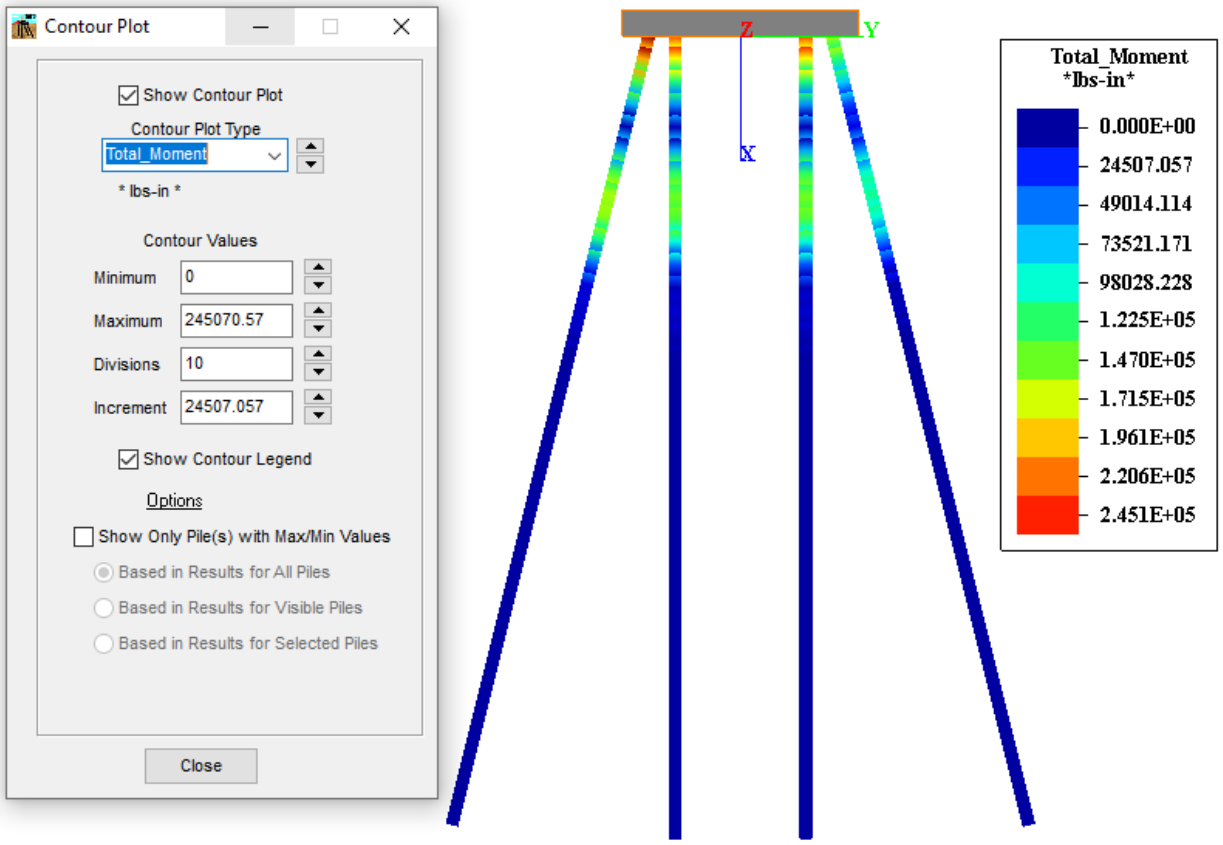


Figure 4.96 Sample graphics screen for Contour Plot command

4.6.3 Show 3D View -Pile Diagrams

This option allows the user to display the diagrams along the piles for the current load case or combination.

Check on **Show Pile Diagrams** to display the diagrams along the piles of the current type. Select the desired type from the **Pile Diagrams Type** drop-down menu.

