
COMPUTER PROGRAM

GeoMat

2025.4.2

***A Program for the Analysis of Soil-Supported Mat
and Structural Slab
(User's Manual)***

by

Luis Vasquez
Shin Tower Wang
Daqing Xu

for

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

February - 2026

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 product 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 in all computers on 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 the United States Copyright Law and International Copyright Treaty.

Copyright 2004-2025 by ENSOFT, INC. All rights reserved.

Printed in the United 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 of our software. As modifications, updates, or new versions are produced, notices will be sent to all subscribed users that keep their address current on ENSOFT, INC.'s files.

No warranty, expressed or implied, is offered as to the accuracy of results from ENSOFT, INC.'s software products. The software products 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. Users are assumed to be knowledgeable of the information in the printed documentations that are distributed with the digital media. Users are assumed to recognize that the input parameters, e.g., subgrade modulus, material properties, geometry and dimension of mat, and many others, can have a significant effect on the solution and must be chosen carefully. Users should have a thorough understanding of the relevant theoretical criteria (appropriate references are suggested in the software documentation).

Contents

1	Introduction	1-1
1.1	General Description.....	1-1
1.2	Method of Analyses.....	1-2
1.3	Program Features and Change Log.....	1-3
1.3.1	GeoMat 1.0 (2004)	1-3
1.3.2	GeoMat 2.0 (2014)	1-3
1.3.3	GeoMat 3.0 (2022)	1-4
1.3.4	GeoMat 3.1 (2022)	1-4
1.3.5	GeoMat 4.1 (2025)	1-4
1.3.6	GeoMat 4.2 (2025)	1-4
1.4	Technical Support.....	1-4
1.4.1	Preferred Methods of Software Support	1-5
1.4.2	Support by Telephone.....	1-5
1.4.3	Upgrade Verification and Internet Site	1-5
1.4.4	Changes of Support Policy.....	1-5
2	Installation and Getting Started.....	2-1
2.1	Installation Procedures.....	2-1
2.1.1	Installation of Single-User Version	2-1
2.1.2	Introduction of Network Version	2-4
2.1.3	Installation of Network Version	2-4
2.1.4	Backup of Original Software.....	2-6
2.2	Software Updates.....	2-6
2.3	Getting Started	2-7
2.3.1	Starting the program.....	2-7
2.3.2	File Management	2-8
2.3.3	View Management of Program Window	2-9
2.3.4	Data Input for Program Operation.....	2-10
2.3.5	Computation Options.....	2-12
2.3.6	Manipulating the Slab Model.....	2-13

2.3.7	Results	2-14
2.3.8	Option Menu	2-15
2.3.9	Selection Menu	2-16
2.3.10	Window Menu	2-16
2.3.11	Help Menu	2-17
3	Data Input	3-1
3.1	File Menu	3-1
3.1.1	File – New	3-1
3.1.2	File – New from template	3-1
3.1.3	File – Open	3-6
3.1.4	File – Save	3-7
3.1.5	File – Save As	3-7
3.1.6	File – Import External Mesh	3-7
3.1.7	File – Exit	3-8
3.2	Input Data Menu	3-9
3.2.1	Numeric Data Entries and Sign Conventions.....	3-10
3.2.2	Input Data -Title	3-10
3.2.3	Input Data – Material Properties	3-11
3.2.4	Input Data – Node Coordinates.....	3-14
3.2.5	Input Data – Time Stepping.....	3-15
3.2.6	Input Data – Soil Stiffness	3-15
3.2.7	Input Data – Nonlinear Spring.....	3-17
3.2.8	Input Data – Element Type.....	3-19
3.2.9	Input Data – Element Connectivity	3-19
3.2.10	Input Data – Integration Points	3-19
3.2.11	Input Data – Nodal Constrains.....	3-19
3.2.12	Input Data – Nodal Loads	3-20
3.2.13	Input Data – Nodal Stiffness	3-20
3.2.14	Input Data – Distributed Loads.....	3-22
3.2.15	Input Data – Distributed Stiffness	3-22
3.2.16	Input Data – Concentrated Loads.....	3-23
3.2.17	Input Data – Equivalent Concentrated Loads.....	3-23

3.2.18	Input Data – Concentrated Stiffness	3-24
3.2.19	Input Data – Edge Load.....	3-25
3.2.20	Input Data – Edge Stiffness.....	3-27
3.2.21	Input Data – Node Sets.....	3-28
3.2.22	Input Data - Element Set	3-29
3.2.23	Input Data – Edge Sets.....	3-30
3.2.24	Input Data – Section Cut.....	3-32
3.2.25	Input Data – Node Path	3-33
4	Finite Element Mesh Adjustment	4-36
4.1	Mesh Refinement	4-36
4.2	Change of Element Type.....	4-39
4.3	Change of Integration Rule.....	4-41
4.4	Manual Modification of Nodes and Elements.....	4-42
4.4.1	Step 1: Modify Nodes.....	4-42
4.4.2	Step 2: Modify Elements	4-42
4.4.3	Update Set Definition.....	4-44
5	Program Execution and Output Reviews	5-1
5.1	Introduction.....	5-1
5.2	Computation Menu	5-1
5.2.1	Computation- Run Analysis	5-1
5.2.2	Computation- View Input File	5-2
5.2.3	Computation- View Output File	5-3
5.2.4	Computation- View Message File.....	5-4
5.3	Results Menu	5-4
5.3.1	Results – Select Step	5-5
5.3.2	Results – Deformed Mesh	5-6
5.3.3	Results – Contour Plot.....	5-6
5.3.4	Results – X-Y Plot.....	5-10
5.3.5	Results – Nodal Result.....	5-11
5.4	Speed Buttons in Graphics Mode	5-11
6	Examples	6-1
6.1	Introduction.....	6-1

6.2	Example 1: Cantilevered square plate simulated by single quadratic element	6-1
6.2.1	Problem Description.....	6-1
6.2.2	Modeling Instruction.....	6-2
6.2.3	Output Results Review	6-3
6.3	Example 2: Fixed square plate subject to concentrated force at center	6-4
6.3.1	Problem Description.....	6-4
6.3.2	Modeling Instruction.....	6-5
6.3.3	Output Results Review	6-6
6.4	Example 3: Square slab on soil - distributed stiffness method	6-7
6.4.1	Problem Description.....	6-7
6.4.2	Modeling Instruction.....	6-7
6.4.3	Output Results Review	6-8
6.5	Example 4: Square slab on soil - Mindlin method	6-9
6.5.1	Problem Description.....	6-9
6.5.2	Modeling Instruction.....	6-9
6.5.3	Output Results Review	6-10
6.6	Example 5: Slab on soil to support four columns (no rotation restraint).....	6-11
6.6.1	Problem Description.....	6-11
6.6.2	Modeling Instruction.....	6-11
6.6.3	Output Results Review	6-12
6.7	Example 6: Slab on soil to support four concentrated forces (restrained nodal rotation)	6-14
6.7.1	Problem Description.....	6-14
6.8	Example 7: Rectangular slab on soil subject to column loads.....	6-16
6.8.1	Problem Description.....	6-16
6.8.2	Modeling Instruction.....	6-16
6.8.3	Output Results Review	6-17
6.9	Example 8: Rectangular raft on soil to support columns and walls	6-19
6.9.1	Problem Description.....	6-19
6.9.2	Modeling Instruction.....	6-20
6.9.3	Output Results Review	6-21
6.10	Example 9: Wind turbine foundation supported by drilled shafts	6-23
6.10.1	Problem Description	6-23

6.10.2	Modeling Instruction	6-25
6.10.3	Output Results Review	6-30
6.11	Example 10: Square slab on soil - Vlasov method	6-33
6.11.1	Problem Description	6-33
6.11.2	Modeling Instruction	6-33
6.11.3	Output Results Review	6-35
6.12	Example 11: Circular wind turbine foundation - Vlasov method	6-38
6.12.1	Problem Description	6-38
6.12.2	Modeling Instruction	6-38
6.12.3	Output Results Review	6-42
6.13	Example 12: Rectangle slab supported by nonlinear spring.....	6-43
6.13.1	Problem Description	6-43
6.13.2	Modeling Instruction	6-43
6.13.3	Output Results Review	6-46
6.13.4	Combination of Nonlinear Spring with Other Soil Stiffness	6-48
6.14	Example 13: Case Study – Pile-Raft Analysis.....	6-51
6.14.1	Problem Description	6-51
6.14.2	Modeling Instruction	6-53
6.14.3	Combination of Nonlinear Spring with Other Soil Stiffness	6-55
6.14.4	Output Results Review	6-56
6.15	Example 14: L-shape raft on soil.....	6-58
6.15.1	Modify slab from example 7.....	6-59
6.16	Example 15: Import external mesh.....	6-61
6.16.1	Problem Description	6-61
6.16.2	Import mesh from an external mesh file.....	6-61
6.16.3	Model Instruction	6-62
6.16.4	Out Results	6-63
7	References	7-64

Figures

Figure 2.1 Main installation screen for ENSOFT software (may change with time)	2-2
Figure 2.2 License-agreement screen for ENSOFT software (may change with time)	2-3
Figure 2.3 File-location screen for ENSOFT software (may change with time)	2-3
Figure 2.4 File-location screen for ENSOFT software (may change with time)	2-5
Figure 2.5 Layout of GeoMat program screen	2-8
Figure 2.6 Options contained in the File menu	2-9
Figure 2.7 Options contained in the View menu	2-10
Figure 2.8 Options contained in the Input Data menu	2-12
Figure 2.9 Options contained in the Computation menu	2-13
Figure 2.10 Options contained in the Transform menu	2-14
Figure 2.11 Options contained in the Results menu	2-14
Figure 2.12 Available items contained in the Options menu	2-16
Figure 2.13 Options contained in the Selection menu	2-17
Figure 2.14 Options contained in the Window menu	2-17
Figure 2.15 Options contained in the Help Menu	2-18
Figure 3.1 Window screen for the File – File menu	3-1
Figure 3.2 Options for various template	3-2
Figure 3.3 Input screen for the Circular option	3-2
Figure 3.4 Input screen for the Rectangular option – Equally Spaced option	3-3
Figure 3.5 Input screen for the Rectangular option – Generally Spaced option	3-3
Figure 3.6 Grid Y direction input screen for the Rectangular option – Generally Spaced option	3-3
Figure 3.7 Parameters of creating ring shape slab	3-4
Figure 3.8 Example of a full ring slab	3-4
Figure 3.9 Template for octagon wind turbine foundation	3-5
Figure 3.10 Template for circular wind turbine foundation	3-6
Figure 3.11 Window screen for the File – Open dialog	3-7
Figure 3.12 Import external mesh (left) and neutral mesh file (right)	3-8
Figure 3.13 Message window advising that changes were not saved to disk	3-9
Figure 3.14 Options contained in the Input Data menu	3-10
Figure 3.15 Window screen for sample Data – Title	3-11
Figure 3.16 Window screen for material selection under Input Data Menu – Material Properties	3-11
Figure 3.17 Window screen for material data Under Material Properties	3-12
Figure 3.18 Window screen for material data Under Material Properties using uniform plate with orthotropic reinforcement	3-13
Figure 3.19 Window screen for material data Under Material Properties using orthotropic plate with enhanced ribs	3-13

Figure 3.20 Window screen for material data Under Material Properties using orthotropic plate with one-way box section	3-14
Figure 3.21 Window screen for material data Under Material Properties using user-defined section ..	3-14
Figure 3.22 Window screen for Input Data Menu – Time Stepping	3-15
Figure 3.23 Window screen for Input Data Menu – Default option	3-16
Figure 3.24 Window screen for Input Data Menu – Vlasov approach.....	3-16
Figure 3.25 Window screen for Input Data Menu – Mindlin equation.....	3-17
Figure 3.26 Define data for a nonlinear spring	3-18
Figure 3.27 Import spring data from an external file.....	3-18
Figure 3.28 Example of external spring data	3-19
Figure 3.29 Window screen for Input Data Menu – Nodal Constraints	3-20
Figure 3.30 Window screen for Input Data Menu – Nodal Loads.....	3-20
Figure 3.31 Window screen for Input Data Menu – Constant Nodal Stiffness.....	3-21
Figure 3.32 Window screen for Input Data Menu – Nonlinear Nodal Stiffness	3-21
Figure 3.33 Define Nonlinear Nodal Stiffness by means of Node Set.....	3-22
Figure 3.34 Window screen for Input Data Menu – Distributed Loads	3-22
Figure 3.35 Window screen for Input Data Menu – Distributed Stiffness.....	3-23
Figure 3.36 Window screen for Input Data Menu – Concentrated Loads	3-23
Figure 3.37 Window screen for Input Data Menu – Equivalent Concentrated Loads	3-24
Figure 3.38 Window screen for Input Data Menu – Concentrated Stiffness.....	3-25
Figure 3.39 Define edge load by using edge set	3-26
Figure 3.40 Define edge load element by element.....	3-26
Figure 3.41 Example of edge loads defined on left side of a square slab.....	3-27
Figure 3.42 Window screen for Input Data Menu – Edge Stiffness.....	3-28
Figure 3.43 Window screen for Input Data Menu – Node Sets	3-28
Figure 3.44 Window screen for selecting nodal points under Input Data Menu – Node Sets	3-29
Figure 3.45 Window screen for Input Data Menu – Element Sets	3-30
Figure 3.46 Window screen for selecting nodal points under Input Data Menu – Element Sets.....	3-30
Figure 3.47 Window screen for sample Input Data Menu – Edge Sets	3-31
Figure 3.48 Window screen for selecting edges for Input Data – Edge Sets	3-31
Figure 3.49 Window screen for sample Input Data Menu – Section Cut	3-32
Figure 3.50 Sample of output data from Edge Sets option	3-33
Figure 3.51 Window screen showing cutting sections by using Input Data Menu – Section Cut.....	3-33
Figure 3.52 Define a node path for X-Y plots about discrete nodal results.....	3-34
Figure 3.53 Define a new node path using start/end node numbers.....	3-34
Figure 3.54 Define a new node path along an arc	3-35
Figure 3.55 Define a new node path by inputting node numbers manually	3-36
Figure 4.1 Mesh adjustment options contained in the Edit menu	4-37
Figure 4.2 Mesh adjustment for conventional circular/rectangular plate	4-37
Figure 4.3 Information message before mesh adjustment is processed.....	4-38
Figure 4.4 Refined mesh in both y and z directions.....	4-38
Figure 4.5 Refine mesh for wind turbine foundation	4-39

Figure 4.6 Change element type from Edit menu	4-40
Figure 4.7 Information message before change of element type is processed.....	4-40
Figure 4.8 Mesh after change of element type from 8-node to 4-node	4-41
Figure 4.9 Change integration rule from Edit menu	4-42
Figure 4.10 Window for node definition.....	4-43
Figure 4.11 Window for element definition	4-43
Figure 4.12 User modified mesh by deleting several middle elements	4-44
Figure 5.1 Options contained in the Computation menu	5-1
Figure 5.2 Sample use of Microsoft Notepad for editing input text of Example Problem 1.	5-2
Figure 5.3 Sample use of Microsoft Notepad for view output text of Example Problem 1.....	5-3
Figure 5.4 Sample use of Microsoft Notepad for view message file of Example Problem 1.	5-4
Figure 5.5 Command options contained in Result menu	5-5
Figure 5.6 Sample screen of the Select Step command option	5-5
Figure 5.7 Sample screen of the Deformed Mesh command option.....	5-6
Figure 5.8 Combined display of deformed and undeformed configuration	5-6
Figure 5.9 Sample screen of the Contour Plot command option	5-7
Figure 5.10 Line contour option for vertical displacement in Contour Plot	5-7
Figure 5.11 Shading contour option for vertical displacement in Contour Plot	5-8
Figure 5.12 Shading contour option for moment distribution in Contour Plot	5-8
Figure 5.13 Shading contour option for stress distribution in Contour Plot	5-9
Figure 5.14 Shading contour option for distributed soil reactions.....	5-9
Figure 5.15 X-Y plot for plate deflection along the cut line	5-10
Figure 5.16 X-Y plot for forces of nonlinear springs along a node path	5-10
Figure 5.17 Nodal output	5-11
Figure 5.18 Speed Buttons for graphics manipulation	5-12
Figure 5.19 Slab model displayed using the YZ View (Top) speed button	5-12
Figure 5.20 Slab model displayed using the ZY View (Bottom) speed button.....	5-13
Figure 5.21 Slab model displayed using the XY View (Front) speed button	5-13
Figure 5.22 Slab model displayed using the XZ View (Side) speed button	5-14
Figure 5.23 Slab model displayed using the XYZ View (3D View) speed button.....	5-14
Figure 5.24 Sample screen of the Rotation Angles command option	5-15
Figure 6.1 Example 1 slab, shown with marked and labeled nodal points.....	6-2
Figure 6.2 Example 1 slab shown with nodal constrains and nodal loads.....	6-3
Figure 6.3 Review output file example1.sou for displacement results	6-3
Figure 6.4 Example 1 slab shown with nodal constrains and nodal loads.....	6-4
Figure 6.5 Quarter model of fixed edge slab with center load	6-5
Figure 6.6 Defining edge nodes (BOTTOM edge nodes highlighted).....	6-6
Figure 6.7 Contour plot of displacement in X for slab in Example 2.....	6-7
Figure 6.8 New Project from Template dialog screen for Example 3	6-8
Figure 6.9 Slab of Example 3 shown with soil stiffness and center point load	6-9
Figure 6.10 Slab of Example 4 shown with center load and edge restrains	6-10
Figure 6.11 Deformed shape of slab of Example 4 shown with center load and edge restrains	6-11

Figure 6.12 Deformed shape of slab of Example 5	6-12
Figure 6.13 Contour plot of moment distribution in the y direction of Example 5	6-13
Figure 6.14 Nodal output for moment M_y at node 39 in Example 5	6-13
Figure 6.15 Apply node rotation constraints to simulate non-rotation at column foot in Example 6	6-14
Figure 6.16 Contour plot of moment distribution in the y direction of Example 6	6-15
Figure 6.17 Moment M_y at node 39 in Example 6	6-15
Figure 6.18 Location of columns on raft of Example 7	6-17
Figure 6.19 Deformed shape of slab with column loads of Example 7	6-18
Figure 6.20 Moment distribution of the slab in the y direction of Example 7	6-18
Figure 6.21 Deflection of the slab along the cutting line of Example 7	6-19
Figure 6.22 Location of walls and columns in Example 8	6-20
Figure 6.23 Location of edge set for U-shape wall in Example 8	6-21
Figure 6.24 Deformation of slab in Example 8	6-22
Figure 6.25 Contour of bending moment M_z	6-22
Figure 6.26 Contour of bending moment M_y	6-23
Figure 6.27 Conceptual sketch of drilled-shaft foundation supporting a wind turbine	6-24
Figure 6.28 Dimension of pile cap and the embedded mounting plate with two rings of anchor bolts	6-25
Figure 6.29 New Project from Template dialog screen for Example 9	6-26
Figure 6.30 Node Set screen for Example 9	6-27
Figure 6.31 Definition of Node Sets P1 and P2 for Example 9	6-27
Figure 6.32 Definition of Node Sets P3 and P4 for Example 9	6-27
Figure 6.33 Definition of Node Sets P5 and P6 for Example 9	6-28
Figure 6.34 Definition of Node Sets P7 and P8 for Example 9	6-28
Figure 6.35 Node numbers on the screen for Example 9	6-29
Figure 6.36 Nodal constrain dialog screen for Example 9	6-29
Figure 6.37 Equivalent concentrated loads screen for Example 9	6-30
Figure 6.38 Contour of deflection of Example 9	6-31
Figure 6.39 Contour of bending moment (M_y) of Example 9	6-31
Figure 6.40 Contour of shear on XY plane of Example 9	6-32
Figure 6.41 Bending moment (M_z) along a cutting line of Example 9	6-32
Figure 6.42 Configuration of slab resting on soil of example 10	6-33
Figure 6.43 Input soil properties in Vlasov approach	6-34
Figure 6.44 Define concentrated load at slab center	6-34
Figure 6.45 View node output for one specific node	6-35
Figure 6.46 Distribution of moment M_y along cut line 1 in example 10	6-35
Figure 6.47 Distribution of shear force Q_y along cut line 1 in example 10	6-36
Figure 6.48 Output of γ , k and $2t$ in example 10	6-36
Figure 6.49 Influence of mesh size to shear stress in example 10	6-37
Figure 6.50 WTB foundation resting on soil of example 11	6-38
Figure 6.51 Template for circular wind turbine foundation in example 11	6-39
Figure 6.52 main window for material definition in example 11	6-40
Figure 6.53 Define Young's modulus and Poisson ratio for each slab sets in example 11	6-40

Figure 6.54 Define soil property using Vlasov approach in example 11	6-41
Figure 6.55 Check equivalent nodal loads in example 11	6-41
Figure 6.56 Apply nodal constraints to remove rigid body motion in example 11.....	6-42
Figure 6.57 Deflection along cut line 1 in example 11.....	6-42
Figure 6.58 3D and side view of pile-supported slab in example 12	6-43
Figure 6.59 Load-settlement curves for nonlinear springs in example 12.....	6-44
Figure 6.60 Define two nonlinear spring groups in example 12	6-44
Figure 6.61 Define node set for normal piles with normal stiffness in example 12	6-45
Figure 6.62 Define two nonlinear stiffness groups in example 12	6-45
Figure 6.63 Define distributed element loads in example 12.....	6-46
Figure 6.64 Error message if nonlinear spring does not have adequate range in example 12	6-46
Figure 6.65 Extend data range to cover larger settlement in example 12	6-47
Figure 6.66 Contour of slab deflection in example 12	6-47
Figure 6.67 Deformed slab mesh in example 12.....	6-48
Figure 6.68 Access nodal reaction force from nonlinear spring in example 12.....	6-48
Figure 6.69 Combine Vlasov soil stiffness with nonlinear spring in example 12	6-49
Figure 6.70 Contour of slab deflection from model B and C in example 12	6-49
Figure 6.71 Foundation plan for the piled raft at Urawa City site in example 13a.....	6-52
Figure 6.72 The nonlinear axial stiffness of piles at the piled raft at Urawa City site in example 13a ...	6-52
Figure 6.73 The load from each column at the piled raft at Urawa City site in example 13a	6-53
Figure 6.74 Extend data range to cover the nonlinear axial springs (examples for Piles P1 and P4)	6-54
Figure 6.75 Define four nonlinear stiffness groups in example 13a	6-54
Figure 6.76 Enter column loads in example 13a	6-55
Figure 6.77 Combine Vlasov soil stiffness with nonlinear spring in example 13a	6-55
Figure 6.78 Contour of slab settlement using column loads in example 13a.....	6-56
Figure 6.79 Slab settlement reported by Yamashita and Yamada (1998)	6-56
Figure 6.80 Contour of slab settlement using averaged distributed loads in example 13b	6-57
Figure 6.81 Define node path for X-Y plots in example 13a	6-57
Figure 6.82 Force in nonlinear springs at the specified nodal path in example 13a	6-58
Figure 6.83 Bending moment of M_z in the slab at the specified cutting line in example 13a.....	6-58
Figure 6.84 Delete elements to change slab shape	6-59
Figure 6.85 Irregular L-shape slab and renamed element number	6-60
Figure 6.86 deflection contour of L-shape slab	6-60
Figure 6.87 M_y on cutline 2 in example 14.....	6-60
Figure 6.88 Geometry and parameters of slab in example 15	6-61
Figure 6.89 Mesh of slab mesh imported from an external mesh file.....	6-61
Figure 6.90 Define slab material in example 15	6-62
Figure 6.91 Define concentrated load on slab in example 15	6-62
Figure 6.92 Display concentrated loads on slab	6-63
Figure 6.93 Contour of deflection in example 15	6-63

Tables

Table 2-1: File extensions used in GeoMat2-7

Table 6-1: Parameters of slab and soil in example 106-33

Table 6-2: Parameters of slab and soil in example 116-38

Table 6-3: Settlements, Reactions and load verification in example 12.....6-48

1 Introduction

1.1 General Description

The program is aimed at the solution under static loading of two classes of problems encountered in structural engineering: a soil-supported mat or a structural slab. The mat or structural slab is modeled with linear finite elements. The shape may be rectangular, round, or irregular and the thickness may vary.

For the soil-supported mat, soil is assumed to have a linear response, defined as the subgrade modulus, and is characterized by a set of springs which can vary in stiffness at points under the mat. The springs can reflect horizontal resistance as well as vertical. The solution follows the classical Winkler model. This method of modeling soil has been widely used in the analysis of flexible beams and mats on elastic materials. Since the Winkler model, often referred to as a “one-parameter model”, cannot represent a continuous medium well, the modified Vlasov model by Vallabhan and Das (1988) was also introduced in *GeoMat* for solving the soil-structure interaction in continuous medium. The modified Vlasov model accounts for the effect of the neglected shear strain energy in the soil and shear forces that come from surrounding soil by introducing an arbitrary parameter, γ , to characterize the distribution of the vertical displacement in an elastic foundation using an iterative procedure. The modified Vlasov model has improved the accuracy of the solution computed based on the Winkler soil model.

When a structural slab is analyzed, the supports can be assumed to exist at the edges, or along the interior of the slab, as for beams. The edges of the slab are assumed to be simply supported or subjected to a moment. The supports for the slab may be assumed to be unyielding or set of deflections may be specified. Iteration may be done externally to get agreement between deflection of the slab and that of the supporting beams. This program has the ability to compute deformation and stresses of isotropic or orthotropic plates based on the first order Mindlin thick plate theory. The orthotropic plates include uniform plates with orthotropic reinforcements, plate with orthotropic enhanced ribs, and plate with one way box section. The program also allows the user to define the orthotropic plates/slab with the user-specified properties.

GeoMat allows the user to specify loadings on the surface of the mat or slab as uniform, or distributed, or concentrated as from columns. Horizontal loads may be applied as well as vertical. The finite-element method employed by this program can take into account the variety of loadings as well as the properties of the material in a structure. With modern methods of characterizing material in the mat or slab and with the capability of desktop computers, solutions to finite-element arrays proceed rapidly and with a degree of accuracy in the control of the user.

The program does not set a limit on the number of finite element meshes for modeling. Only the size of RAM in the user's computer will limit the available meshes. The element types include 4-node linear elements, 8-node quadratic elements, and 9-node Lagrangian elements. The soil is assumed to

behave in the linear range, but the user may assign different values of subgrade stiffness at desired locations of the foundation.

1.2 Method of Analyses

The engineer must make several initial steps in performing the analysis and design of a soil-supported mat or a structural slab: 1. estimate the allowable bearing capacity of the mat or the resistance provided by the structural members supporting the slab; 2. estimate settlement and differential movement of the mat or slab; and 3. estimate the moments and shears for the structural design of the mat or slab. Assuming that a geotechnical engineer has provide information on the soil, leading to an estimate of the bearing capacity of the mat, the experience of the structural engineer will allow for the sizing of the mat or slab for the initial analyses.

With regard to the soil-supported mat, the initial data provided by the geotechnical engineer may show a range of values because settlement and differential movement are difficult to estimate because the settlement is dependent on the stiffness of the soil and on the rigidity of the mat.

Loadings on a mat or a slab can vary widely in nature and magnitude. The finite-element method has been the best tool to take into account the variety of loadings as well as the properties of the material in a structure. With modern methods of characterizing material in the mat or slab and with the capability of desktop computers, solutions to finite-element arrays proceed rapidly and with a degree of accuracy in the control of the user.

For a given set of loadings, deformations and movements within the mat and slab can be computed, along with bending moment and shear stress at any point with the material. These results provide the engineer with information on which to base the design of the system. The method can be applied to a mat or slab that is circular, rectangular, or an irregular shape, leading to a powerful analytical tool interpreted comparably. Furthermore, in only a few instances have measurements been made, using the necessary instrumentation, that reveal the detailed manner in which the foundation interacts with the supporting soil.

For the mat on foundation, information is available in technical literature on subgrade modulus. Plainly, the reaction of soil against the base of a foundation is dependent on the deflection of the foundation; therefore, the value of the subgrade modulus is not constant. For many problems of a mat on foundation, however, a constant value of subgrade modulus will lead to acceptable solutions because deflections in most cases are relatively small. The ability to compute the settlement of the mat at all points will allow the engineer to use judgment about the value of modulus being used and adjustment in the value can be made where indicated. The ability to study the influence of the various parameters that entered in the problem gives the engineer powerful tools for analysis and design.

1.3 Program Features and Change Log

1.3.1 GeoMat 1.0 (2004)

The initial version was able to consider the soil-structure-interaction by using a generalized Finite-element thick-plate model, and then analyze the behavior of slab or mat supported by soil springs as subgrade. The first version of Program *GeoMat* has the following features that are designed to enhance the ability of the engineer to obtain usable results.

- The program employs well-established analytical solutions for soil-structure interaction under static loading.
- The program does not set a limit on the number of finite element meshes for modeling. Only the size of RAM in the user's computer will limit the available meshes. The element types include 4-node linear elements, 8-node quadratic elements, and 9-node Lagrangian elements.
- The soil is assumed to behave in the linear range, but the user may assign different values of subgrade stiffness at desired locations of the foundation.
- The user may input various concentrated loads at nodal points, distributed loads at any element, and uniform loads for an entire mat or slab.
- The user may specify a mat or slab with any shape. The program has the capability to generate automatically finite meshes for mats or slabs with rectangular shapes or circular shapes.
- The program will display the layout of finite-element meshes with nodal numbers and element numbers.
- The program will generate contour graphics of slab deformation and stress distribution.

A Windows GUI (Graphic User Interface) was used for input screens and file operation. All variables have on-line text description or graphical symbols for definition. Also, all input data can be saved and retrieved from disk. Finally, graphical output of pressure distribution, deflection moment, and shear could be printed.

1.3.2 GeoMat 2.0 (2014)

The second version was developed in 2014 with several significant upgraded features and is fully compatible with the latest operation systems including Microsoft Windows 7 and 8.

- The program does not set a limit on the number of finite element meshes for modeling. Only the size of RAM in the user's computer will limit the available meshes. The element types include 4-node linear elements, 8-node quadratic elements, and 9-node Lagrangian elements.
- The program enhanced the contour graphics of slab deformation and stress distribution.
- The program added function for "Section Cut" which is a useful tool to plot deflection, bending moment, or stress components along one or more straight cut lines.
- This version has the ability to compute deformation and stresses of space orthotropic plates. Three common types covered in this version are: uniform plates with orthotropic reinforcements, plate with orthotropic enhanced ribs, and plate with one way box section.
- This software also provides the option for inputting user-defined orthotropic properties.

1.3.3 GeoMat 3.0 (2022)

The third version was developed in 2021 and 2022 with several significant upgraded features and is fully compatible with the latest operation systems including Microsoft Windows 10 and 11.

- The program includes the modified Vlasov model by Vallabhan and Das (1988) for solving the soil-structure interaction in continuous medium. The modified Vlasov model has improved the accuracy of the solution computed based on the Winkler soil model.
- The program enhanced the performance of 4-node linear elements in addition to the available choices on a high-order quadratic elements (8-node) and Lagrangian elements (9-node).
- The program added a practical example which is a useful reference for modeling the mat foundation supported by piles (pile-raft foundations).
- This version has the ability to assign the external loads from anchor bolts, which may be arranged in circular rings, directly onto the associated nodes as inputted concentrated loads.

1.3.4 GeoMat 3.1 (2022)

This version was implemented in 2022. Version 3.1 adds the capability to handle nonlinear response of piles to support a slab. Piles are simulated as nonlinear springs while the load-settlement curves are assigned to these springs. Nonlinear behavior of piles can be in tension/compression.

- Add user defined nonlinear spring
- Add capability to use nonlinear spring to provide vertical support to a slab foundation
- Improve user experience to manipulate objects in view
- Add output of soil reactions from various soil stiffness/ nonlinear springs

1.3.5 GeoMat 4.1 (2025)

This version adds the ability to use nonlinear spring under the slab.

1.3.6 GeoMat 4.2 (2025)

This version adds ability to create a ring mesh or create a model from an external mesh file. This version also updates the ability to let users modify an existing mesh.

1.4 Technical Support

Although computer program *GeoMat* was designed to be distinguished by its ease of use and by its helpful User's Manual, some users may still have questions. The technical staff at ENSOFT strongly supports all registered users with questions related to the installation or use of *GeoMat*, according to the stipulations presented below. The software is provided with free maintenance service for the first year. After the first year the user is encouraged to purchase the yearly maintenance services. The yearly maintenance services include free download of the latest version and free technical support as described below.

1.4.1 Preferred Methods of Software Support

Software support is given, in order of preference, by the following methods:

- Electronic mail to: support@ensoftinc.com
- Fax to: (512) 244-6067
- Telephone call to: (512) 244-6464, extension 2

Users are strongly encouraged to utilize electronic means of support via email. In any event, users should include the following information when requesting for software support: software version (obtain from the Help/About menu), description of the problem copy, input-data file attachment, contact person and licensed company or user name (or name and office location of the licensed company).

Although immediate answers (less than two hours during regular working hours) are offered on most technical support requests, up to three business days should be expected in case of difficulties or conflict of schedules. The users will be informed of ENSOFT's reception of their electronic mails requesting software support if the problem is complicated and cannot be resolved within two business days.

1.4.2 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 15 minutes of each call if the user has a current maintenance policy. A charge of \$90.00 US dollars per hour (\$1.50 US dollars per minute) may come into effect for every additional minute after the initial 15 minutes. Applicable charges will be billed to the user.

Software support by any of the methods (fax, email or telephone) is free for all users that have a current annual maintenance policy.

1.4.3 Upgrade Verification and Internet Site

Starting from *GeoMat* v.2 (2014) the software provides options for the user to check the most recent maintenance release through an internet connection by selecting Help/Check for Updates. This command starts the default internet browser and will display the user's maintenance expiration date, the user's software release number and the most recent release number that is available for downloading.

If the user's version is not the latest version and the maintenance has not been expired, the user can download the latest version from our web server directly (www.ensoftinc.com). Users may also consult our internet site for additional information on software updates, demos, and new applications; technical news; and company information.

1.4.4 Changes of 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.

2 Installation and Getting Started

2.1 Installation Procedures

Program *GeoMat* is distributed with an associated USB key (hardware key or dongle). The hardware key consists of a device that is attached to an empty USB port (or USB hub) of the computer in use (or in the designated software server in the case of local network licenses). This method of software protection has been found to provide compatibility with existing operating systems, better stability than other alternatives, and allows users to obtain software updates or replacements via downloads from the internet.

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

- An open USB port
- At least 30MB of free space
- Windows 7,8,10,11

The file, contained in the distribution media, is a Windows-based program which will assist the user in installing all program modules into a user-selected directory with the proper settings used for the Windows environment. The user is assumed to have basic knowledge in running applications under MS Windows. The following steps are recommended for a successful installation.

2.1.1 Installation of Single-User Version

Users are normally supplied with CDROM (software files and user manual) and hardware (USB) lock. This version of *GeoMat* has been tested to be compatible with the following versions of Windows 7/8/10/11.

The following guidelines are recommended to be followed during the installation process of *GeoMat* for single-user licenses.

1. It is our recommendation that the hardware lock (USB dongle) should not be plugged in the computer prior to the software-installation process. The USB dongle can be plugged into the computer after the software has been installed successfully and the computer should recognize the USB device immediately.
2. Start Windows®. Insert the distribution CD-ROM in the appropriate tray. If the main installation program does not start automatically, click on the Start button and select Run. On the command line, type *d:\setup.exe* or *e:\setup.exe*, where *d:* or *e:* represents the drive that contains the distribution CD. Press *e* or click OK to execute the command and start the main installation program for ENSOFT's software. A screen similar to the one in Figure 2.1 should appear.
3. Click anywhere on the *GeoMat* icon and then click on the **Install Standard** button to start the installation of the single-user version of *GeoMat*.

4. The user should read the license agreement shown in Figure 2.2. Please click "Yes" if you agree and would like to proceed.
5. If your license is for a single-user, select the "Single-User License" option and click Next.
6. The user will be provided with an option to select a drive and directory for the installation of *GeoMat* ; the default is *c:\Ensoft\GEOMAT2025*. If the desired directory does not exist, the installation program will automatically create a new directory in the chosen hard drive.
7. Most distribution files will be copied to the installation directory (either the default of the one selected by the user). However, certain files are copied to system directories in Windows and some Windows services are started during installation. The name of the system files, their exact locations, and the type of services that are started may change. Up to date settings are usually included in the *readme* file copied to the selected installation directory. The *readme* file is named according to the operating system in use.
8. Follow the dialog boxes until the installation finishes.
9. Plug the hardware lock (USB device) into one of the available USB ports in your computer. Typically the computer will recognize the USB device automatically by finding the USB driver from the Ensoft CD-ROM. If the computer cannot recognize the USB key automatically the hardware wizard prompt from the operation system will guide the user through the required driver installation.



Figure 2.1 Main installation screen for ENSOFT software (may change with time)

After the installation is finished, it is usually not necessary to reboot Windows for the program to run. The user may run the program by double-clicking its icon in the **ENSOFT/GEOMAT** folder located in the **Start/Programs** menu.



Figure 2.2 License-agreement screen for ENSOFT software (may change with time)

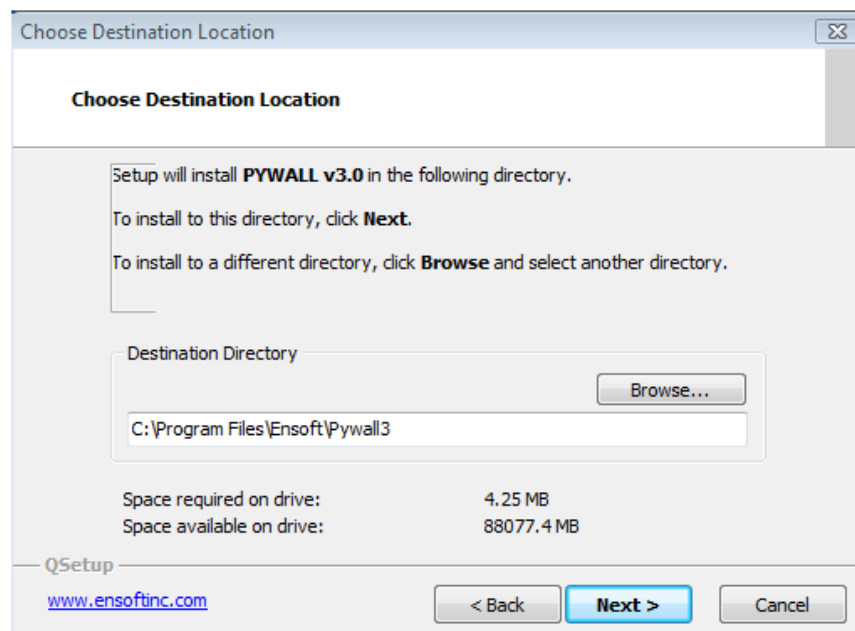


Figure 2.3 File-location screen for ENSOFT software (may change with time)

2.1.2 Introduction of Network Version

Special network licenses and hardware locks are available for users that desire to operate *GeoMat* on a Windows network. The network version is limited to users with two or more licenses at the same physical site. Discounted rates apply for users purchasing multiple licenses for the same site.

Network versions of *GeoMat* have special subroutines written for installations in “software servers” and for installations of “individual clients”. 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 *GeoMat* as long as the network lock is attached to its parallel port and the “server” version of the software is installed on its hard drive. Software “clients” may be all other computers of the network that have the program installed as clients. Software clients do not need any hardware lock attached to their computers. The program installed in “software clients” will be allowed to run as long as the computer designated as “software server” is running on the network with the proper operating system and with its network lock secured in place.

Users of the network version of *GeoMat* are allowed to have the software installed in as many computers as desired within their specified IP ranges of their local Windows network. However, only a number of users equal to the total number of purchased licenses will be able to operate the program at the same time.

2.1.3 Installation of Network Version

To install this program, the user must have network privileges to modify the Windows registry. In large computer networks, this is usually only available to network administrators. They are the ones recommended to proceed with these installations.

1. The network administrator must first define the computer that will be carrying the network lock for this software. The computer carrying the network lock is known as the “server” for this particular computer program. Notice that the “server” may be any computer in the network, and not necessarily the existing file server or network server.
2. INSTALLATION TO A “SERVER”: The “server” computer must be turned on and logged into the network to enable other users (i.e., “clients”) to access this computer program. Please attach the network lock (USB device) to the software server after the installation is completed.
3. INSTALLATION TO “CLIENTS”: It is not necessary to connect the network lock to client computers during installation or during program executions. Once the program is executed, the installed network protocols will automatically recognize the computer where the network key is residing and the number of simultaneous users.

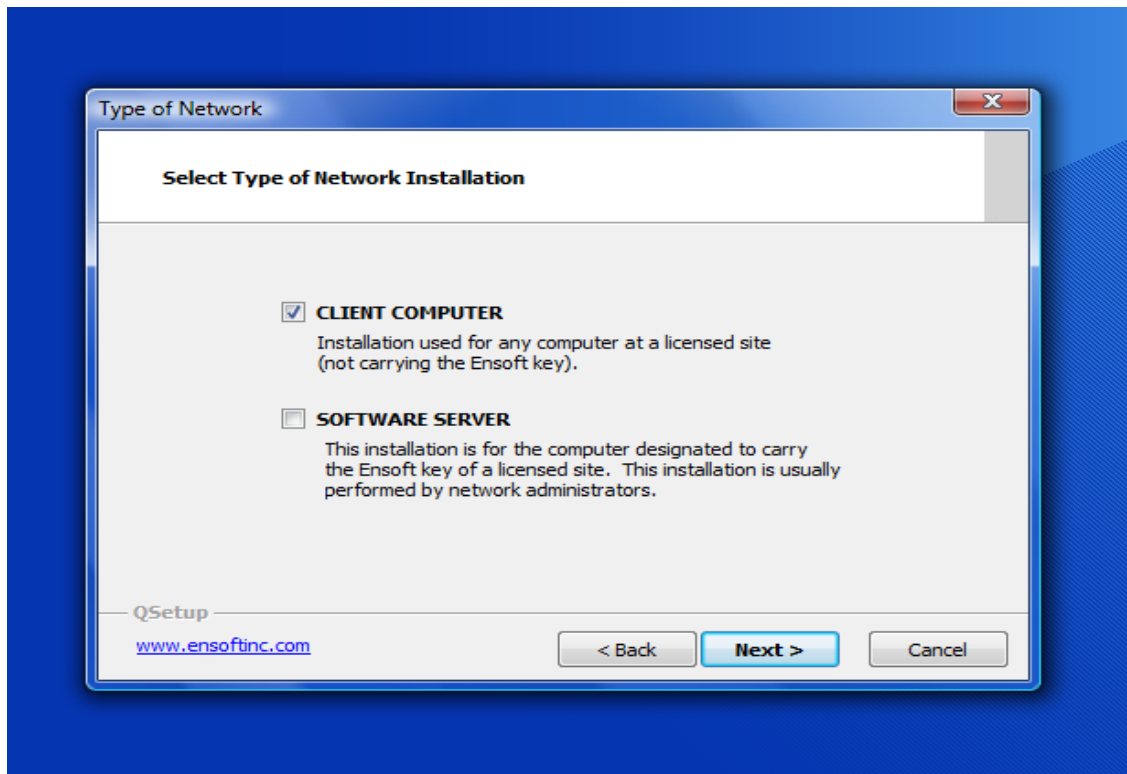


Figure 2.4 File-location screen for ENSOFT software (may change with time)

4. Start Windows.
5. Insert the CD-ROM in the appropriate tray.
6. If the main installation program does not start automatically, click on the **Start** button and select **Run**. On the command line, type `d:\setup.exe` or `e:\setup.exe`, where *d:* or *e:* represents the drive that contains the distribution CD. Press the Enter key or click **OK** to execute the command and start the main installation program for ENSOFT's software. A screen similar to the one previously shown in Figure 2.1 should appear.
7. Click anywhere on the *GeoMat* icon and then click on the Install Network button to start the installation of the network version of *GeoMat*.
8. The user will be provided with an option to select a drive and directory for the installation of *GeoMat*; the default is `c:\Ensoft\GEOMAT2025`. If the desired directory does not exist, the installation program will automatically create a new directory in the chosen hard drive.
9. The installation program will automatically recognize the version of Windows that is running on the destination computer. The user will be prompted to select an installation to a "Client" or to a "Server" computer. If desired, the network administrator may install the program to a single directory in a file server. In this case, the software must first be installed in the file server as a "software server" and icons pointing to the `GEOMAT.exe` file (residing in the server) can later be created in each individual client computer.
10. Most distribution files will be copied to the installation directory (either the default or the one selected by the user). However, certain files are copied to system directories in Windows and some Windows services are started during installation. The name of the system *readme* files,

their exact locations, and the type of services that are started may change. Up to date settings are usually included in the *readme* file copied to the selected installation directory.