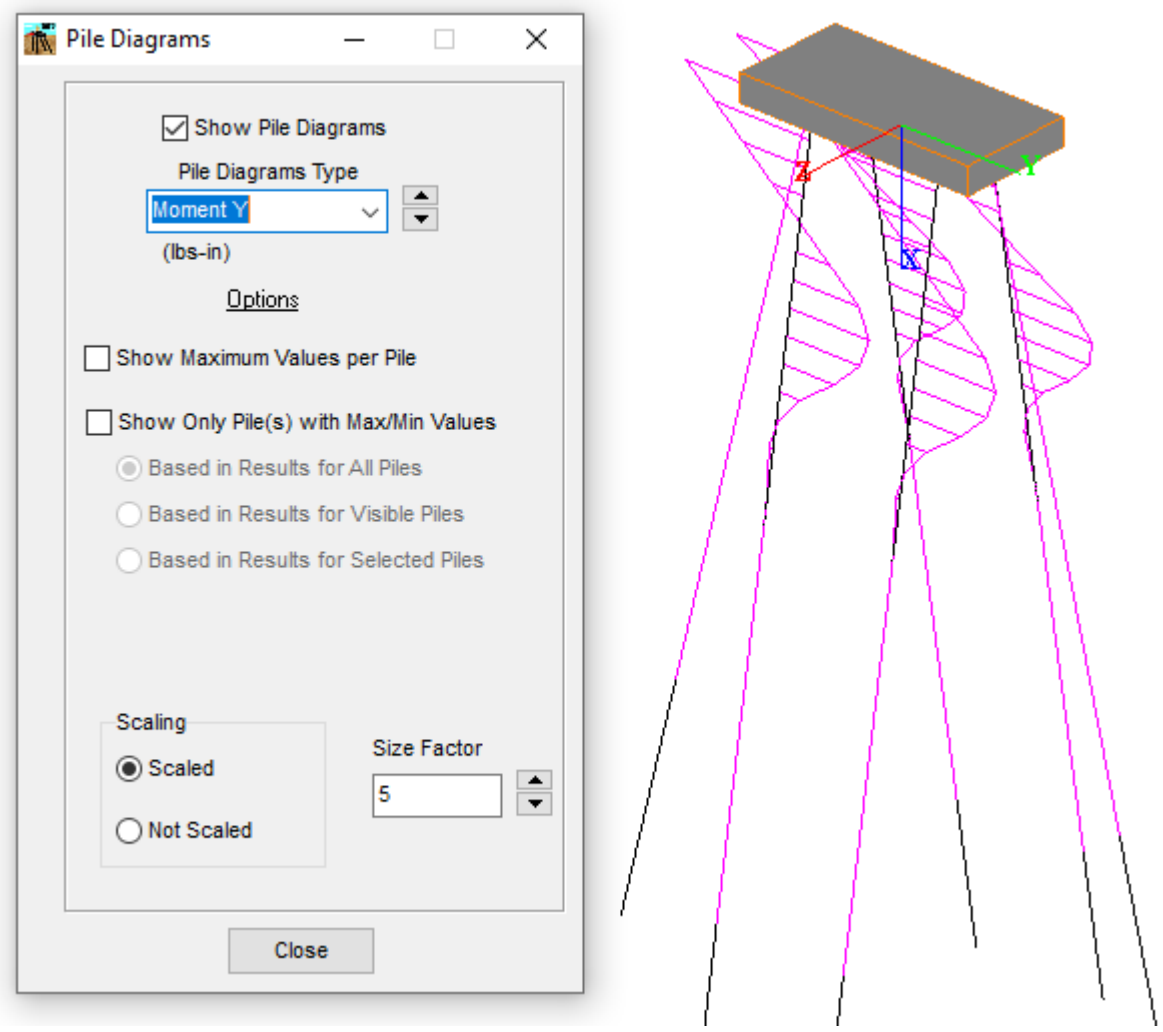


Figure 4.97 Sample graphics screen for Pile Diagrams command

4.6.4 Show 3D View -Pile Heads Response

This option allows the user to present graphically the results at the pile head level (pile cap) for the current load case or combination.

Check on **Show Cap Response** to display the cap response of the current type. Select the desired type from the **Type** drop-down menu. Choose between local axes and global axes. Check on **Show Bar Plot** to display the response using bars normal to the pile cap. Enter the length and width factor or click on the Up and Down buttons of each value to increase or decrease it. Check on **Show Values** to display the numerical values. Check on **Only Maximums** to display only maximum and minimum values. Check on **Show Pile Numbers** to display pile numbers.