11. After the installation is finished, it is usually not necessary to reboot Windows for the program to run. The user may run the program by double-clicking its icon in the folder located in the menu.

Installers of network licenses also can refer to a separate booklet with installation instructions for the Network version of this product. The document can be downloaded from the ENSOFT web site ([http://www.ensoftinc.com/updates/Ensoft Network Installation Notes.pdf](http://www.ensoftinc.com/updates/Ensoft%20Network%20Installation%20Notes.pdf)) or can be requested via email to support@ensoftinc.com

For installation of network licenses on local client computers there is an option for command-based installations that are completely silent (performed without other user input). Instructions for silent installations on client computers can be downloaded from the ENSOFT web site using the following link:

[http://www.ensoftinc.com/updates/Silent Install on Client Computers.pdf](http://www.ensoftinc.com/updates/Silent%20Install%20on%20Client%20Computers.pdf)

Alternatively, the document can be requested via email to support@ensoftinc.com

2.1.4 Backup of Original Software

The distributed software may be copied for backup purposes. The program may be installed in several computers at the same time. However, unless network licenses are purchased, the program will only operate in computers that carry the appropriate hardware lock.

2.2 Software Updates

Occasionally, ENSOFT will produce software improvements and/or fixes and place the latest software programs on ENSOFT's internet site. Software users may freely download the latest program update from the following site:

<http://www.ensoftinc.com>

The following steps are recommended for a successful installation of software updates.

1. Download appropriate file specified above (either *single-user version* or *network version*) to a local directory. If the download was not made to the computer carrying the hardware lock, the downloaded file must be copied to the appropriate computer.
2. Run the downloaded file in the appropriate computer and follow instructions. This can be done by double clicking within Windows Explorer, or through the Start/Run/Browse menu.
3. The downloaded file is a compressed "zip" file and normally decompresses into the temporary Windows directory (usually *c:\windows\temp*). After decompression of files, the installation program (*setup.exe*) is started automatically from within the temporary directory. All files in the temporary directory are deleted automatically after completing the installation.

2.3 Getting Started

A first windows screen showing the menu choices, speed buttons, and operational controls of program *GeoMat* is presented in Figure 2.5. The following paragraphs provide a short description of the operational features of *GeoMat* and should quickly enable the user to get started with the program.

Several additional files are created in every new *GeoMat* run. A general description of these files is presented in Table 2-1. Every run of *GeoMat* thus generates four text files in the same drive and directory where the input-data file was saved or opened. Any of these files may be opened with standard text editors or word-processing programs.

Table 2-1: File extensions used in GeoMat

File Name Extension	Usage Description	File Format	Example Files
*.slb	Input data file	GeoMat file	example1.slb
*.sin	Input data file	Text file	example1.sin
*.sou	Output data file	Text file	example1.sou
*.sms	Message file	Text file	example1.sms
*.sds	Timestep data file	Text file	example1.sds
*.cut	Data for section cut	Text file	Example1.cut

2.3.1 Starting the program

The program is started by double clicking the left mouse button anywhere in the *GeoMat* icon. A new window will appear on the screen, with the following top-menu choices: **File, Edit, View, Input Data, Assign Data, Computation, Transform, Results, Options, Selection, Window, Help**. As a standard Windows feature, all underlined letters of top menu operations indicate that choices may also be accessed by keyboard combinations of ALT + [letter] (where [letter] represents the desired underlined letter).

- **New:** Create a new data file.
- **Open...:** Open an existing data file. If a partially-completed *GeoMat* input file, or an invalid data file is opened, an information window reporting that an “invalid or incomplete” file is being opened. Clicking OK dismisses the message, and the previously saved data should be available. If a complete input file is loaded, an information window reporting that “Data File: (name of file), has been read by GeoMat ” should appear, and the user should click OK.
- **Save:** Save input data under the current file name.
- **Save As...:** Save input data under a different file name.
- **Exit:** Exit the program. If the input file was modified but unsaved, a prompt will appear asking if the user would like to save changes.

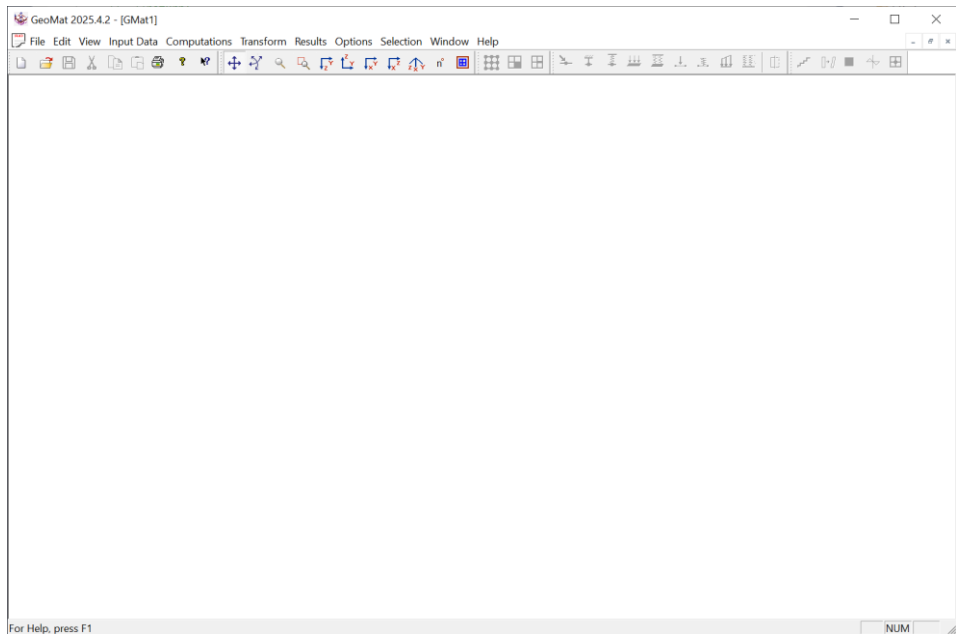


Figure 2.5 Layout of GeoMat program screen

A list of recently opened files is displayed in between **Save As...** and **Exit**.

2.3.2 File Management

The **File** menu option contains five submenus, as shown in Figure 2.6; they are:

- New** to create a new data file.
- New from Template** ... to create a new mesh data file with the template.
- Open** to open an existing data file.
- Close...** to close an existing data file.
- Save** to save input data under the current file name.
- Save As...** to save input data under a different file name.
- Import External Mesh** to create a new mesh from external mesh file
- Print** to print the current graphics.
- Print Preview...** to view the graphic layout before sending to the printer.
- Print Setup** to setup printer configuration.
- Page Setup** to setup page border and configuration.
- Save As Bitmap** to save graphics using Bitmap format.
- Exit** to exit Program GeoMat.

Several additional files are created in every new *GeoMat* run. A general description of these files is presented in Table 2.1. Every run of *GeoMat* thus generates four text files in the same drive and directory where the input-data file was saved or opened. Any of these files may be opened with standard text editors or word-processing programs.

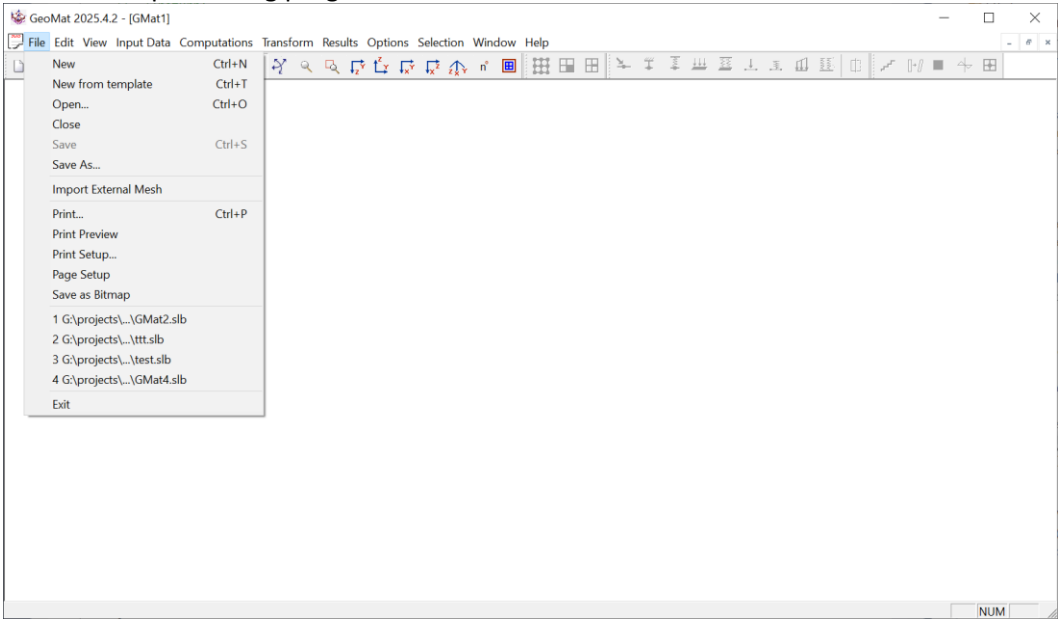


Figure 2.6 Options contained in the File menu

2.3.3 View Management of Program Window

The **View** menu contains several toolbar options, as shown in Figure 2.7. By default, these options are turned on when the program is launched. The choices are listed below, along with a general description of their use.

- File Toolbar** this option displays the toolbar associated with the options contained in the File and Help menus.
- View Toolbar** this option displays the toolbar associated with the options contained in the Transform menu.
- Select Toolbar** this option displays the toolbar associated with the options contained in the Selection menu.
- Actions Toolbar** this option displays the toolbar associated with the options contained in the Input Data menu.
- Status Bar** this option displays the status bar at the bottom of the screen.

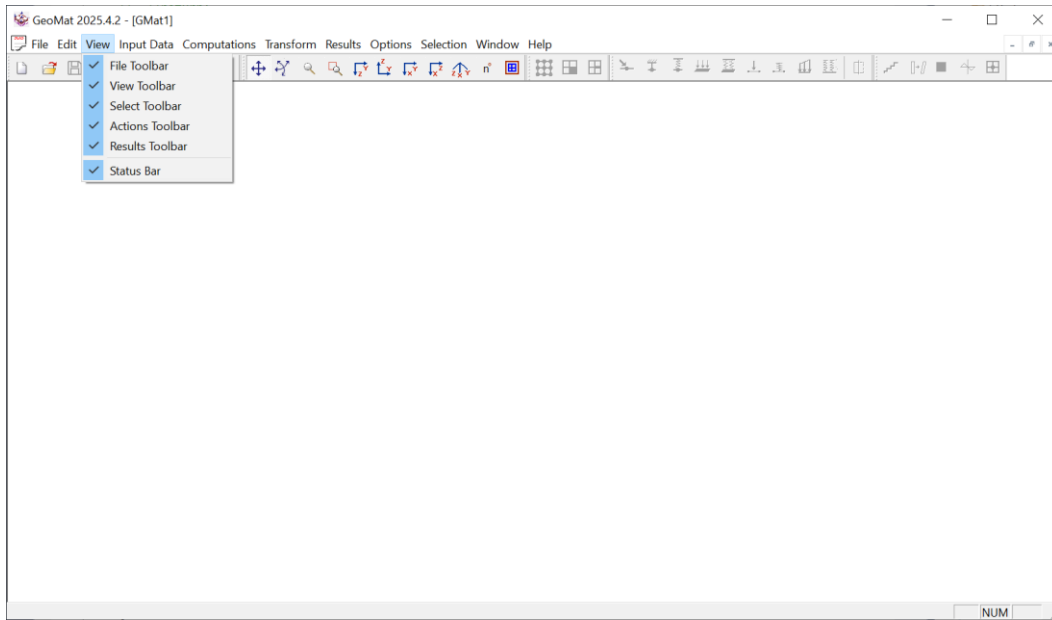


Figure 2.7 Options contained in the View menu

2.3.4 Data Input for Program Operation

The **Data Input** menu is provided for the control of program variables and preferences. The listing of menu options is shown in Figure 2.8 and the different submenu choices are briefly described below. The user may select any of the options without concern for the sequential order.

- Title** this option allows the user to title the current evaluation. It is displayed in the Input File and Output File.
- Material Properties** ... as a default, the program considers the cross-section of the slab to be uniform. Young's modulus, Poisson's ratio, and slab thickness may be modified as needed. This version has the ability to compute deformation and stresses of space orthotropic plates. Four types are covered in this version: uniform plates with orthotropic reinforcements, plate with orthotropic enhanced ribs, plate with one way box section and general orthotropic plate.
- Node Coordinates** the user should input in this box the coordinates of the mesh to be drawn on the slab. This action is already performed for the user if the problem was designed using the New from template... option in the File menu.
- Time Stepping** the user should input in this box the coordinates of the mesh to be drawn on the slab.
- Soil Stiffness** this option allows the user to solve soil-slab interaction using Mindlin's Equation or Vlasov approach.
- Nonlinear Spring** this option allows the user to define one or more nonlinear springs to simulate vertical behavior of piles in tension/compression
- Element Type** three element types are available for use in the analysis: Linear 4-noded, Quadratic 8-noded, and Lagrangian 9-noded.

Element Connectivity ..	this window allows the user to specify the connecting edges of the elements. This action is already performed for the user if the problem was designed using the New from template... option in the File menu.
Integration Points	this option allows the user to select the number of integration points to be used in the analysis: 1x1, 2x2, or 3x3.
Nodal Constraints	this box allows the user to apply constraints at nodal points.
Nodal Loads	this option allows the user to apply loads at nodal points.
Nodal Stiffness	this option allows the user to apply constant stiffnesses or nonlinear stiffness at nodal positions
Distributed Loads	this option allows the user to select the element location and intensity of distributed loads.
Distributed Stiffness ..	this option allows the user to select the element location and intensity of distributed stiffnesses.
Concentrated Loads ..	this option allows the user to place a concentrated load at any location on the slab.
Equivalent Concentrated Loads ..	this option allows the user to place concentrated loads from anchor bolts of the mounting plate to the top of the foundation. The program will calculate the location of each anchor bolts arranged in a circular pattern and the distributed load on each anchor bolt based on the user-specified data.
Concentrated Stiffness	this option allows the user to place a concentrated stiffness at any location on the slab.
Edge Loads	this option allows the user to place a load along an edge.
Edge Stiffness	this option allows the user to place a stiffness along an edge.
Node Sets	this option allows the user to create node sets in order to facilitate the ease of defining nodal loads, stiffnesses, and constraints.
Element Sets	this option allows the user to create node sets in order to facilitate the ease of defining distributed loads, distributed stiffnesses, and material properties.
Edge Sets	this option allows the user to create node sets in order to facilitate the ease of defining edge loads and stiffness
Section Cut	This option allows users to define cut lines for XY curve plots.
Node Path	This option lets users define node paths to output discrete nodal reactions in X-Y plots

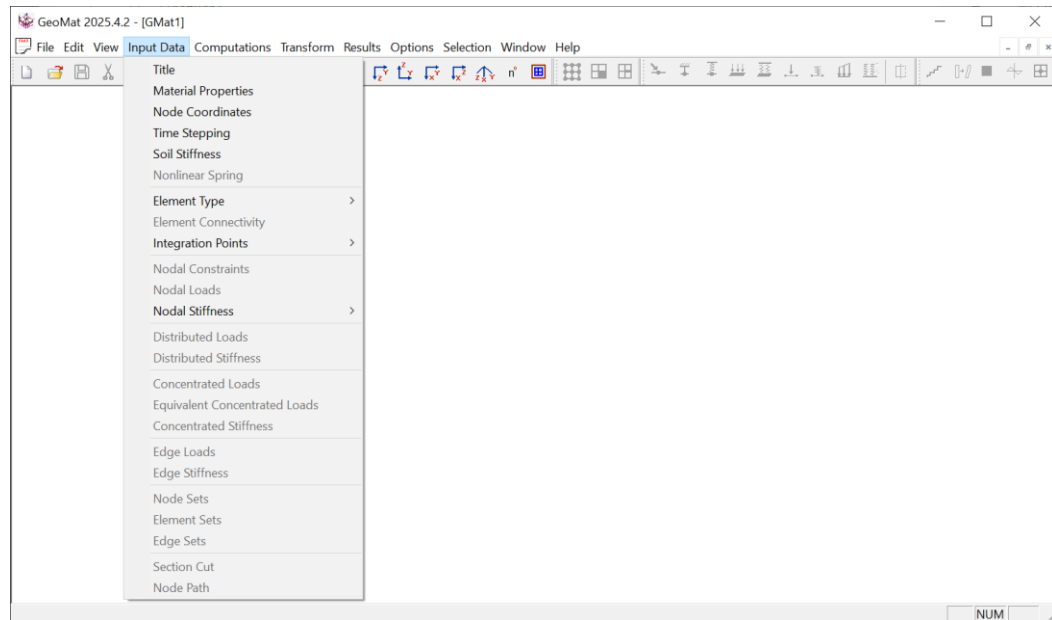


Figure 2.8 Options contained in the Input Data menu

2.3.5 Computation Options

The **Computations** menu is provided to run the analytical computations after all data are entered and saved. After the computation is executed successfully, this menu also provides options for the reviews of plain-text input data, notes produced during computation, and output data. Submenu choices, shown in Figure 2.9, are briefly described below.

- Run Analysis** this option is chosen to run the analytical computations. This option should be selected after all data have been entered and saved.
- View Intput File** this option calls Microsoft Notepad to observe and/or edit the analytical input data in plain-text format. The option becomes available after the input data has been saved to disk, or when opening an existing input-data file.
- View Output File** this option calls Microsoft Notepad to observe, format, and/or print the analytical-output data. The option becomes available only after a successful run has been made. Certain output files may be too large for the Microsoft Notepad editor, so other text editors would have to be used (Microsoft WordPad should be able to open most text files).
- View Message File** this option calls Microsoft Notepad to observe the processor run notes which may include error messages.

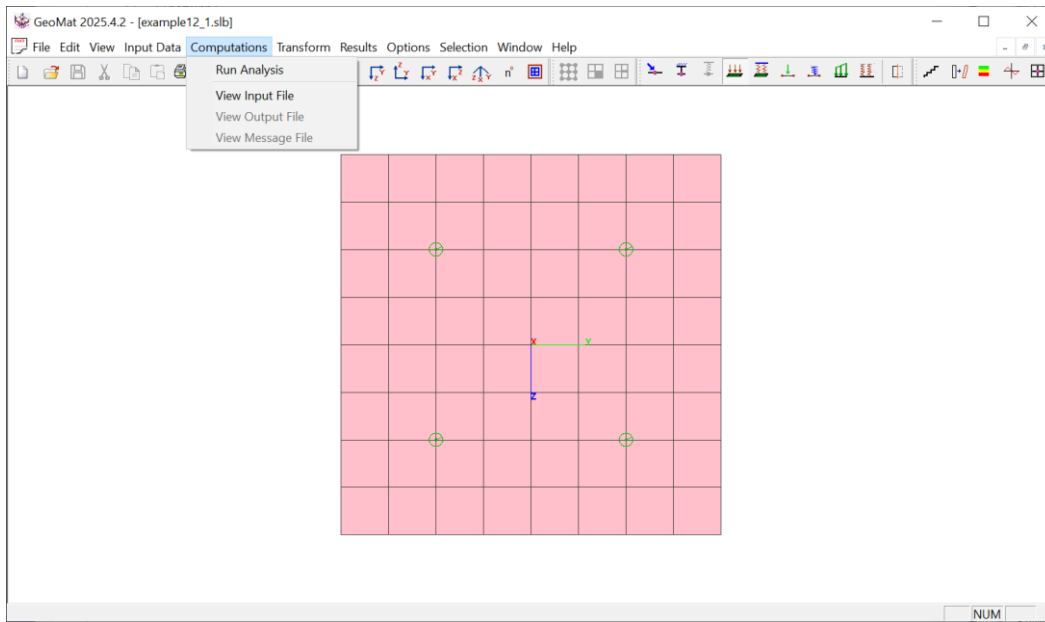


Figure 2.9 Options contained in the Computation menu

2.3.6 Manipulating the Slab Model

The **Transform** menu is used to manipulate and transform the slab model displayed in the program window. The user may also access these operations via the View Toolbar shortcuts. Submenu choices, shown in Figure 2.10, are briefly described below.

Panning	this function allows the slab model to be moved in the window.
Rotate	this function allows the slab model to be rotated around the origin.
Zoom In/Out	this function zooms in and out on the slab model.
Restart	this function resets the view to the original YZ plane view.
YZ View (Top)	this orients the slab in the YZ plane, looking at the top of the slab.
YZ View (Bottom) ...	this orients the slab in the YZ plane, looking at the bottom of the slab.
XY View (Front)	this orients the slab in the XY plane, looking at the front of the slab
XZ View (Side)	this orients the slab in the XZ plane, looking at the top of the slab.
XYZ View (3Dview) ..	this orients the slab in an orthographic 3D view.
Rotation Angles	this option allows the user to specify the desired rotation angle.

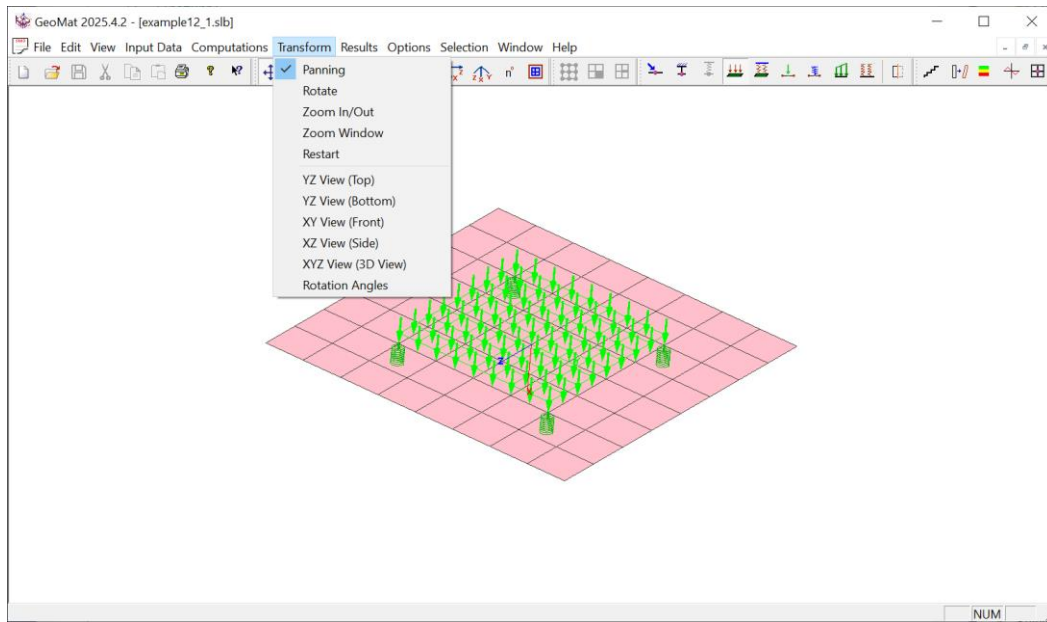


Figure 2.10 Options contained in the Transform menu

2.3.7 Results

The **Results** menu provides additional options for viewing the slab after an analysis has been completed. The submenu options, shown in Figure 2.11, are described below.

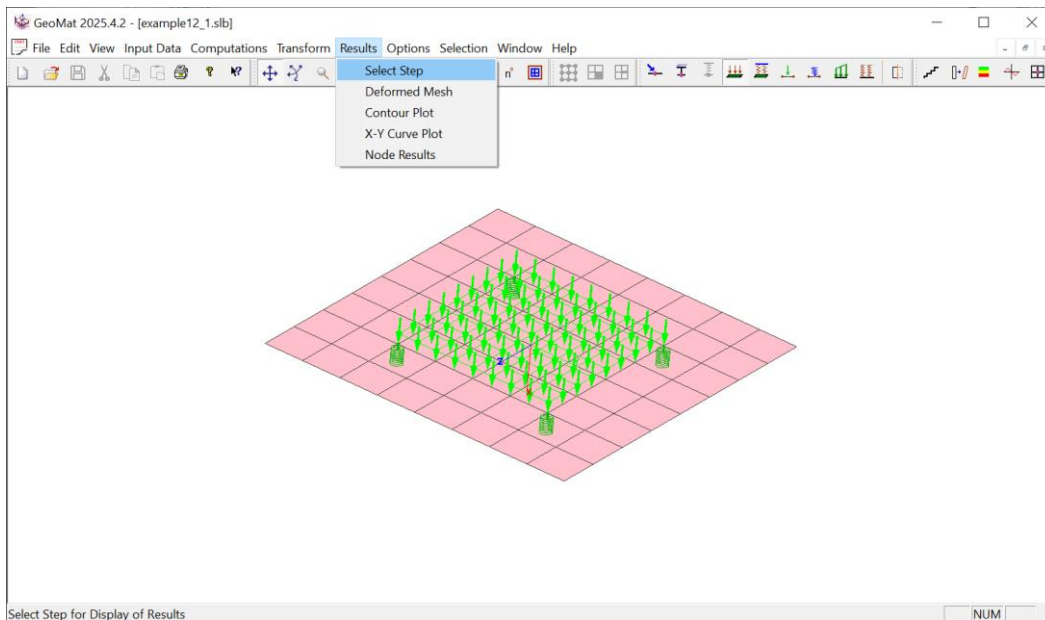


Figure 2.11 Options contained in the Results menu

Select Step this option allows the user to choose the number of steps used in the solution.

<u>D</u>eformed Mesh	this option displays the deformed shape of the slab, based on the user's specified shape deformation factor. The undeformed shape may also be displayed alongside the deformed shape.
<u>C</u>ontour Plot	this option creates a contour plot of various output information, such as deflection, rotation, shear, moment, and stress.
<u>X</u>-Y Plot	this option creates X-Y plots of various output information along the user-selected cutting lines or node paths, such as deflection, rotation, moment, stress components, soil reactions or forces in nonlinear springs.
<u>N</u>ode Results	this option displays deformation, plate forces, reactions, and plate stresses of one specific node.

2.3.8 Option Menu

The **Options** menu provides various options for displaying data about the slab on-screen. Submenu options, shown in Figure 2.12, are briefly described below.

<u>M</u>ark Nodes	this option marks the nodal points.
<u>N</u>ode Numbers	this option displays the node numbers.
<u>E</u>lement Numbers ...	this option displays the element number.
<u>B</u>oundary Only	this option shows the slab and its edge boundary.
<u>L</u>ocal Axes	this option displays the axes associated with the slab.
<u>G</u>lobal Axes	this option displays the global axes of the workspace.
<u>S</u>olid Face	this option shows the elements as solid units.
<u>S</u>how Thickness	this option displays the thickness of the slab.
<u>N</u>odal <u>C</u>onstraint	this option displays the location of the user input nodal constraints
<u>N</u>odal <u>L</u>oads	this option displays the location of the user input nodal loads.
<u>N</u>odal <u>S</u>tiffness	this option displays the location of the user input nodal stiffnesses.
<u>C</u>oncentrated <u>L</u>oads	this option displays the location of the user input concentrated loads.
<u>C</u>oncentrated <u>S</u>tiffness	this option displays the location of the user input concentrated stiffnesses.
<u>C</u>oncentrated <u>T</u>otal Stiffness	this option displays the nodal, concentrated, and distributed stiffnesses.
<u>D</u>istributed <u>L</u>oads	this option displays the user input distributed loads.
<u>D</u>istributed <u>S</u>tiffness .	this option displays the user input distributed stiffness.
<u>E</u>dge <u>L</u>oads	this option displays the user input edge loads.
<u>E</u>dge <u>S</u>tiffness	this option displays the user input edge stiffnesses.

Font and Color this option lets users control fonts and colors used for node/element number.

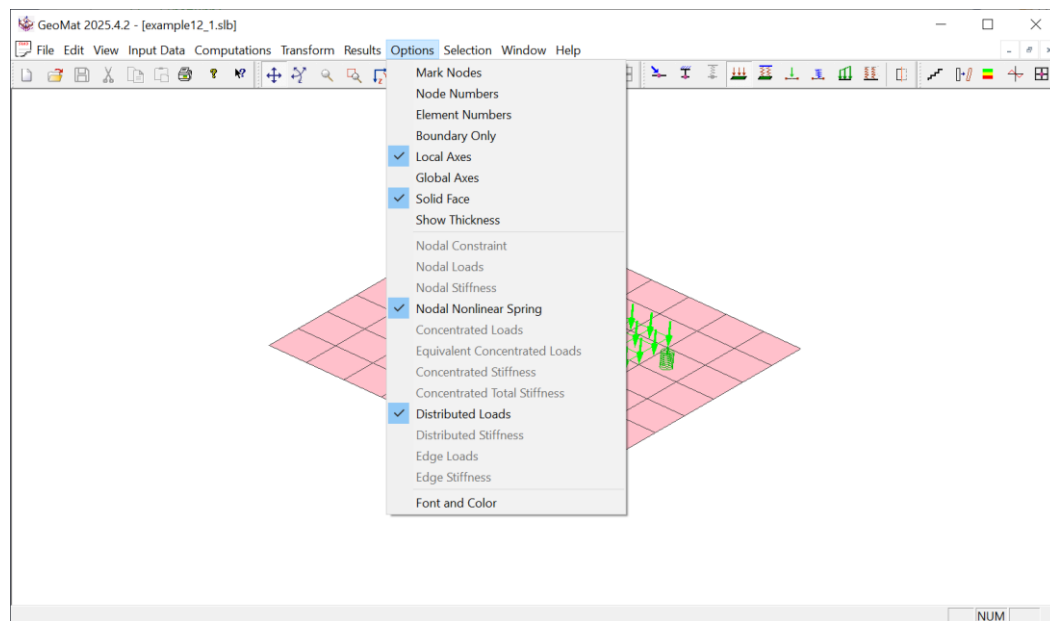


Figure 2.12 Available items contained in the Options menu

2.3.9 Selection Menu

The Selection menu provides options for selecting different members of the slab. Submenu options, shown in Figure 2.13, are briefly described below.

Select Node this option allows the user to select a node using the mouse button.

Selection Element ... this option allows the user to select an element using the mouse button.

Select Edge this option allows the user to select an edge using the mouse button.

2.3.10 Window Menu

The Window menu provides the user with various ways to view the active documents. Submenu options, shown in Figure 2.14, are briefly described below. The open windows are listed at the bottom of the menu, after the divide line.

New Window this option launches a copy of the current open window.

Cascade this option stacks the open windows on top of one another.

Tile Horizontal this option tiles the open windows horizontally.

Tile Vertical this option tiles the open windows vertically.

Arrange Icons this arranges the minimized windows

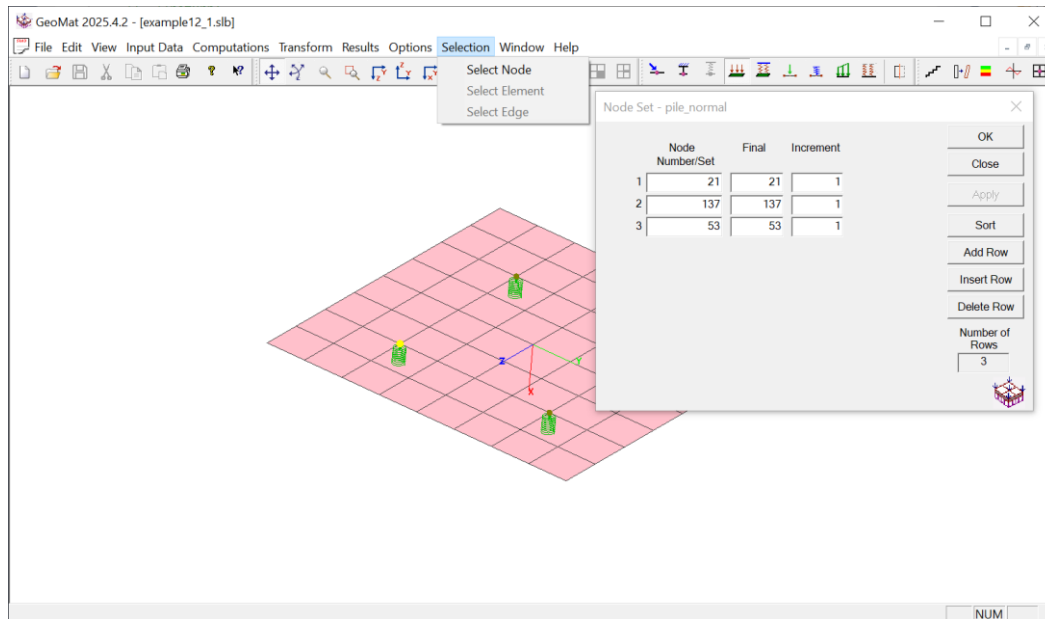


Figure 2.13 Options contained in the Selection menu

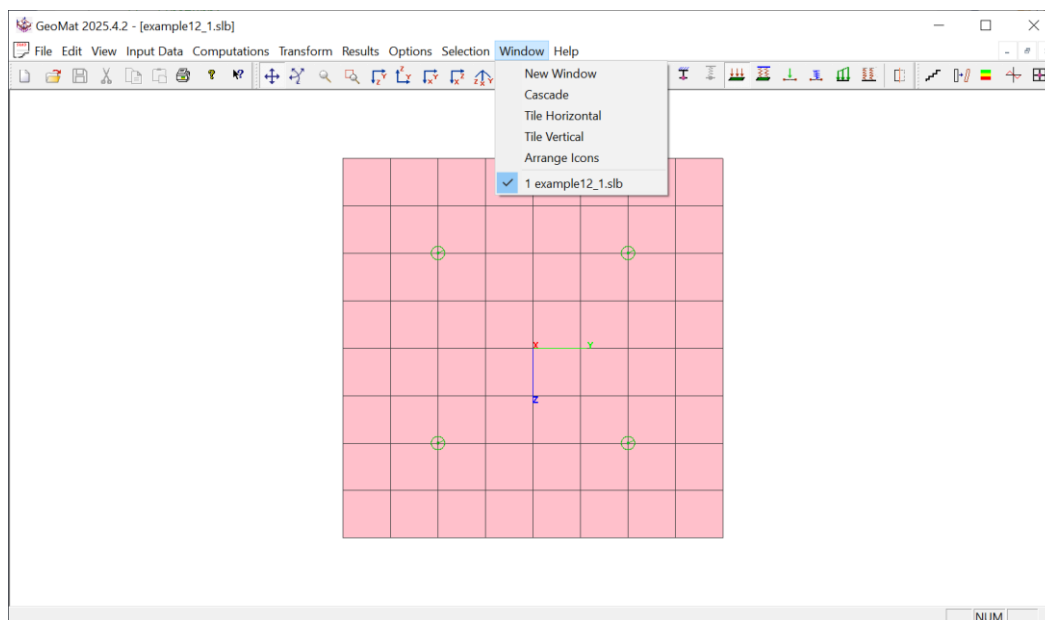


Figure 2.14 Options contained in the Window menu

2.3.11 Help Menu

The **Help** menu provides an online help reference on topics such as: using the program, entering data, information about variables used in the program, and methods of analyses. The **Help** menu may be

accessed at any time while in *GeoMat*. Submenu options, shown in Figure 2.15, are briefly described below.

- Help Topics** the main reference files for help are accessed through this submenu option. Clicking on this option provides a screen with reference help for the following topics: File Menu, Edit Menu, View Menu, Input Data Menu, Computations Menu, Transform Menu, Results Menu, Options Menu, Selection Menu, Window Menu, and Help Menu. Under each topic are additional subtitles that corresponds to data-entry headings. If the user selects one of those, a help screen on the topic is displayed.
- About** this displays a screen describing the program version, date, and methods for accessing technical support.

Help Topics ... the main reference files for help are accessed through this submenu option. Clicking on this option provides a screen with reference help for the following topics: File Menu, Edit Menu, View Menu, Input Data Menu, Computations Menu, Transform Menu, Results Menu, Options Menu, Selection Menu, Window Menu, and Help Menu. Under each topic are additional subtitles that correspond to data-entry headings. If the user selects one of those, a help screen on the topic is displayed.

About this displays a screen describing the program version, date, and methods for accessing technical support.

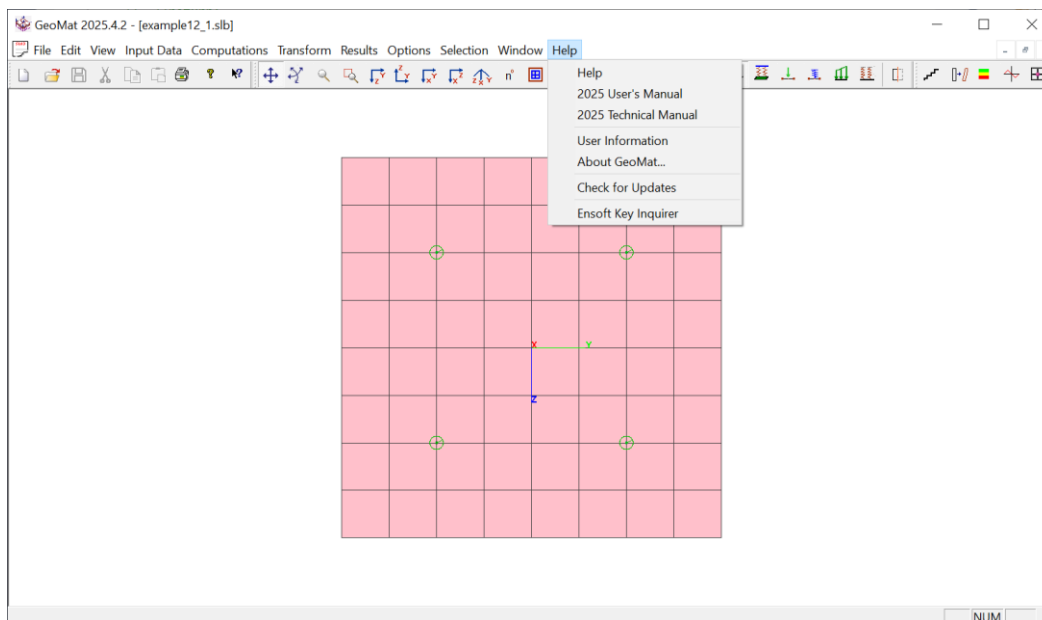


Figure 2.15 Options contained in the Help Menu

3 Data Input

3.1 File Menu

This menu, shown in Figure 3.1, contains options related to the management of input data files and to exit the program. Input data files created for *GeoMat* are provided with a standard file-name extension in the form of *.slb (where * represents any allowable file name). All input-data files are standard text files and may be edited with any text editor or word-processing program.

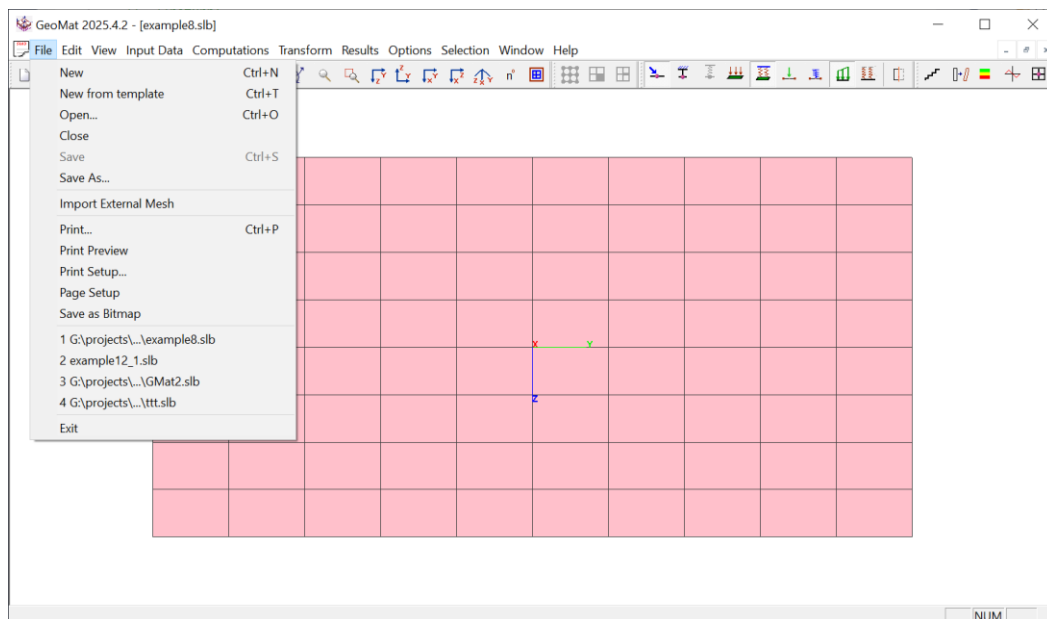


Figure 3.1 Window screen for the File – File menu

3.1.1 File – New

Once the program is started, default values are used for certain operating parameters and a blank, input data file is created. Selecting **New** under the **File** menu resets all *GeoMat* variables to either default or blank values, as appropriate. This option should be selected when a new data file is desired to be created from a blank form. This menu option may also be accessed with the Ctrl+N keyboard combination

3.1.2 File – New from template

3.1.2.1 slab with conventional circular/rectangular shape

To assist the user in creating an input file, the **New from template** option may be selected. *GeoMat* provides four templates: conventional rectangular/circular plate, ring shape, octagonal wind turbine foundation, or circular wind turbine foundation (Figure 3.2). Later two options are specifically used for soil-slab interaction of wind turbine foundations by means of Vlasov or Mindlin approach.

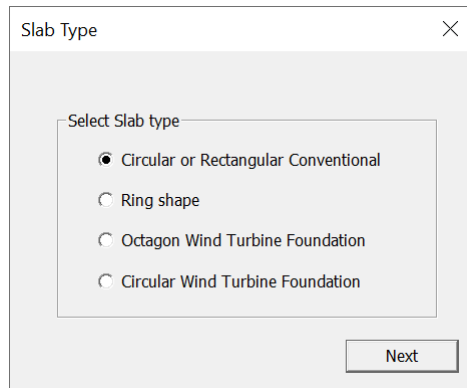


Figure 3.2 Options for various template

Option 1 is used for conventional circular or rectangular plate (Figure 3.3). A new template dialog box prompts the user for the geometric and computational properties of the slab. The shape, number of nodes for computation, dimensions, and number of grid divisions may be easily entered with this interface. This automatically fills the Node Coordinate, Element Type, and Element Connectivity data in the **Input Data** menu.

There are three types of shapes to choose from: Circular, Rectangular - Equally Spaced, and Rectangular - General Spacing. With each option, the bottom portion of the window reflects the needed input to draw the slab model. Examples of each screen associated with these choices are shown in Figure 3.3 to Figure 3.5. The Rectangular General Spacing allows users to input the grid locations, an example of which is shown in Figure 3.6. The user may also select the element type to be used in the analysis. Three types are available: Linear 4 noded, Quadratic 8 noded, and Lagrangian 9 noded.

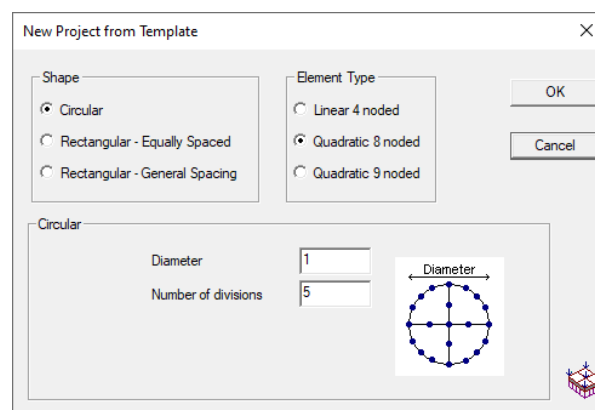


Figure 3.3 Input screen for the Circular option

Figure 3.4 Input screen for the Rectangular option – Equally Spaced option

Figure 3.5 Input screen for the Rectangular option – Generally Spaced option

	Number Divisions	Span Length	Total Length
1	1	5	5
2	2	3	8
3	3	4	12
4	4	5	12

Number of Spans: 4

Figure 3.6 Grid Y direction input screen for the Rectangular option – Generally Spaced option

3.1.2.2 slab with ring shape

If the user wants to create a ring shape slab, select the second option “Ring Shape” as shown in Figure 3.2. The parameters of a ring shape slab are shown in Figure 3.7. Input the internal radius RI , external radius RO and the circumferential angle to define the slab geometry. Then input the divisions in the radial and circumferential directions. The default element is quadratic 8 node element with reduced integration

rule. Click “OK”. To complete the ring mesh as shown in Figure 3.8. Users can create a partial ring slab by inputting the circumferential angle smaller than 360 °.