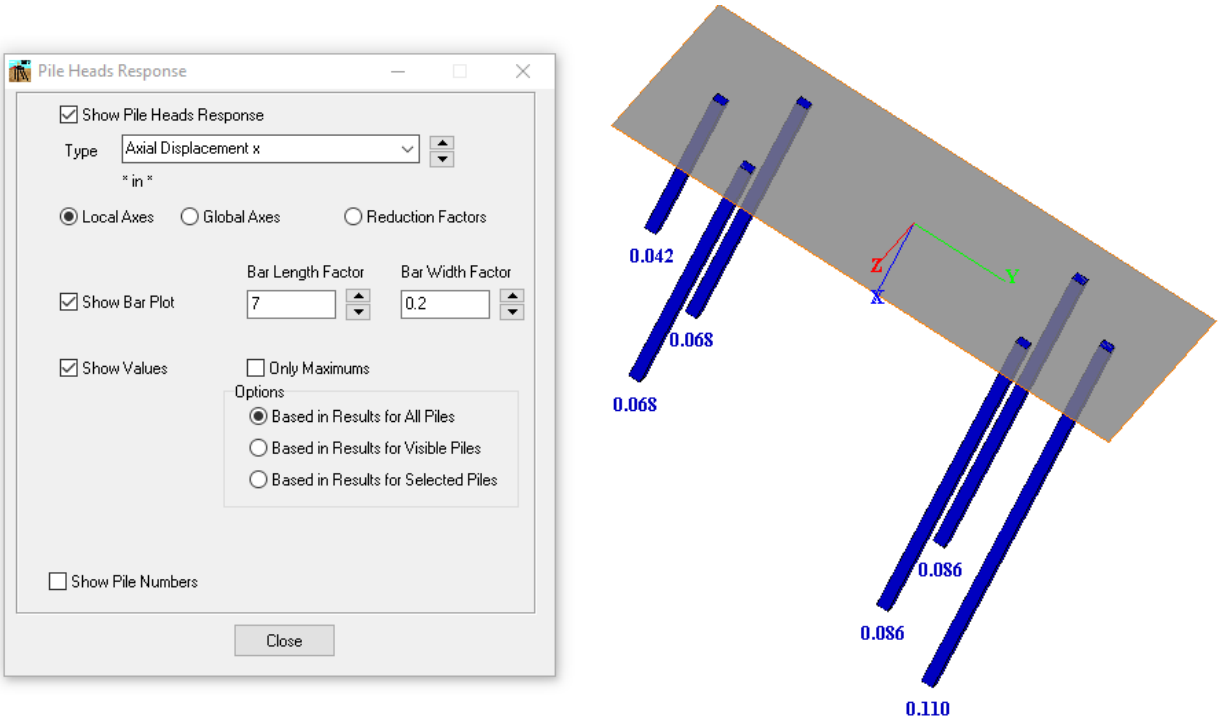


Figure 4.98 Sample graphics screen for Pile Cap Response command

4.6.5 Show 3D View -Pile Top Effects

This option allows the user to display the numerical results at the pile cap and at the pile heads for the current load case or combination.

Enter the pile number, desired to get the numerical results, in the **Pile Group** entry field or click on the Up and Down buttons to increase or decrease the pile number (the pile will be highlighted in yellow). Choose between local axes and global axes cap response.

Check on **Cap Results** to display the displacements and applied forces at the pile cap (origin of coordinates). Also checking this option allows the user to display the values stiffness matrix and/or flexibility matrix for load cases that are estimated.

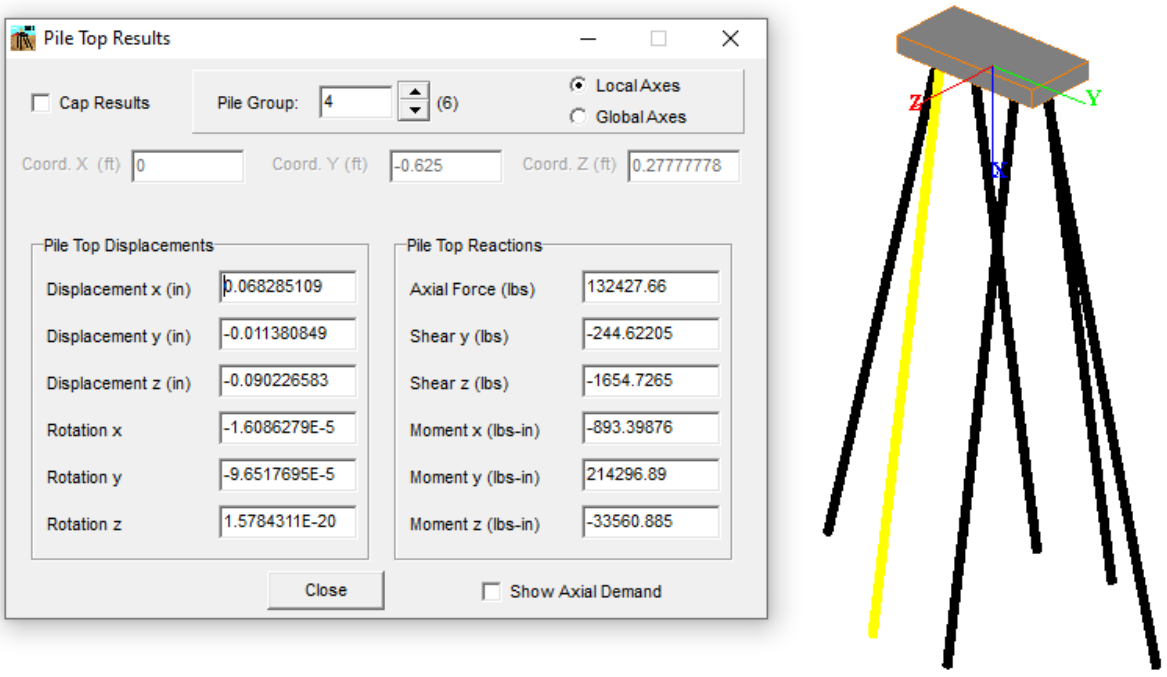


Figure 4.99 Sample graphics screen for Pile Top Effects command

Check on **Show Axial Demand** to plot the load-displacement cir ve for the current pile along the axial force demanded for the current load case or combination. Please note that the Axial Demand Plot is only available while displaying results in local ccordinates.