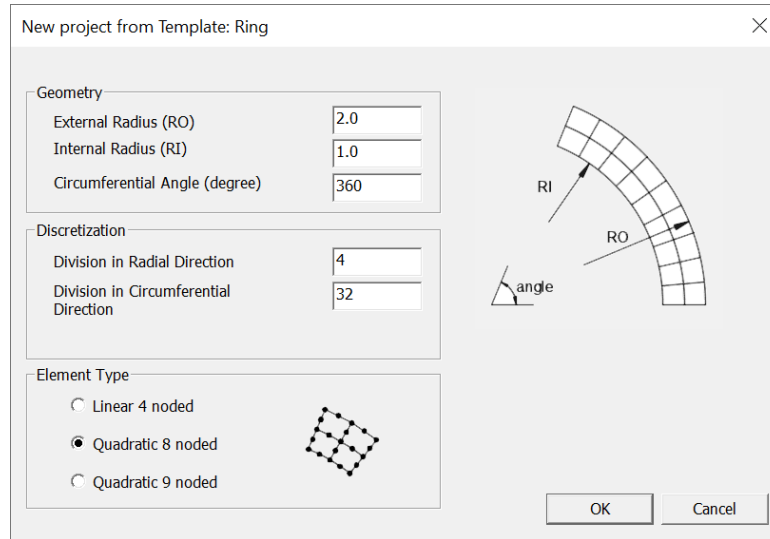


Figure 3.7 Parameters of creating ring shape slab

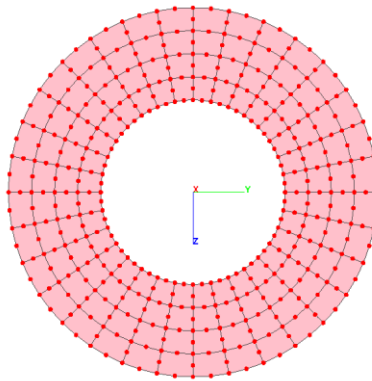


Figure 3.8 Example of a full ring slab

3.1.2.3 Octagon/Circular wind turbine

If option 2 in Figure 3.2 is selected, a new template for octagon wind turbine foundation (WTB) is shown as in Figure 3.9. L_1 , L_2 are plane dimensions for this octagon WTB. D is the diameter for the circular pedestal. $D_1 \sim D_3$ are step sizes in the thickness direction. In this template, loads P , V and M are optional. P is the vertical thrust force from tower weight, instrument weight or vertical component of wind loads. V is the shear force applied to the top of the foundation. Note that if the shear force V is non-zero, appropriate nodal constraints should be defined to avoid rigid body motion of the foundation. Non-zero loads P , V and M will be discretized to equivalent nodal loads by *GeoMat* automatically. The default finite element is set to quadratic eight-noded element with 2x2 integration rule.

Element refinement is determined by typical element size. Majority of elements will be generated in a square shape and each edge will follow this size approximately. The default typical element size is equal to dimension L_1 divided by default division 12 as shown in Figure 3.9. Users can override the default division number to generate a coarser mesh or a finer mesh. Angle α is the angle between load plane of V , M and horizontal y axis. Positive angle is in the clockwise direction. As an example, input 90 will define a wind load in the downward direction.

Template of Octagon Wind Turbine Foundation

Plane Dimension

L1 = 21.0

L2 = 17.0

D = 14.0

Angle(deg) = 0

Section Dimension

D1 = 1.5

D2 = 5.0

D3 = 3.5

Load

P = 600.0

V = 0.0

M = 50000.0

Finite Element Option

☐ Linear 4 noded

☒ Quadratic 8 noded

☐ Quadratic 9 noded

Division of L1 = 12

Typical FE Size = 1.75

Integration Rule

☐ 1x1

☒ 2X2

☐ 3X3

OK Cancel

Figure 3.9 Template for octagon wind turbine foundation

The third option for the new template is also a wind turbine foundation in circular shape (Figure 3.10). R_1 is the radius of the pedestal. R_2 is the radius of the second circle. R_3 is the radius at the foundation bottom. The default typical element size is equal to circumference of the circle divided by the default division number as shown in Figure 3.10. Users can override the default division to generate a coarser or finer mesh. Other parameters have the same meanings as the octagon wind turbine foundation.

Template of Circular Wind Turbine Foundation

Plane Dimension

R1 = 3.5

R2 = 5.0

R3 = 14.0

Angle(deg) = 0

Section Dimension

D1 = 1.5

D2 = 5.0

D3 = 3.5

Load

P = 600.0

V = 0.0

M = 50000.0

Finite Element Option

☐ Linear 4 noded

☒ Quadratic 8 noded

☐ Quadratic 9 noded

Division of $2\pi \times R3$ = 40

Typical FE Size = 1.10

Integration Rule

☐ 1x1

☒ 2X2

☐ 3X3

OK Cancel

Figure 3.10 Template for circular wind turbine foundation

3.1.3 File - Open

This option is used to open a file that has been previously prepared and saved to disk. The **File - >Open** window dialog, shown in Figure 3.11, is used to search for an existing input-data file. By default, the file is initially searched in the directory where *GeoMat* was installed. Standard, windows-navigation procedures may be used to locate the name and directory of the desired project file. This menu option may be accessed with the Ctrl+O keyboard combination.

Every analytical run of *GeoMat* produces several additional files, as described in Table 2.1. The name of the input-data file indicates the names of all related files produced by a successful execution (output, graphics, and processor-text files). All the additional program files will be created in the same directory as the input file. Input-data files that are partially completed may be saved and later opened for completion, run, and view of results.

Opening partially-completed, 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.

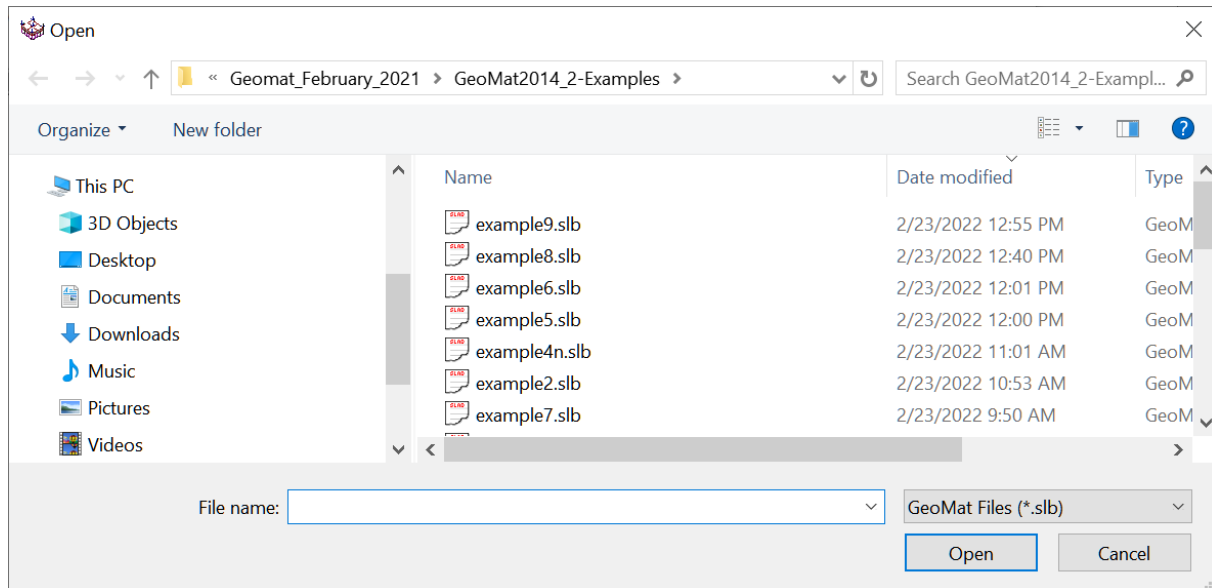


Figure 3.11 Window screen for the File – Open dialog

3.1.4 File – Save

This option is used to save input data under the current file name. With this method of storing data to disk, any input data that was previously saved with the same file name is replaced with the current parameters. Input-data files should be saved every time before proceeding with runs for analytical computation. This menu option may also be accessed with the Ctrl+S keyboard combination.

3.1.5 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. Any input data file saved under an existing file name will replace the contents of the existing file.

3.1.6 File – Import External Mesh

If users intend to build a slab model which cannot be created from the template inside *GeoMat*, the new slab can be created from an external mesh file. The external mesh file should have “.mesh” as the file suffix. It also should follow the neutral mesh file format:

- Global data: total number of nodes, total number of elements, number of nodes inside one element. This is the first data line.
- Node data: node number, y coordinate, z coordinate. Repeat node data line until all nodes are defined
- Element data: element number, four corner nodes in counter clock wise, four middle nodes in counter clockwise, center node if quadratic Q9 is used. Repeat the element data line until all elements are defined
- Both period “,” and white space “ ” are accepted to separate two input values

- Number of nodes and elements should be input sequentially which starts from 1 to the last node/element. Numbers of nodes/elements should be continuous without gap.
- No blank lines between data lines

One example of an external mesh is shown in Figure 3.12. The left is the slab creating from an external mesh file. The mesh file is shown on the right (nodes 12~33 are omitted for smaller image size).

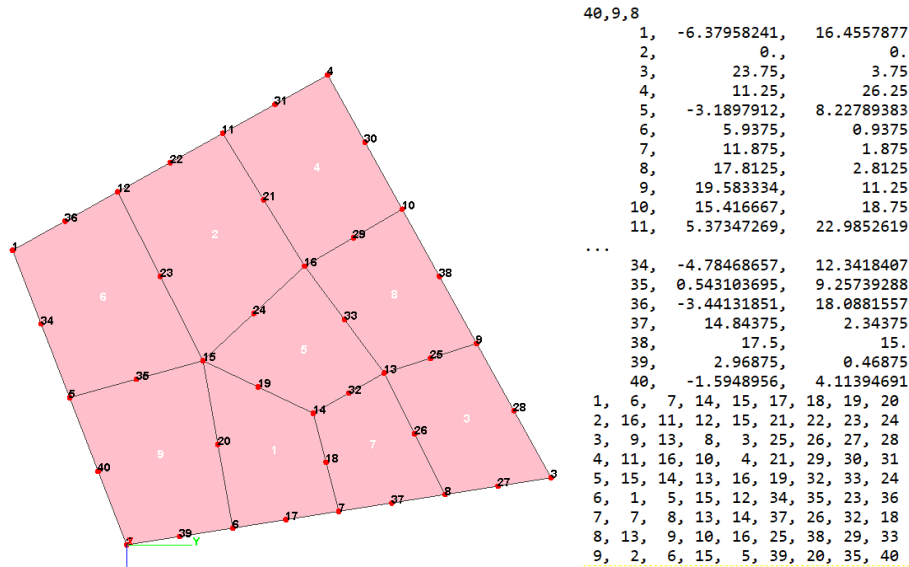


Figure 3.12 Import external mesh (left) and neutral mesh file (right)

3.1.7 File - Exit

This is selected to exit *GeoMat*. Any input-data file that was modified and not yet saved to disk will produce a confirmation window before exiting the program (see Figure 3.13).

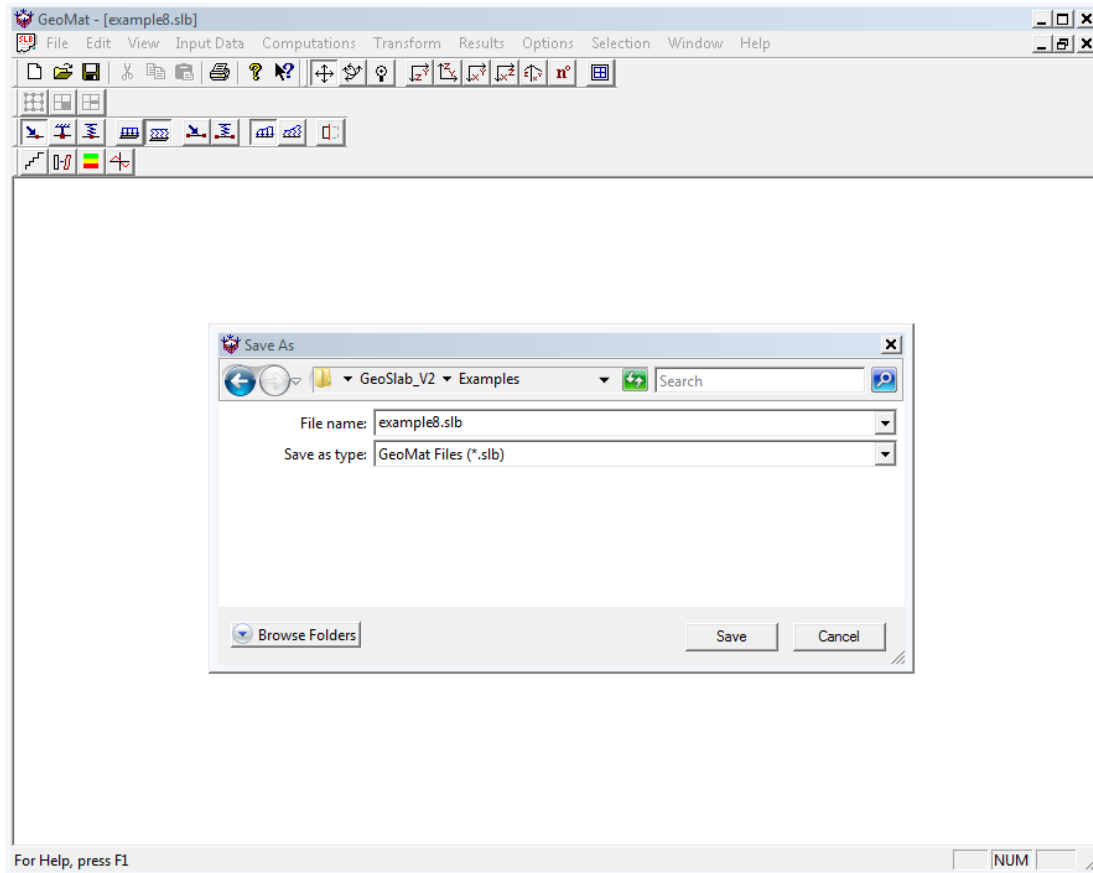


Figure 3.13 Message window advising that changes were not saved to disk

3.2 Input Data Menu

The input of specific parameters for an application is controlled under options contained within this menu, shown in Figure 3.14. It is recommended that the user chooses each submenu and enter parameters in a consecutive manner starting from the top option.

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), or
- clicking the  box on the upper-right corner of the window, or
- double-clicking the *GeoMat* icon on the upper-left corner of the window, or
- using the Ctrl + O keyboard combination, or
- clicking once on the *GeoMat* icon on the upper-left corner of the window and then choosing Close.

Open 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 sub-windows of the **Data** menu will show an **Add Row**, **Insert Row** and/or **Delete Row** buttons. The **Add Row** button always adds new rows at the end after all existing rows. The **Insert Row** button always inserts a new row right after an existing row highlighted by the mouse. Clicking on the **Delete Row** button deletes the row where the cursor is located.

3.2.1 Numeric Data Entries and Sign Conventions

Cells that require numeric data only accept simple numeric entries. Entering a mathematical expression works similarly to normal numeric data. If a non-valid number is inputted, the program will display an error message.

As units are not explicitly defined in the program, the user is reminded to keep consistent units. The sign conventions in this program is based on the right-hand rule, which use X direction for the load or displacement perpendicular to the plate and the plate planar dimension is defined in Y and Z directions.

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.

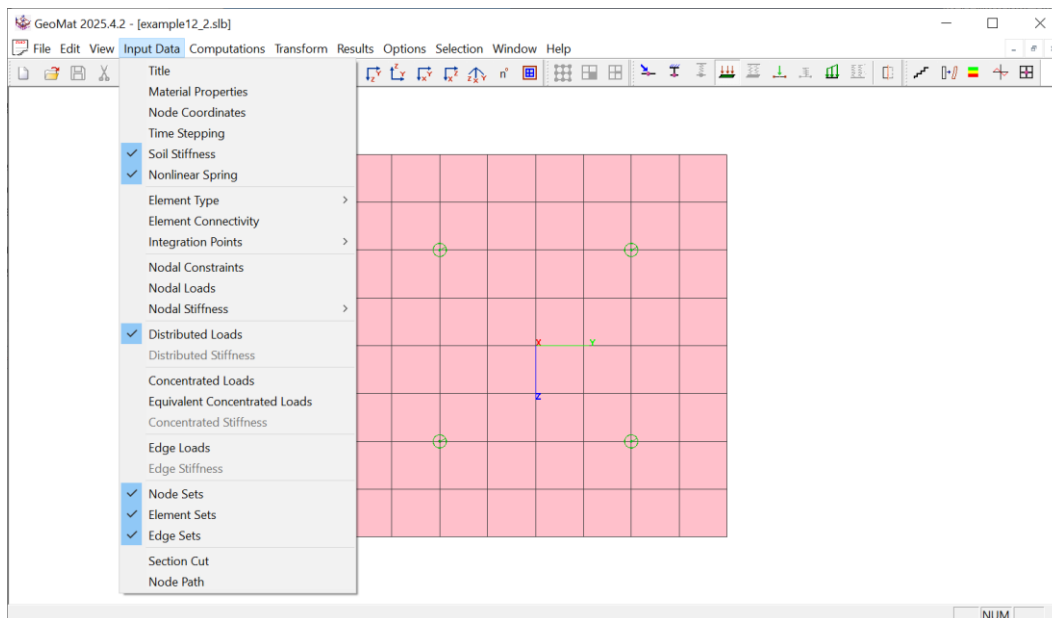


Figure 3.14 Options contained in the Input Data menu

3.2.2 Input Data -Title

This option activates the window shown in Figure 3.15, where the user can enter a line of text containing a general description for 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. This is done to prevent the user from going beyond the maximum permissible length of characters allowed for the title line.

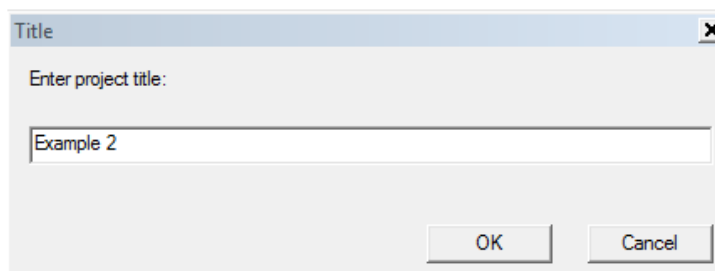


Figure 3.15 Window screen for sample Data – Title

3.2.3 Input Data – Material Properties

This menu option launches the window shown in Figure 3.16, where the user can specify the type of materials. The current version of *GeoMat* allows the user to select Isotropic plate/slab and Orthotropic plate/slab with various configuration.

Isotropic Properties

After the material selection and clicking **Define** button, the screen for material properties will be shown for data entry (Figure 3.17), including the location of the material properties of the slab. The input requested is the Element Number/Set, Young's Modulus, Poisson's Ratio, and Slab Thickness. The properties may be varied across different sections by using the Add Row function. If the **New from template** option is used, the material properties are set to the default values of ALL for the Element Sets, 3000 for Young's Modulus, 0.3 for Poisson's Ratio, and 1 for Slab Thickness. The program will ask the user to select proper representation from the following five different categories.

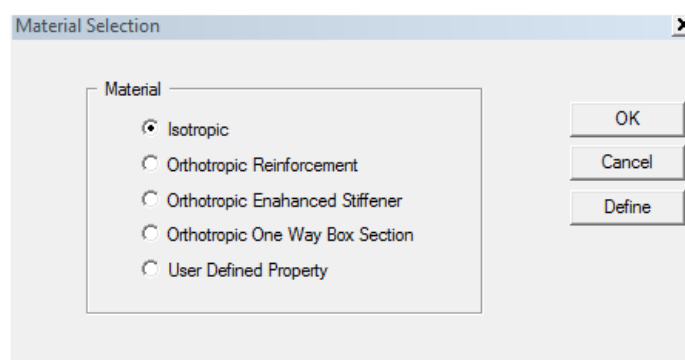


Figure 3.16 Window screen for material selection under Input Data Menu – Material Properties

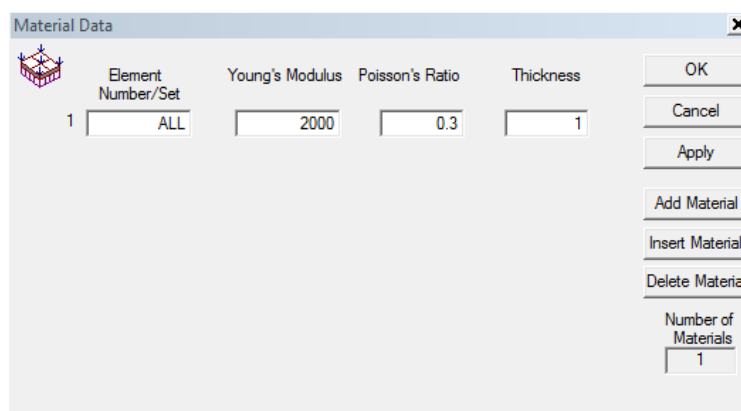


Figure 3.17 Window screen for material data Under Material Properties

Orthotropic Properties

This version has the ability to compute deformation and stresses of space orthotropic plates. Three types of orthotropic plates/slabs available in this version are: uniform plates with orthotropic reinforcements, plate with orthotropic enhanced ribs, and plate with one way box section. This version also provides the option for inputting user-defined orthotropic properties.

Type 1: Uniform plate with orthotropic reinforcement

The input interface is shown in Figure 3.18. A_s1 and A_s2 are areas of reinforcement above or below the plate neutral axis. $X1$ and $x2$ are distances from the center of reinforcement to the neutral axis.

Type 2: Plates with orthotropic enhanced ribs

The input interface is shown in Figure 3.19. The dimension and location of each component on the orthotropic plate are sketched and shown in the input screen for the user to follow.

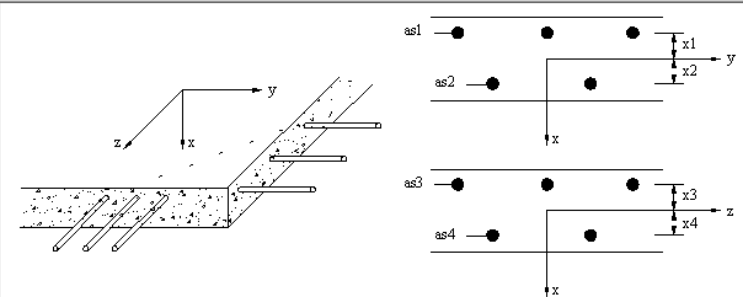
Type 3: Plates with one-way box section

The input interface is shown in Figure 3.20. The dimension and location of each component on the orthotropic plate are sketched and shown in the input screen for the user to follow, Where $t1$ =thickness of top plate, $t2$ =thickness of vertical plates, $t3$ =thickness of bottom plates.

Type 4: User defined orthotropic plates

This option allows the user to define orthotropic plates. There are six input parameters that are needed for defining the orthotropic properties as shown in Figure 3.21. The flexural and shear rigidities are defined in the technical manual. (Note the x, y and z coordinate axes are defined differently than those used in user input GUI.)

Orthotropic Reinforcement



Element Set: Young's Modulus of Concrete: Young's Modulus of Reinforcement: Poisson's Ratio: Thickness:

Rebar parallel to z axis

Reinforcement As1=
 x1=
 Reinforcement As2=
 x2=

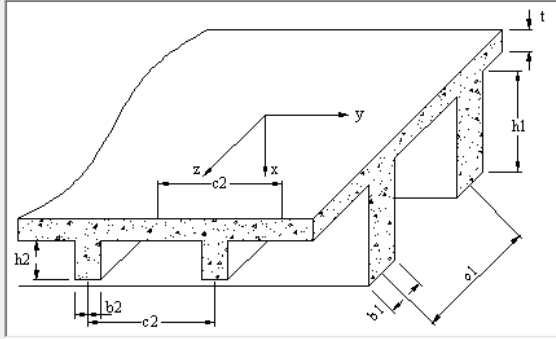
Rebar parallel to y axis

Reinforcement As3=
 x3=
 Reinforcement As4=
 x4=

OK Cancel

Figure 3.18 Window screen for material data Under Material Properties using uniform plate with orthotropic reinforcement

Orthotropic Enhanced Stiffener



Element Set = Young's Modulus =
 Poisson's Ratio = Thickness =

Stiffener Parallel to y axis

Stiffener Web Height h1=
 Stiffener Web Width b1=
 Effective Width c1=

Stiffener Parallel to z axis

Stiffener Web Height h2=
 Stiffener Web Width b2=
 Effective Width c2=

Torsional Stiffness Reduction =

OK Cancel

Figure 3.19 Window screen for material data Under Material Properties using orthotropic plate with enhanced ribs

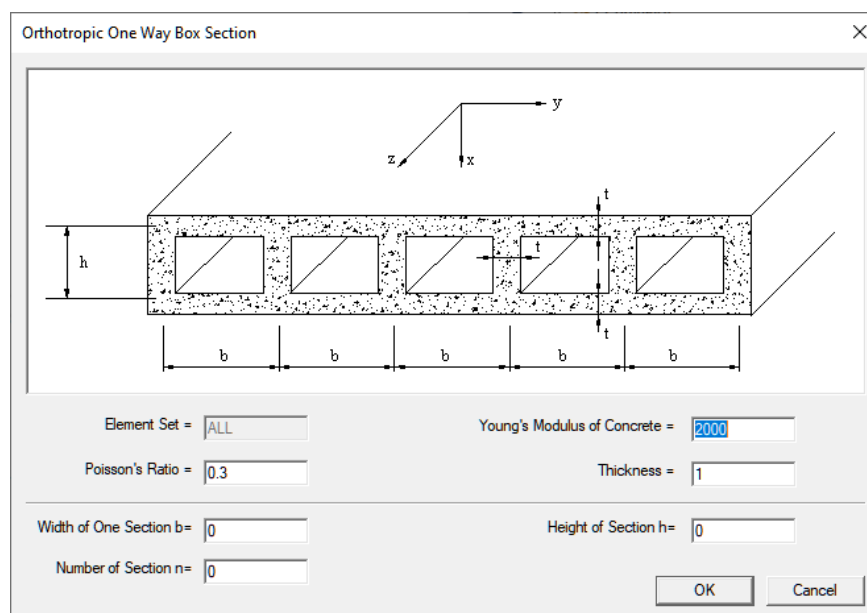


Figure 3.20 Window screen for material data Under Material Properties using orthotropic plate with one-way box section

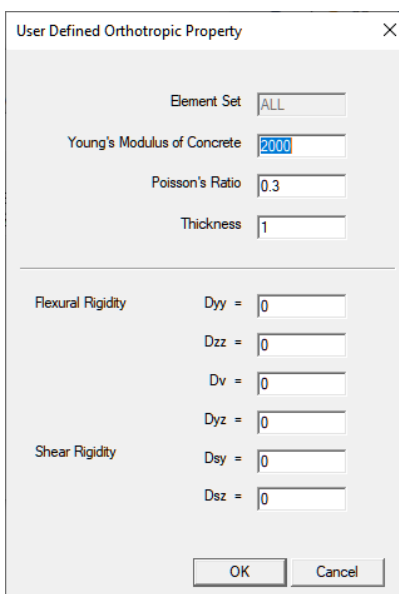


Figure 3.21 Window screen for material data Under Material Properties using user-defined section

3.2.4 Input Data – Node Coordinates

This submenu allows the user to modify coordinates of nodal points to change the element shape. If the **New from template** option is used, the coordinates are automatically determined and entered here for the user. For a mat that is neither rectangular nor circular, the user may combine deleting unwanted elements and modifying node coordinates to obtain irregular slab shape.

3.2.5 Input Data – Time Stepping

If the user chooses to ask the program to estimate the soil subgrade stiffness based on the Young's modulus of the foundation soil. The program will estimate the soil stiffness using iterative procedures until the compatibility between the mat movement and soil stiffness converges. The process uses non-linear iteration for achieving the convergence and the user needs to specify the parameters associated with the iteration process shown in Figure 3.22.

If the user will enter the soil stiffness and support externally, "Time Stepping" is not needed and may leave default values for this option.

Figure 3.22 Window screen for Input Data Menu – Time Stepping

3.2.6 Input Data – Soil Stiffness

This submenu is disabled as a default but may be enabled if the user chooses to ask the program to estimate the soil subgrade stiffness based on the Young's modulus of the foundation soil. The program will estimate the soil stiffness using iterative procedures until the compatibility between the mat movement and soil stiffness converges. The associated screen is shown in Figure 3.23. Option 1 does not consider soil stiffness. This is the default option. Users may also select option 2 to estimate soil stiffness using Vlasov approach. Corresponding input window is shown as in Figure 3.24. The Young's modulus at top and bottom can be different. The default initial gamma=1 works well for most cases. If soil depth is significantly high, the initial gamma can be greater than 1 and smaller than 5. Option 3 uses Mindlin equation to estimate soil stiffness (Figure 3.25). Mindlin's equations are also used to calculate the stress-strain behavior in the soil. The detailed iteration procedures are discussed in the Technical Manual.

Soil Stiffness

☒ Do not consider Soil Stiffness

☐ Estimate Soil Stiffness using Vlasov Approach

☐ Estimate Soil Stiffness using Mindlin Equation

Young Modulus

1000

Poisson's Ratio

0.3

Initial Stiffness

500

OK

Cancel



Figure 3.23 Window screen for Input Data Menu – Default option

Soil Stiffness

☐ Do not consider Soil Stiffness

☒ Estimate Soil Stiffness using Vlasov Approach

☐ Estimate Soil Stiffness using Mindlin Equation

Young Modulus at Top

1000

Young Modulus at Bottom

1000

Poisson's Ratio

0.3

Soil Depth

10

Initial Gamma

1

OK

Cancel



Figure 3.24 Window screen for Input Data Menu – Vlasov approach

Soil Stiffness

☐ Do not consider Soil Stiffness
☐ Estimate Soil Stiffness using Vlasov Approach
☒ Estimate Soil Stiffness using Mindlin Equation

Young Modulus: 1000
 Poisson's Ratio: 0.3
 Initial Stiffness: 500

OK Cancel

Figure 3.25 Window screen for Input Data Menu – Mindlin equation

3.2.7 Input Data – Nonlinear Spring

This submenu lets user to define behavior of nonlinear springs to simulate vertical piles to support a slab.

Click **New** to add one new nonlinear spring. Use buttons **Add Row**, **Insert Row** and **Delete Row** to input/modify spring data (Figure 3.26). Positive spring deformations represent slab settlement in the positive X direction. Positive forces represent compression forces in the spring. Tension forces and tension deformations have negative sign convention. Note that data should follow ascending order. The load-deformation curve will be displayed on the left-bottom.

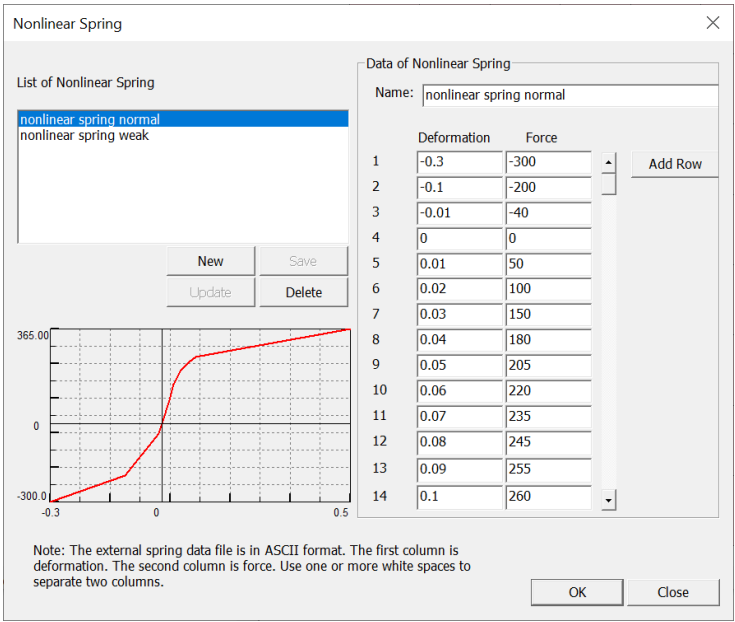


Figure 3.26 Define data for a nonlinear spring

Alternatively, users can import spring data from an external file (Figure 3.27). After clicking **New**, the button “**Open File**” will be available. The external file should be saved in ASCII format without blank lines. Each line has two data columns: deformation and force. One or more white spaces are used to separate two columns. One example of the external nonlinear spring data is shown in Figure 3.28. Spring data should have adequate deformation range to cover expected slab settlements. It is also recommended to include data for both tension and compression statuses.

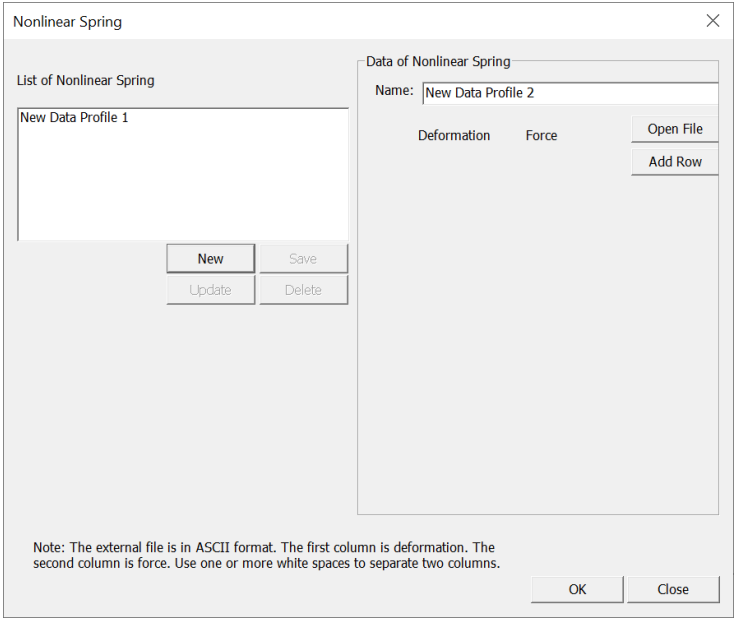


Figure 3.27 Import spring data from an external file

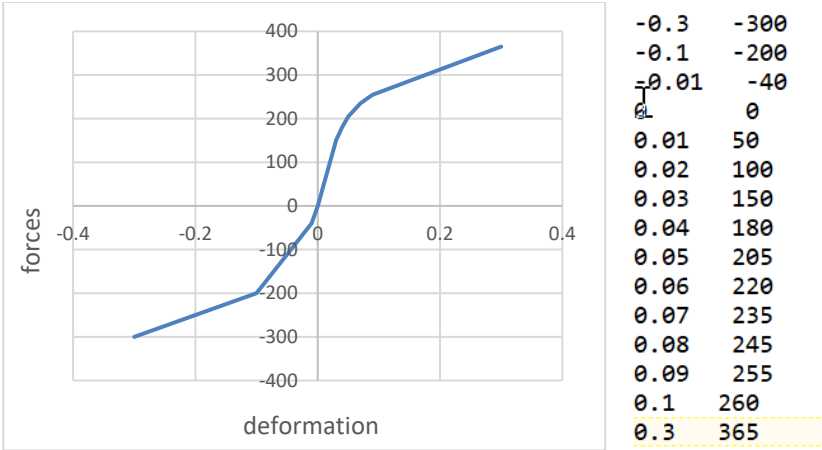


Figure 3.28 Example of external spring data

After defining one new spring, click **Save** to save data to the model. When users click one predefined spring in the spring list window, users can modify it and click **Update** to refresh model database. Click the button **Delete** will delete selected spring data from the model.

3.2.8 Input Data – Element Type

This option allows the user to choose the type of element to be used in the analysis. There are three options: Linear 4-noded, Quadratic 8-noded, and Lagrangian 9-noded. If the **New from template** option is used, this choice is automatically made for the user.

3.2.9 Input Data – Element Connectivity

This selection launches the Element Connectivity window. This allows the user to define the nodes for each element number. If the New from template option is used, these values are automatically filled in for the user, these values are automatically filled in for the user. The user can modify the data if an irregular shape is required for the meshes by deleting unwanted elements.

3.2.10 Input Data – Integration Points

This submenu allows the user to select either a reduced or normal integration procedure to be used for the analysis. For linear 4-noded elements, only one integration point in each element is available. Otherwise, the user can select 2x2 or 3x3 for the integration points.

3.2.11 Input Data – Nodal Constrains

This submenu item opens the Nodal Constraints dialog box, as shown in Figure 3.29. Here, the user inputs the node(s) where the restraint is located. Both translational and rotational restraints at nodal points may be restricted using this feature.

Node Number	Displacement X	Displacement Y	Displacement Z	Rotation X	Rotation Y	Rotation Z
1 LEFT	☐	☐	☐	☒	☐	☒
2 BOTTOI	☐	☐	☐	☒	☒	☐
3 TOP	☒	☒	☒	☒	☒	☒
4 RIGHT	☒	☒	☒	☒	☒	☒

Figure 3.29 Window screen for Input Data Menu – Nodal Constraints

3.2.12 Input Data – Nodal Loads

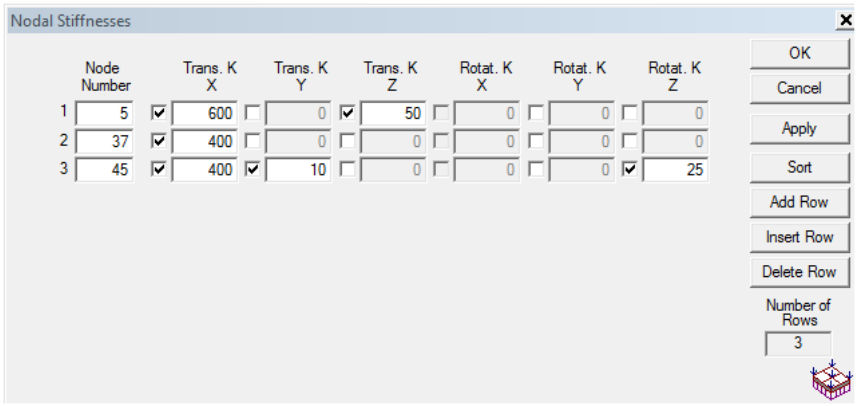
This window allows the user to specify the location of a point load applied at a nodal point. A sample screen of this box is shown in Figure 3.30. The units for nodal loads are F.

Node Number	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1 1	☒	0.25	☐	☐	☐	☐

Figure 3.30 Window screen for Input Data Menu – Nodal Loads

3.2.13 Input Data – Nodal Stiffness

Nodal stiffness has two options: use constant stiffness or nonlinear springs. Figure 3.31 shows the input user interface for constant stiffness. This window allows the user to specify the location of a concentrated stiffness applied at a nodal point. A sample screen of this box is shown in Figure 3.31. The units of nodal stiffness are F/L.

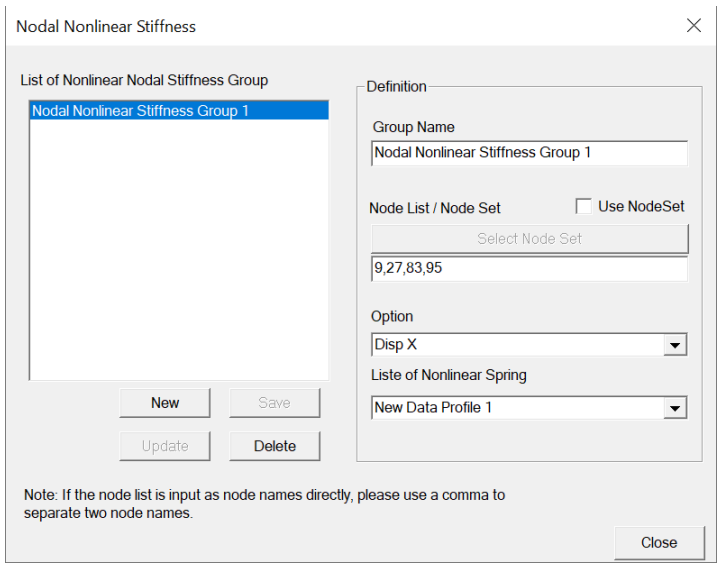


The 'Nodal Stiffnesses' dialog box contains a table with 7 columns: Node Number, Trans. K X, Trans. K Y, Trans. K Z, Rotat. K X, Rotat. K Y, and Rotat. K Z. It lists three nodes with their respective stiffness values and checkboxes. On the right, there are buttons for OK, Cancel, Apply, Sort, Add Row, Insert Row, Delete Row, and a 'Number of Rows' field set to 3.

	Node Number	Trans. K X	Trans. K Y	Trans. K Z	Rotat. K X	Rotat. K Y	Rotat. K Z
1	5	☒ 600	☐ 0	☒ 50	☐ 0	☐ 0	☐ 0
2	37	☒ 400	☐ 0	☐ 0	☐ 0	☐ 0	☐ 0
3	45	☒ 400	☒ 10	☐ 0	☐ 0	☐ 0	☒ 25

Figure 3.31 Window screen for Input Data Menu – Constant Nodal Stiffness

The second option to use nonlinear nodal stiffness is shown in Figure 3.34~Figure 3.33. Click the button **New** to add one new nodal nonlinear stiffness. Users may change the default group name if necessary. If the option node set is not checked, users will input node list. A comma is used to separate two node numbers. If the option node set is checked, click the button **Select Node Set**. Then click to select a predefined node set from the “Select Set” dialog. Click **OK** to finish the node list definition (Figure 3.33).



The 'Nodal Nonlinear Stiffness' dialog box is divided into two main sections. The left section, 'List of Nonlinear Nodal Stiffness Group', shows a list with 'Nodal Nonlinear Stiffness Group 1'. The right section, 'Definition', includes fields for 'Group Name' (Nodal Nonlinear Stiffness Group 1), a 'Node List / Node Set' field with the value '9,27,83,95', and an 'Option' dropdown set to 'Disp X'. There are also buttons for 'New', 'Save', 'Update', 'Delete', and 'Close'. A note at the bottom states: 'Note: If the node list is input as node names directly, please use a comma to separate two node names.'

Figure 3.32 Window screen for Input Data Menu – Nonlinear Nodal Stiffness

The stiffness option is set to “Disp X” automatically because this nonlinear nodal stiffness is applied in the slab settlement direction. Click to select one predefined nonlinear spring curve. Click **Save** to save the new nodal stiffness into model.

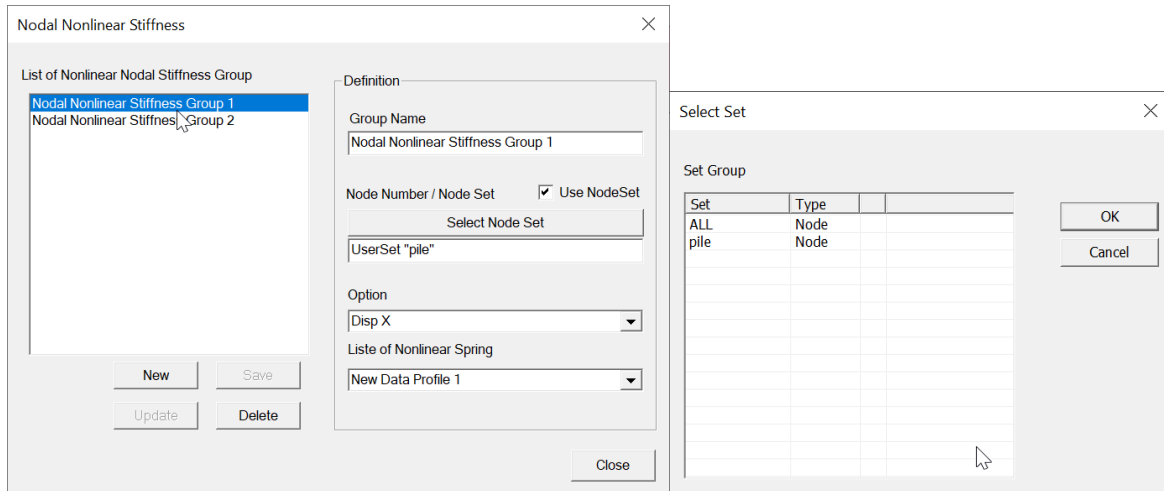


Figure 3.33 Define Nonlinear Nodal Stiffness by means of Node Set

Select one predefined nonlinear stiffness group from the list window to view/modify. After modification, click **Update** to refresh data in the model database.

Select one predefined stiffness group, then click **Delete** to remove it from the model.