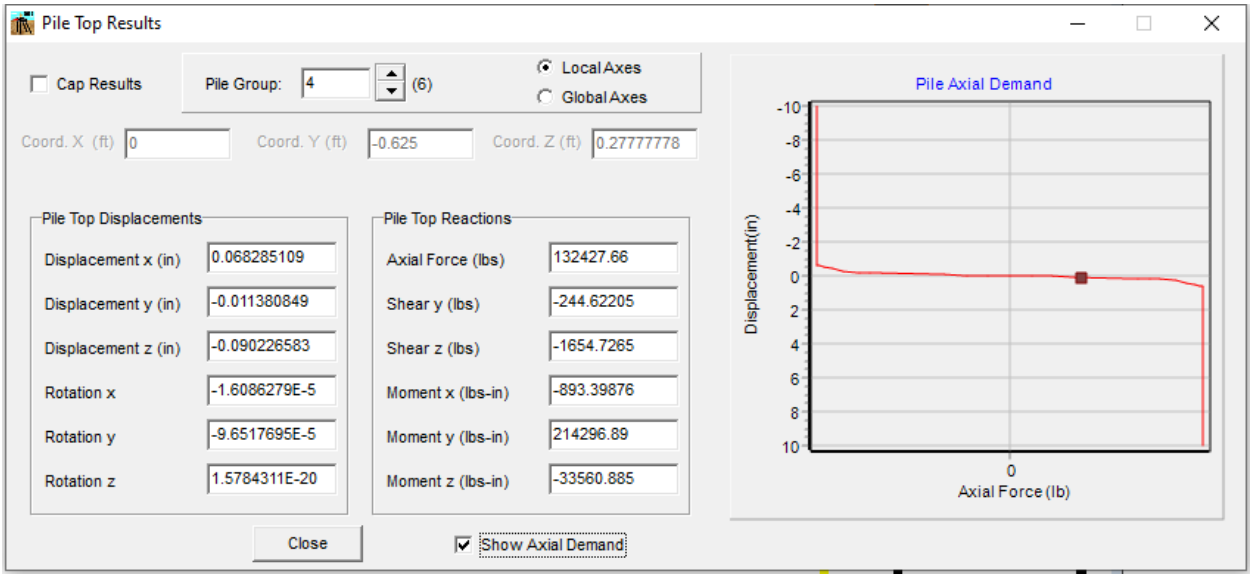


Figure 4.100 Sample screen for Pile Top Results Window including Axial Demand Plot

4.6.6 Show 3D View -Lateral Effects in Piles

This option allows the user to display the numerical results for lateral effects along the piles for the current load case or combination.

Enter the pile number, desired to get the numerical results, in the **Pile Group** entry field or click on the Up and Down buttons to increase or decrease the pile number (the pile will be highlighted in yellow). Enter the pile node number, desired to get the numerical results, in the **Pile Node** entry field or drag the horizontal Bar, right or left, to increase or decrease the pile number. The pile node will be highlighted in red

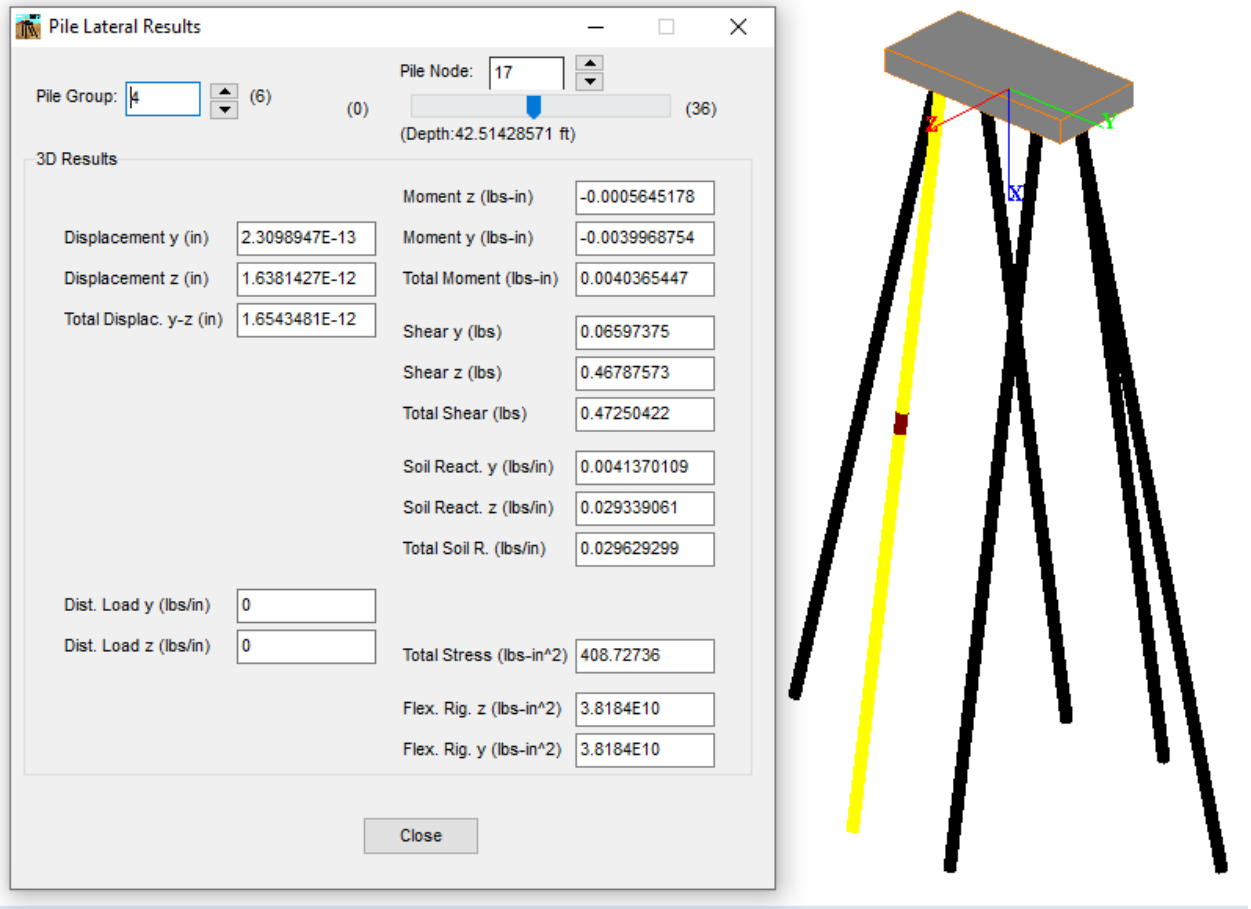


Figure 4.101 Sample graphics screen for Pile Top Effects command

4.6.7 Show 3D View -Flexible Cap Contour Plot - Flexible Cap Version

This option allows the user to display a contour plot of a selected variable for plate elements that belong to the flexible pile cap for the current load case or combination.

Check on **Show Contour Plot** to display the contour plot of the current type. Select the desired variable type from the **Variable Type** options and select the desired type from the **Contour Plot Type** drop-down menu. The program will assign maximum, minimum, division and increments by default. Click on the Up and Down buttons of each value to increase or decrease it. Check on **Show Contour Legend** to display the legend of the contour.

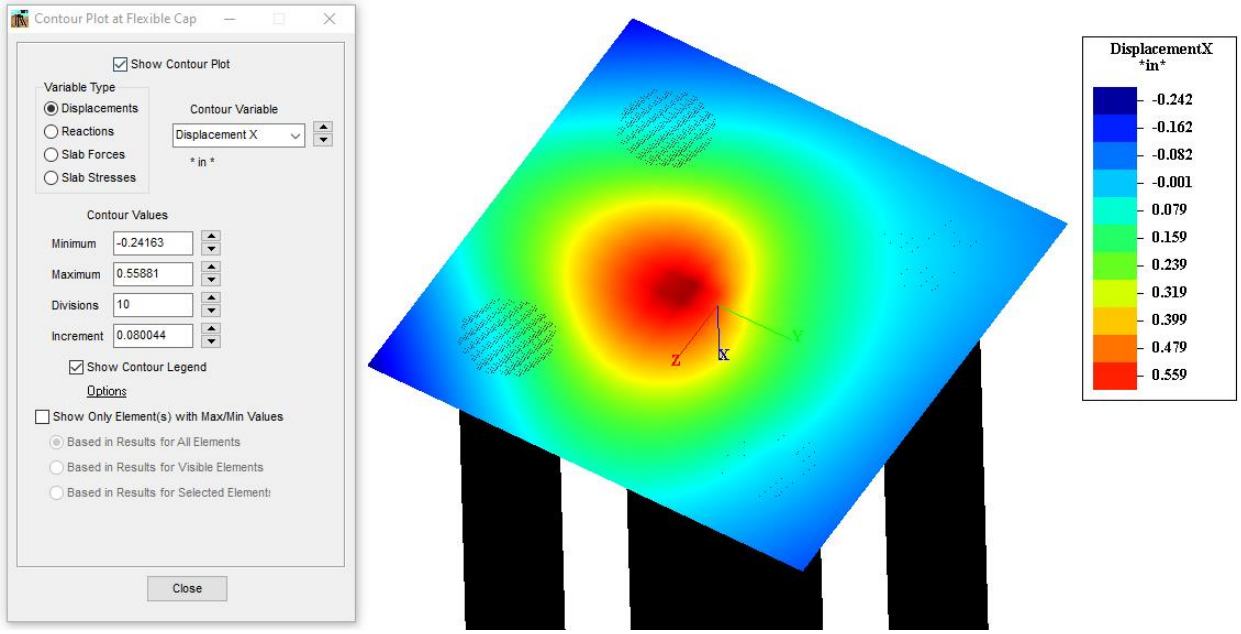


Figure 4.102 Sample graphics screen for Flexible Cap Contour Plot command – Flexible Cap Version