3.2.14 Input Data – Distributed Loads

This option displays the window shown in Figure 3.34. The user may enter pressures which are distributed over an element or elements. Both distributed force (the units are F/L^2) and distributed moment (the units of are $F-L/L^2$) can be specified.

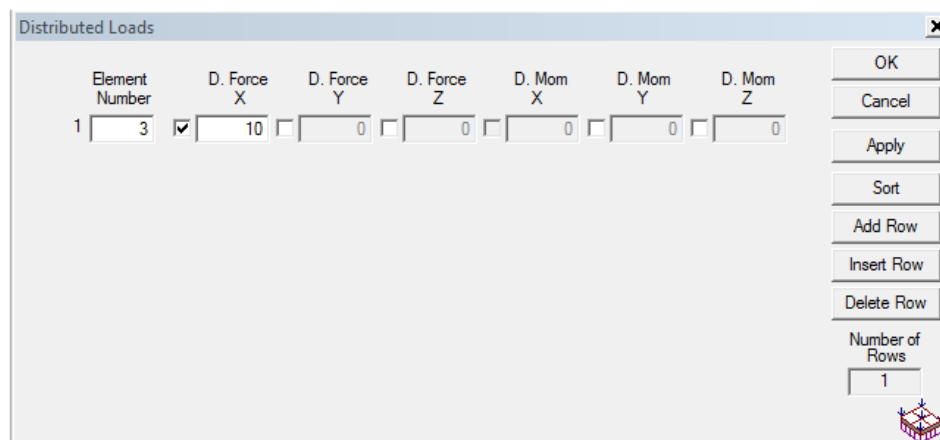


Figure 3.34 Window screen for Input Data Menu – Distributed Loads

3.2.15 Input Data – Distributed Stiffness

This option displays the Distributed Stiffness input box, shown in Figure 3.35. The user may enter here stiffnesses which are distributed over an element or elements. Both rotational and translational

stiffnesses can be specified. The units for distributed translational stiffness are F/L^3 . The units for distributed rotational stiffness are $F-L/Rad./L^2$.

Element Number	D. Force X	D. Force Y	D. Force Z	D. Mom X	D. Mom Y	D. Mom Z
1 3	☒ 10	☐ 0	☐ 0	☐ 0	☐ 0	☐ 0

Buttons: OK, Cancel, Apply, Sort, Add Row, Insert Row, Delete Row, Number of Rows (1)

Figure 3.35 Window screen for Input Data Menu – Distributed Stiffness

3.2.16 Input Data – Concentrated Loads

This option displays the window shown in Figure 3.36. The user may specify the exact coordinates of a point load which is not located at a nodal point. The units for concentrated force are F. The units for concentrated moment are F-L.

	Coord. X	Coord. Y	Coord. Z	Force	Force	Force	Moment	Moment	Moment
1	0	-7.6	3.9	1.1	0	0	0	0	0
2	0	7.6	3.9	1.09	0	0	0	0	0
3	0	7.6	-3.9	1.34	0	0	0	0	0
4	0	-7.6	-3.9	1.11	0	0	0	0	0

Buttons: OK, Cancel, Apply, Add Row, Insert Row, Delete Row, Number of Rows (4)

Figure 3.36 Window screen for Input Data Menu – Concentrated Loads

3.2.17 Input Data – Equivalent Concentrated Loads

This option displays the window shown in Figure 3.37. This option allows the user to place concentrated loads from anchor bolts of the mounting plate to the top of the foundation. The user can specify the number of circular rings for anchor bolt arrangement and the diameters of each ring. The user also needs to specify the loads (six-degree of loading) applied on the foundation through the anchor bolts.

The program will calculate the location of each anchor bolts arranged in a circular pattern and the distributed load on each anchor bolt based on the user-specified data.

Note that equivalent concentrated load points cannot locate outside of the slab. A warning message will popup if such load points are found.

These equivalent concentrated loads locate on an arc which has the default center ($y=0, z=0$). If the point of application is not zero for “y” or “z”, these equivalent loads will shift to an arc with center (y, z) as users defined.

Loading

Fx: 1108
Fy: 0
Fz: 0
Mx: 0
My: 21373.4
Mz: 0

Ring 1 Data

Radius (ri): 1.839
Points (ni): 48
Angle (fi): 0

Ring 2 Data

Radius (re): 1.959
Points (ne): 48
Angle (fe): 0

Point of Application

X: 0
Y: 0
Z: 0

Number of Rings

☐ One Ring
☒ Two Rings

☐ Show Values

Equivalent Loads

Ring	N	X	Y	Z	Fx
Int 1	1	0.000000	1.839000	0.000000	11.541667
Int 2	2	0.000000	1.823267	0.240038	41.150992
Int 3	3	0.000000	1.776338	0.475968	70.253694
Int 4	4	0.000000	1.699014	0.703755	98.351816
Int 5	5	0.000000	1.592621	0.919500	124.96459
Int 6	6	0.000000	1.458977	1.119512	149.63667
Int 7	7	0.000000	1.300369	1.300369	171.94590
Int 8	8	0.000000	1.119512	1.458977	191.51058
Int 9	9	0.000000	0.919500	1.592621	207.99593
Int 10	10	0.000000	0.703755	1.699014	221.11990
Int 11	11	0.000000	0.475968	1.776338	230.65793
Int 12	12	0.000000	0.240038	1.823267	236.44682
Int 13	13	0.000000	0.000000	1.839000	238.38752
Int 14	14	0.000000	-0.240038	1.823267	236.44682
Int 15	15	0.000000	-0.475968	1.776338	230.65793
Int 16	16	0.000000	-0.703755	1.699014	221.11990

Visible Variables

☒ X
☒ Y
☒ Z
☒ Fx
☒ Fy
☒ Fz

OK Cancel Apply

Figure 3.37 Window screen for Input Data Menu – Equivalent Concentrated Loads

3.2.18 Input Data – Concentrated Stiffness

This option displays the Concentrated Stiffness input box, shown in Figure 3.38. The user may specify the exact location of a concentrated stiffness which is not located at a nodal point. The units of concentrated translation stiffness are F/L. The units of concentrated rotational stiffness are F-L/Radian.

	Coord. X	Coord. Y	Coord. Z	Tra. K	Tra. K	Tra. K	Rot. K	Rot. K	Rot. K
1	0	5	2	0.200	0	0	0	0	0

Figure 3.38 Window screen for Input Data Menu – Concentrated Stiffness

3.2.19 Input Data – Edge Load

This option displays the Edge Load input box, shown in Figure 3.40. The user may enter a line load to be evenly distributed along an edge.

Element: the user needs to input the number of a single element, or input the name of an edge set. The edge set should be defined before this load operation. (see 3.2.21 Input Data – Edge Sets).

Side No.: each element has 4 sides. The user needs to specify which side with the edge stiffness.

DOF Index: there are 6 degrees of freedom on each edge node. The degree freedom index is 1 for translation in X direction, 2 for translation in Y direction, 3 for translation in Z direction, 4 for rotation about X axis (not available in this program), 5 for rotation about Y axis, and 6 for rotation about Z axis.

Load Node 1: each side of an element usually has either 2 nodes or 3 nodes. Load Node 1 is for one of the edge nodes. Enter the load (based on the DEF index) on this nodal point.

Load Node 2: each side of an element usually has either 2 nodes or 3 nodes. Load Node 2 is for the other one of the edge nodes. Enter the load (based on the DOF index) on this nodal point.

Load Node 3: each side of an element usually has either 2 nodes or 3 nodes. Stiff Node 3 is for the middle node if quadratic element is used. Enter the load (based on the DEF index) on this nodal point.

The units for edge translation load are F/L. The units for edge bending moment are F-L/ L.

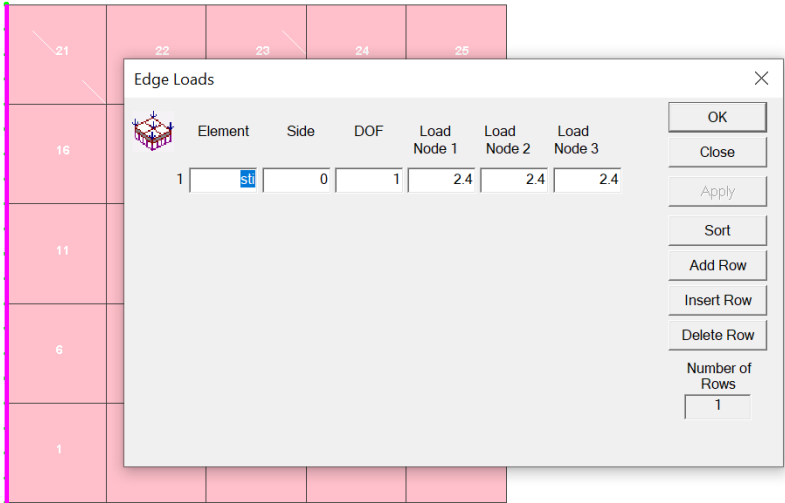


Figure 3.39 Define edge load by using edge set

One example to define the edge load by using available edge set is shown in Figure 3.39. Note the number under the “Side” input is not meaningless because side information has been already included in the edge set. Alternatively, users can define edge load element by element here.

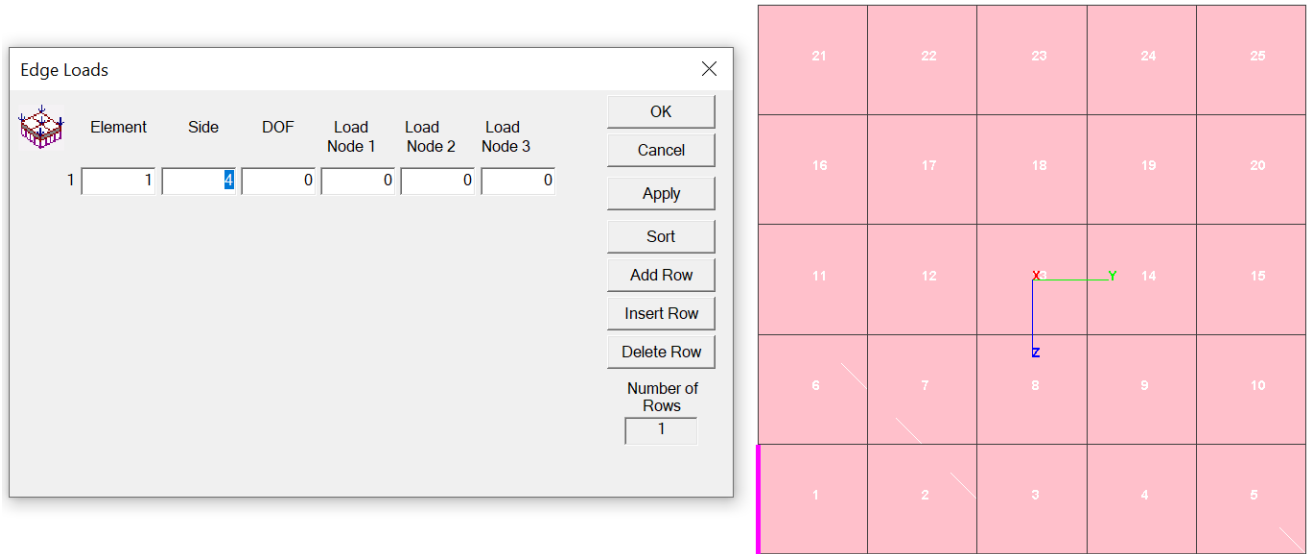


Figure 3.40 Define edge load element by element

Click “**Add Row**” to add one new data line. Except for inputting element and edge number manually, *GeoMat* provides a more convenient way to pick an edge by mouse. When the edge action window is opened, the icon “Edge Selection”  will become available. Click the icon  to activate it, LEFT-CLICK on the desired edge, element and side number will be filled automatically.

After all edge loads are defined, click “**Options->Edge Loads**” to turn on display of edge loads. One example is shown in Figure 3.41.

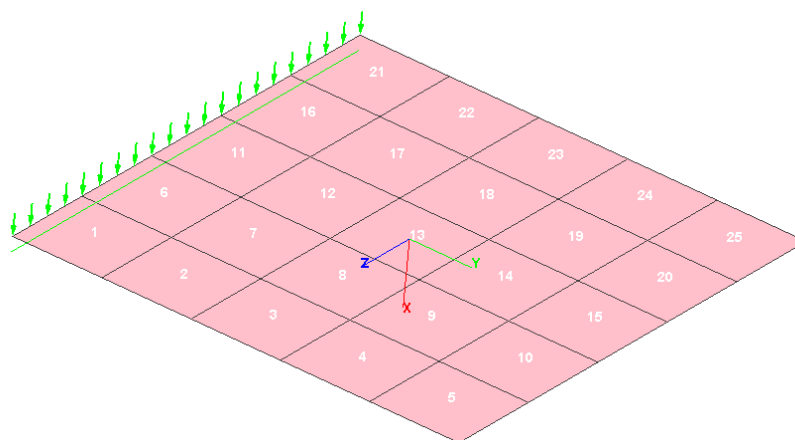


Figure 3.41 Example of edge loads defined on left side of a square slab

3.2.20 Input Data – Edge Stiffness

This option displays the Edge Stiffness input box, shown in Figure 3.42. The user may enter stiffnesses which are distributed along an element edge.

Element: the user can select a single element or a group of elements with the same edge stiffness (see 3.2.21 Input Data – Edge Sets).

Side No.: each element may have 3 sides or 4 sides. The user needs to specify which side with the edge stiffness.

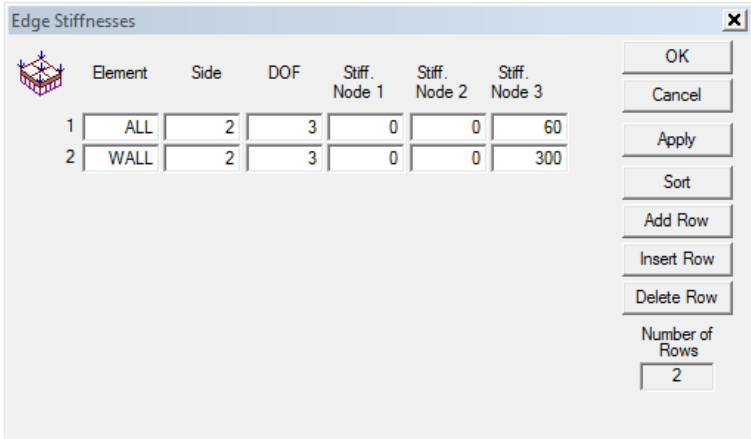
DOF Index: there are 6 degrees of freedom on each edge node. The degree freedom index is 1 for translation in X direction, 2 for translation in Y direction, 3 for translation in Z direction, 4 for rotation about X axis (not available in this program), 5 for rotation about Y axis, and 6 for rotation about Z axis.

Stiff Node 1: each side of an element usually has either 2 nodes or 3 nodes. Stiff Node 1 is for one of the edge nodes. Enter the translation or rotational stiffness (based on the DEF index) on this nodal point.

Stiff Node 2: each side of an element usually has either 2 nodes or 3 nodes. Stiff Node 2 is for the other one of the edge nodes. Enter the translation or rotational stiffness (based on the DEF index) on this nodal point.

Stiff Node 3: each side of an element usually has either 2 nodes or 3 nodes. Stiff Node 3 is for the middle node if quadratic element is used. Enter the translation or rotational stiffness (based on the DEF index) on this nodal point.

The units for edge translation stiffness are F/L^2 . The units for edge rotational stiffness are $F-L/Rad./L$.



The 'Edge Stiffnesses' window contains a table with the following data:

	Element	Side	DOF	Stiff. Node 1	Stiff. Node 2	Stiff. Node 3
1	ALL	2	3	0	0	60
2	WALL	2	3	0	0	300

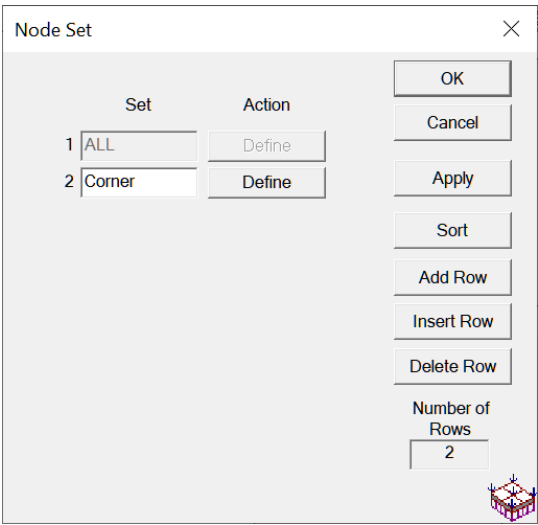
Buttons on the right: OK, Cancel, Apply, Sort, Add Row, Insert Row, Delete Row, Number of Rows (2).

Figure 3.42 Window screen for Input Data Menu – Edge Stiffness

3.2.21 Input Data – Node Sets

This window, shown in Figure 3.43, allows the user to define groups of nodes. This option facilitates the assignment of nodal properties by grouping nodes together into node sets. The user enters an alpha-numeric name for the set, then selects the nodes to be placed in that group. To use a defined node set, the name of the node set is entered in lieu of the node number. By default, the set ALL is defined as the node set containing all of the nodes in the slab.

To select the nodes to be included in a named group, click the Define button. This opens the screen shown in Figure 3.44. Using the Add Row button, enter the range of nodes to be added to the set. For greater ease in selecting nodes, use the **Selection - Select Node** function to “point and click” on the desired nodes.



The 'Node Set' window contains a table with the following data:

	Set	Action
1	ALL	Define
2	Corner	Define

Buttons on the right: OK, Cancel, Apply, Sort, Add Row, Insert Row, Delete Row, Number of Rows (2).

Figure 3.43 Window screen for Input Data Menu – Node Sets

Node Number/Set	Final	Increment
1	15	3

Number of Rows: 1

Figure 3.44 Window screen for selecting nodal points under Input Data Menu – Node Sets

Alternatively, users can define this node set by mouse clicking. Click “**Add Row**” to start a new node set data line. Check the node selection icon  to be available. Click this icon, LEFT-CLICK to pick one node or drag a window to select multiple nodes. Node number, final and increment will be filled by *GeoMat* automatically. If multiple nodes are selected, multiple new data lines will be added that each line has only one node.

3.2.22 Input Data - Element Set

This window, shown in Figure 3.45, allows the user to define groups of elements. This option facilitates the assignment of element properties by grouping elements together into element sets. The user enters an alpha-numeric name for the set, then selects the elements to be placed in that group. To use a defined element set, the name of the element set is entered in lieu of the element number. By default, the set ALL is defined as the element set containing all of the elements in the slab.

To select the elements to be included in the group, click the Define button. This launches a window similar to that seen in Figure 3.46. For greater ease in selecting elements, use the **Selection - Select Element** function to “point and click” on the desired elements.

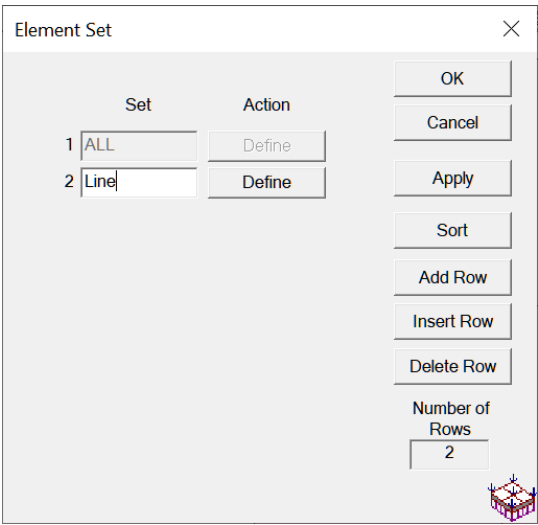


Figure 3.45 Window screen for Input Data Menu – Element Sets

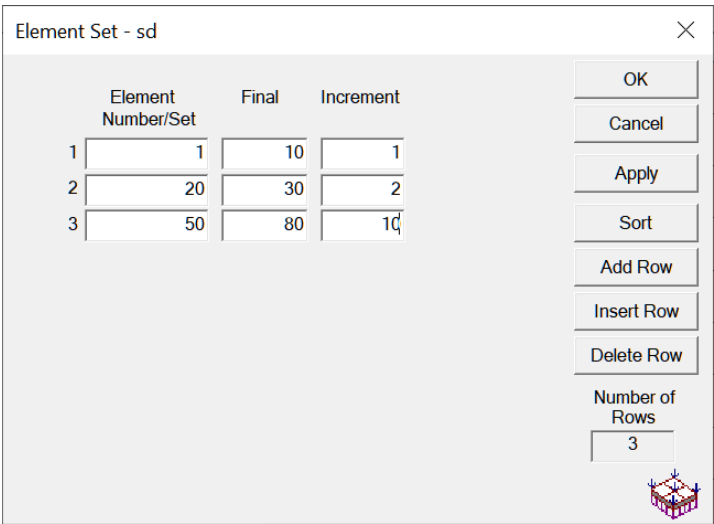


Figure 3.46 Window screen for selecting nodal points under Input Data Menu – Element Sets

Alternatively, users can define this element set by mouse clicking. Click “**Add Row**” to start a new element set data line. Check the element selection icon  to be available. Click this icon, LEFT-CLICK to pick one element or drag a window to select multiple elements. Element number, final and increment will be filled by *GeoMat* automatically. If multiple elements are selected, multiple new data lines will be added that each line has only one element.

3.2.23 Input Data – Edge Sets

This window, shown in Figure 3.47, allows the user to define groups of element edges. This option facilitates the assignment of edge properties by grouping edges together into edge sets. The user enters

an alpha-numeric name for the set, then selects the edges to be placed in that group. To use a defined edge set, the name of the edge set is entered in lieu of the element and edge number. By default, the set ALL is defined as the set containing all of the edges in the slab.

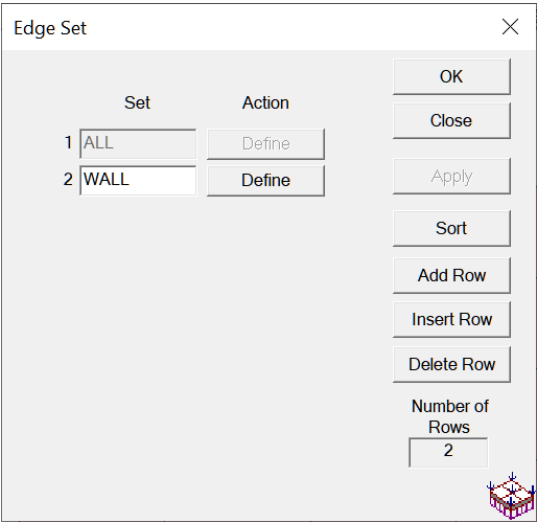


Figure 3.47 Window screen for sample Input Data Menu – Edge Sets

To select the edges to be included in a named group, click the Define button. This opens the screen shown in Figure 3.48. Using the Add Row button, enter the edges to be added to the set.

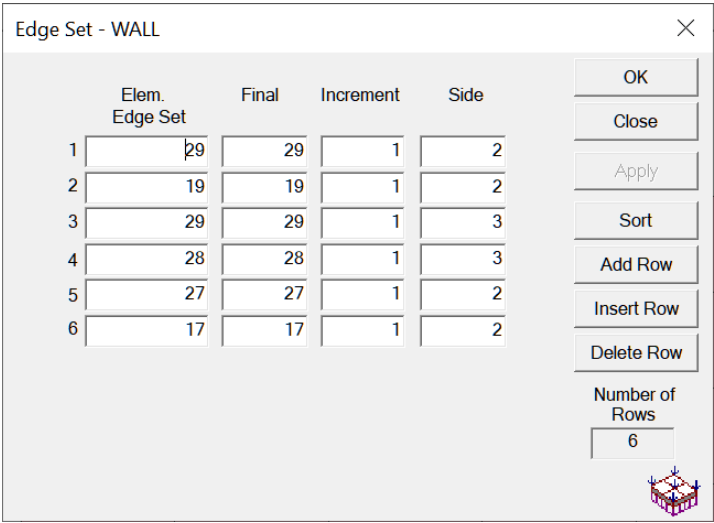


Figure 3.48 Window screen for selecting edges for Input Data – Edge Sets

Alternatively, users can define this edge set by mouse click. Click **“Add Row”** to start a new edge data line. Check the edge selection icon  to be available. Click it, then LEFT-CLICK the edge to be added into the set. Element number and side number will be completed by *GeoMat* automatically. If multiple

edges are added into the set, users have to add them one-by-one manually. Selecting multiple edges by dragging a window is not available to the edge set. This is different from the operation to define a node set or an element set.

3.2.24 Input Data – Section Cut

Section cut is a useful tool to plot stress components along one or more straight cut lines. The input interface is shown as in Figure 3.49.

1. Number of sample division of each segment.

When one cut line passes through the region of a mesh element, it will be divided into several line segments by intersected elements. The default segment division is 4 with equal spaces.

Outputs are given at 5 points: two locate on finite element edges, three locate inside this element. The default division number can be modified by user.

2. Source of stress points:

Two options are available to select the stress source for interpolation. Option one use stresses from Gaussian integration points. Another option uses stresses at the nodal position. Generally stresses are accurate at Gaussian integration points. The nodal values are averaged from adjacent elements and are less accurate. But at region with sharp deformation gradient, stresses are often not continuous from one element to another. Using the nodal value option is helpful to smooth the curve plot.

3. Define cut lines

The user inputs the end coordinates to define one cut line. Two default cut lines are defined automatically: one horizontal and one vertical. Both default cut lines pass through the origin point (0, 0). Note that the coordinates of end nodes of one cut line shall locate outside of the plate.

	Start Point		End Point	
	coordinate y	coordinate z	coordinate y	coordinate z
1	-5.5	0	5.5	0
2	0	-5.5	0	5.5
3	-5.5	-5.5	5.5	5.5

Figure 3.49 Window screen for sample Input Data Menu – Section Cut

The cut line can be horizontal, vertical or in an arbitrary angle. Section cut results can be found from the output file *.sou. One sample is shown in Figure 3.50:

Section cut output

Cut line=	1	Start at point: y=	-5.1000	z=	0.0000	End at point: y=	5.1000	z=	0.0000			
Element	S	Deflection	Theta_y	Theta_z	My	Mz	Myz	Q_xy	Q_xz			
181	0.0000E+00	0.0000E+00	0.1343E-20	-0.1430E-04	-0.1018E+01	-0.2250E+01	0.1340E+00	-0.2062E+02	0.9410E+01			
181	0.1250E+00	0.1799E-05	0.1187E-20	-0.1428E-04	-0.1156E+01	-0.2321E+01	0.1217E+00	-0.2028E+02	0.6337E+01			
181	0.2500E+00	0.3598E-05	0.1031E-20	-0.1426E-04	-0.1295E+01	-0.2392E+01	0.1095E+00	-0.1993E+02	0.3263E+01			
181	0.3750E+00	0.5398E-05	0.8753E-21	-0.1424E-04	-0.1433E+01	-0.2462E+01	0.9728E-01	-0.1959E+02	0.1890E+00			
181	0.5000E+00	0.7197E-05	0.7195E-21	-0.1422E-04	-0.1572E+01	-0.2533E+01	0.8505E-01	-0.1925E+02	-0.2885E+01			
182	0.5000E+00	0.7197E-05	0.7195E-21	-0.1422E-04	-0.1572E+01	-0.2533E+01	0.8505E-01	-0.1925E+02	-0.2885E+01			
182	0.6250E+00	0.9216E-05	0.1152E-20	-0.1413E-04	-0.1901E+01	-0.2916E+01	0.8801E-01	-0.1871E+02	-0.1420E+01			
182	0.7500E+00	0.1123E-04	0.1585E-20	-0.1404E-04	-0.2230E+01	-0.3299E+01	0.9097E-01	-0.1817E+02	0.4531E-01			
182	0.8750E+00	0.1325E-04	0.2018E-20	-0.1395E-04	-0.2559E+01	-0.3682E+01	0.9392E-01	-0.1764E+02	0.1510E+01			
182	0.1000E+01	0.1527E-04	0.2451E-20	-0.1386E-04	-0.2888E+01	-0.4064E+01	0.9688E-01	-0.1710E+02	0.2975E+01			
183	0.1000E+01	0.1527E-04	0.2451E-20	-0.1386E-04	-0.2888E+01	-0.4064E+01	0.9688E-01	-0.1710E+02	0.2975E+01			
183	0.1125E+01	0.1671E-04	0.2585E-20	-0.1374E-04	-0.3139E+01	-0.4536E+01	0.9802E-01	-0.1747E+02	0.1302E+01			
183	0.1250E+01	0.1814E-04	0.2720E-20	-0.1363E-04	-0.3389E+01	-0.5007E+01	0.9916E-01	-0.1784E+02	-0.3709E+00			
183	0.1375E+01	0.1958E-04	0.2854E-20	-0.1352E-04	-0.3639E+01	-0.5479E+01	0.1003E+00	-0.1821E+02	-0.2044E+01			
183	0.1500E+01	0.2102E-04	0.2988E-20	-0.1341E-04	-0.3890E+01	-0.5950E+01	0.1014E+00	-0.1859E+02	-0.3717E+01			
184	0.1500E+01	0.2102E-04	0.2988E-20	-0.1341E-04	-0.3890E+01	-0.5950E+01	0.1014E+00	-0.1859E+02	-0.3717E+01			
184	0.1625E+01	0.2322E-04	0.2377E-20	-0.1326E-04	-0.4187E+01	-0.6457E+01	0.1022E+00	-0.1921E+02	-0.1880E+01			
184	0.1750E+01	0.2543E-04	0.1766E-20	-0.1311E-04	-0.4485E+01	-0.6965E+01	0.1029E+00	-0.1984E+02	-0.4263E-01			
184	0.1875E+01	0.2764E-04	0.1154E-20	-0.1297E-04	-0.4783E+01	-0.7472E+01	0.1037E+00	-0.2047E+02	0.1795E+01			

where “s” is the distance to the most left or top mesh edge.

Figure 3.50 Sample of output data from Edge Sets option

Default color for the cut lines are blue. Currently selected cutline will be highlighted in red as in Figure 3.51. Users can edit the coordinates of the start/end point of a cut line. Then click **Apply** to make it effective.

Default two cut lines can not be edited or deleted.

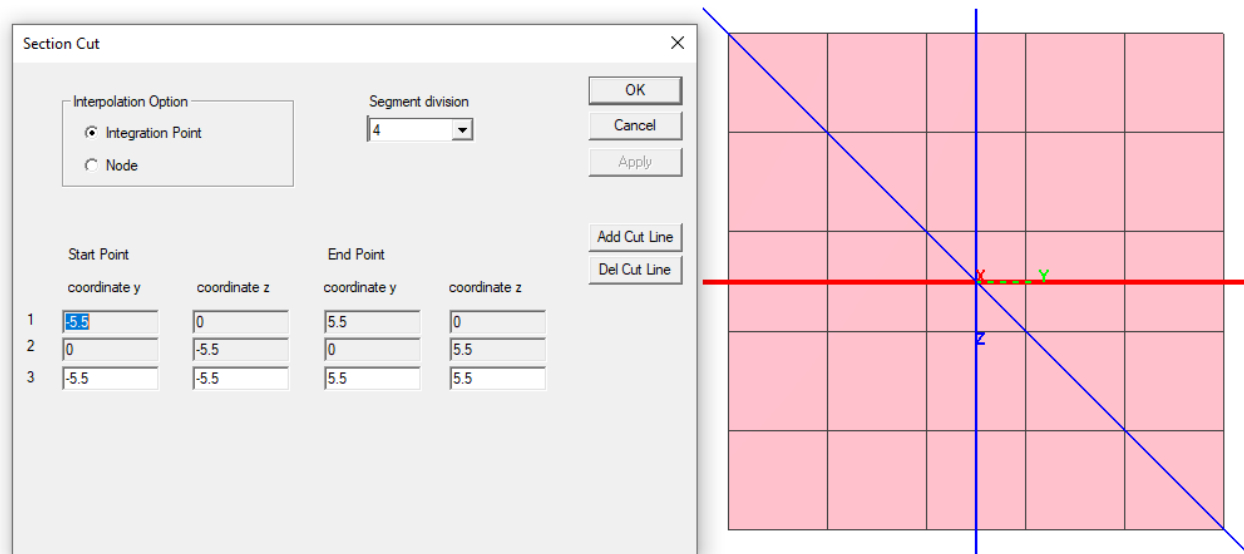


Figure 3.51 Window screen showing cutting sections by using Input Data Menu – Section Cut

3.2.25 Input Data – Node Path

Section cuts are used to plot X-Y curves of deformations, plate forces/stresses or distributed soil reactions. These results can be interpolated inside a finite element. In Geomat, these are some results available at nodal positions, such as nodal reactions from nodal soil stiffnesses or forces of a nonlinear spring. These nodal results cannot be interpolated inside a finite element. These

results can be plotted as discrete points along a node path. The dialog to define node paths is shown in Figure 3.52.

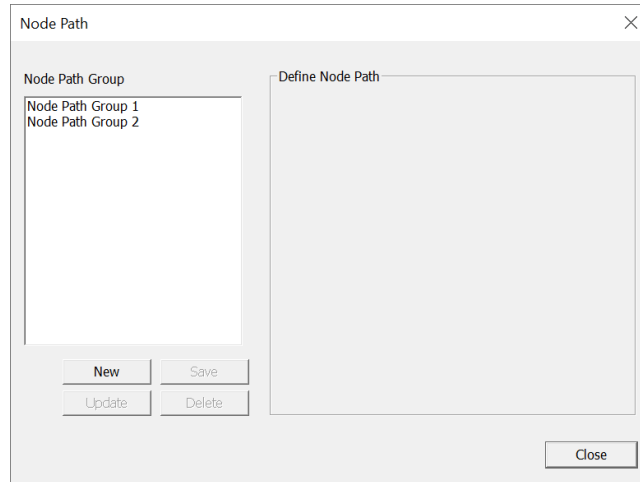


Figure 3.52 Define a node path for X-Y plots about discrete nodal results

Click **New** to create a new Path node path. Geomat provides three options to define a new node path. Users can input the start/end node numbers to create a straight path (Figure 3.53). Internal nodes are found by the program automatically. Alternatively, users can input node numbers directly in the node list window. A comma should be used to separate two node numbers.

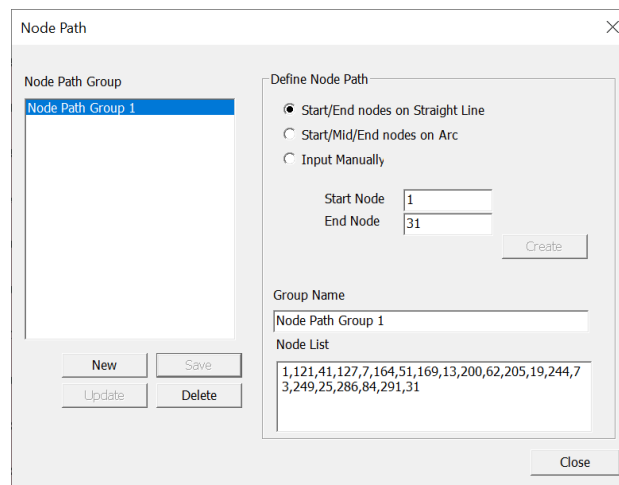


Figure 3.53 Define a new node path using start/end node numbers

The second option is to define node path along an arc (Figure 3.54). The path will start from the start node, pass through the mid node and end the path at end node. The mid node controls the path to be counter clockwise or clockwise. If three nodes don't locate on the same arc, an error message will popup to remind users that this path cannot be created.

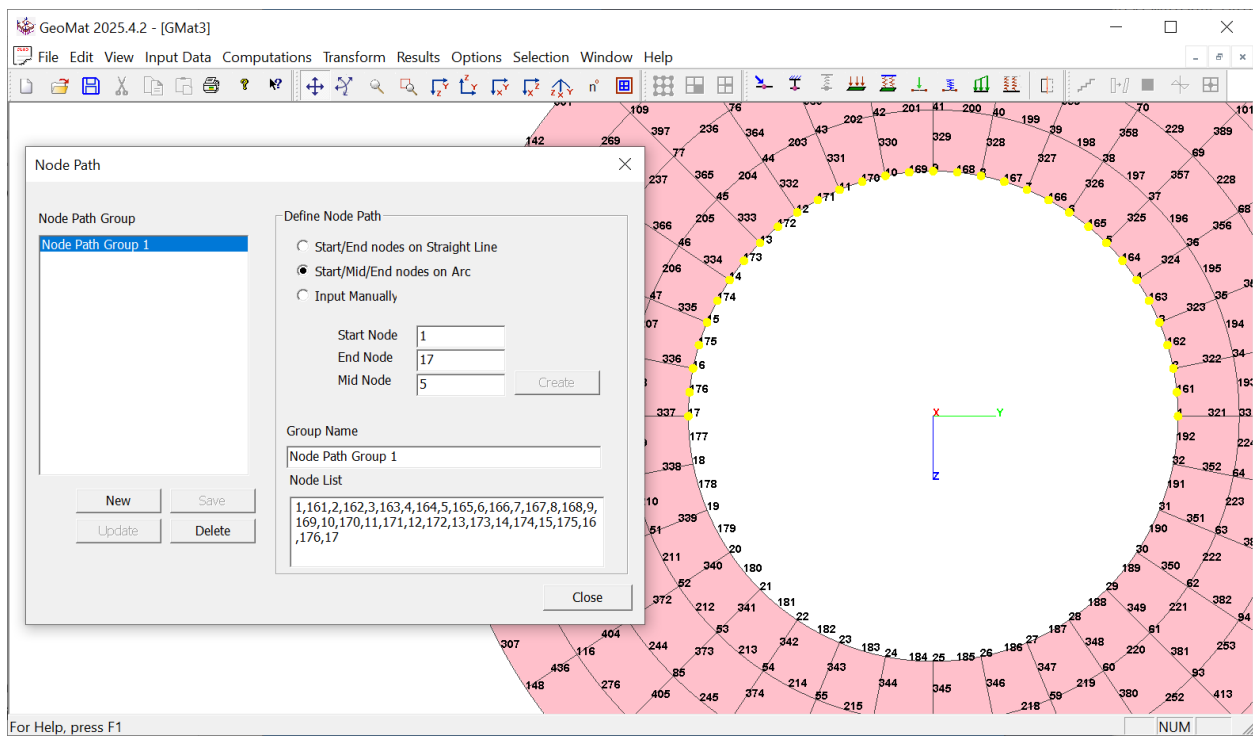


Figure 3.54 Define a new node path along an arc

The third method can generate a non-straight path, or skip some nodes by inputting nodes directly. Note the nodes should be sequential along the path. Nodes in a path will be highlighted in yellow as in Figure 3.55.

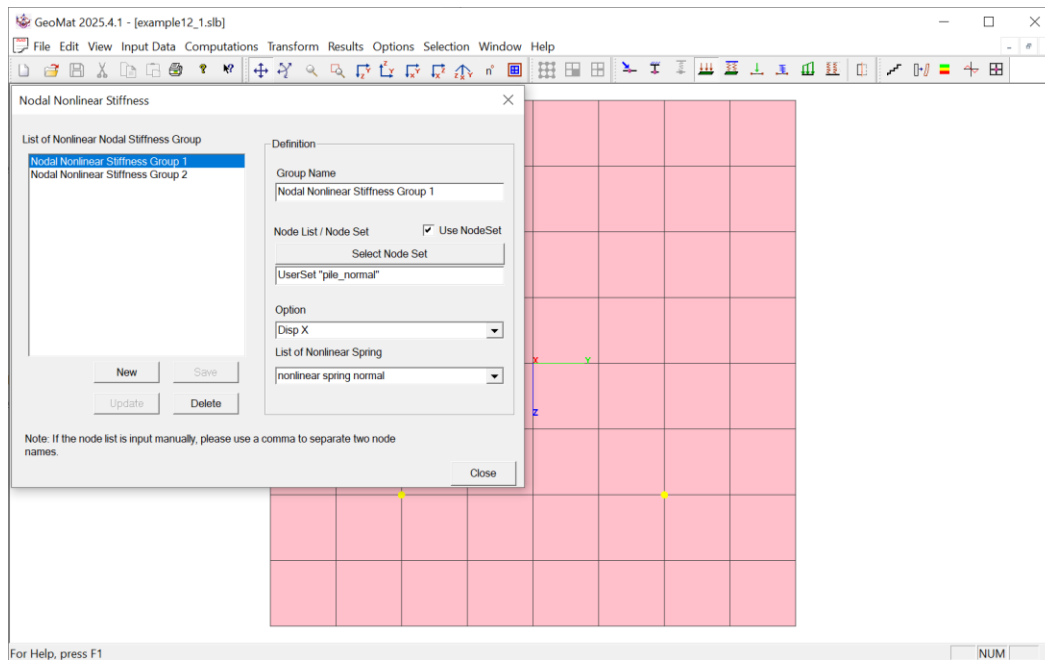


Figure 3.55 Define a new node path by inputting node numbers manually

After selecting a predefined node path from the group window, users can change the path definition and click **Update** to refresh the database.

Select one node path and click **Delete** will remove it from the model database.

4 Finite Element Mesh Adjustment

Capability of mesh adjustment is added into version *GeoMat 2022*. Users may refine current mesh for better convergence performance or make mesh coarser to reduce computing time. However, this operation may delete previously defined materials, loads, constrains or stiffnesses. Users shall check consistency before the new analysis.

4.1 Mesh Refinement

Mesh refinement option is incorporated into the Edit menu as shown in Figure 4.1. Based on the template used to build previous mesh, different interface window will prompt. If previous mesh is built from conventional circular/rectangular plate template, the mesh refinement window will look like that Figure 4.2. The default refinement factor for both y and z directions are set to 2 which means element number in y and z directions will be doubled. Refinement factor should be an integer. Any float number will be round up to an integer. If the refinement factor is smaller than 1, no new mesh is regenerated. If users set the division in one direction to be 1, there will be no mesh refinement in that direction.

Note this refinement approach is also applicable to unequally spaced mesh.

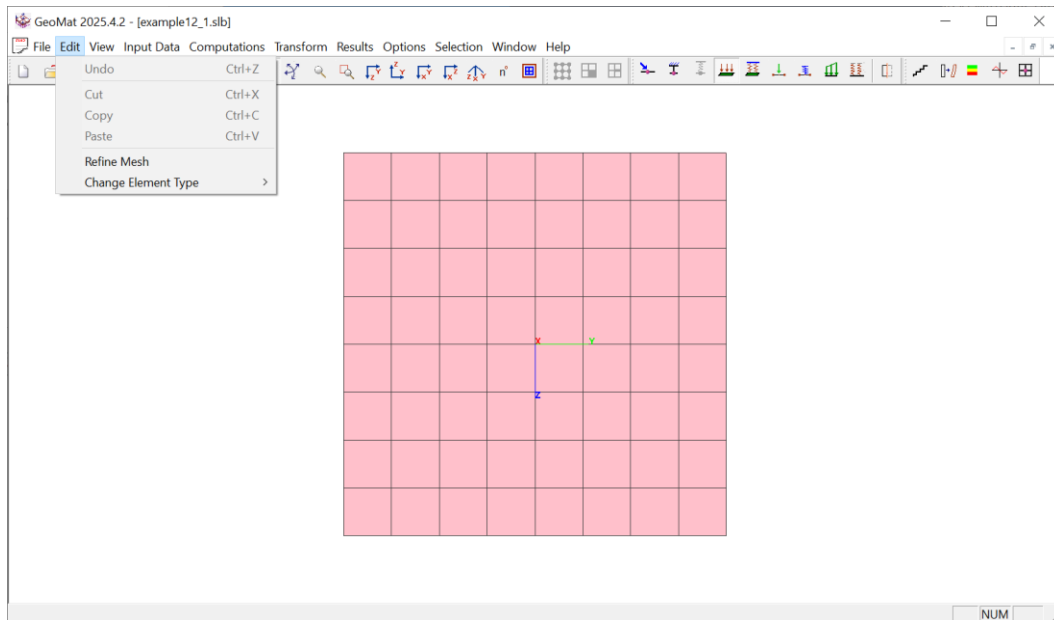


Figure 4.1 Mesh adjustment options contained in the Edit menu

Click button **OK** to close mesh refinement window. A message window will pop up to remind users to check consistency because previous boundary conditions or materials may be not available (Figure 4.3). If **Yes** is clicked, new mesh will be generated as shown in Figure 4.4. The division in Y or Z direction can be different if the mesh is created from conventional circular/rectangular or ring template. If the mesh belongs to user-modified or imported from an external mesh file, divisions in Y and z directions are limited to “2”. In this case, one element is split to four sub-elements as refinement.

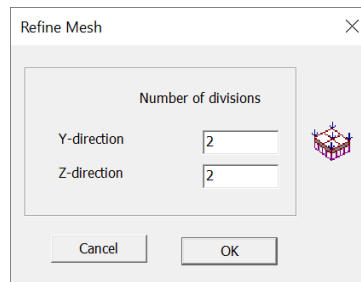


Figure 4.2 Mesh adjustment for conventional circular/rectangular plate

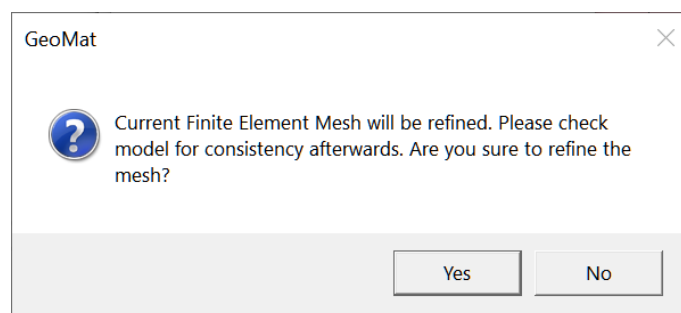
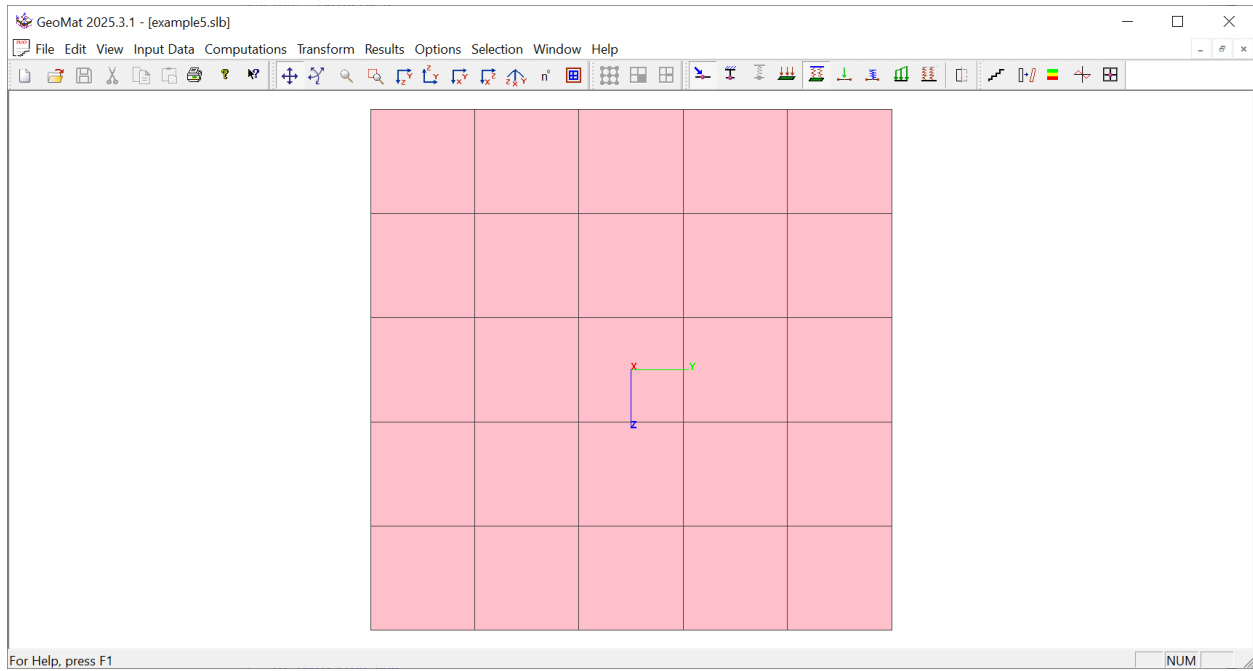


Figure 4.3 Information message before mesh adjustment is processed

If previous mesh is built from the template of octagon/circular wind turbine foundation, the mesh refinement input window will look like that in Figure 4.5. Users may change the typical finite element size to a new value for the new mesh. Previous geometry data and load data will be kept and convert to new mesh. Users still should check consistency of materials, node constrains or other boundary conditions defined in previous mesh.

**Figure 4.4 Refined mesh in both y and z directions**

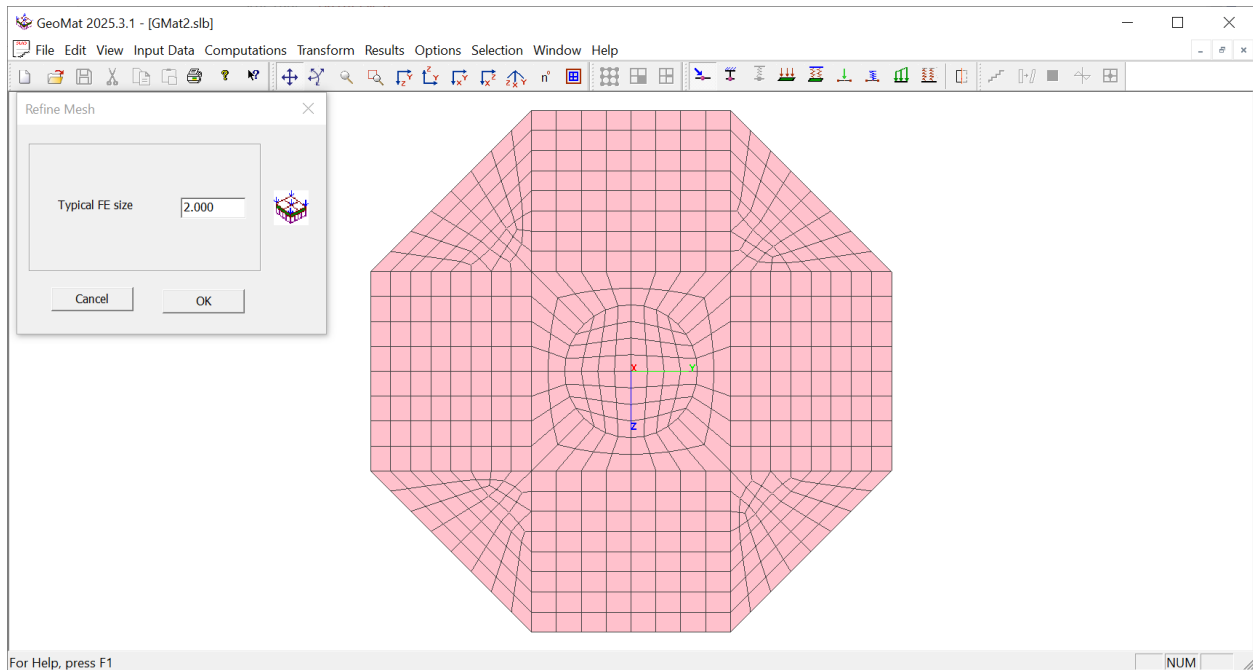


Figure 4.5 Refine mesh for wind turbine foundation

4.2 Change of Element Type

It is recommended in *GeoMat* to use quadratic eight-node element combined with 2x2 integration rule. This element type provides good accuracy and efficiency so that it is appropriate for most cases. If another element type is desired, users may click menu **Edit->Change Element Type** and select another one. If quadratic eight-node is the current element type, only conversion to linear four-node or Lagrangian nine-node element type is available. A message window will pop up to remind users to check consistency after conversion (Figure 5.7). Since converting to a new element type will regenerate the mesh, all previously defined boundaries/materials may be invalid. After a successful conversion, nodes will be highlighted in red as in Figure 4.8. Click **Options->Mark Nodes** to disable node mark.

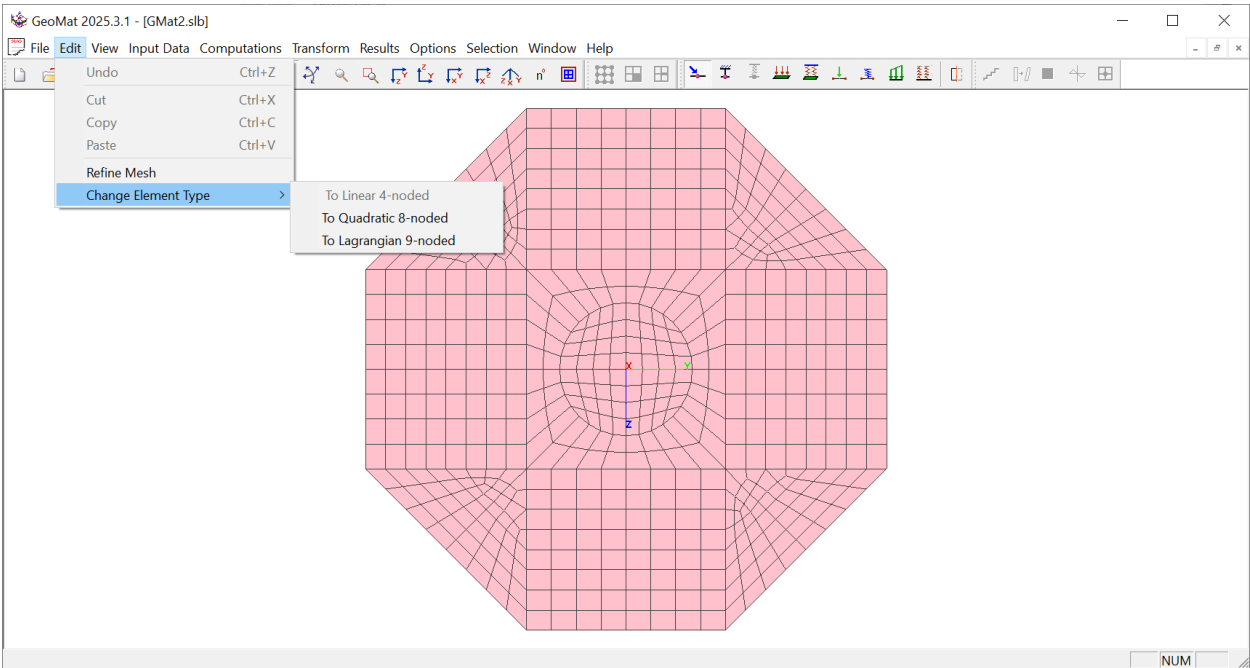


Figure 4.6 Change element type from Edit menu

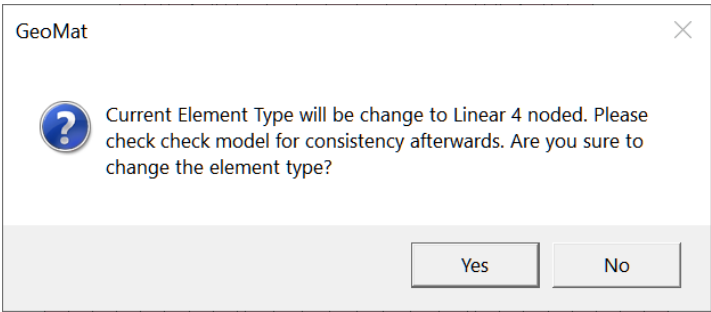


Figure 4.7 Information message before change of element type is processed

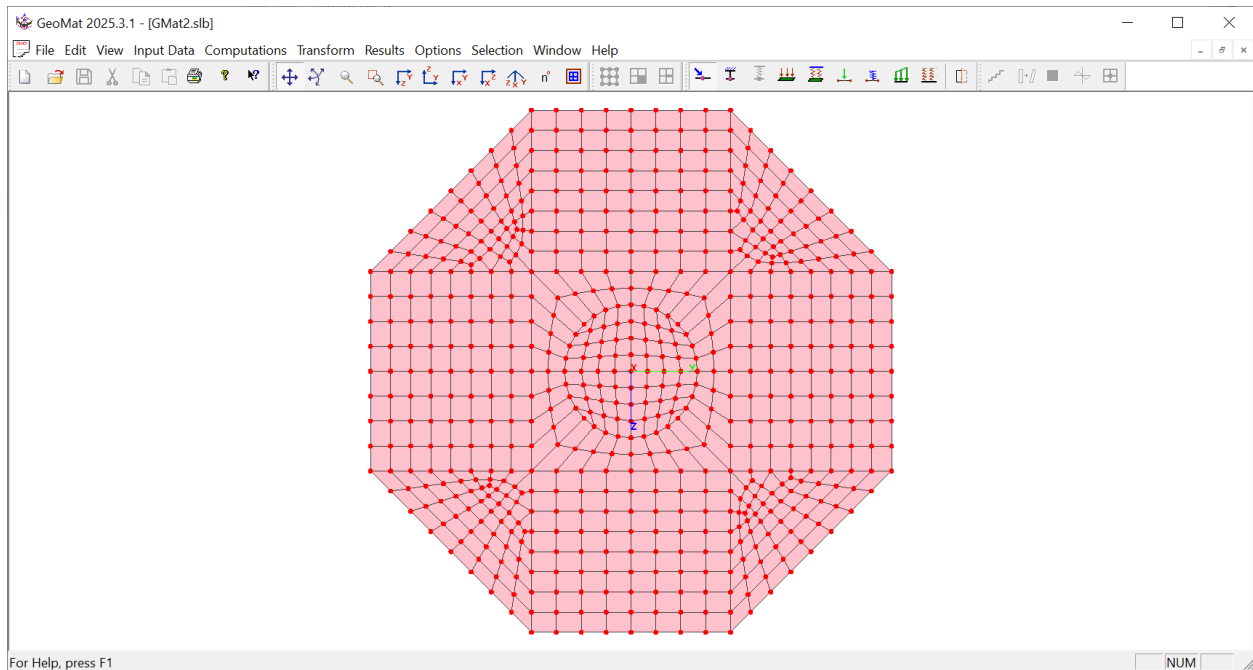


Figure 4.8 Mesh after change of element type from 8-node to 4-node

4.3 Change of Integration Rule

Default integration rules in *GeoMat* are defined as following:

- 1x1 or 2x2 are available to four-node element. 2x2 is recommended
- 2x2 or 3x3 are available to eight-node element. 2x2 is recommended
- 3x3 is available to nine-node Lagrangian element. It is the only integration rule for this element type.

Users may change integration rule by clicking menu **Edit->Input Data->Integration Points** then select one rule(Figure 5.9). Changing integration does not regenerate the mesh. Previously defined material, boundaries, various sets can be kept.

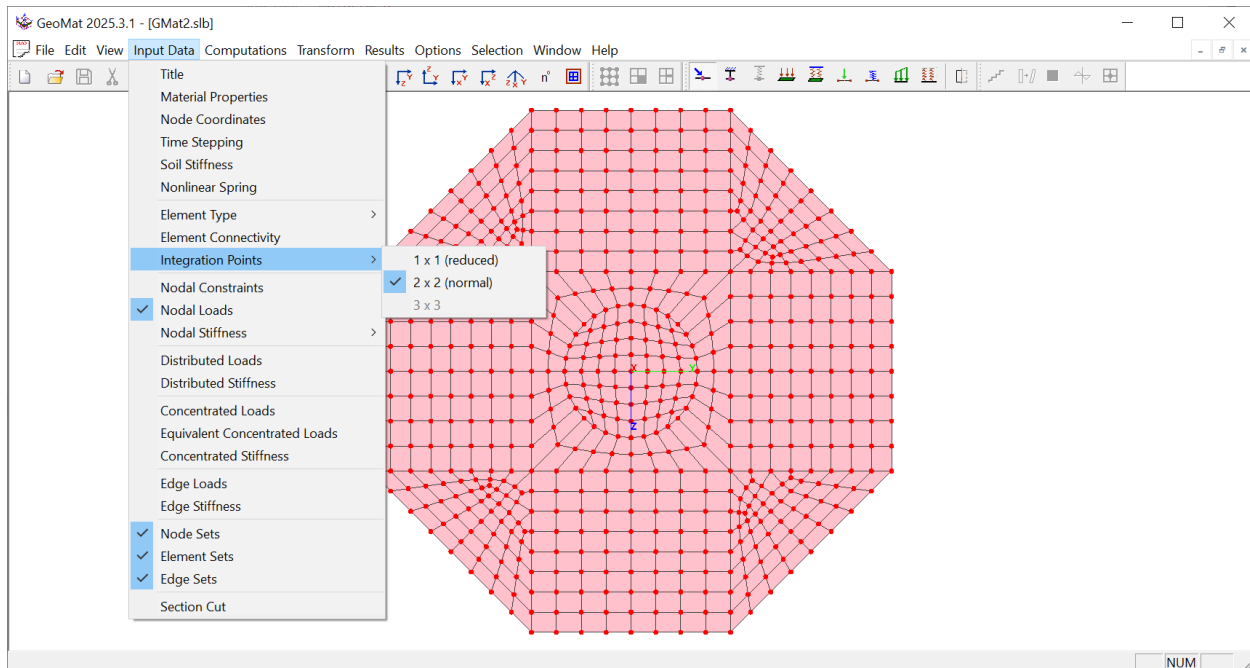


Figure 4.9 Change integration rule from Edit menu

4.4 Manual Modification of Nodes and Elements

GeoMat provides the ability to let users modify a mesh generated from mesh template. Users can add/delete a node, change coordinates of a node, add/delete an element manually. In order to create a valid user-defined mesh, it is recommended to follow steps in this section.

4.4.1 Step 1: Modify Nodes

Click the menu “**Option->Node Number**” to turn on display of node numbers. Then click the menu “**Input Data->Node Coordinates**” to open the node definition window as shown in Figure 4.10. Users can change coordinates of nodes to change the geometry of the slab. Note that mid nodes should be modified to locate at the middle position of edges after coordinates of corner nodes are changed.

4.4.2 Step 2: Modify Elements

Click the menu “**Option->Element Number**” to turn on display of element numbers. Click the menu “**Input Data->Element Connectivity**” to open the element definition window as shown in Figure 4.11. Click “**Delete Element**” will delete the element at current mouse cursor position which is high-lighted as cyan color. After deleting unwanted elements, orphan nodes will be deleted automatically. Number of nodes and elements are also reordered to keep continuity.

Node Coordinates

	Node Number	Coord. X	Coord. Y	Coord. Z
82	82	0	-0.1	-0.4
83	83	0	0.1	-0.4
84	84	0	0.3	-0.4
85	85	0	0.5	-0.4
86	86	0	-0.5	-0.5
87	87	0	-0.4	-0.5
88	88	0	-0.3	-0.5
89	89	0	-0.2	-0.5
90	90	0	-0.1	-0.5
91	91	0	0	-0.5
92	92	0	0.1	-0.5
93	93	0	0.2	-0.5
94	94	0	0.3	-0.5
95	95	0	0.4	-0.5
96	96	0	0.5	-0.6

OK

Close

Apply

Sort

Number of Nodes
96



Figure 4.10 Window for node definition

Element Connectivity

	Element Number	Node 1	Node 2	Node 3	Node 4	Node 5	Node 6	Node 7	Node 8	Node 9
1	1	1	2	8	7	37	43	48	42	
2	2	2	3	9	8	38	44	49	43	
3	3	3	4	10	9	39	45	50	44	
4	4	4	5	11	10	40	46	51	45	
5	5	5	6	12	11	41	47	52	46	
6	6	7	8	14	13	48	54	59	53	
7	7	8	9	15	14	49	55	60	54	
8	8	9	10	16	15	50	56	61	55	
9	9	10	11	17	16	51	57	62	56	
10	10	11	12	18	17	52	58	63	57	
11	11	13	14	20	19	59	65	67	64	
12	12	14	15	21	20	60	66	68	65	
13	13	19	20	26	25	67	73	78	72	
14	14	20	21	27	26	68	74	79	73	
15	15	21	22	28	27	69	75	80	74	

OK

Close

Apply

Sort

Delete Element

Number of Elements
22



Figure 4.11 Window for element definition

Figure 4.12 shows one example of a user-modified mesh. Slabs with special shapes can be created by deleting unwanted elements.

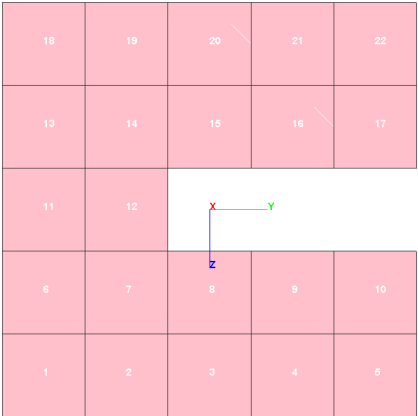


Figure 4.12 User modified mesh by deleting several middle elements

4.4.3 Update Set Definition

After users modify the mesh, previous nodal sets and the element sets usually need to be updated to remove deleted nodes/elements or add into new nodes/elements.

5 Program Execution and Output Reviews

5.1 Introduction

Chapter 4 presents options related to execution of the program and includes methods of addressing run-time errors. This chapter also includes suggestions for reviewing input, output, and processor text files. The final section of this chapter includes descriptions about the slab properties that may be observed in graphical form. The commands covered in this chapter are contained in the top menu, under the **Computation** and the **Results** titles.

5.2 Computation Menu

This menu is used to execute the program with the parameters that were saved in the input-data file. Within the options contained under this menu, shown in Figure 5.1, there are commands that facilitate the reviews of the text files produced for storing input data, output results, and processor notes. Detailed description of the submenu options contained under the **Computation** menu is explained in the following sections.

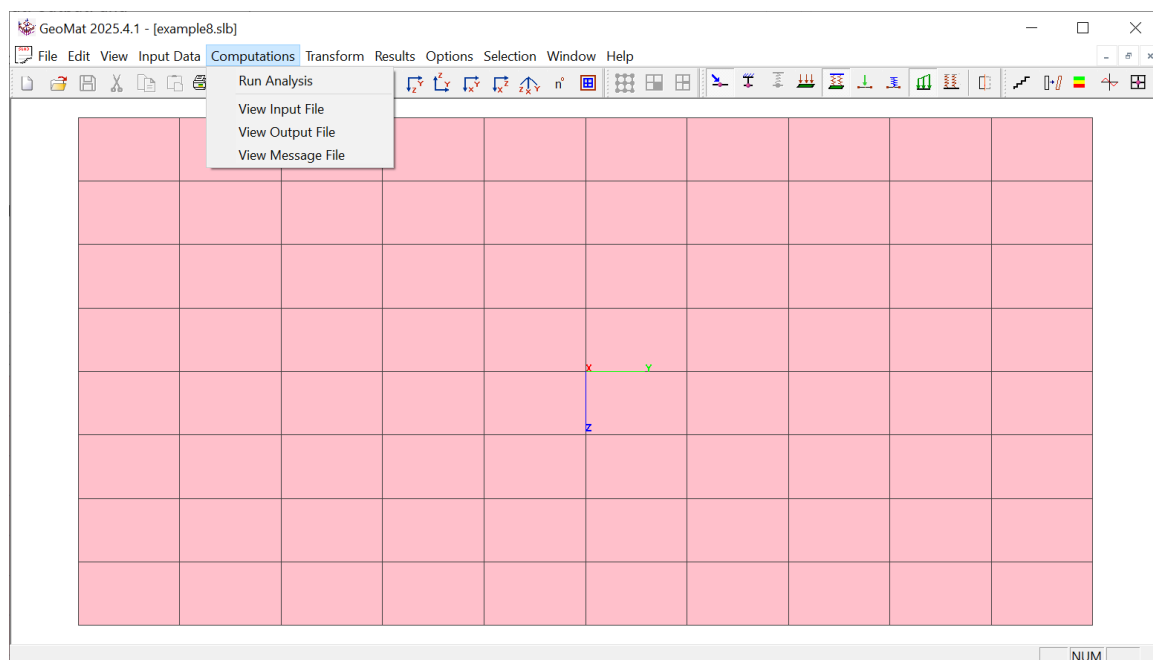


Figure 5.1 Options contained in the Computation menu

5.2.1 Computation- Run Analysis

An input file, after preparation or modification, must be saved to disk before selecting the **Computation -Run Analysis** submenu option, which executes the analytical portion of program *GeoMat*. The module, a stand-alone program routine, is called for execution by the "shell" process of the environment in Microsoft Windows[®].

The user should remember to save the input data under a user-specified name before executing the analytical module. When saving data to disk, *GeoMat* will automatically add an extension of the type **.slb* to the name of the input file. A sub-window is usually produced during the execution of the module. A finished execution of the module is determined when the sub-window closes.

At the beginning of the run, the analytical module will read the saved input data progressively, showing the line number which is being read. If an input-data format is incorrect during reading, the analytical module will stop immediately and in many cases it saves an error message and a status report in a file with the extension **.sms*. This file may be accessed by selecting **View Message File** in the **Computation** menu.

Within the processor-run notes, if all input data was read correctly, the analytical module will show the ending time of the analysis. The analytical module automatically creates an output file with the same name as the input and with the extension **.sou*. Once a successful run is produced, the user may proceed to the next items for observation of results.

5.2.2 Computation- View Input File

This submenu option is used to edit the input-data file in plain 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. This submenu automatically invokes a text editor. The default setting is to use the utility program named *notepad.exe* provided by Microsoft Windows[®]. Input-data text files are automatically saved to disk with the extension of **.sin* by program *GeoMat*. Use of the notepad for editing the input data for Example Problem 1 is shown in Figure 5.2.

```

example8.slb - Notepad
File Edit Format View Help
V4.1
TITLE
Example 8
UNIT_DATA
*
0
END_UNIT_DATA
NODE_DATA
*
1      0      -8      4
*
2      0      -7.2    4
*
3      0      -6.4    4
*
4      0      -5.6    4
*
5      0      -4.8    4
*
6      0      -4      4
*
7      0      -3.2    4
*
8      0      -2.4    4
*
9      0      -1.6    4
*
10     0      -0.8    4
*
11     0      0      4
*
12     0      0.8     4
*
13     0      1.6     4
*
14     0      2.4     4
*
15     0      3.2     4
*
16     0      4      4
*
17     0      4.8     4
*
18     0      5.6     4
*
19     0      6.4     4
*
20     0      7.2     4
*
21     0      8      4

```

Figure 5.2 Sample use of Microsoft Notepad for editing input text of Example Problem 1.

5.2.3 Computation- View Output File

This submenu option is used to edit 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 successfully executed, or when opening previously-executed data files.

The submenu automatically invokes a text editor. The default setting is to use the utility program named *notepad.exe* provided by Microsoft Windows[®]. Output files are automatically saved to disk with the same file name as the input-data file but with the extension **.sou*. Use of Microsoft Notepad[®] for editing the output file for Example Problem 1 is shown in Figure 5.3.

The output file contains all input information in this model, such as nodal coordinates, element connections, fixed boundary and various stiffness definitions. It also contains results such as deformations at nodes, plate stresses at element integration points, averaged soil reaction pressure on elements, reactions as fixed boundary nodes and results along a section cut line.

```

example1.sou - Notepad
File Edit Format View Help

Example 1

npoin = 8      nelelem = 1      nvfix = 7      ncase = 1      ntype = 0      nnode = 8      ndofn = 6
nmats = 1      nprop = 3      ngaus = 2      ngaus2 = 2      ndime = 3      nstre = 8      nevab = 48
nnlod = 3      nnsti = 0      ndlod = 0      ndsti = 0      nclod = 0      ncsti = 0      nctsti = 0
dt = 1.0000      tini = 0.0000      tend = 1.0000      dtmin = 1.0000      maxiter = 10

element material node numbers
1 1 1 3 8 6 2 5 7 4

nodal point coordinates
node x y z
1 0.0000 -5.0000 5.0000
2 0.0000 0.0000 5.0000
3 0.0000 5.0000 5.0000
4 0.0000 -5.0000 0.0000
5 0.0000 5.0000 0.0000
6 0.0000 -5.0000 -5.0000
7 0.0000 0.0000 -5.0000
8 0.0000 5.0000 -5.0000

restrained nodes
nodf codf fixed values
1 000101 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
3 000101 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
4 000101 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
5 000101 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
6 100111 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
7 100111 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
8 100111 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00

material properties

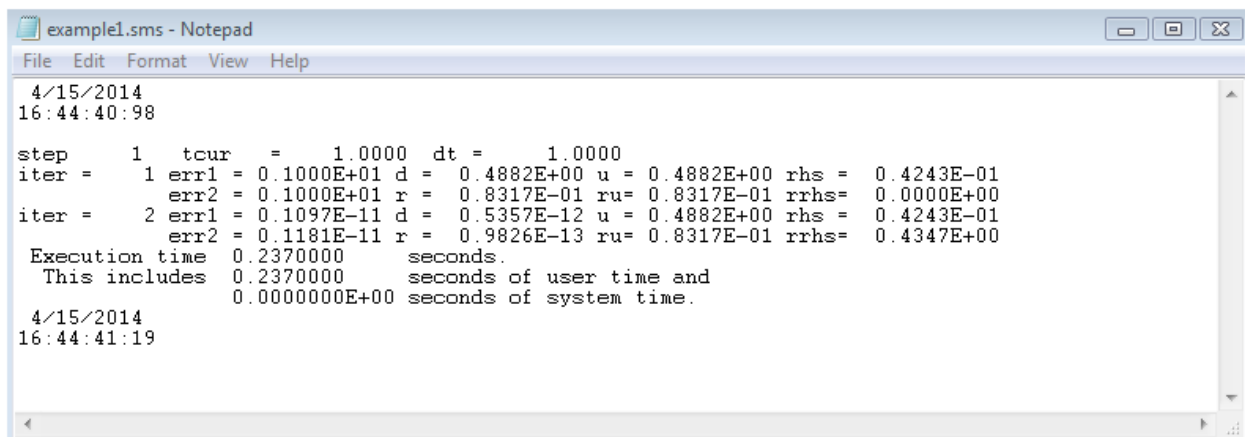
```

Figure 5.3 Sample use of Microsoft Notepad for view output text of Example Problem 1.

5.2.4 Computation- View Message File

This submenu option is used to review messages created in analysis. Before *GeoMat* will check validity of the model and print messages if the model has errors. This file only includes notes produced during the processing of the input data. This submenu option becomes active after new-data files have been saved to disk and executed, or when opening previously-executed data files. This submenu automatically invokes a text editor. The default setting is to use the utility program named *notepad.exe* provided by Microsoft Windows[®].

Files containing processor-run notes are automatically saved to disk with the same file name as the input-data file but with the extension **.sms*. Using of the Microsoft Notepad[®] for editing the processor-run notes for Example Problem 1 is shown in Figure 5.4. Observation of the notes produced during a processor run may become helpful to debug a data file that did not produce a successful run. A successful run usually produces a file of processor-run notes containing similar lines of text as those in Figure 5.4.



```

example1.sms - Notepad
File Edit Format View Help
4/15/2014
16:44:40:98

step    1    tour    =    1.0000    dt =    1.0000
iter =    1    err1 = 0.1000E+01 d = 0.4882E+00 u = 0.4882E+00 rhs = 0.4243E-01
        err2 = 0.1000E+01 r = 0.8317E-01 ru= 0.8317E-01 rrhs= 0.0000E+00
iter =    2    err1 = 0.1097E-11 d = 0.5357E-12 u = 0.4882E+00 rhs = 0.4243E-01
        err2 = 0.1181E-11 r = 0.9826E-13 ru= 0.8317E-01 rrhs= 0.4347E+00
Execution time 0.2370000 seconds.
This includes 0.2370000 seconds of user time and
              0.0000000E+00 seconds of system time.

4/15/2014
16:44:41:19

```

Figure 5.4 Sample use of Microsoft Notepad for view message file of Example Problem 1.

5.3 Results Menu

This menu option is selected to observe the different graphical representations of the program results contained in the output file. Submenu options contained under this menu are shown in Figure 5.5. All of the graphical representation of output data that may be produced by the program are contained in the following commands of the **Results** menu:

Select Step

Deformed Mesh

Contour Plot

X-Y Plot

The observation of any of the above-listed options will activate the graphics mode of *GeoMat*. Several changes occur during use of the graphics mode: new mouse commands are enabled and a new top-menu option becomes available.

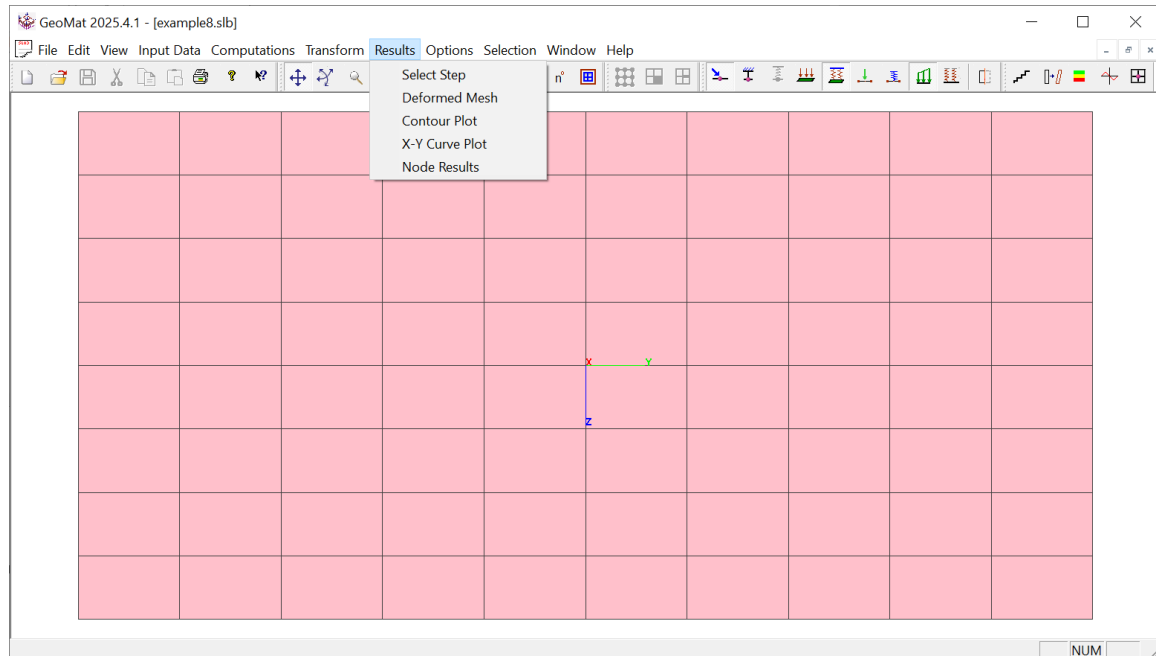


Figure 5.5 Command options contained in Result menu

5.3.1 Results – Select Step

The user may select this command option to observe a graphical representation of the time steps used to compute the solution. A sample screen of the **Select Step** command option is shown in Figure 5.6. This is ideally used in conjunction with the other two available menu options, **Deformed Mesh** and **Contour Plot**.

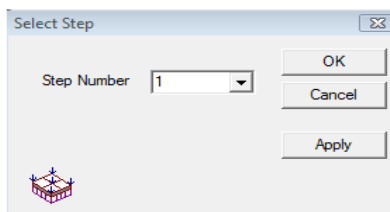


Figure 5.6 Sample screen of the Select Step command option

5.3.2 Results – Deformed Mesh

The user may select this command option to observe a graphical representation of the deformed shape for the modeled slab. A sample screen of the **Deformed Mesh** command option is shown in Figure 5.7. The shape factor for the amount of deformation may be varied by the user. The user also may choose to display undeformed shapes together with the deformed shape, as seen in Figure 5.8

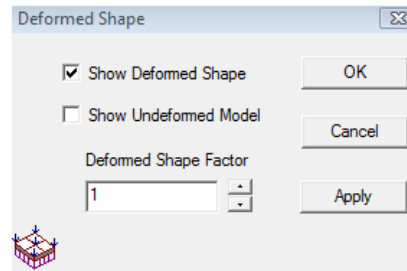


Figure 5.7 Sample screen of the **Deformed Mesh** command option

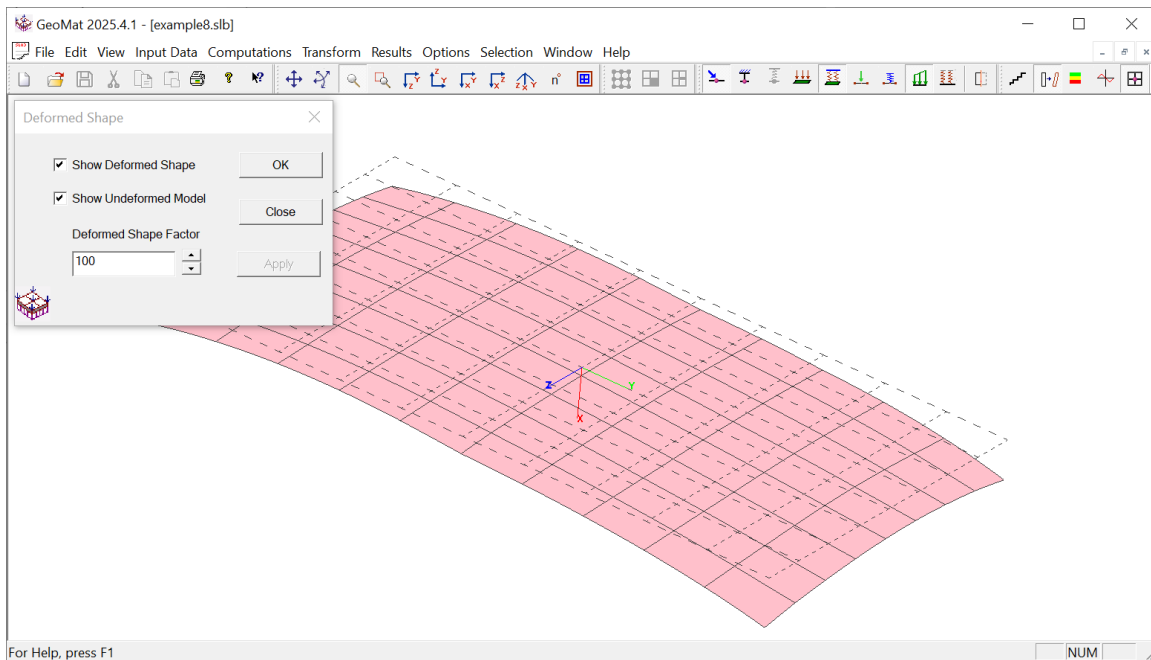


Figure 5.8 Combined display of deformed and undeformed configuration

5.3.3 Results – Contour Plot

The user may select this command option to observe a contour plot of the output on the surface of the slab model. A sample screen of the **Contour Plot** command option is shown in Figure 5.9. The user may choose from a variety of contour distributions: displacements, reactions at fixed boundary, plate forces, plate stresses and distributed soil reaction pressure. The user may select the default contour limits or modify them. Contour types available are shading, lines, and shading and lines together. Figure 5.10 demonstrates the appearance of the line contour option, and Figure 5.11 the shading contour option for

vertical displacement. Figure 5.12 and Figure 5.13 demonstrate the shading contour option for moment and stress distributions, respectively. Figure 5.14 shows the contour of distributed soil reactions in case of the soil stiffness is defined in the model.