4.6.8 Show 3D View -Flexible Cap Section Cut - Flexible Cap Version

This option allows the user to display the distribution of a selected variable on plates elements that belong to the flexible pile cap through a given section cut for the current load case or combination.

Enter the coordinates for the Start Point and End Point that define the Section Cut. Select the desired variable type from the **Variable Type** options and select the desired variable from the **Variable Type**.

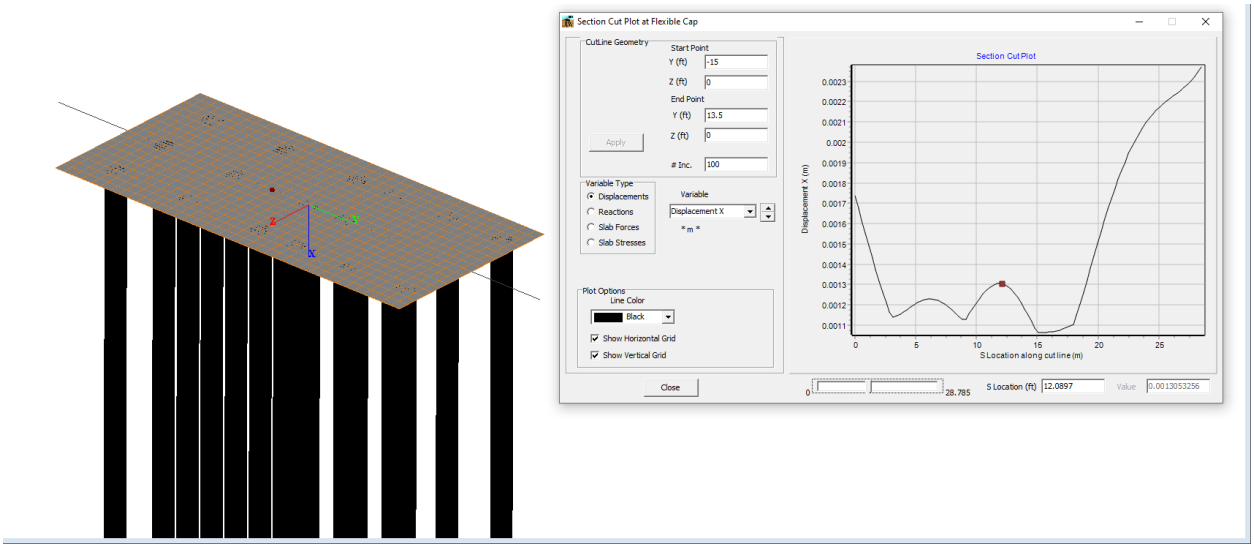


Figure 4.103 Sample graphics screen for Flexible Cap Section Cut command – Flexible Cap Version

4.6.9 Show 3D View – Flexible Cap Beam Diagrams - Flexible Cap Version

This option allows the user to display the diagrams along the beams that belong to the flexible pile cap for the current load case or combination.

Check on **Show Beam Diagrams** to display the diagrams along the piles of the current type. Select the desired type from the **Beam Diagrams Type** drop-down menu.

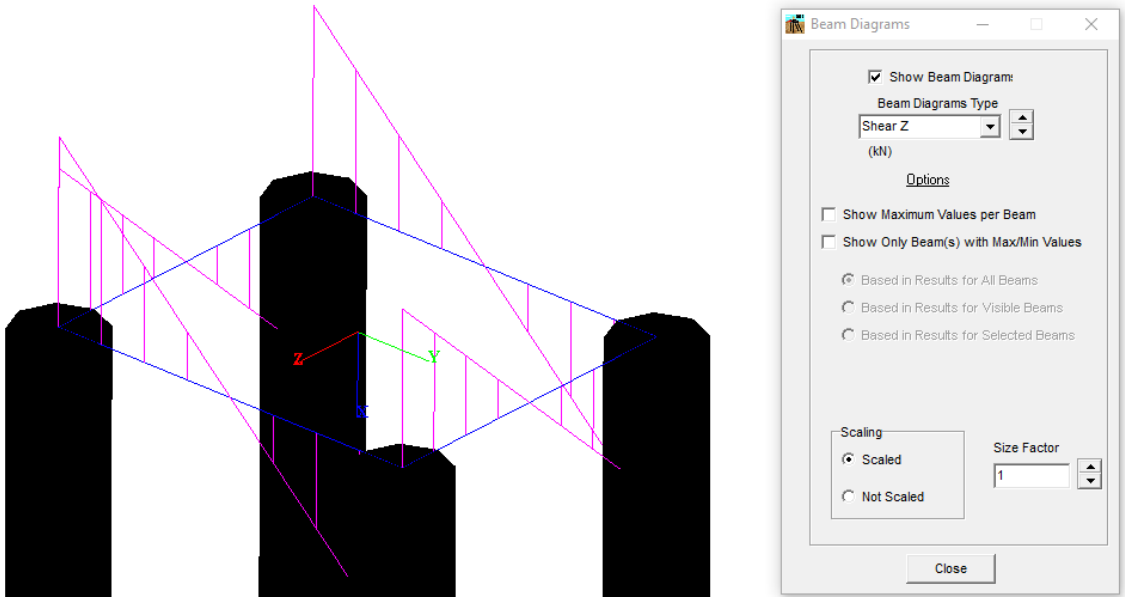


Figure 4.104 Sample graphics screen for Beam Diagrams command – Flexible Cap Version

4.6.10 Show 3D View -Flexible Cap Node Results - Flexible Cap Version

This option allows the user to display values of a selected variable at nodes on the flexible pile for the current load case or combination.

Enter the Node Number to inquire. Also, the node number can be picked-up graphically by clicking on the node after setting Select Nodes Mode. The inquired node will be displayed with yellow color.

Select the desired variable type from the **Variable Type** options.

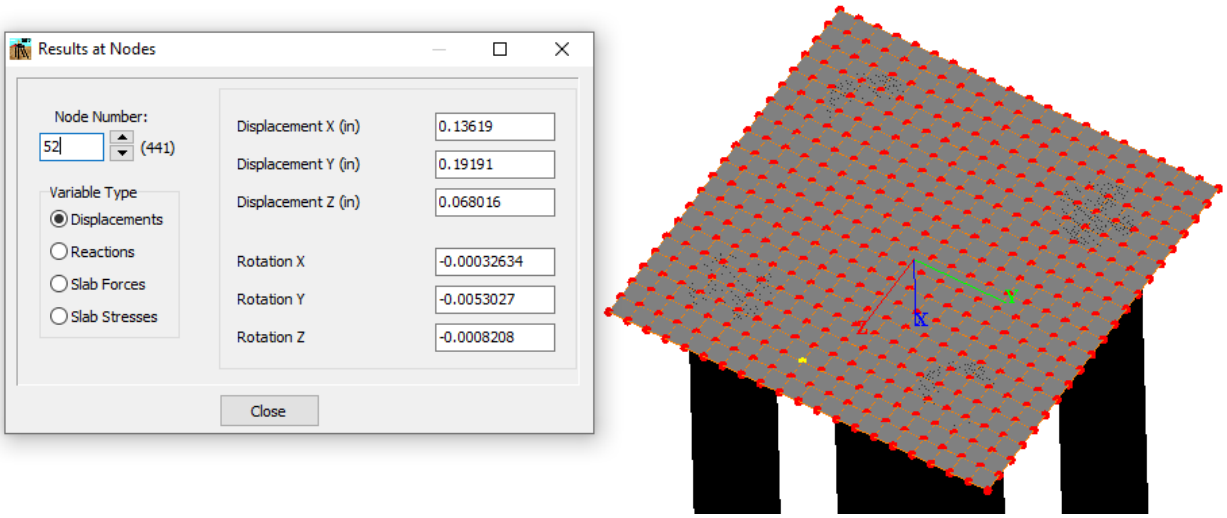


Figure 4.105 Sample graphics screen for Flexible Cap Node Results – Flexible Cap Version

4.6.11 Show 3D View -Flexible Cap Plate Results - Flexible Cap Version

This option allows the user to display values of a selected variable at plates on the flexible pile for the current load case or combination.

Enter the plate Number to inquire. Also, the plate number can be picked-up graphically by clicking on the plate after setting Select Plates Mode. The average value for the whole element will be displayed by default, but the user can select the values for any integration point. The inquired plate will be displayed with light blue color and the integration point will be displayed as dark gray.

Select the desired variable type from the **Variable Type** options.