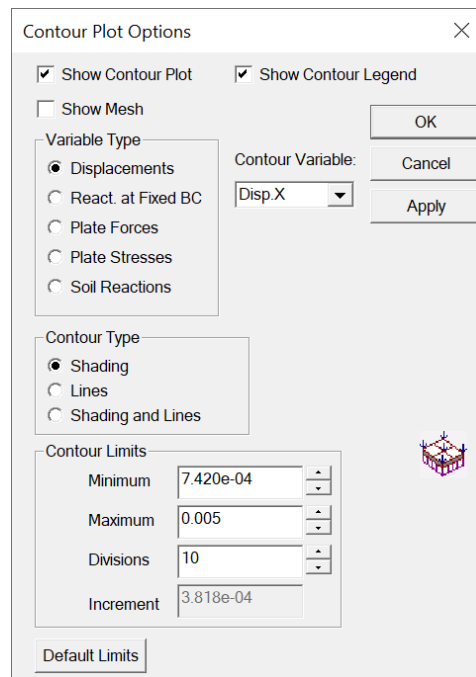


Figure 5.9 Sample screen of the Contour Plot command option

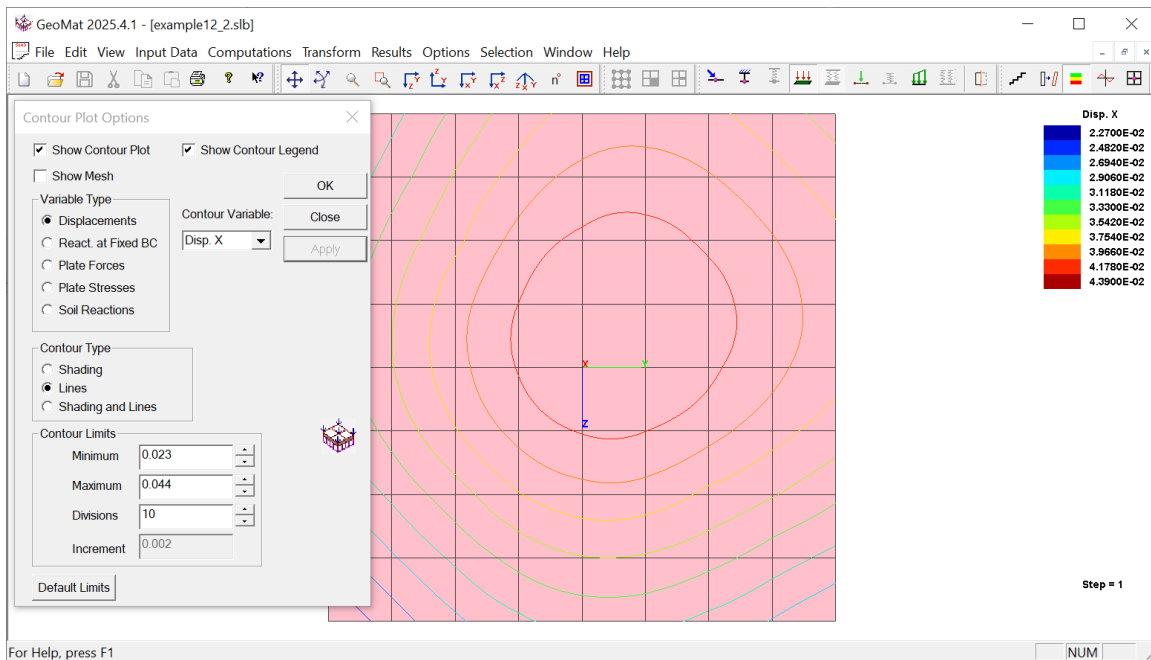


Figure 5.10 Line contour option for vertical displacement in Contour Plot

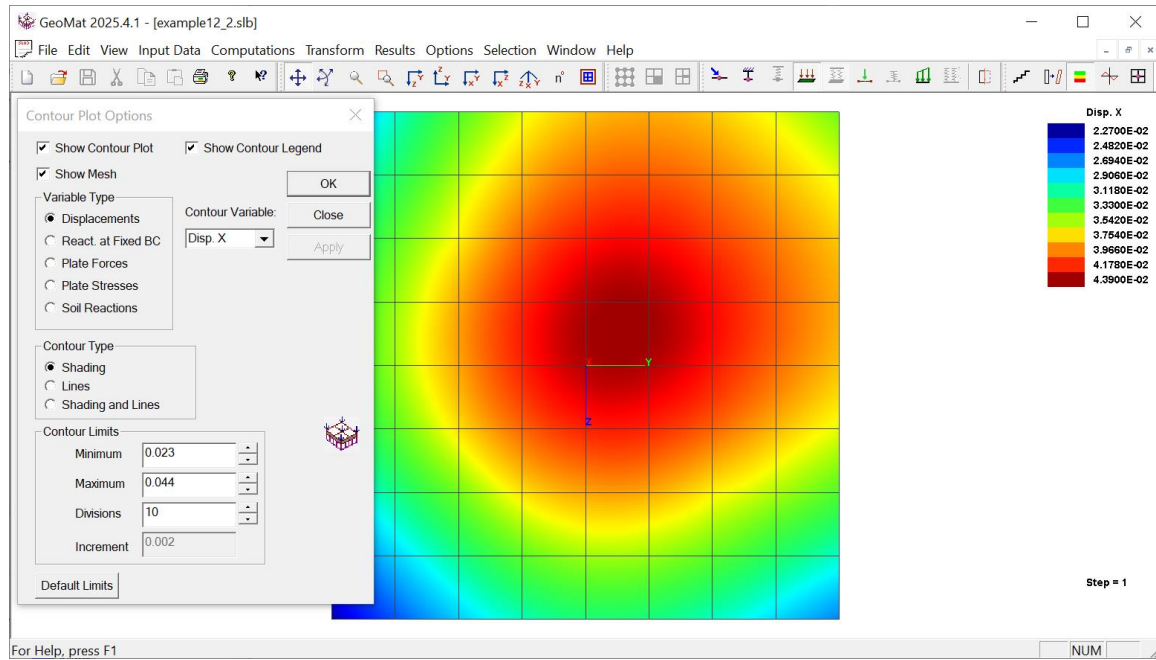


Figure 5.11 Shading contour option for vertical displacement in Contour Plot

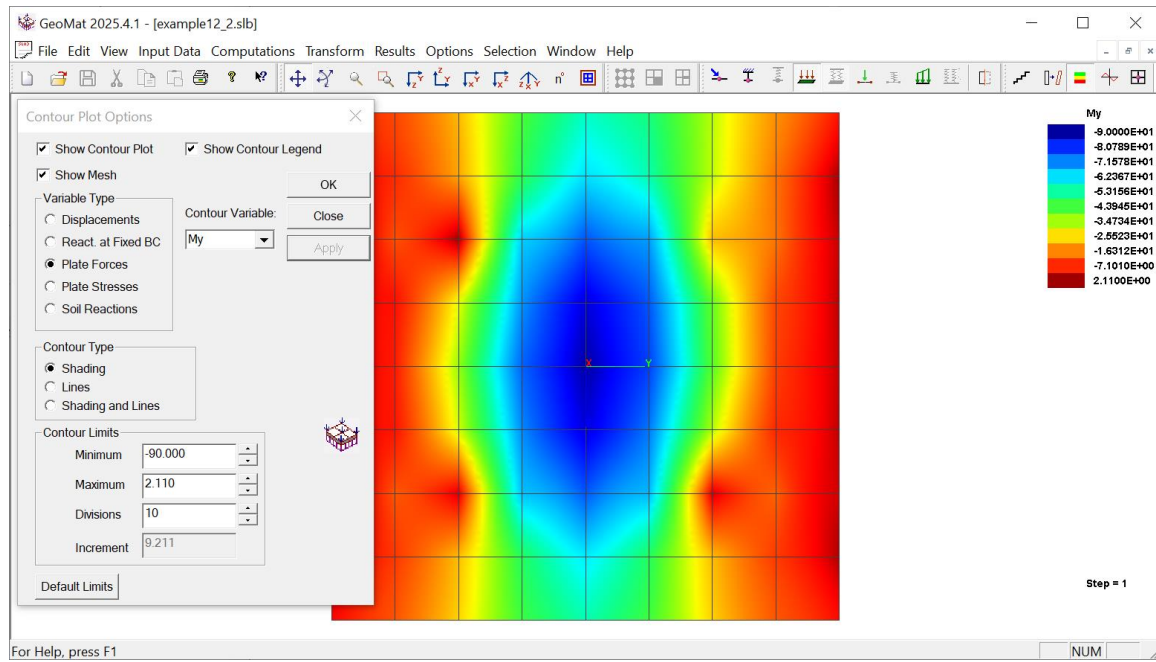


Figure 5.12 Shading contour option for moment distribution in Contour Plot

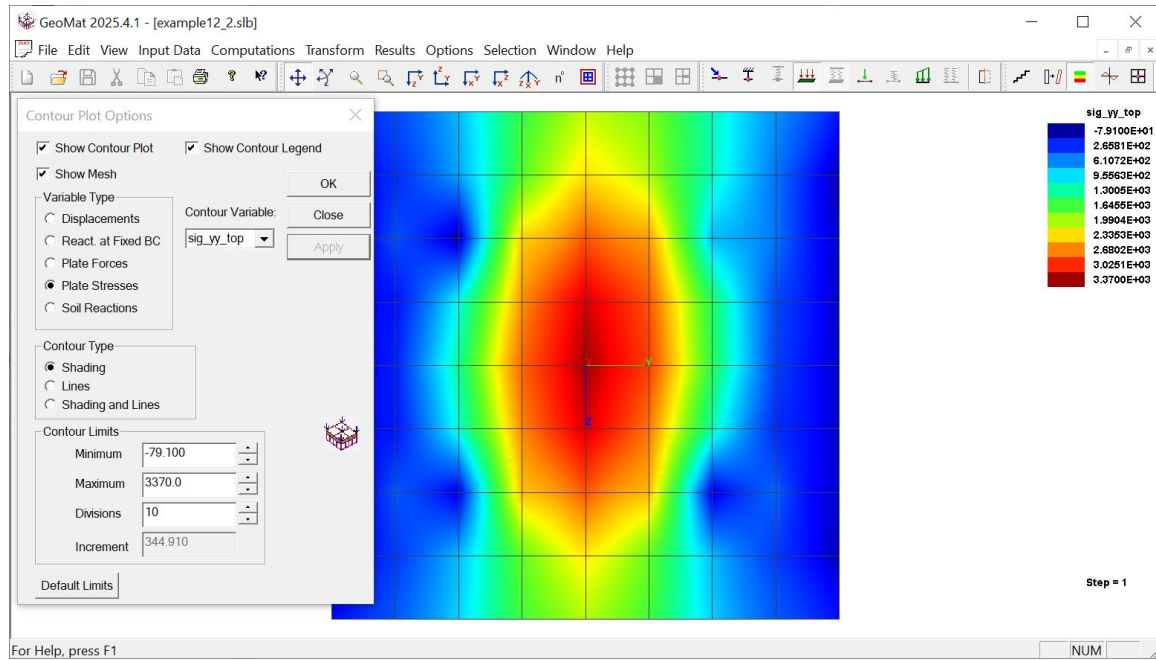


Figure 5.13 Shading contour option for stress distribution in Contour Plot

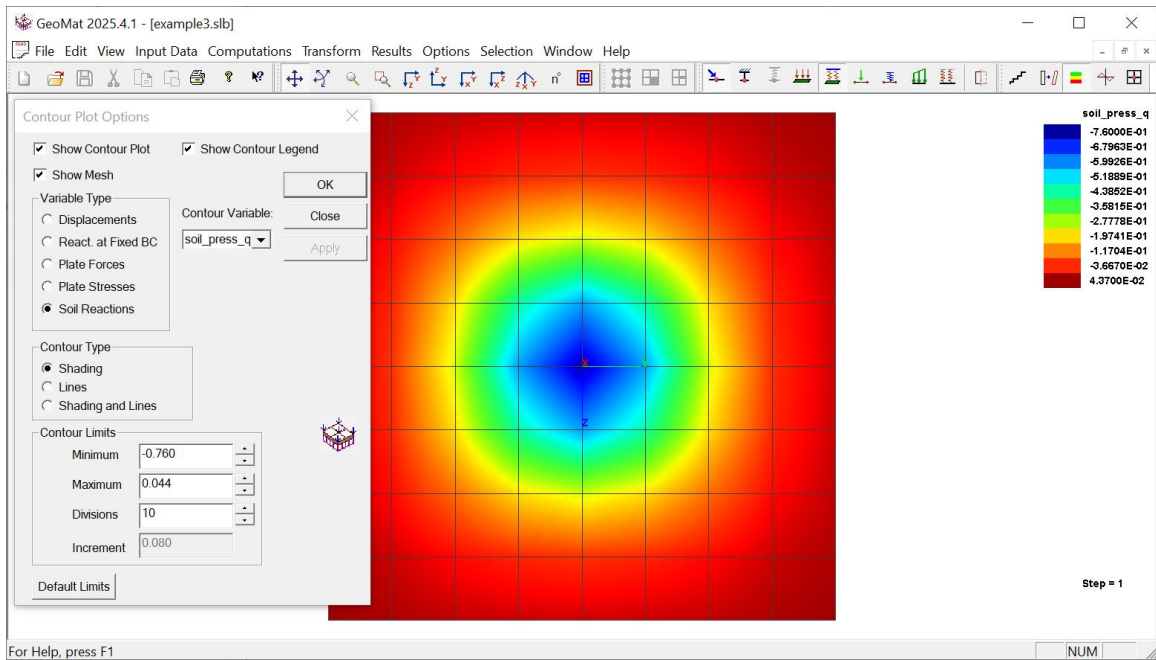


Figure 5.14 Shading contour option for distributed soil reactions

“Soil Reactions” option has one output “soil_react_q”. This is the distributed soil pressure calculated from nodal reaction forces. The direction of the soil pressure is parallel with the x axis.

5.3.4 Results – X-Y Plot

X-Y curve plots help users to understand the change of one physical variable along a path. The X-Y curve in *GeoMat* can be plotted in two difference ways: plot along a section cut line, or plot along a node path. Deformations, plate forces, plate stresses and distributed soil reactions are continuous variables on the plate. They can be interpolated at any point inside a finite element. The option “On Cutline” is appropriate for these variables. The second option is used for discrete variables, such as nodal reaction forces from nodal stiffness, forces of nonlinear springs, or sum of nodal reactions and spring forces. The option “On Node Path” should be selected for these variables. Figure 5.15 shows the X-Y curve for deflection w along a cutline. Figure 5.16 shows the X-Y curve for forces of nonlinear springs. Since two nodes along the node path have been assigned nonlinear springs, the curve shows two non-zero peaks at the spring location. Users can also change line color or line width, or turn on/turn off line makers.

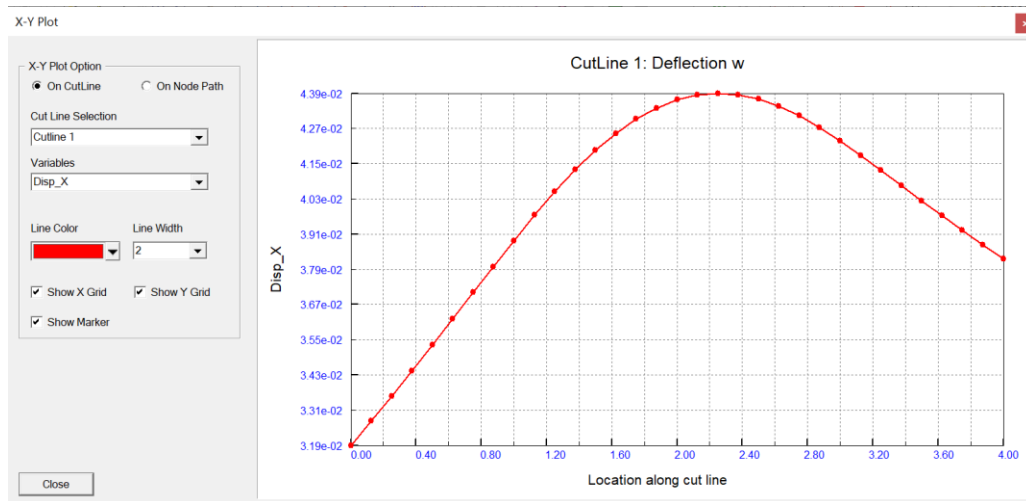


Figure 5.15 X-Y plot for plate deflection along the cut line

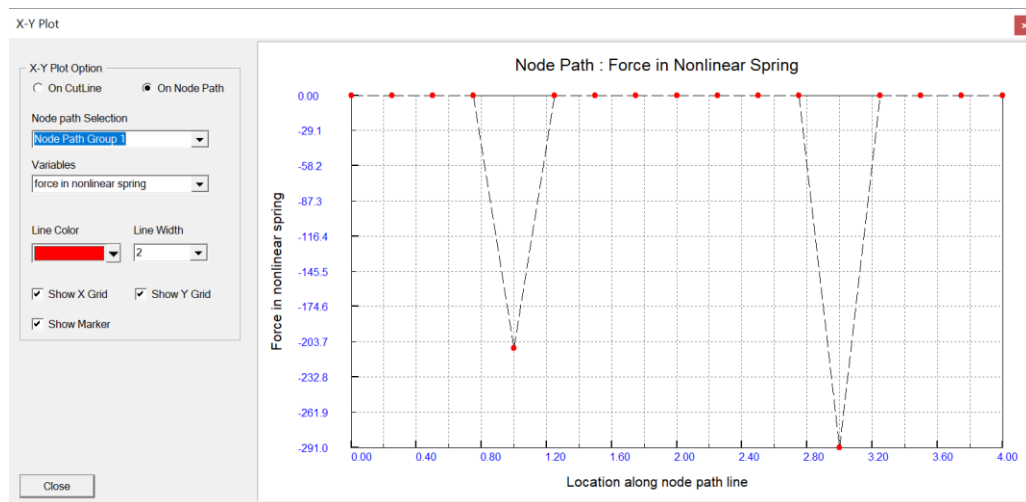


Figure 5.16 X-Y plot for forces of nonlinear springs along a node path

5.3.5 Results – Nodal Result

This command lets users investigate results at any node quickly. After users Input manually or turn the spin button to a specific node number, available nodal results will be shown according to different categories. Figure 5.17 shows the soil reaction of node 21 in example 12_1. “*nodal_Rx*” is the discrete nodal reaction. In *GeoMat*, the soil reaction pressure is assumed to be uniform in a finite element. However, soil pressures in different elements may be different. “*soil react.q*” is the averaged soil pressure if this node is shared by more than one finite elements. The force of the nonlinear spring “*F NL spring*” is also available since a nonlinear spring is assigned to node 21 in this model.

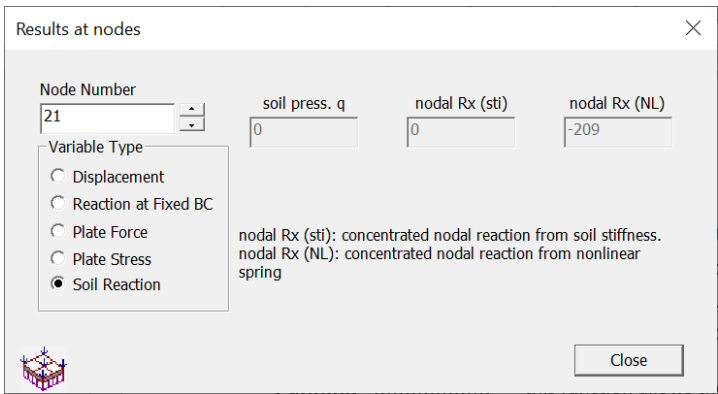


Figure 5.17 Nodal output

5.4 Speed Buttons in Graphics Mode

The user is provided with a set of speed buttons available in the View toolbar, which is associated with the commands in the **Transform** menu. Submenu choices, in order of their appearance in the speed button toolbar (Figure 5.18), are briefly described below.

- Panning** this function allows the slab model to be moved in the window.
- Rotate** this function allows the slab model to be rotated around the origin.
- Zoom In/Out** this function zooms in and out on the slab model.
- Restart** this function resets the view to the original YZ plane view.
- YZ View (Top)** this orients the slab in the YZ plane, looking at the top of the slab. This is demonstrated in Figure 5.19.
- YZ View (Bottom)** this orients the slab in the YZ plane, looking at the bottom of the slab. This is demonstrated in Figure 5.20.

- XY View (Front)** this orients the slab in the XY plane, looking at the front of the slab. This is demonstrated in Figure 5.21.
- XZ View (Side)** this orients the slab in the XZ plan, looking at the side of the slab. This is demonstrated in Figure 5.22.
- XYZ View (3D View)** . this orients the slab in an orthographic 3D view. This is demonstrated in Figure 5.23.
- Rotation Angles** this option allows the user to specify the desired rotation angle. The input window is shown in Figure 5.24.

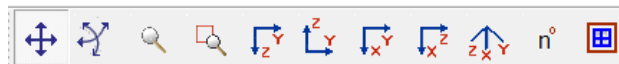


Figure 5.18 Speed Buttons for graphics manipulation

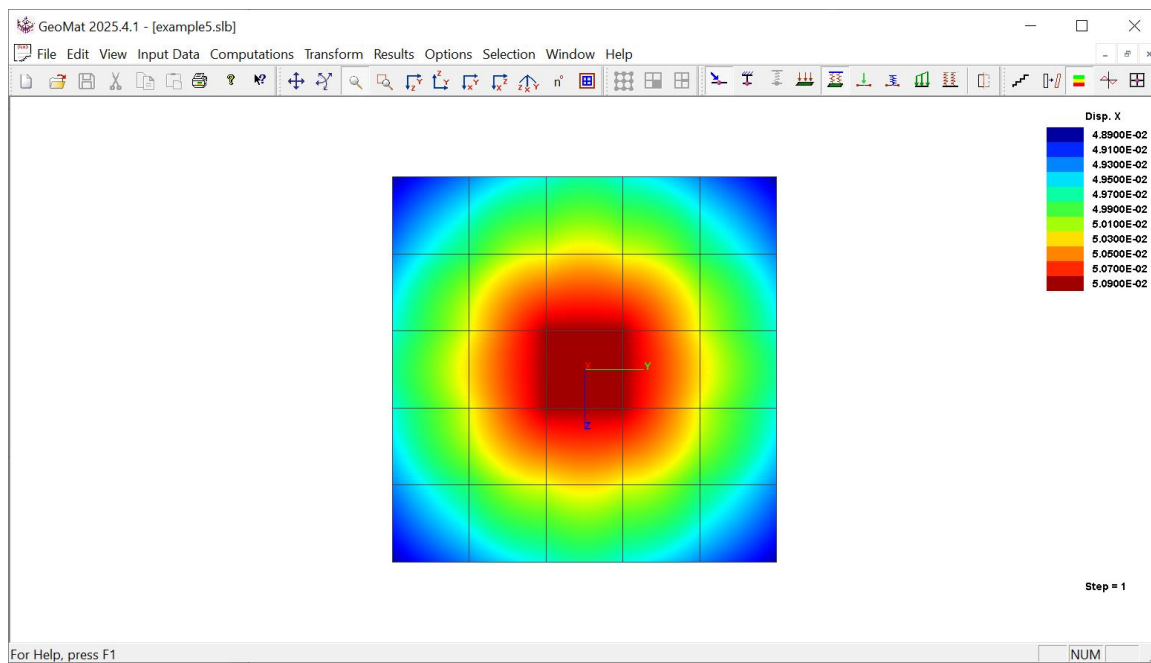


Figure 5.19 Slab model displayed using the YZ View (Top) speed button

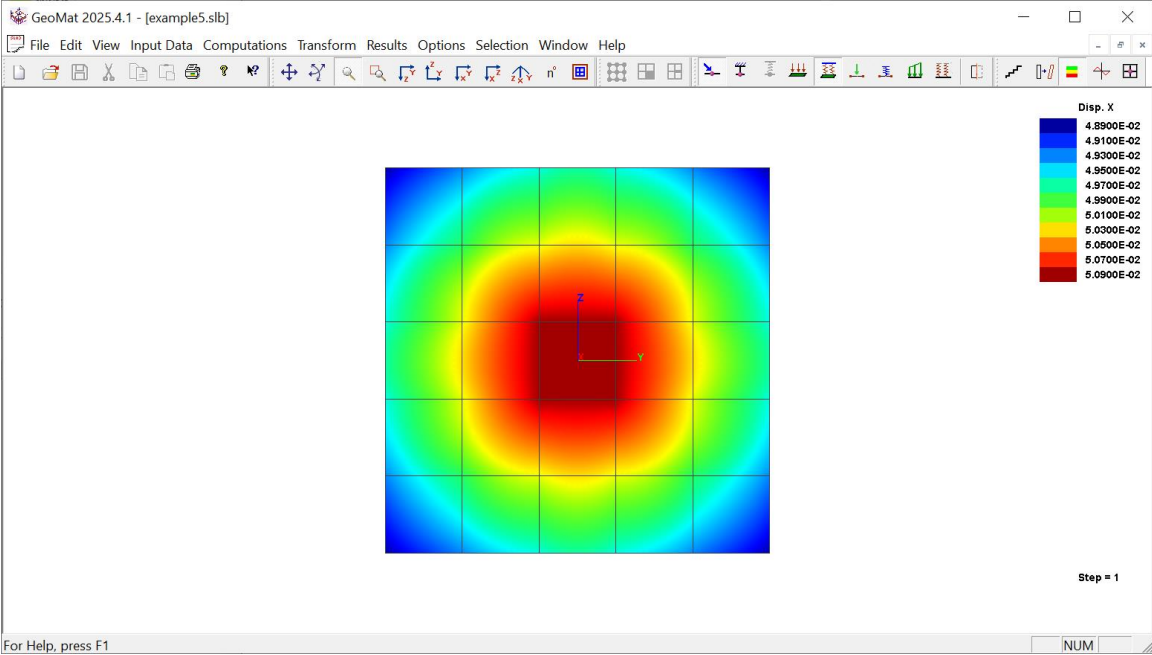


Figure 5.20 Slab model displayed using the ZY View (Bottom) speed button

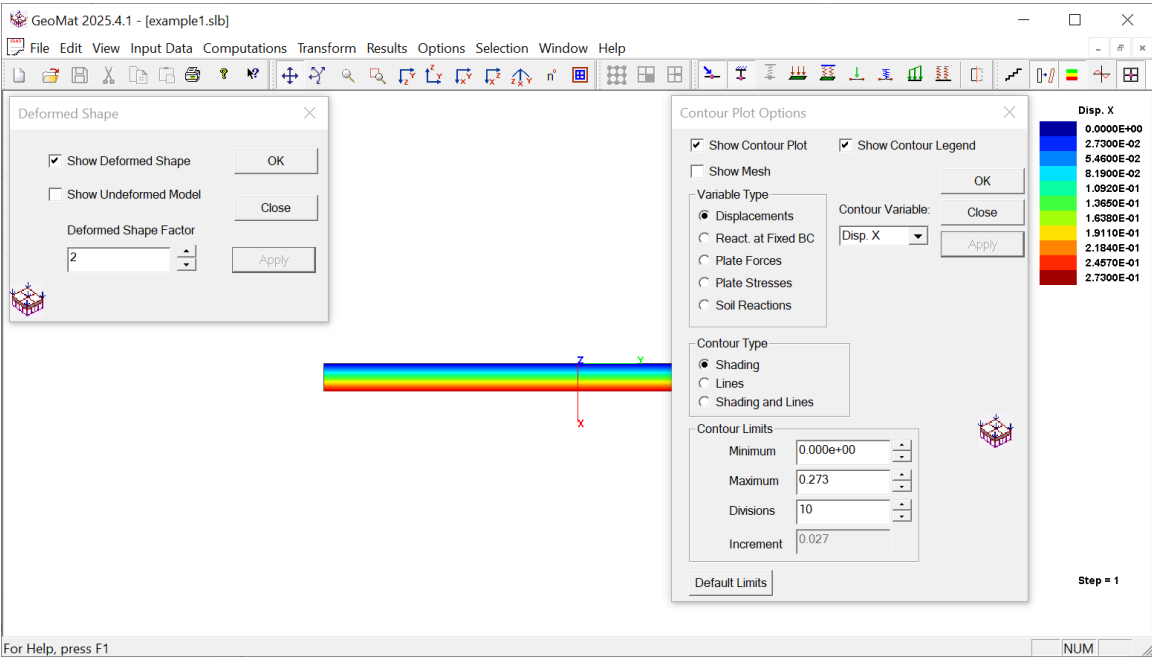


Figure 5.21 Slab model displayed using the XY View (Front) speed button

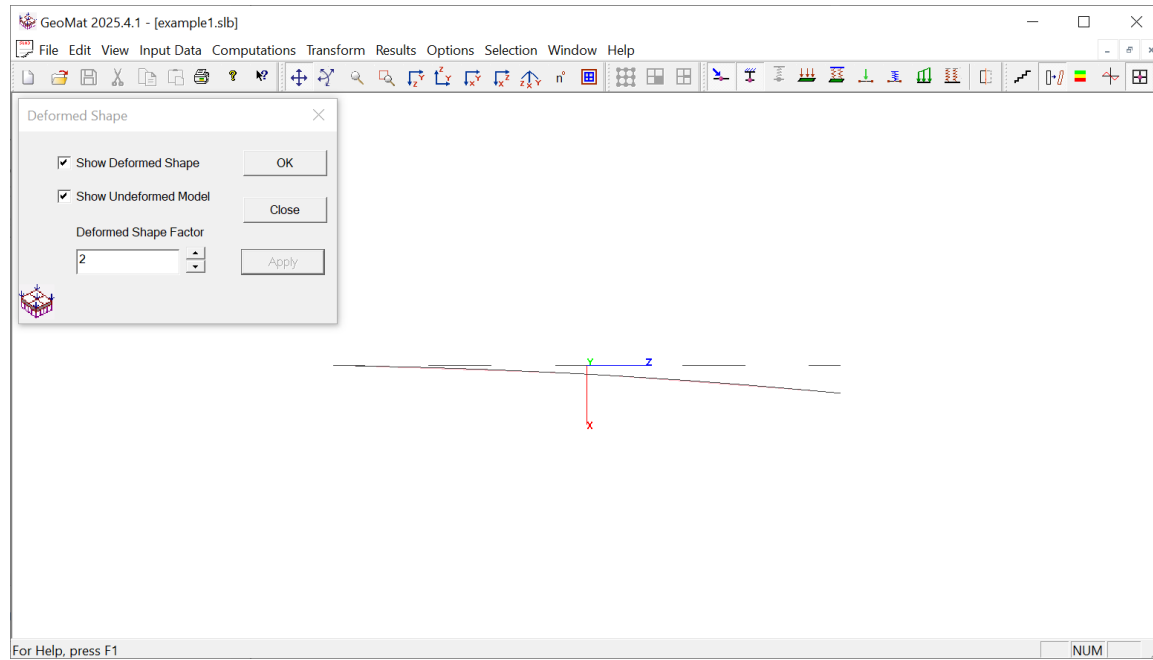


Figure 5.22 Slab model displayed using the XZ View (Side) speed button

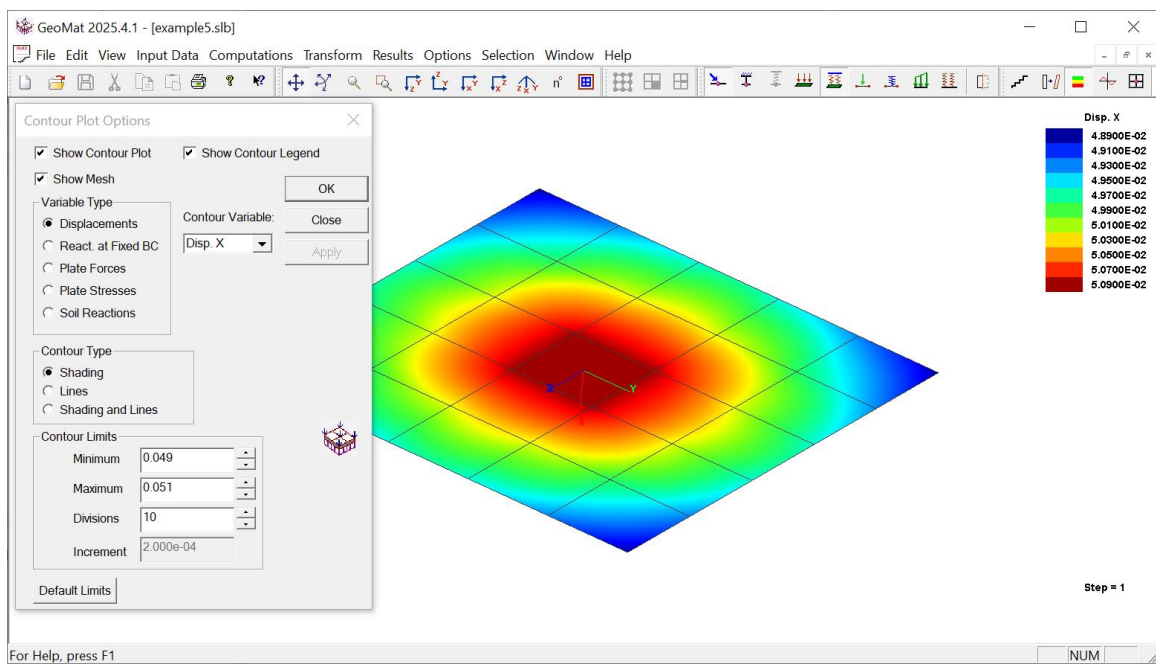


Figure 5.23 Slab model displayed using the XYZ View (3D View) speed button

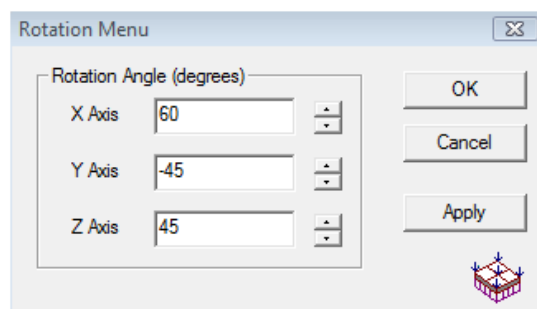


Figure 5.24 Sample screen of the Rotation Angles command option

6 Examples

6.1 Introduction

The chapter presents several examples studied by using *GeoMat*. In order to assure accuracy from the computer results, some examples have been compared with the results from close-form solutions or calculated by other analytical tools presented in the technical literature. Although examples presented in this chapter validate the output generated by this software, no warranty, expressed or implied, is offered as to the accuracy of results from the program. The program should not be used for design unless caution is exercised in interpreting the results and independent calculations are helpful to verify the general correctness of the results.

There are three types of output data provided by the computer. The first type is the output file giving the formatted text that consists of an echo-print of the input data, nodal coordinates, boundary constraints, load distributions, deflection at each nodal point, and, most of all, the bending moment and stress distribution at the integration points of each element.

The second type of output presents the data for a graphics file that allows the code to produce plots. All of the data are saved with ASCII format and mainly for internal use to present the graphics and contour plots based on the user's selection. The third type of output presents the status of execution with the user's data. It contains information about the iteration number, convergence condition, and the information about unsuccessful computation.

Example problems provide the user information on input and output of various cases, and present a quick tutorial for real-world applications. The user is encouraged to study these examples and, with modifications, may even use them to solve similar problems. However, by no means can these limited examples explore the full functions and features provided by *GeoMat*.

6.2 Example 1: Cantilevered square plate simulated by single quadratic element

6.2.1 Problem Description

A square cantilever plate is modeled by a single 8-node isotropic plate element. Rotational restraints in the z direction are applied to nodes 1, 3, 4, and 5. Along the restrained edge, nodes 6, 7, and 8, translations in the x direction as well as rotational restraints in the y and z directions are applied. The length in the y and z direction is 10 in. and the thickness of the plate is 0.2 in. Young's modulus is taken as 10,000 kip/in² (ksi) and Poisson's ratio as 0.3. To model a uniform edge load, concentrated loads of P, 4P, and P are applied in the x direction at nodes 1, 2, and 3, where P = 0.01kip.

6.2.2 Modeling Instruction

To model this slab in *GeoMat*, first go to **File -> New from template**. In the Template dialog box, choose “Rectangular – Equally Spaced, Quadratic 8 Noded” for the Shape and Element Types. Enter the “Length” and “Width” as 10, and the “Number of Divisions” as 1. Click **OK** to draw the slab on the screen (Figure 6.1).

Go to **Input Data -> Title** and enter a name for the current project. The name used here is *Example 1*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default “Element Number/Set” value as **ALL**, and modify the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

Before setting the nodal constraints and loads, it is recommended that the node numbers and marks are turned on. To do so, go to menu **Options -> Mark Nodes** and **Options -> Node Numbers**. Now that the nodes are marked, return to the **Input Data** menu and select **Nodal Constraints**. Use the **Add Row** button to create a new line. In the Node Number column, enter the first node, 1. Check the box in the “Rotation Z” column to restrain the rotations in the z direction. Repeat for nodes 3, 4, and 5. For nodes 6, 7, and 8, check the “Displac. X”, “Rotation Y”, and “Rotation Z” columns. Click **Apply** and then **OK**. Next, proceed to menu **Input Data Nodal Loads**. To set the load of 0.01kip to node 1, click **Add Row**, check the box next to the “Force X” column to activate the input box, and enter 0.01 in the Input box. Repeat this for nodes 2 and 3. Click **Apply** and then **OK**. Figure 6.2 shows the slab with applied load and the nodal constraints.

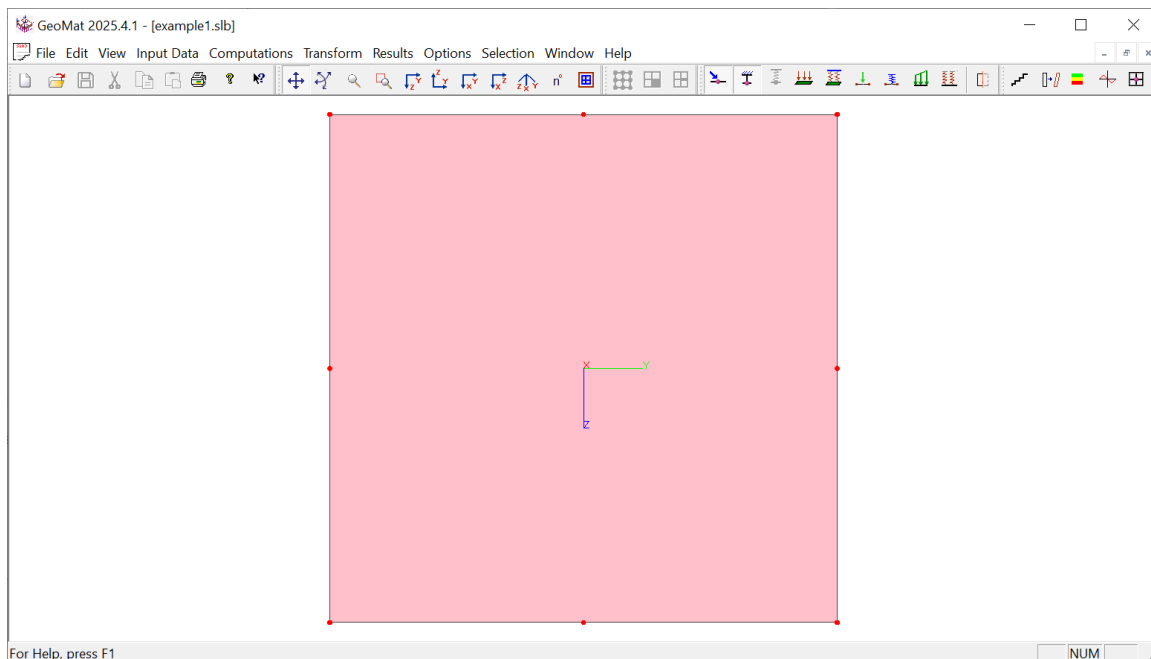


Figure 6.1 Example 1 slab, shown with marked and labeled nodal points

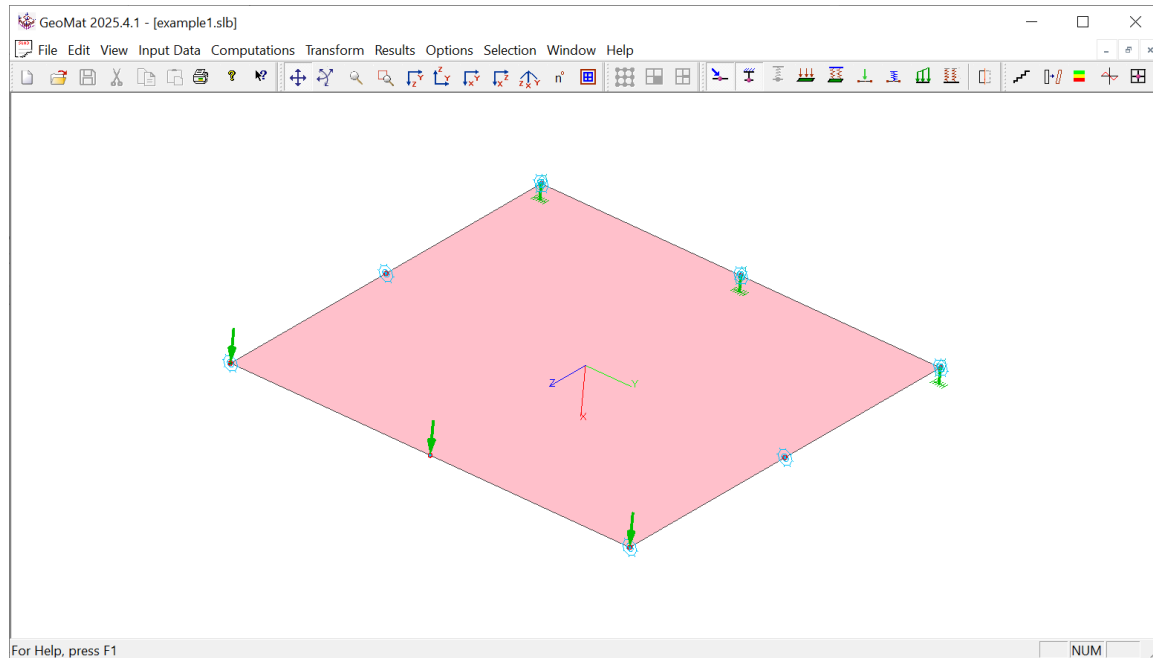


Figure 6.2 Example 1 slab shown with nodal constraints and nodal loads

Before executing the file, go to **File -> Save As...** and save the file as example1.slb. Once this is completed, go to menu **Computations -> Run Analysis** to launch the analysis. When the analysis is completed, go to menu **Computations -> View Output File**.

6.2.3 Output Results Review

When the analysis is completed, go to menu **Computations -> View Output File**. Figure 6.3 shows part of the result file. This output compares well to the results given in the text *Finite Elements for Structural Analysis* for this problem.

Displacements

Node	Disp X	Disp Y	Disp Z	Rot. X	Rot. Y	Rot. Z
1	0.273094E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.409500E-01	0.000000E+00
2	0.273094E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.409500E-01	0.110496E-13
3	0.273094E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.409500E-01	0.000000E+00
4	0.853593E-01	0.000000E+00	0.000000E+00	0.000000E+00	0.307125E-01	0.000000E+00
5	0.853593E-01	0.000000E+00	0.000000E+00	0.000000E+00	0.307125E-01	0.000000E+00
6	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
7	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
8	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00

Reactions

Node	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-0.176148E-12
3	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-0.293655E-12
4	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.600000E-01
5	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-0.600000E-01
6	-0.100000E-01	0.000000E+00	0.000000E+00	0.000000E+00	-0.100000E+00	0.300000E-01
7	-0.400000E-01	0.000000E+00	0.000000E+00	0.000000E+00	-0.400000E+00	0.642866E-12
8	-0.100000E-01	0.000000E+00	0.000000E+00	0.000000E+00	-0.100000E+00	-0.300000E-01

Figure 6.3 Review output file example1.sou for displacement results

Figure 6.4 shows the deformed shape of the slab. This graphic is produced using the **Results -> Deformed Mesh** option. This output compares well to the results given in the text *Finite Elements for Structural Analysis* for this problem.

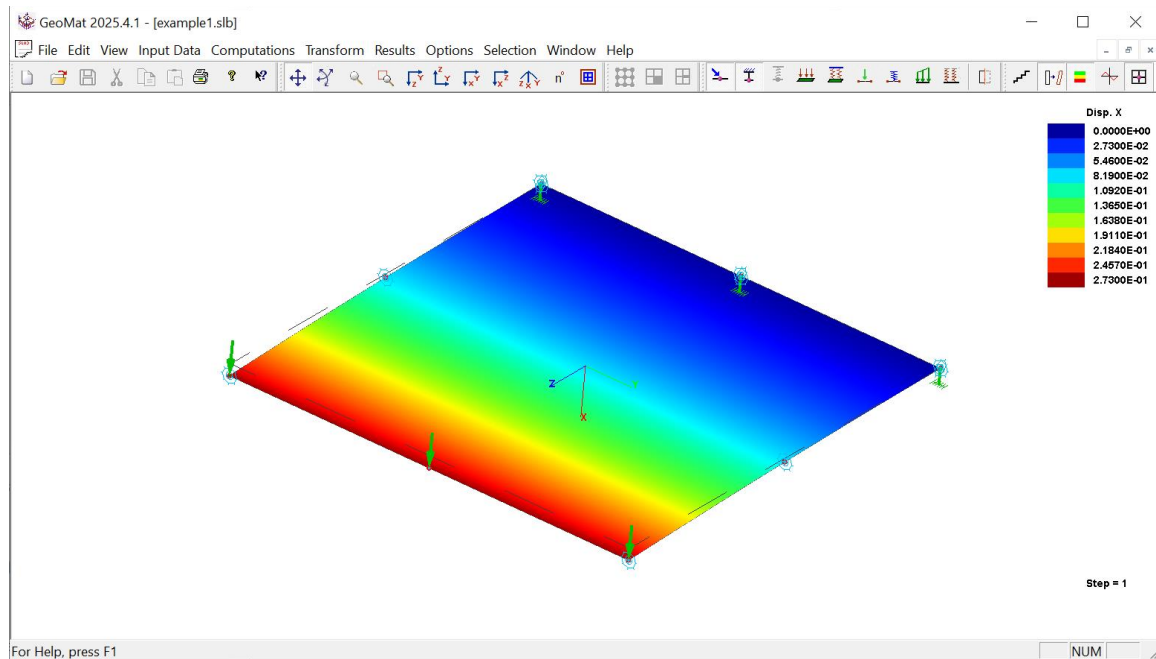


Figure 6.4 Example 1 slab shown with nodal constraints and nodal loads

6.3 Example 2: Fixed square plate subject to concentrated force at center

6.3.1 Problem Description

This example models a fixed, square plate using eight-node isotropic plate elements with a concentrated load P applied at the center. Taking advantage of symmetry, only a quarter of the plate will be modeled in this example.

Due to symmetry, restraints against rotations in the z direction are taken along an edge parallel to the z axis, and restraints against rotations in the y direction are taken along an edge parallel to the y axis. The fixed boundary is restrained against all translations and rotations. The length of the quarter plate is 8 in. in both the y and z direction, with a thickness of 0.25 in., and is divided into a 4×4 network. Young's modulus is 30,000 ksi and Poisson's ratio is 0.3. The center load, P , is 1 kip; however, due to symmetry, the load applied to the edge node will be 0.25 kip. Figure 6.5 shows the configuration of the nodal constraints on the slab.

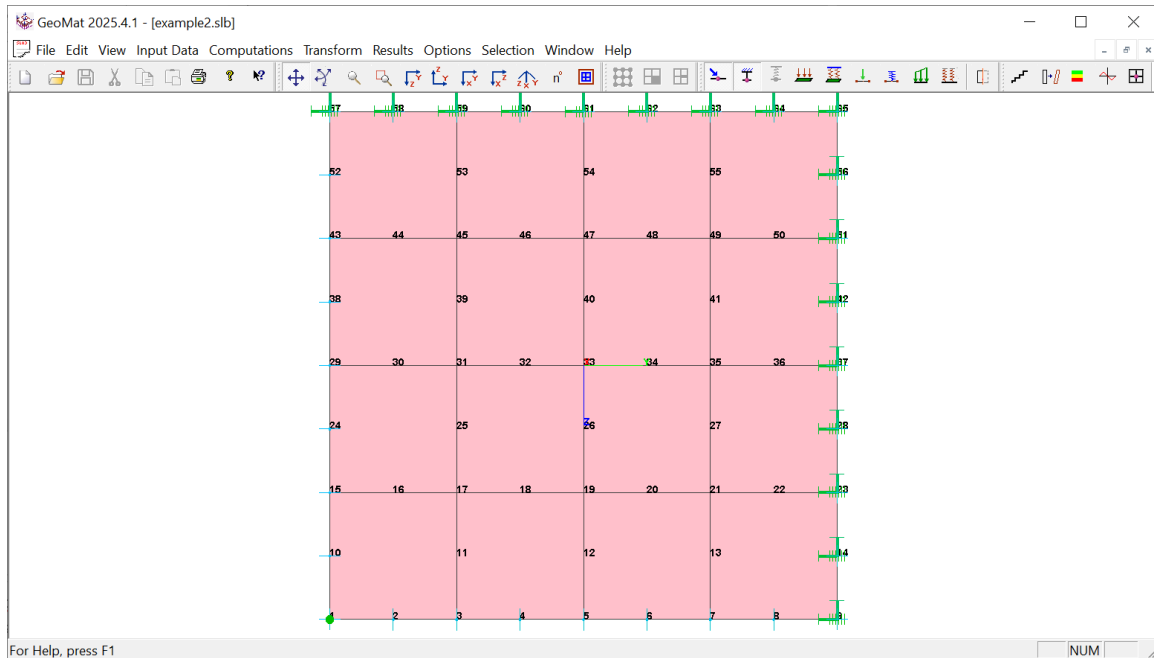


Figure 6.5 Quarter model of fixed edge slab with center load

6.3.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose “Rectangular – Equally Spaced, Quadratic eight-Noded” for the Shape and Element Types. Enter the “Length” and “Width” as 8, and the “Number of Divisions” as 4. Click **OK** to draw the slab on the screen.

Go to menu **Input Data -> Title** and enter a name for the current project; here it is *Example 2*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default Element Number/Set value as ALL, and modify the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

Before setting the nodal constraints and loads, it is recommended that the node numbers and marks are turned on. To do so, go to menu **Options -> Mark Nodes** and **Options -> Node Numbers**.

In addition, it is helpful to define node sets to aid in the assigning of restraints and loads. For this problem, four nodal sets were created using the **Input Data -> Node Sets** dialog. To create the first node set, click on the **Add Row** button. Enter the name BOTTOM into the edit box and click the **Define** button to launch the Node Set dialog, as shown in Figure 6.6. Using the **Add Row** button, select the nodes to be defined in this nodal set (nodes 1 through 8). For nodal set LEFT, nodes 1, 10, 15, 24, 29, 38, 43, and 52 are selected. For nodal set TOP, nodes 57 through 65 are selected. For nodal set RIGHT, nodes, 9, 14, 23, 28, 37, 42, 51, and 56 are selected. Click **OK** to save these settings

Now that the nodes are marked and node sets defined, return to the **Input Data** menu and select **Nodal Constraints**. Use the **Add Row** button to create a new line. In the Node Number column, enter the name of one of the nodal set. As seen in Figure 6.5, nodal set **BOTTOM** corresponds to a rotational restraint in the y direction. Check the box in the “Rotation Y” column to restrain the rotations in the y direction. Repeat for the remaining node sets, using Figure 6.5 as a guide. Click **Apply** and then **OK**. Next, proceed to menu **Input Data -> Nodal Loads**. To set the load of 0.25kip to node 1, the center node, click **Add Row**, check the box next to the “Force X” column to activate the input box, and enter 0.25 in the input box. Click **Apply** and then **OK**.

Before executing the file, go to menu **File -> Save As...** and save the file as example2.slb. Once this is completed, go to **Computations -> Run Analysis** to launch the analysis.

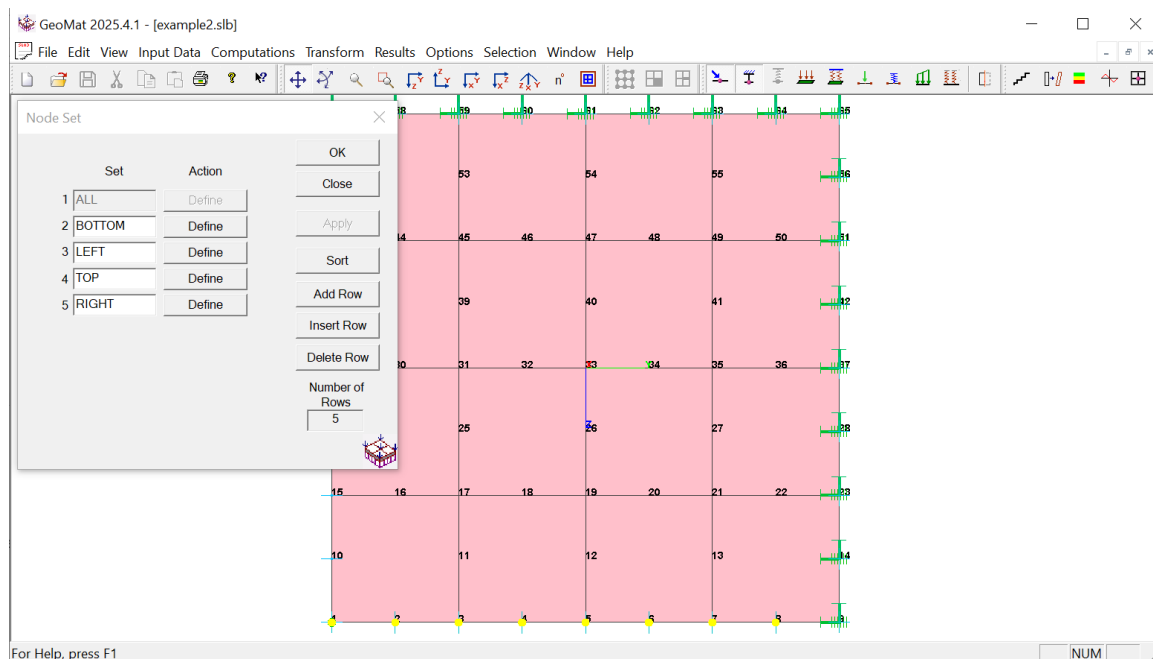


Figure 6.6 Defining edge nodes (BOTTOM edge nodes highlighted)

6.3.3 Output Results Review

When the analysis is completed, go to menu **Computations -> View Output File**. The deflection at node 1, the center deflection for the fixed square plate, is 0.0335. Figure 6.7 shows the contour plot for the slab deflection. This result corresponds exactly to the computer generated numerical result for a 4x4 network as reported in *Finite Elements for Structural Analysis*, which is within 0.4% of the classical solution.

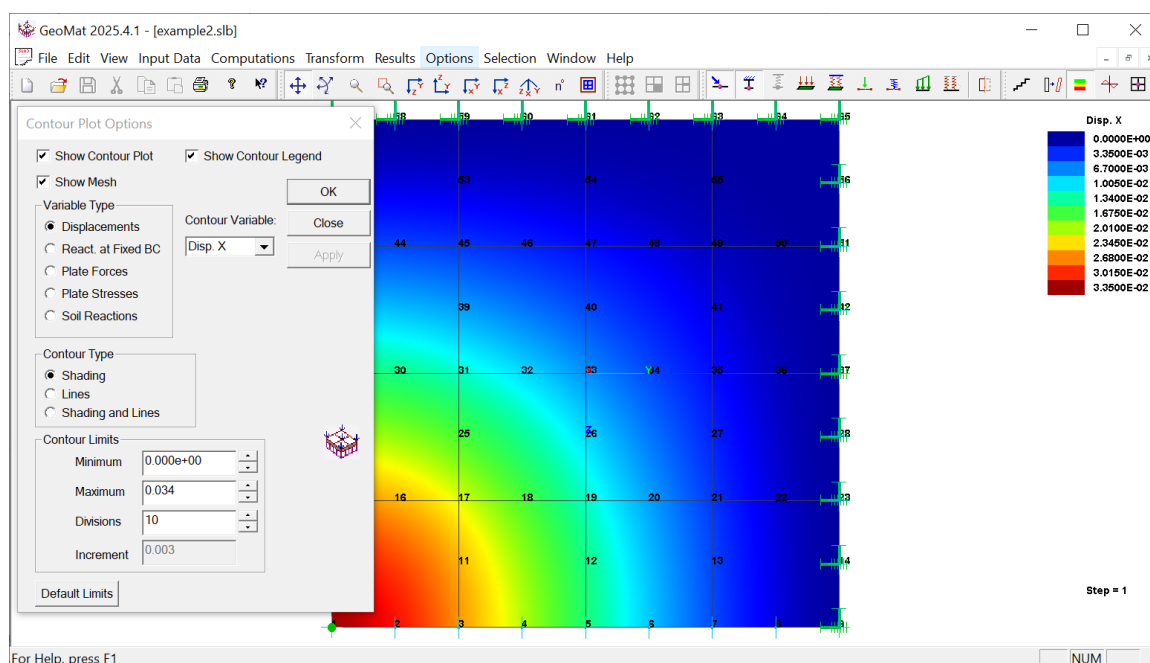


Figure 6.7 Contour plot of displacement in X for slab in Example 2

6.4 Example 3: Square slab on soil - distributed stiffness method

6.4.1 Problem Description

This example is included to model a square concrete slab resting on soil using 8-node isotropic plate elements. The slab is 24 ft x 24 ft and 10 in. thick. Young's modulus is taken as 3,000,000 psi and Poisson's modulus is 0.2. The subgrade modulus of the soil is assumed to be 200 psi per inch of deflection. A concentrated load of 10 kips is applied at the center of the slab.

6.4.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose "Rectangular – Equally Spaced, Quadratic 8 Noded" for the Shape and Element Types. Enter the "Length" and "Width" as 8, and the "Number of Divisions" as 4, as shown in Figure 6.8. Click **OK** to draw the slab on the screen.

Go to Input **Data** → **Title** and enter a name for the current project; this one is *Example 3*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default "Element Number/Set" value as **ALL**, and modify the remaining parameters to those given in the problem statement.

To set the soil stiffness, go to menu **Input Data** -> **Distributed Stiffness**. Add a row using the **Add Row** button. Under the "Element Number" heading, enter **ALL** in the edit box, so the soil stiffness is

distributed evenly beneath all of the slab elements. Next, check the box under the “D. Tr. K X” column and enter 200 in the associated edit box. Click **Apply** and then the **OK** button.

Next, go to menu **Input Data -> Nodal Loads**. Add a row and enter the number of the middle node, 33. Check off the “Force X” box, and enter 10000 for the load. Click **Apply** and then **OK**. Figure 6.9 shows the slab with the point load applied at the center and the distributed soil stiffness.

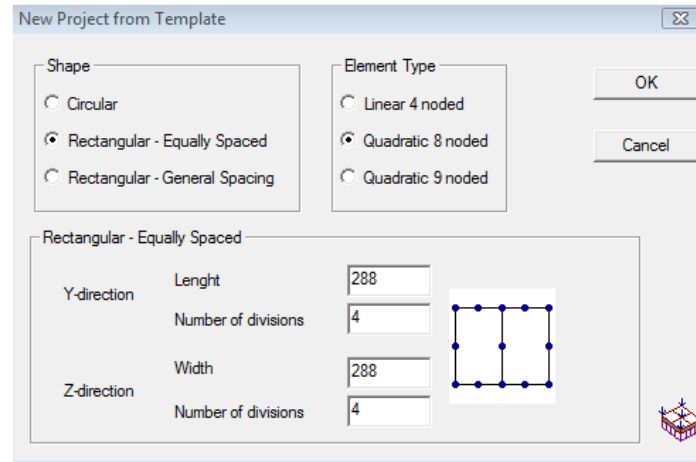


Figure 6.8 New Project from Template dialog screen for Example 3

Before executing the file, go to **File -> Save As...** and save the file as *example3.slb*. Once this is completed, go to **Computations -Run Analysis** to launch the analysis.

6.4.3 Output Results Review

When the analysis is completed, go to menu **Computations -> View Output File**. The center deflection at node 33 is 0.00587. This result is very close to the Westergaard solution for this problem, 0.0057 in., as reported in the text *Discontinuous Orthotropic Plates and Pavement Slabs*.

Now let us consider a different case using the same slab. This time, the 10 kip point load is applied along the edge of the slab. Save a copy of *example3.slb* as *example3b.slb*. Change the project title to *Example 3b*. Move the nodal load from node 33 to node 5, positioning the load at the middle of the lower edge of the slab. Save the file and run the analysis. The maximum deflection under the edge load is 0.0220. The Westergaard closed form solution for this case is 0.019 in., as reported in the text *Discontinuous Orthotropic Plates and Pavement Slabs*.

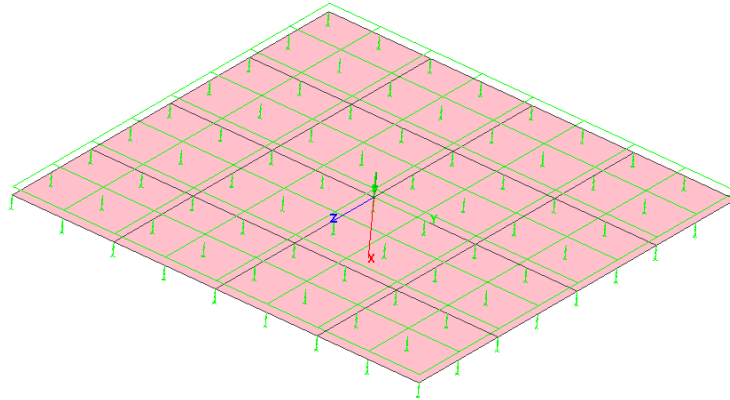


Figure 6.9 Slab of Example 3 shown with soil stiffness and center point load

6.5 Example 4: Square slab on soil - Mindlin method

6.5.1 Problem Description

This example uses the same slab and loading condition as Example 3 to estimate the deformation of the slab based on Young's modulus of the soil which is entered externally. Instead of using the given soil subgrade modulus as soil springs, Young's modulus of soil is used to derive the soil stiffness beneath each nodal point.

6.5.2 Modeling Instruction

To solve this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose "Rectangular – Equally Spaced, Linear 4 Noded" for the Shape and Element Types. Click **OK** to draw the slab on the screen.

Go to menu **Input Data -> Title** and enter a name for the current project; this one is **Example 3**. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default "Element Number/Set" value as **ALL**, and modify the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

The nodal edges must be restrained since there is no nodal stiffness applied for this case. Following the procedure described in Example 2, create a node set named **EDGE** and set the edge nodes to it. Next, go to **Input Data -> Nodal Constraints** and restrain the **EDGE** set in the y and z directions. Click **Apply** and then **OK**.

To input the center load, go to **Input Data -> Nodal Loads**. Add a row and enter the number of the middle node, 33. Check off the "Force X" box, and enter 10000 for the load. Click **Apply** and then **OK**. Figure 6.10 shows the slab with the point load applied at the center and the edge restraints. Next, go to **Input Data -> Soil Stiffness**. The Soil Stiffness with Mindlin Equation dialog box appears.

To input data, check the box next to Estimate soil stiffness. This activates the input boxes. For this example, a Young's modulus of 38000, a Poisson's ratio of 0.4, and an initial stiffness of 200 are assumed. Click **OK**, and then save the file as *example3.slb*.

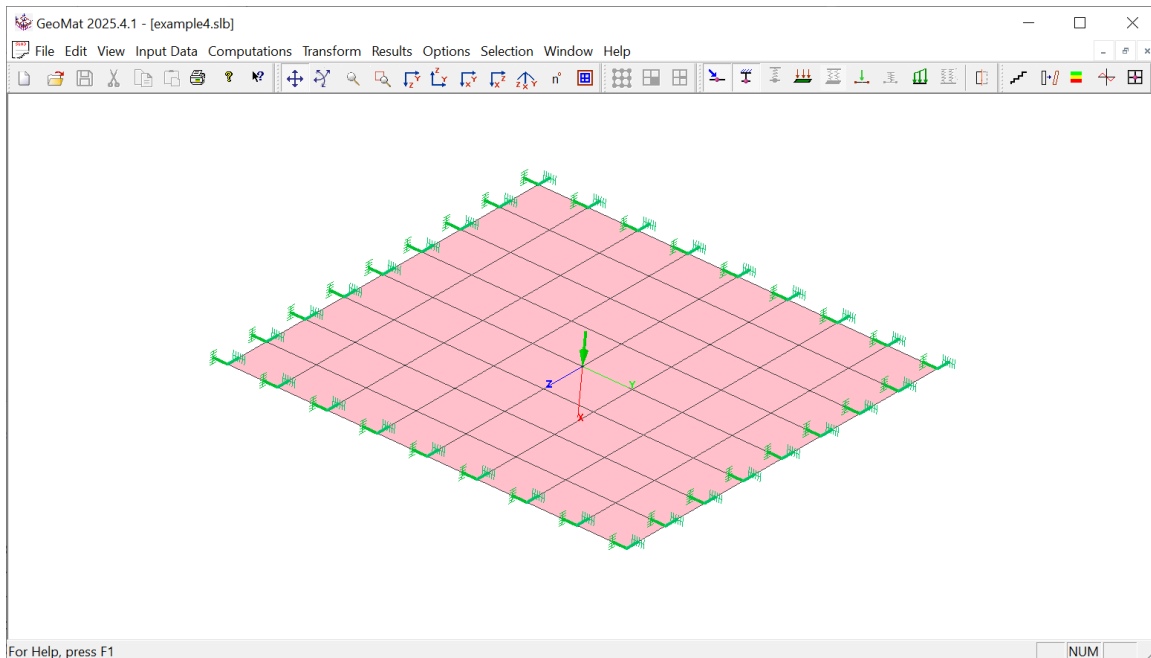


Figure 6.10 Slab of Example 4 shown with center load and edge restrains

6.5.3 Output Results Review

Execute the file and view the output data. To view the deformed shape, go to **Results** -> **Deformed Mesh** and check both options "Show Deformed Shape" and "Show Undeformed Mesh". The graphic shown in Figure 6.11 was produced using a deformation factor of 1000. The center deflection is reported to be 0.00456 inch. The Young's modulus and initial stiffness values will need to be varied until the output deflection approaches that of the known value. A Young's modulus of 17000 and an initial stiffness of 700 were used in this example.

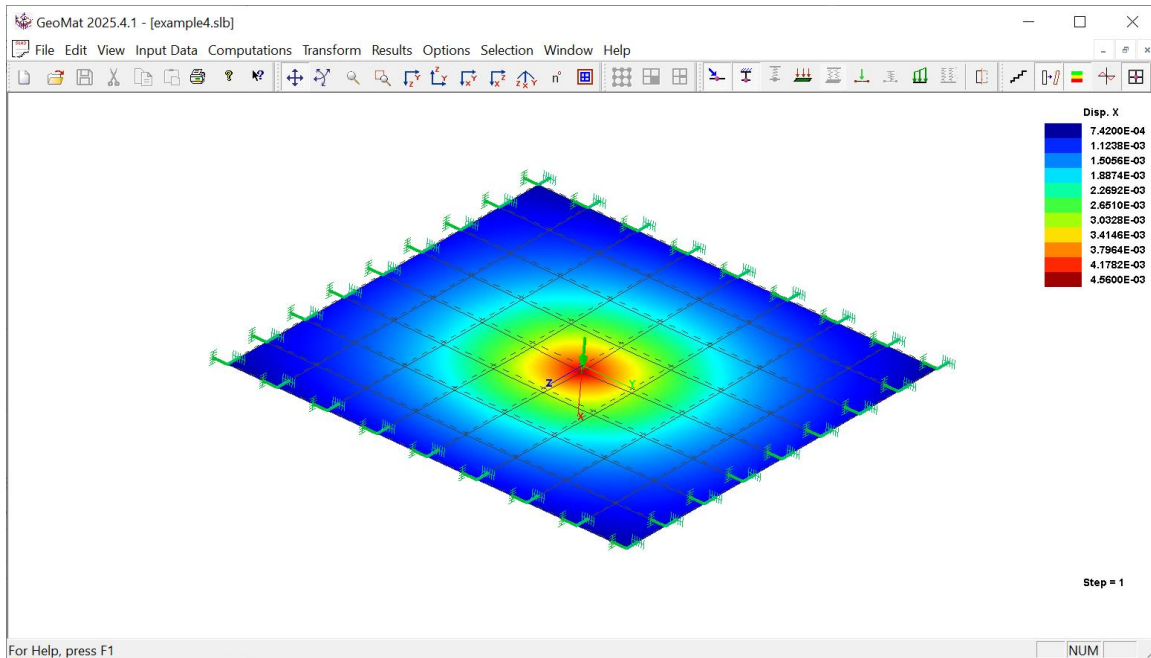


Figure 6.11 Deformed shape of slab of Example 4 shown with center load and edge restrains

6.6 Example 5: Slab on soil to support four columns (no rotation restraint)

6.6.1 Problem Description

This problem models a square concrete slab resting on soil using 8-node isotropic plate elements. The slab is 10 ft. square and 2 ft. thick. Young's modulus is taken as 468,000 kip/ft² (ksf) and Poisson's modulus is 0.15. The subgrade modulus of the soil is assumed to be 100 kcf. A concentrated load of 125 kips is applied to four nodes around the plate center.

6.6.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose "Rectangular – Equally Spaced, Quadratic 8 Noded" for the Shape and Element Types. Enter the "Length" and "Width" as 10, and the "Number of Divisions" as 5. Click **OK** to draw the slab on the screen.

Go to menu **Input Data -> Title** and enter a name for the current project; this one is *Example 5*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default "Element Number/Set" value as *ALL*, and reset the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

To set the soil stiffness, go to menu **Input Data -> Distributed Stiffness**. Add a row using the **Add Row** button. Under the "Element Number" heading, enter *ALL* in the edit box, so the soil stiffness is

distributed evenly beneath all of the slab elements. Next, check the box under the “D. Tr. K X” column and enter 100 in the associated edit box. Click **Apply** and then **OK** button.

Next, go to **Input Data -> Nodal Loads**. Because the load is applied in the center, and no node exists at that location, the load will be evenly divided among the four nodes of element 13 (39, 41, 56, and 58). Add four rows and enter a load of 125 for each node. Click **Apply** and then **OK**.

Before executing the file, go to **File -> Save As...** and save the file as *example5.slb*. Once this is completed, go to **Computations -> Run Analysis** to launch the analysis.

6.6.3 Output Results Review

When the analysis is completed, go to menu **Computations -> View Output File**. The deflection at node 39 is 0.1861 ft. Moments M_y at corner node of element 13, are 52.06 kip*ft/ft. These results are close to the solution reported in the text *Foundation Analysis and Design* for this slab. To view the how the slab deforms, go to **Results -> Deformed Mesh**, or to view the moment distribution, go to **Results -> Contour Plot** and select the desired moment.

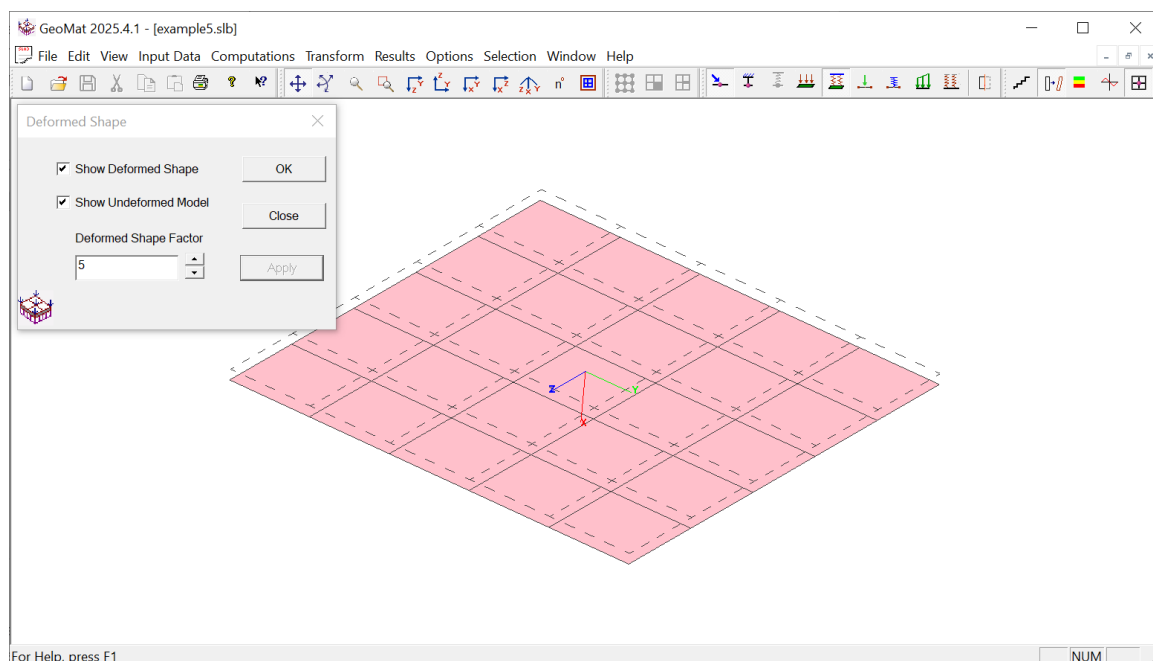


Figure 6.12 Deformed shape of slab of Example 5

Figure 6.12 shows the deformed slab with a deformation shape factor of 5 with the undeformed outline. Figure 6.13 shows the contour plot of the moment in the y direction. Figure 6.14 shows moment M_y of node 39 using nodal output option.

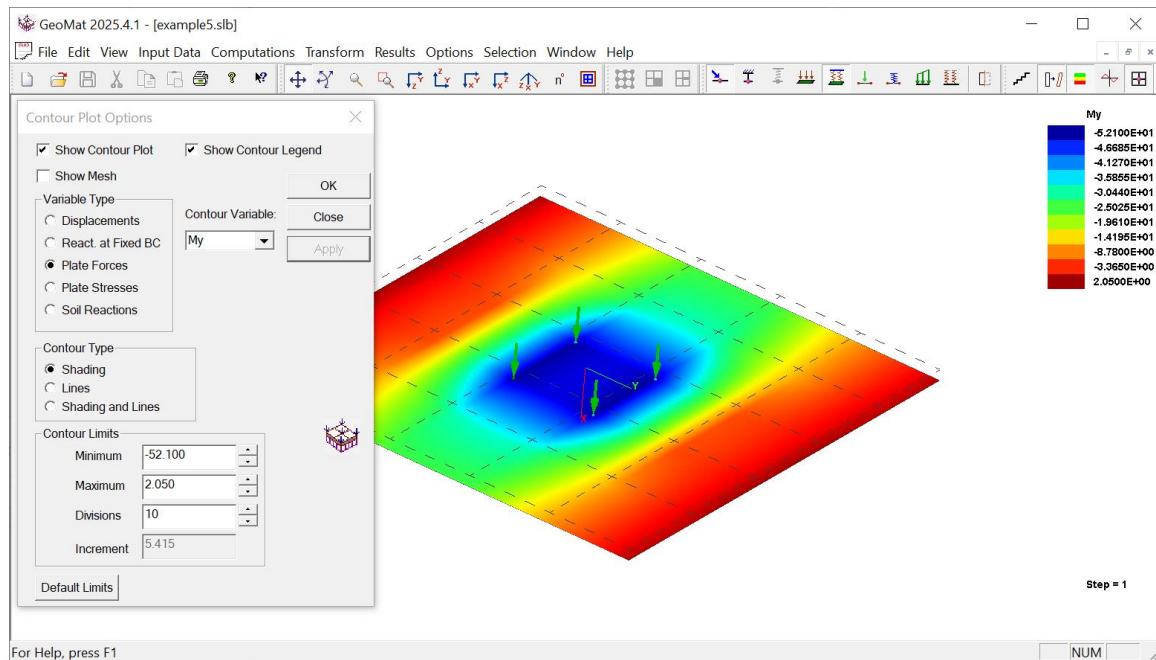
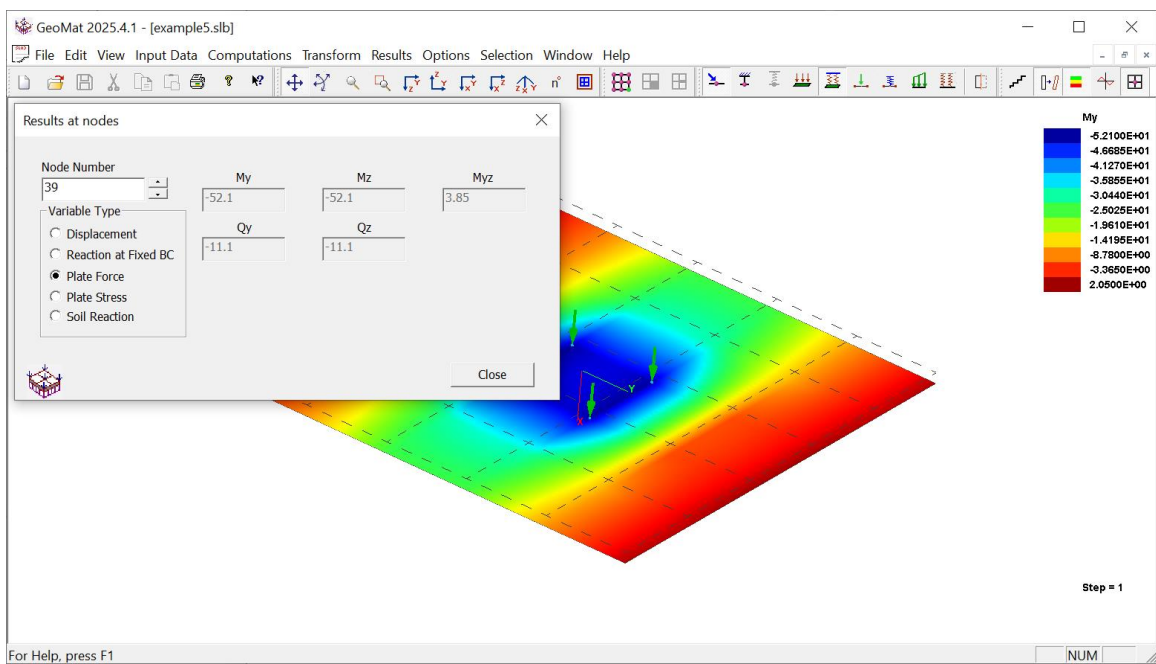


Figure 6.13 Contour plot of moment distribution in the y direction of Example 5

Figure 6.14 Nodal output for moment M_y at node 39 in Example 5

6.7 Example 6: Slab on soil to support four concentrated forces (restrained nodal rotation)

6.7.1 Problem Description

This example uses the slab of Example 5, but this time assumes that nodes 39, 41, 56, and 58 are held against rotation in both y and z directions, simulating the presence of a fixed concrete column. The deformation at node 39 drops to 0.0507750 ft., and the moment under the column becomes 44.19 kip*ft/ft. The deformed shape and the contour plot of the moment in the y direction are shown in Figure 6.15 and Figure 6.16, respectively.

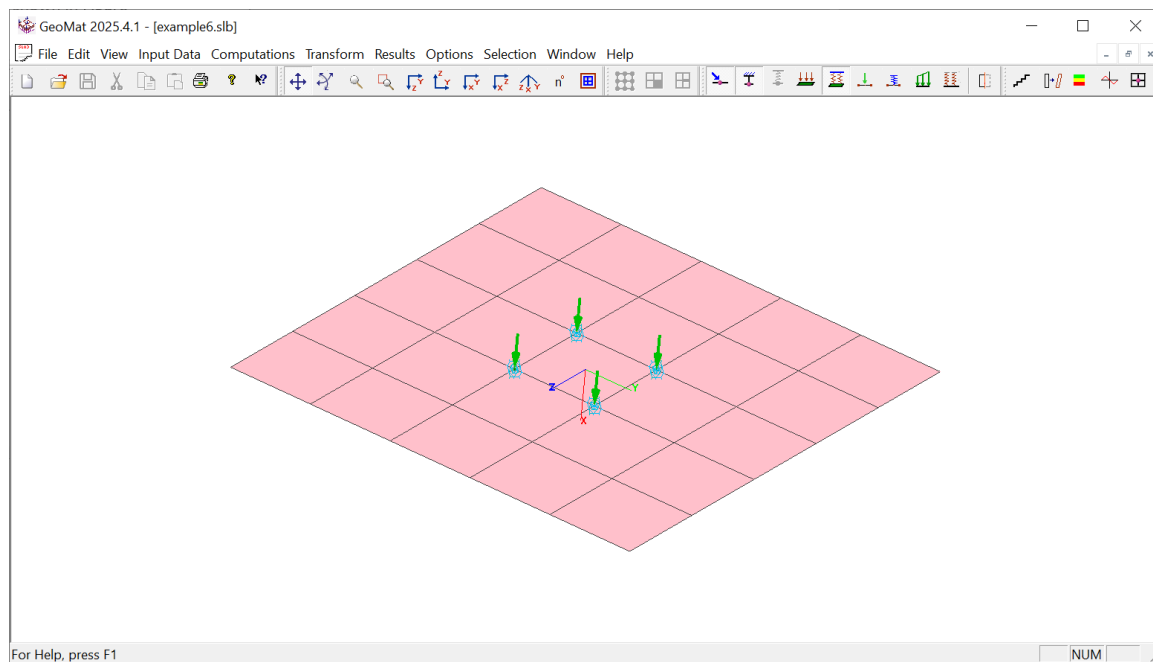


Figure 6.15 Apply node rotation constraints to simulate non-rotation at column foot in Example 6

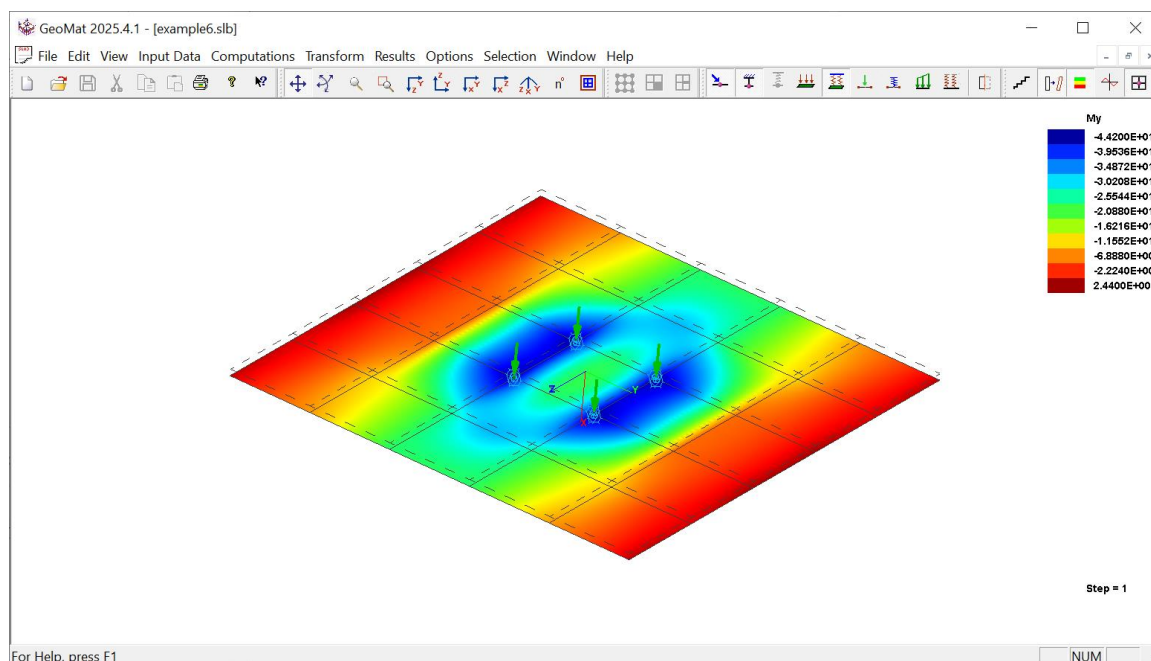


Figure 6.16 Contour plot of moment distribution in the y direction of Example 6

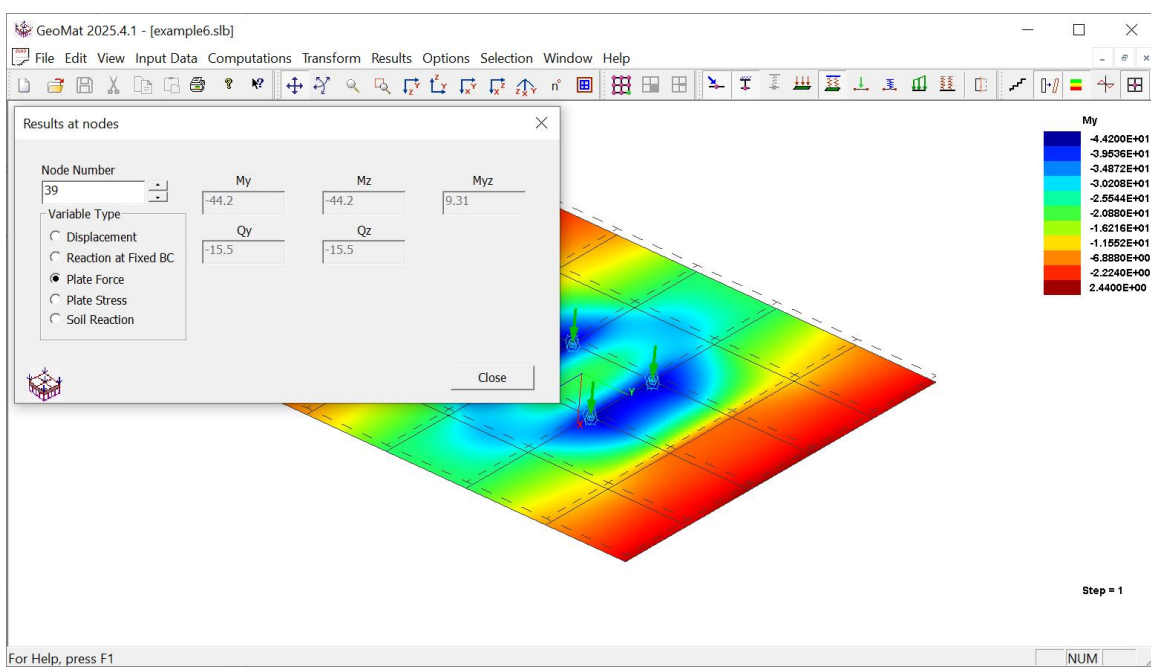


Figure 6.17 Moment My at node 39 in Example 6

6.8 Example 7: Rectangular slab on soil subject to column loads

6.8.1 Problem Description

This example considers a rectangular concrete raft supporting four columns. The slab is 16m by 12 m, and is 0.8m thick. The concrete has a Young's modulus of 30 GPa and a Poisson's ratio of 0.2. The subgrade modulus of the soil is 4 MN/m^3 . The columns provide a concentrated load of 5MN, 3MN, 6MN, and 8MN, respectively. The column locations are shown in Figure 6.18.

6.8.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose "Rectangular – Equally Spaced, Quadratic 8 Noded" for the Shape and Element Types. Enter the "Length" as 16 and "Width" as 12, and the "Number of Divisions" as 1. Click **OK** to draw the slab on the screen.

Go to **Input Data -> Title** and enter a name for the current project; this one is *Example 7*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default "Element Number/Set" value as **ALL**, and reset the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

To set the soil stiffness, go to **Input Data -> Distributed Stiffness**. Add a row using the **Add Row** button. Under the Element Number heading, enter **ALL** in the edit box, so the soil stiffness is distributed evenly beneath all of the slab elements. Next, check the box under the "D. Tr. K X" column and enter 4 in the associated edit box. Click **Apply** and then the **OK** button.

Next, go to **Input Data -> Nodal Loads**. The columns are located at nodes 109, 129, 409, and 429. At node 109, enter a load of 5MN in the x direction; at node 129, a load of 3 MN; at node 409 a load of 8MN; and at node 429 a load of 6MN.

Before executing the file, go to **File -> Save As...** and save the file as *example7.slb*. Once this is completed, go to **Computations -> Run Analysis** to launch the analysis.

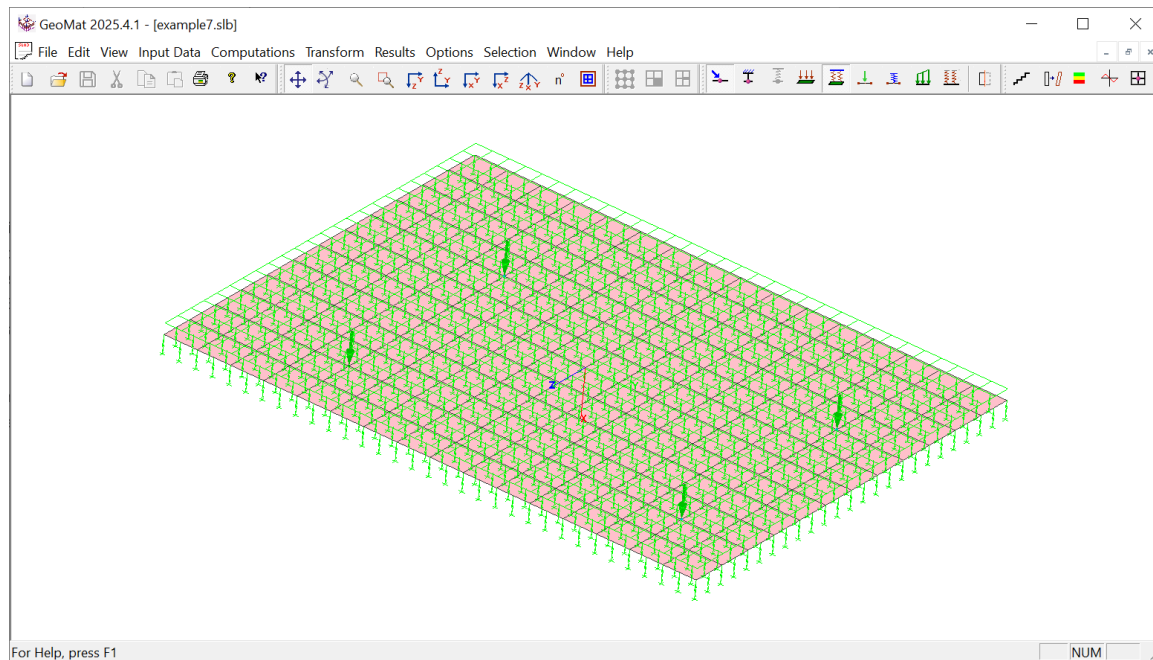


Figure 6.18 Location of columns on raft of Example 7

6.8.3 Output Results Review

When the analysis is completed, go to **Computations** → **View Output File**. To view the how the slab deforms, go to **Results** → **Deformed Mesh**. Figure 6.19 shows the deformed slab with a deformation shape factor of 10 together with the undeformed outline. To view the moment, go to **Results** → **Contour Plot** and select the desired moment axis. The maximum moment in the y direction is 1767.2 m-kN at the 8 MN column. The maximum moment in the x direction also occurs at the 8 MN column and is 1778.1 m-kN. Figure 6.20 shows the moment distribution of the raft along the y axis.

For viewing the section-cut plot, the user should click Section Cut item under **Input Data** menu and then select the cutting lines by default or by the user-specified. To view the deflection along the cutting line, go to **Results** → **Section Cut Plot** and select the desired cutting line and deflection as shown in Figure 6.21 for required display.