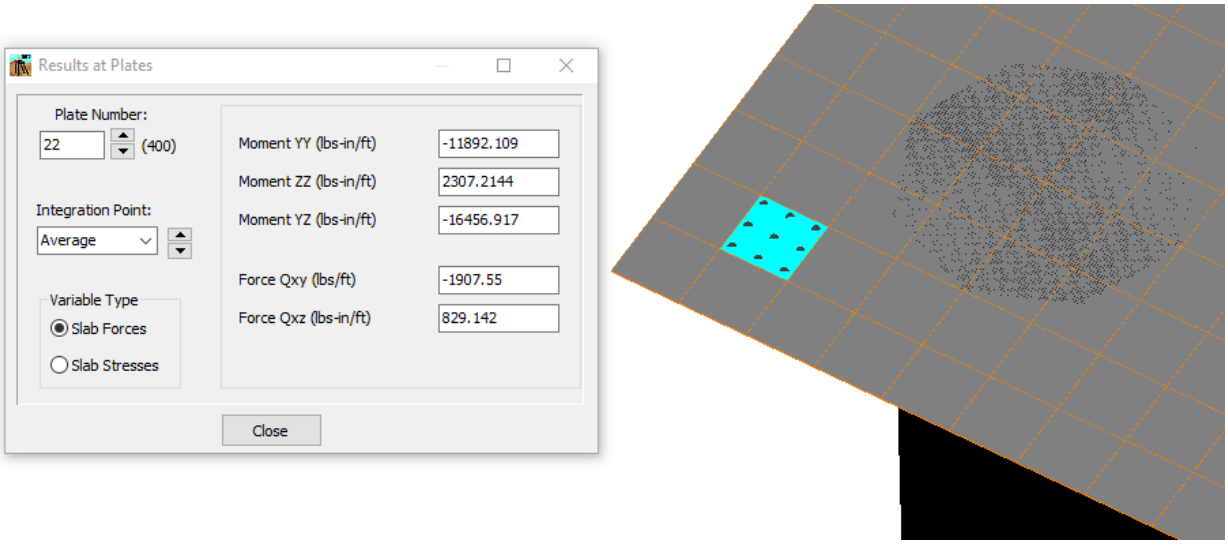


Figure 4.106 Sample graphics screen for Flexible Cap Plate Results – Flexible Cap Version

4.6.12 Show 3D View -Flexible Cap Beam Results - Flexible Cap Version

This option allows the user to display values of a selected variable at beams on the flexible pile for the current load case or combination.

Enter the beam Number to inquire. Also, the beam number can be picked-up graphically by clicking on the plate after setting Select Beams Mode. The average value for the whole element will be displayed by default, but the user can select the values for a point (from a 9 points available) along the length of the beam. The inquired beam will be displayed with yellow color and the point will be displayed as balck.

Select the desired variable type from the **Variable Type** options.

Check the **Plot Variable Along Length** to display a xy plot of any variable along length, then select the variable as desired..

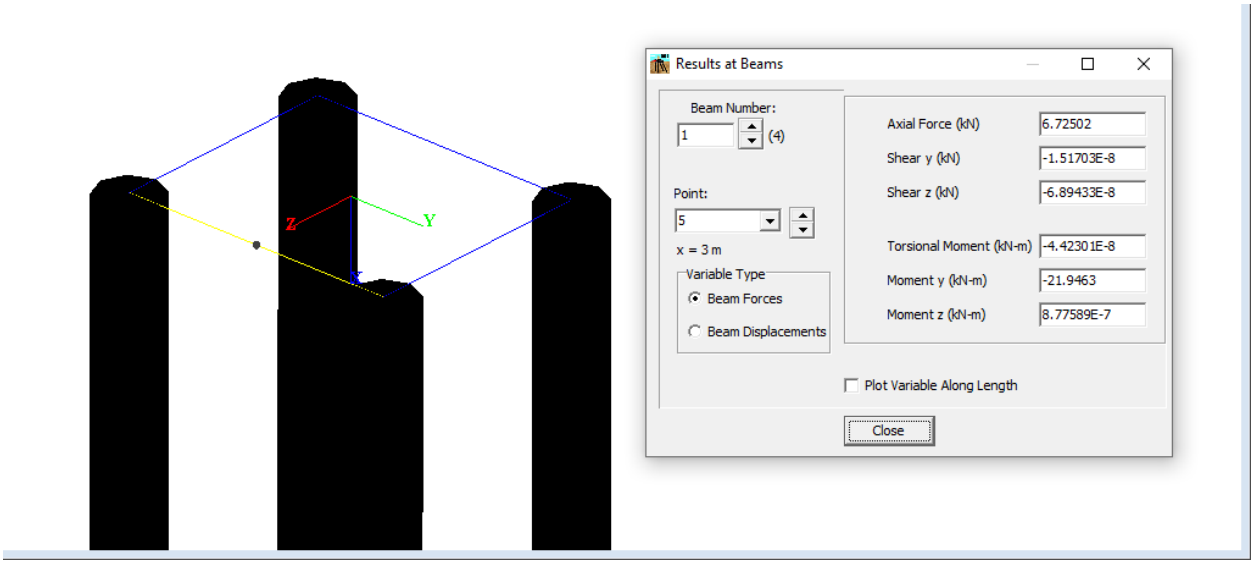


Figure 4.107 Sample graphics screen for Flexible Cap Beam Results – Flexible Cap Version