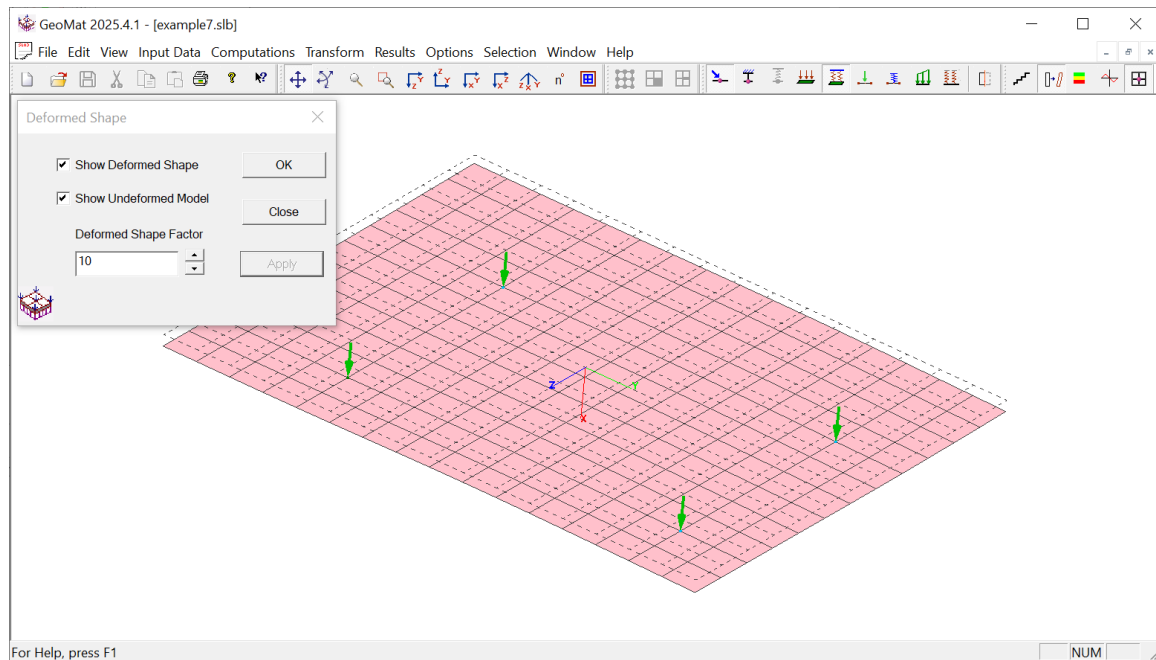


Figure 6.19 Deformed shape of slab with column loads of Example 7

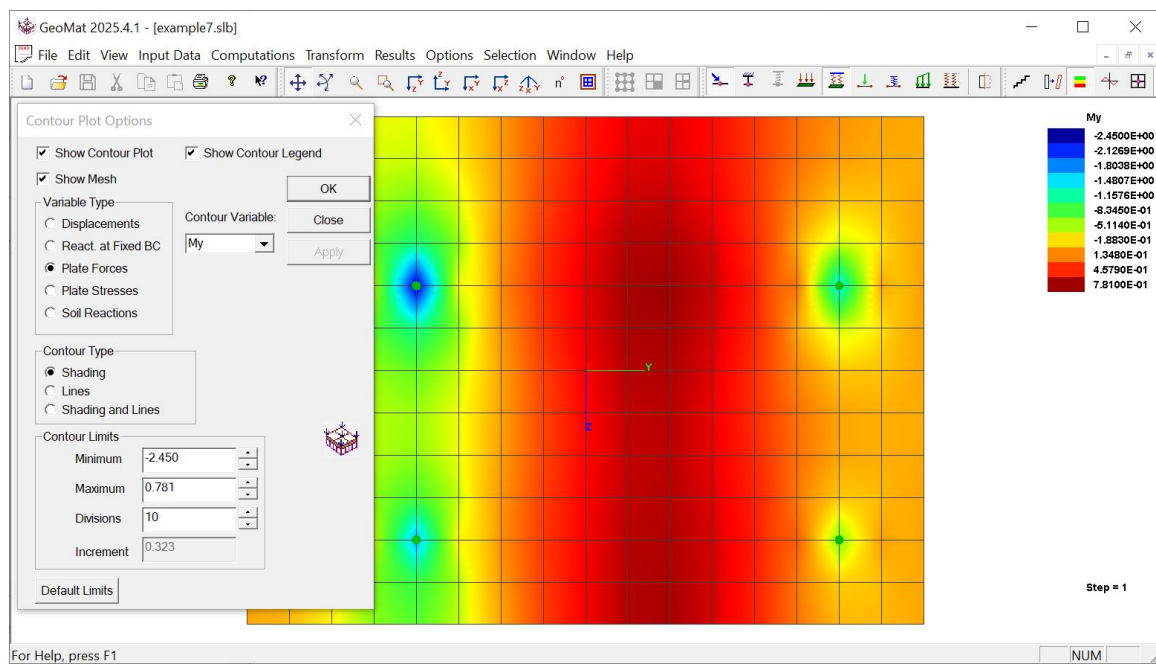


Figure 6.20 Moment distribution of the slab in the y direction of Example 7

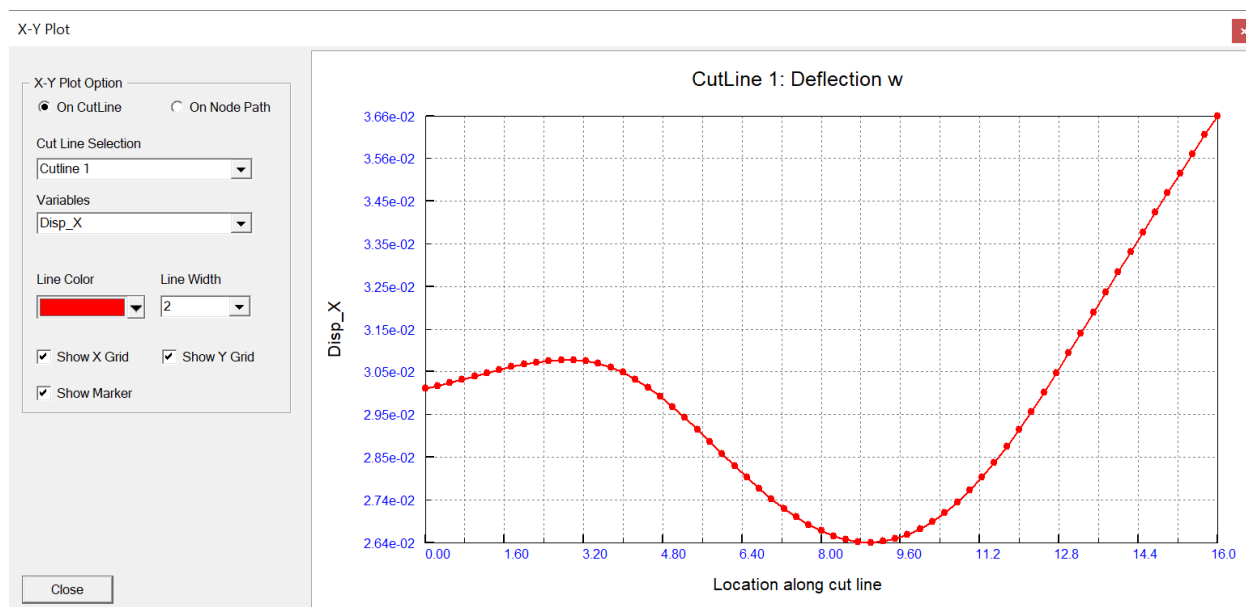


Figure 6.21 Deflection of the slab along the cutting line of Example 7

6.9 Example 8: Rectangular raft on soil to support columns and walls

6.9.1 Problem Description

This example considers a rectangular concrete raft supporting 10 columns and three wall sections. The slab is 16m by 8 m, and is 0.8m thick. The concrete has a Young's modulus of 30000 MPa and a Poisson's ratio of 0.2. The subgrade modulus of the soil is 30 MN/m^3 . The location of the walls and columns and the mesh size are shown in Figure 6.22.

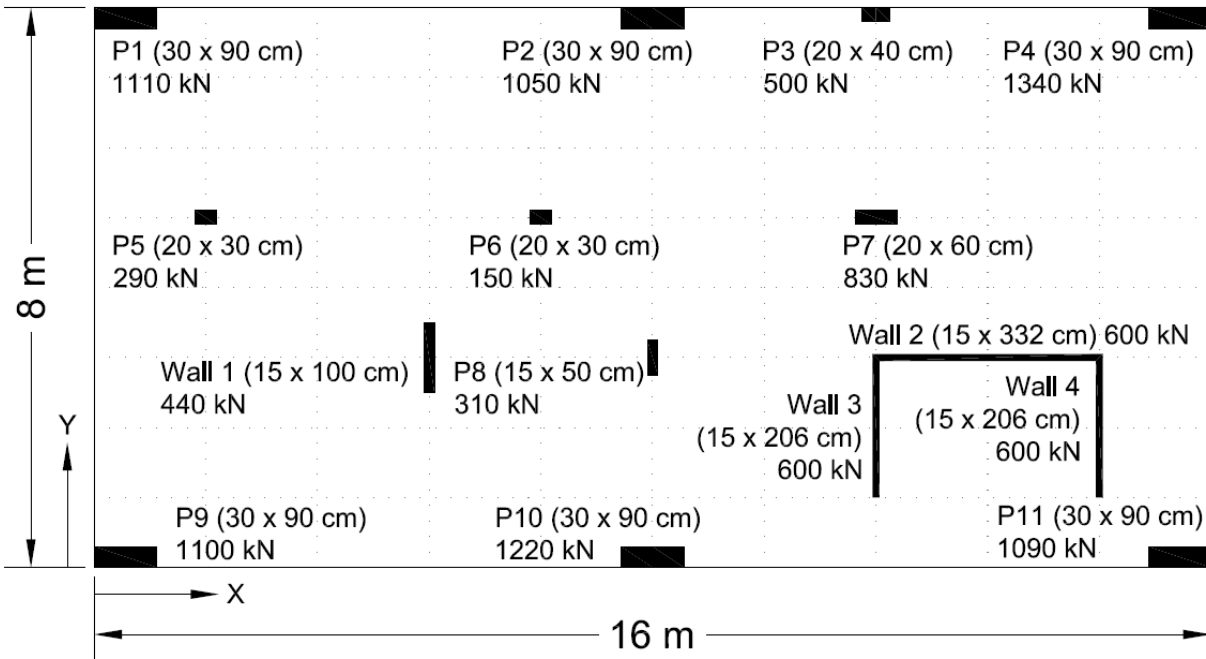


Figure 6.22 Location of walls and columns in Example 8

6.9.2 Modeling Instruction

The soil stiffness and column loads at nodal points are input into *GeoMat* as described in previous examples. The U-shape wall load is created as follows. Go to **Input Data** → **Edge Sets**. Create a new edge set named **WALL**, and select the appropriate element edges using the Select Edge function from the speed button or using the Selection menu. The edge set should look similar to that shown in Figure 6.23. To assign the load to the wall, go to **Input Data** → **Edge Loads**. Set the Edge column to **WALL** and place the 0.600 MN load in the Load Node 3 column.

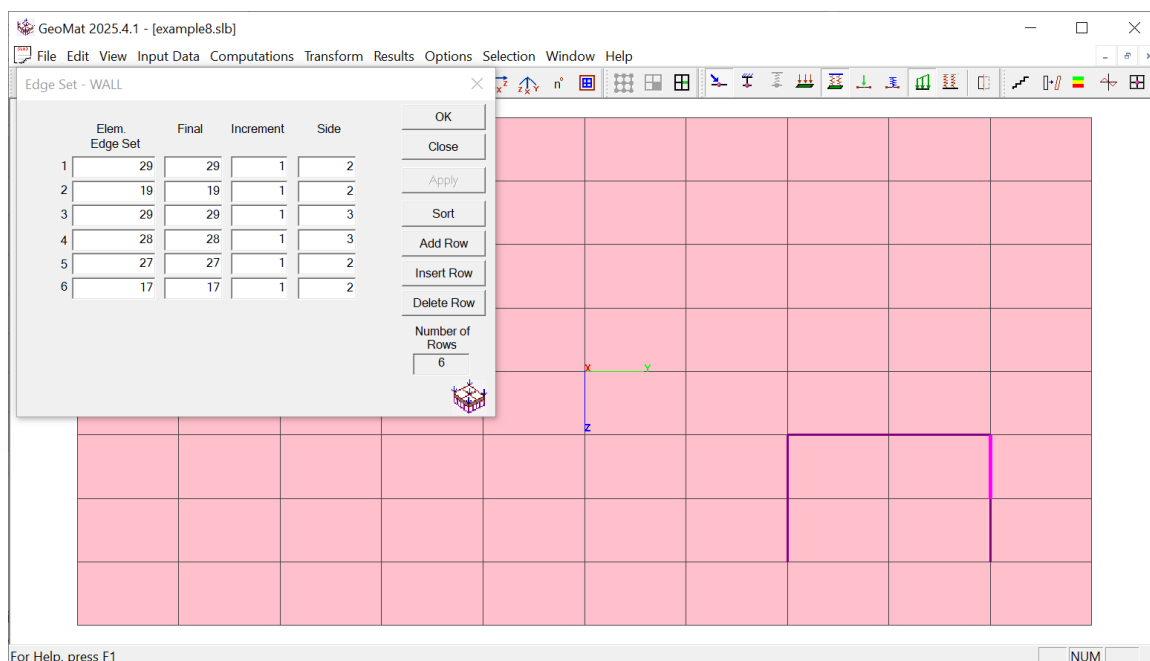


Figure 6.23 Location of edge set for U-shape wall in Example 8

6.9.3 Output Results Review

When the analysis is completed, go to **Computations** → **View Output File**. To view the how the slab deforms, go to **Results** → **Deformed Mesh**. The deformed shape of the slab is shown in Figure 6.24. Moment distributions in the y and z directions are shown in Figure 6.25 and Figure 6.26. Note the locations of positive and negative bending

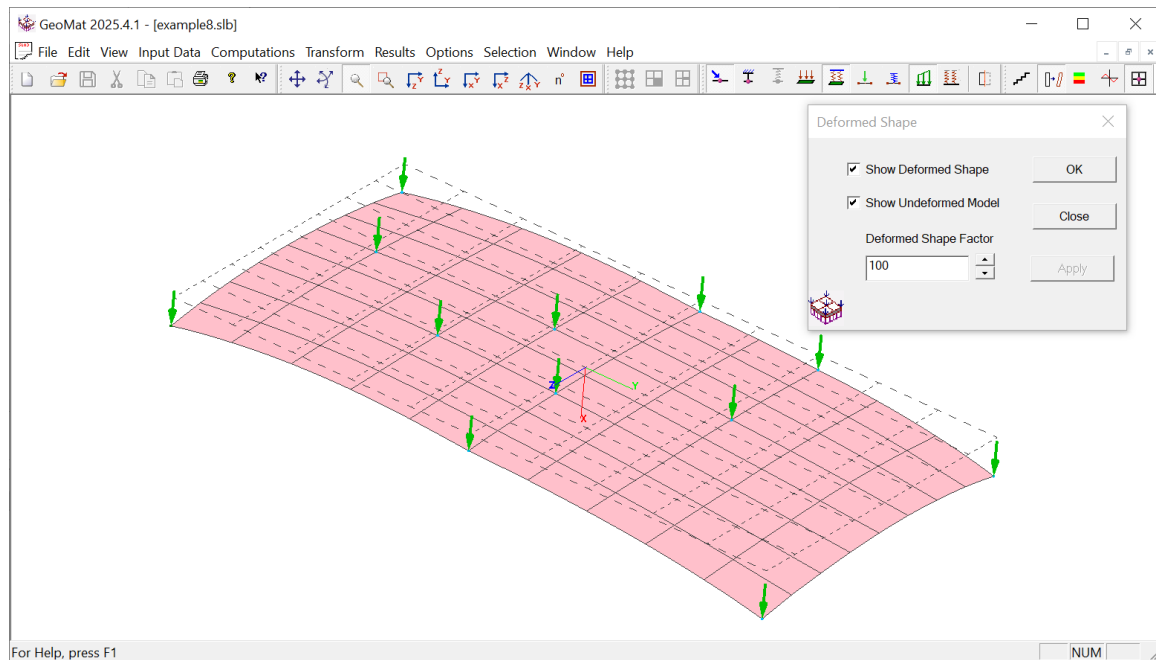
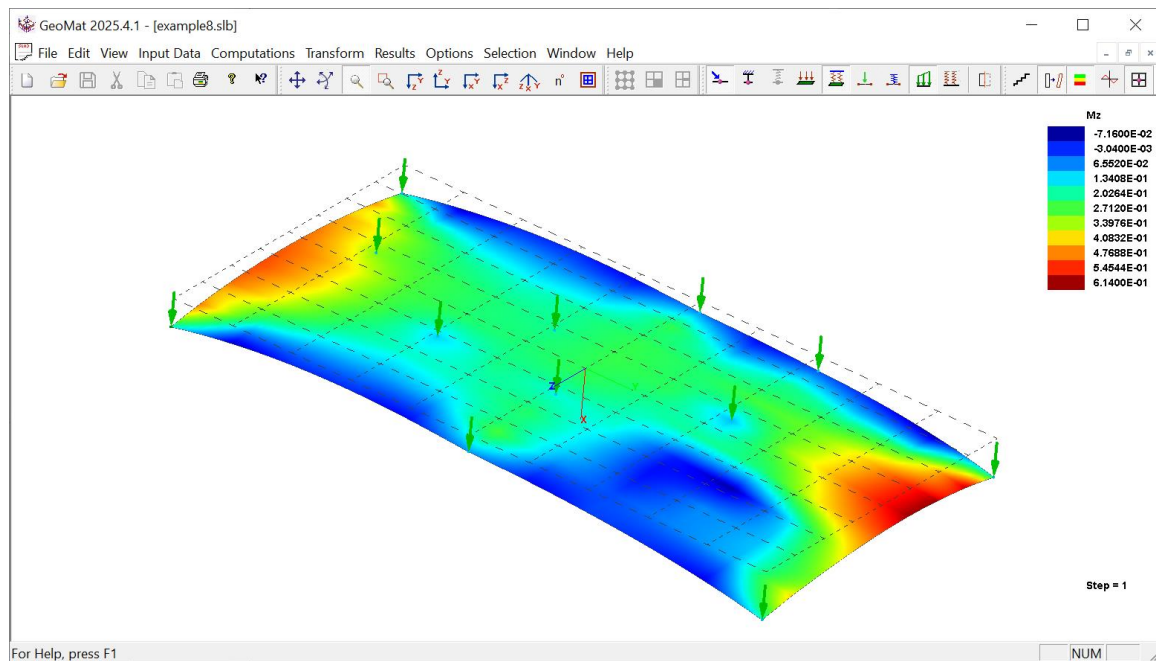


Figure 6.24 Deformation of slab in Example 8

Figure 6.25 Contour of bending moment M_z

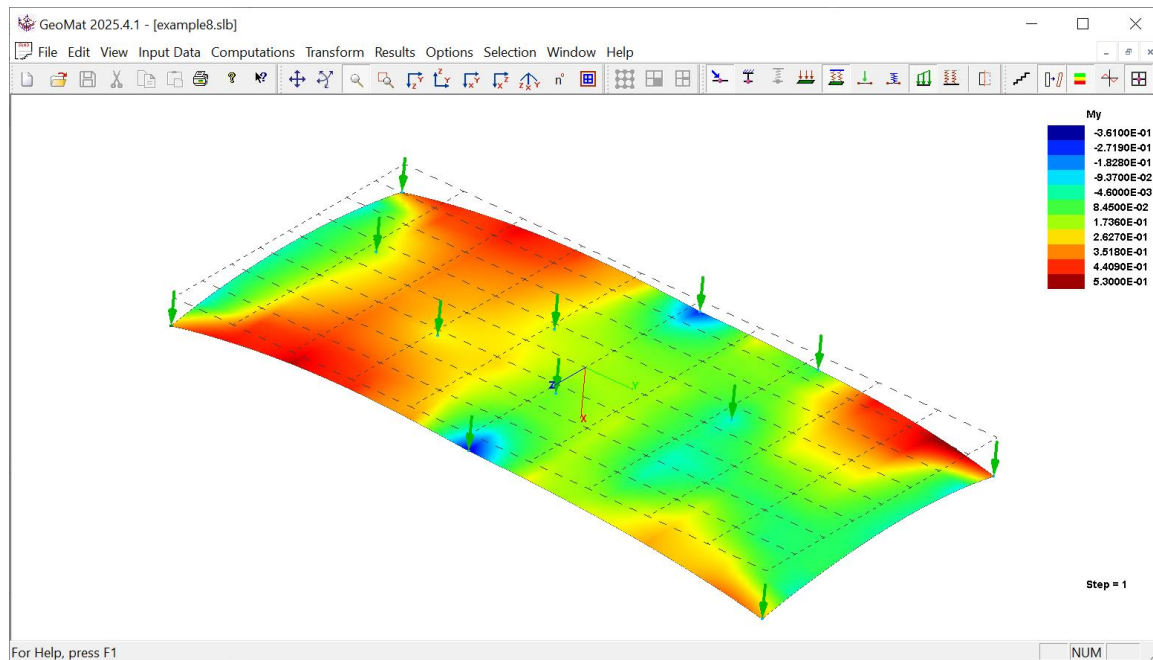


Figure 6.26 Contour of bending moment M_y

6.10 Example 9: Wind turbine foundation supported by drilled shafts

6.10.1 Problem Description

A foundation design utilizing drilled shafts, hereafter referred to as a *drilled shaft foundation*, for a wind turbine generator would conceptually consist of a circular or square array of drilled shafts connected to a reinforced concrete cap (Figure 6.27). The wind turbine tower would then be attached to the cap with the use of post-tensioned anchor bolts anchored to an embedment plate within the cap, or an embedded portion of the wind turbine tower, commonly referred to as an Embedded Stub or Foundation Mounting Part (FMP) as shown in Figure 6.28. Drilled shafts would extend to a sufficient depth to resist design loads from overturning, torsion, and horizontal shear, as well as gravity loads from the wind turbine and associated components.

The unique requirements of structural design on a pile cap for a wind-turbine foundation include: (1) integrating the drilled shafts and the pier cap together so that the pile cap may reliably distribute the loads to each drilled shaft; and (2) integrating the base cap and the embedded mounting plate of the wind turbine tower to substantially reduce or eliminate the risk of pullout failure of the embedded mounting plate. The punching-shear failure near the heavily loaded shaft as well as the surface reinforcement for delivering the required bending capacity of the pile cap should also be evaluated in the pile-cap design.

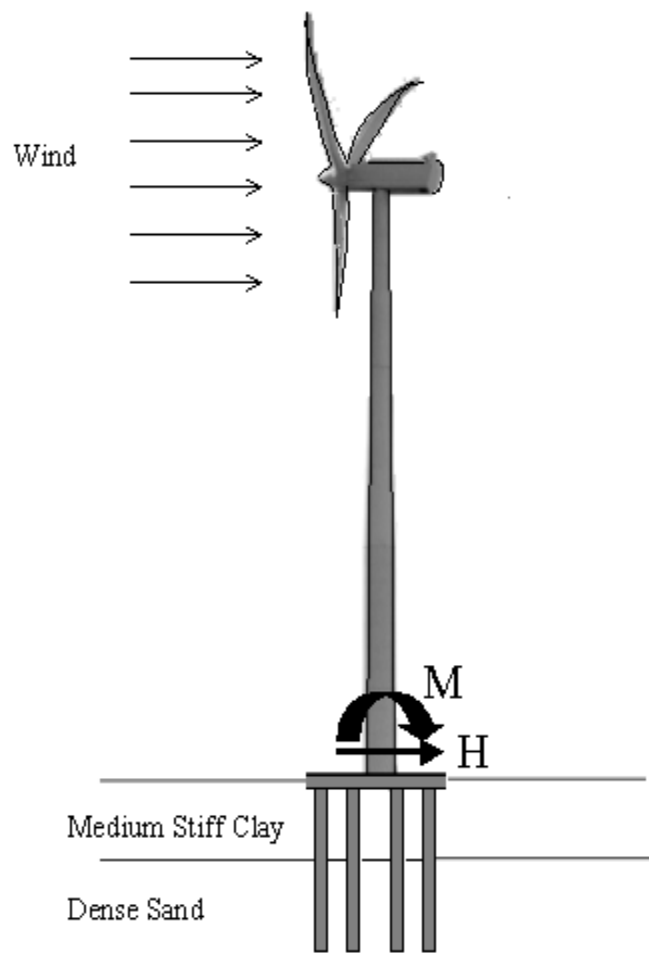


Figure 6.27 Conceptual sketch of drilled-shaft foundation supporting a wind turbine

The diameter of the concrete pile cap is 6.835 m and the thickness of the pile cap is 1.5 m. The loads on the pile cap include (1) vertical load of 1108 kN, horizontal load of 550 kN, and bending moment of 21373.4 kN-m. The embedded mounting plate hosts a total of 96 anchor bolts which are arranged in two circular rings (Figure 6.28). The diameter of the outer ring is 1.959 m and the diameter of the inner ring is 1.839 m.

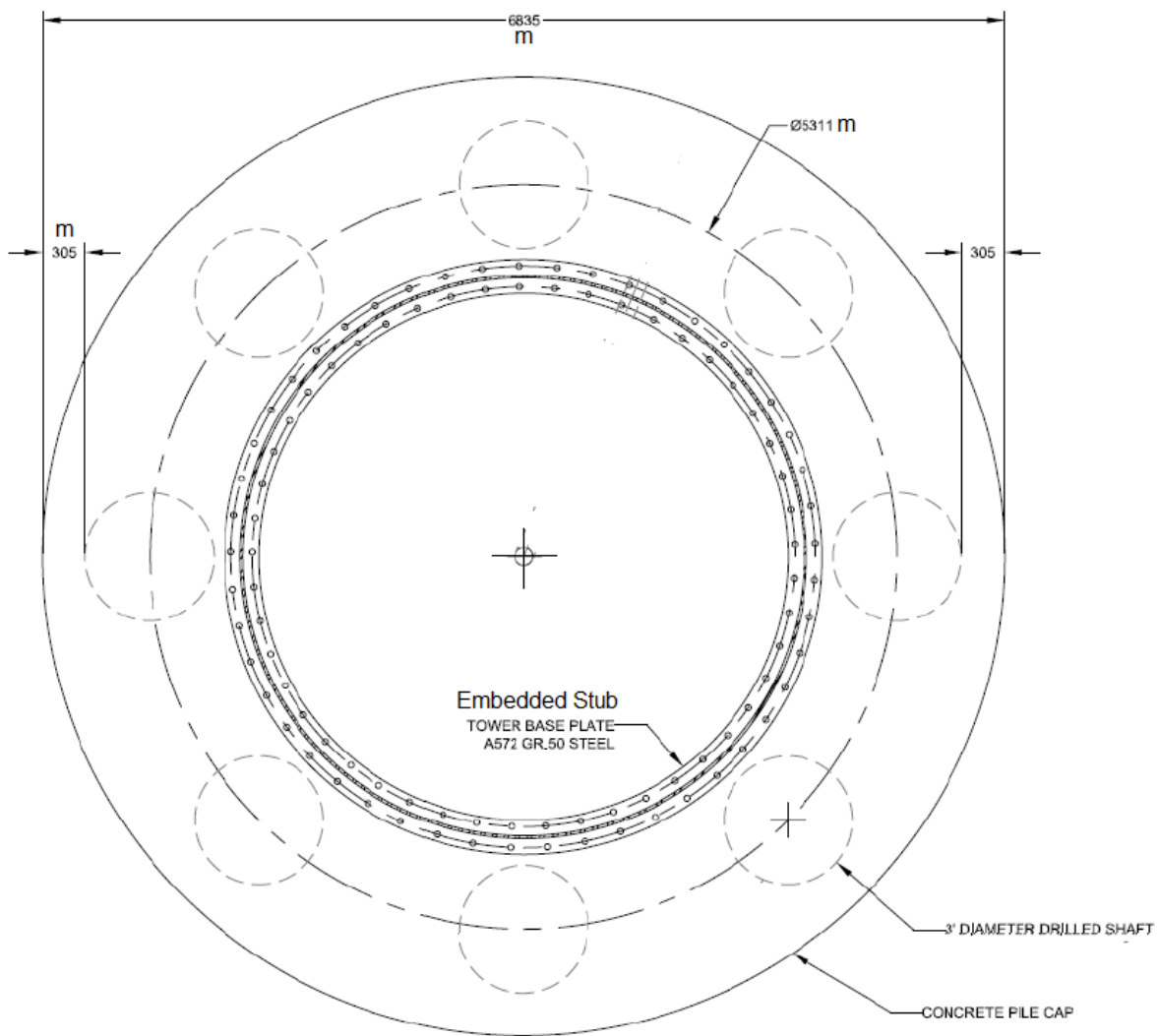


Figure 6.28 Dimension of pile cap and the embedded mounting plate with two rings of anchor bolts

6.10.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose “Circular – Equally Spaced, Linear 4 Noded” for the Shape and Element Types. Enter the diameter as 6.835 m, and the “Number of Divisions” as 64, as shown in Figure 6.29. Click **OK** to draw the slab on the screen.

Go to Input **Data** → **Title** and enter a name for the current project; this one is *Example 9*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. The material can be specified as isotropic type. Leave the default “Element Number/Set” value as **ALL**,

and modify the remaining parameters such as Young's modulus of $21,000,000 \text{ kN/m}^2$, Poisson's ratio of 0.3 , and the thickness of 1.5 m .

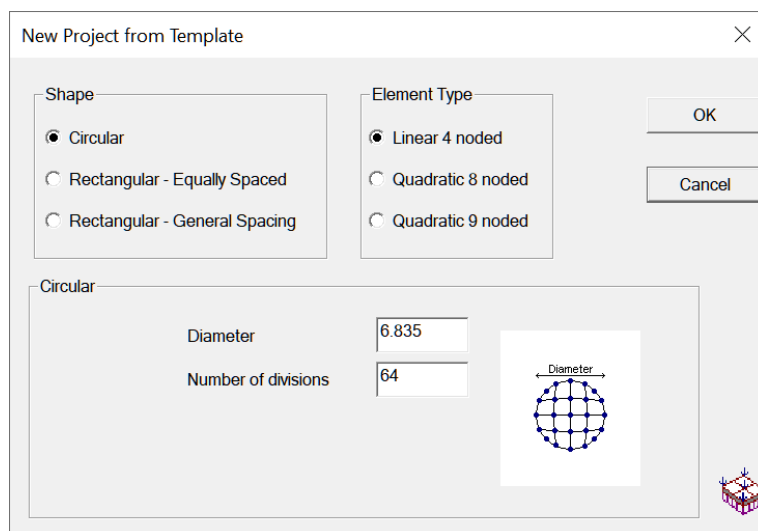


Figure 6.29 New Project from Template dialog screen for Example 9

It can be assumed that the pile cap is restrained completely (not allowing any translations) at the top of drilled shafts. The drilled shafts are assigned as node sets P1, P2, P3, P4, P5, P6, P7, and P8 based on the location.

To define the nodal sets (P1, P2, P3, P4, P5, P6, P7, and P8) to be use later for Nodal Constraints, click **Node Set** under Input Data Menu and bring a dialog box as shown in Figure 6.30. Add 8 new rows and assign the names P1, P2, P3, P4, P5, P6, P7, and P8. Click the **Define** button under "Action" for each set name. As shown in Figure 6.31 to Figure 6.34, the nodal points which are covered by the contact area were entered with some convenient arrays. Those nodal points can be easily identified on the mesh plot by the Zone tool (Figure 6.35).

Element Set

Set	Action
1 ALL	Define

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

1

Node Set

Set	Action
1 ALL	Define
2 P1	Define
3 P2	Define
4 P3	Define
5 P4	Define
6 P5	Define
7 P6	Define
8 P7	Define
9 P8	Define

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

9

Figure 6.30 Node Set screen for Example 9

Node Set - P1

Node Number/Set	Final	Increment
1 2543	2546	1
2 2607	2612	1
3 2671	2677	1
4 2736	2742	1
5 2800	2807	1
6 2865	2872	1
7 2930	2937	1
8 2995	3001	1
9 3060	3065	1
10 3125	3129	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

10

Node Set - P2

Node Number/Set	Final	Increment
1 3465	3470	1
2 3529	3536	1
3 3593	3601	1
4 3657	3666	1
5 3722	3731	1
6 3787	3795	1
7 3852	3859	1
8 3917	3922	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

8

Figure 6.31 Definition of Node Sets P1 and P2 for Example 9

Node Set - P3

Node Number/Set	Final	Increment
1 3486	3491	1
2 3550	3557	1
3 3615	3623	1
4 3680	3689	1
5 3745	3754	1
6 3811	3819	1
7 3877	3884	1
8 3944	3949	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

8

Node Set - P4

Node Number/Set	Final	Increment
1 2590	2593	1
2 2654	2659	1
3 2719	2725	1
4 2784	2790	1
5 2849	2856	1
6 2914	2921	1
7 2979	2986	1
8 3045	3051	1
9 3111	3116	1
10 3177	3181	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows

10

Figure 6.32 Definition of Node Sets P3 and P4 for Example 9

Node Set - P5

	Node Number/Set	Final	Increment
1	1680	1683	1
2	1614	1619	1
3	1549	1555	1
4	1484	1490	1
5	1419	1426	1
6	1354	1361	1
7	1289	1296	1
8	1225	1231	1
9	1161	1166	1
10	1097	1101	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows
10

Node Set - P6

	Node Number/Set	Final	Increment
1	756	761	1
2	690	697	1
3	625	633	1
4	560	569	1
5	495	504	1
6	431	439	1
7	367	374	1
8	304	309	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows
8

Figure 6.33 Definition of Node Sets P5 and P6 for Example 9

Node Set - P7

	Node Number/Set	Final	Increment
1	735	740	1
2	669	676	1
3	603	611	1
4	537	546	1
5	472	481	1
6	407	415	1
7	342	349	1
8	277	282	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows
8

Node Set - P8

	Node Number/Set	Final	Increment
1	1633	1636	1
2	1567	1572	1
3	1501	1507	1
4	1436	1442	1
5	1370	1377	1
6	1305	1312	1
7	1240	1247	1
8	1175	1181	1
9	1110	1115	1
10	1045	1049	1

OK

Close

Apply

Sort

Add Row

Insert Row

Delete Row

Number of Rows
10

Figure 6.34 Definition of Node Sets P7 and P8 for Example 9

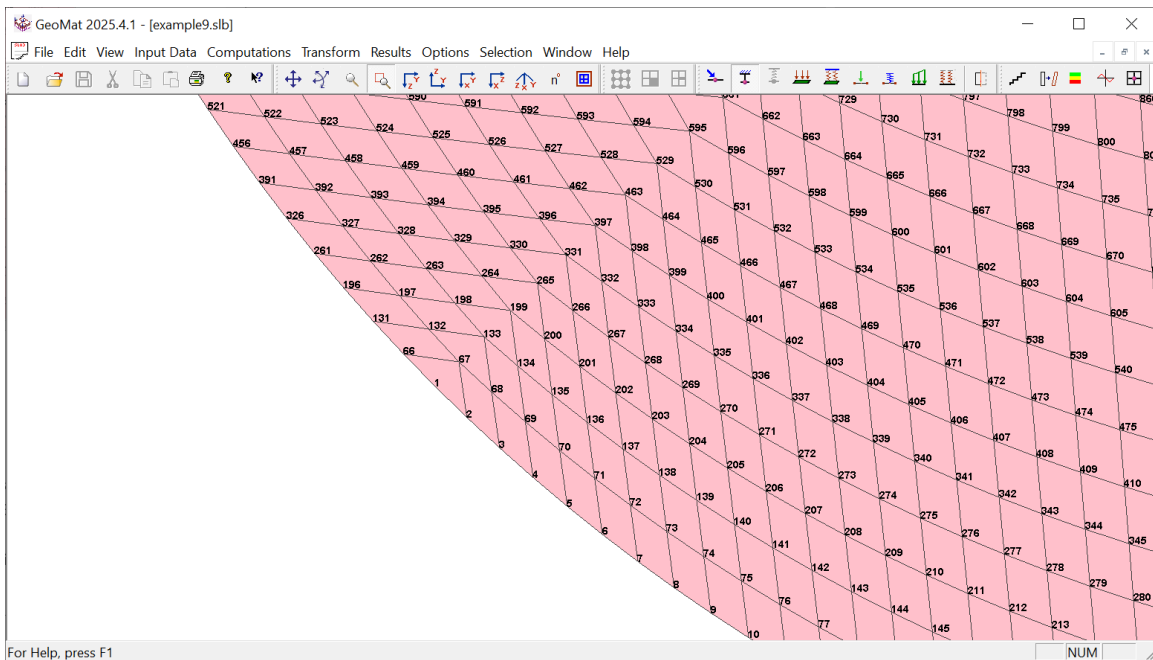


Figure 6.35 Node numbers on the screen for Example 9

To set the nodal constraints, go to **Input Data → Nodal Constraint**. Add one row for each drilled shaft location using the **Add Row** button. Under the Nodal Number heading, enter (P1, P2, P3, P4, P5, P6, P7, and P8) in the edit box, so the each represent the nodal points within the contact area of drilled shaft. Next, check the box under “Displac. X”, “Displac. Y”, and “Displac. Z” columns and enter 0 in the associated edit box (Figure 6.36). Click **Apply** and then the **OK** button.

Specified Nodal Displacements

Node Number	Displac. X	Displac. Y	Displac. Z	Rotation X	Rotation Y	Rotation Z
1 P1	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
2 P2	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
3 P3	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
4 P4	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
5 P5	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
6 P6	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
7 P7	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0
8 P8	☒ 0	☒ 0	☒ 0	☒ 0	☐ 0	☐ 0

Number of Rows: 8

Buttons: OK, Close, Apply, Sort, Add Row, Insert Row, Delete Row

Figure 6.36 Nodal constrain dialog screen for Example 9

Next, go to **Input Data → Equivalent Concentrated Loads**. The user can specify two circular rings for hosting 48 anchors at each ring and enter the diameters of each ring. The user also needs to specify the loads ($F_x = 1108$ kN, and $M_y = 21373.4$ kN) applied on the pile cap through the anchor bolts. Note that for this example the horizontal load was not applied since only actions out of plane were analyzed.

The program will calculate the location of each anchor bolts arranged in a circular pattern and the concentrated loads on each anchor bolt based on the anchor-bolt location and loading data as shown in Figure 6.37.

R...	N	X	Y	Z	Fx	Fy	Fz
Int 1	0.000...	1.839...	0.000...	0.000...	-215...	5.729...	0.000...
Int 2	0.000...	1.823...	0.240...	0.000...	-213...	5.729...	0.000...
Int 3	0.000...	1.776...	0.475...	0.000...	-207...	5.729...	0.000...
Int 4	0.000...	1.699...	0.703...	0.000...	-198...	5.729...	0.000...
Int 5	0.000...	1.592...	0.919...	0.000...	-184...	5.729...	0.000...
Int 6	0.000...	1.458...	1.119...	0.000...	-168...	5.729...	0.000...
Int 7	0.000...	1.300...	1.300...	0.000...	-148...	5.729...	0.000...
Int 8	0.000...	1.119...	1.458...	0.000...	-126...	5.729...	0.000...
Int 9	0.000...	0.919...	1.592...	0.000...	-101...	5.729...	0.000...
Int 10	0.000...	0.703...	1.699...	0.000...	-75.2...	5.729...	0.000...
Int 11	0.000...	0.475...	1.776...	0.000...	-47.1...	5.729...	0.000...
Int 12	0.000...	0.240...	1.823...	0.000...	-18.0...	5.729...	0.000...
Int 13	0.000...	0.000...	1.839...	0.000...	11.54...	5.729...	0.000...
Int 14	0.000...	-0.24...	1.823...	0.000...	41.15...	5.729...	0.000...
Int 15	0.000...	-0.47...	1.776...	0.000...	70.25...	5.729...	0.000...
Int 16	0.000...	-0.70...	1.699...	0.000...	98.35...	5.729...	0.000...
Int 17	0.000...	-0.91...	1.592...	0.000...	124.9...	5.729...	0.000...
Int 18	0.000...	-1.11...	1.458...	0.000...	140.6...	5.729...	0.000...

Figure 6.37 Equivalent concentrated loads screen for Example 9

Before executing the file, go to **File → Save As...** and save the file as *example9.slb*. Once this is completed, go to **Computations → Run Analysis** to launch the analysis.

6.10.3 Output Results Review

When the analysis is completed, go to **Computations → View Output File**. The maximum vertical displacement is approximately 0.113 mm under compression and 0.097 mm under tension as shown in Figure 6.38. The contour of bending moment along the Y-axis is shown in Figure 6.39. The contour of shear along the XY-planes is shown in Figure 6.40.

For viewing the section-cut plot, the user should click Section Cut item under Input Data menu and then select the cutting lines by default or by the user-specified. To view the deflection along the cutting line, go to **Results** → **Section Cut Plot** and select the desired cutting line and deflection as shown in Figure 6.41 for required display.

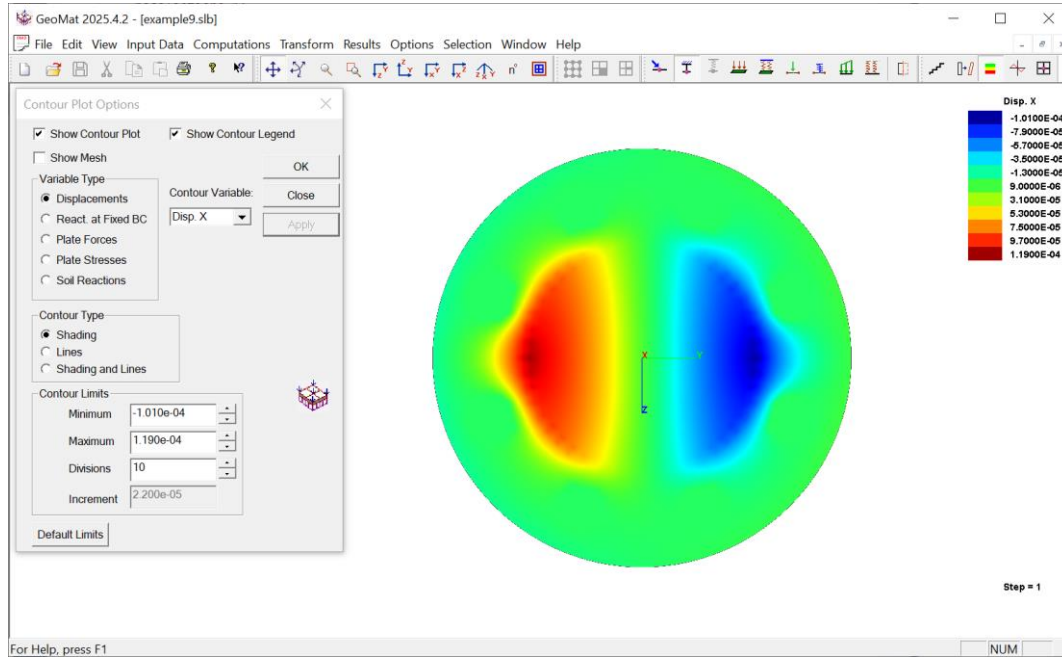


Figure 6.38 Contour of deflection of Example 9

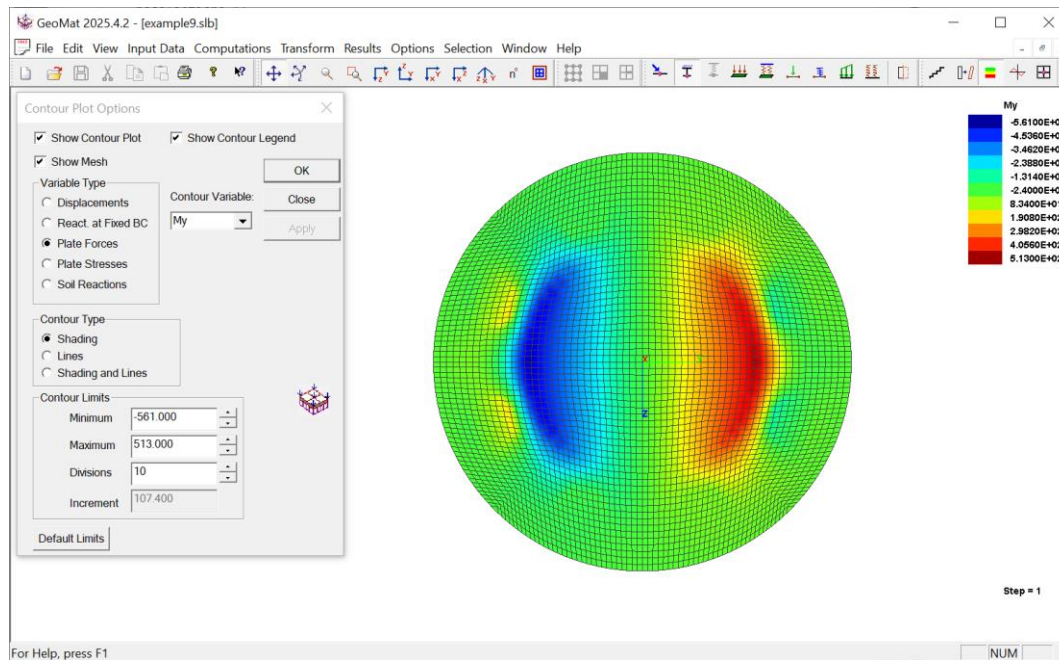


Figure 6.39 Contour of bending moment (My) of Example 9

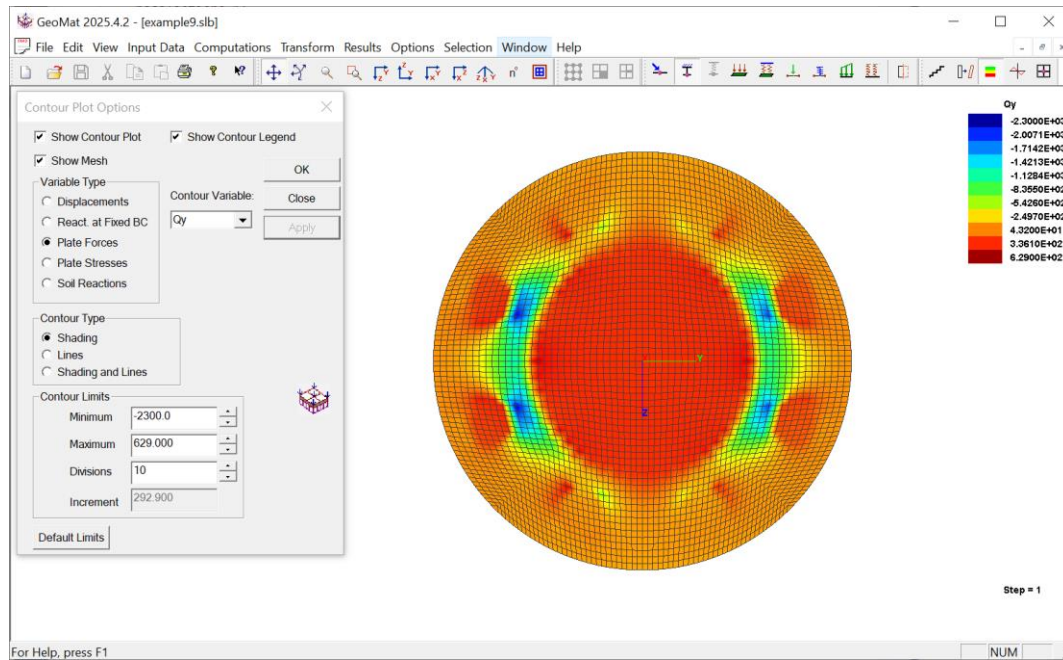


Figure 6.40 Contour of shear on XY plane of Example 9

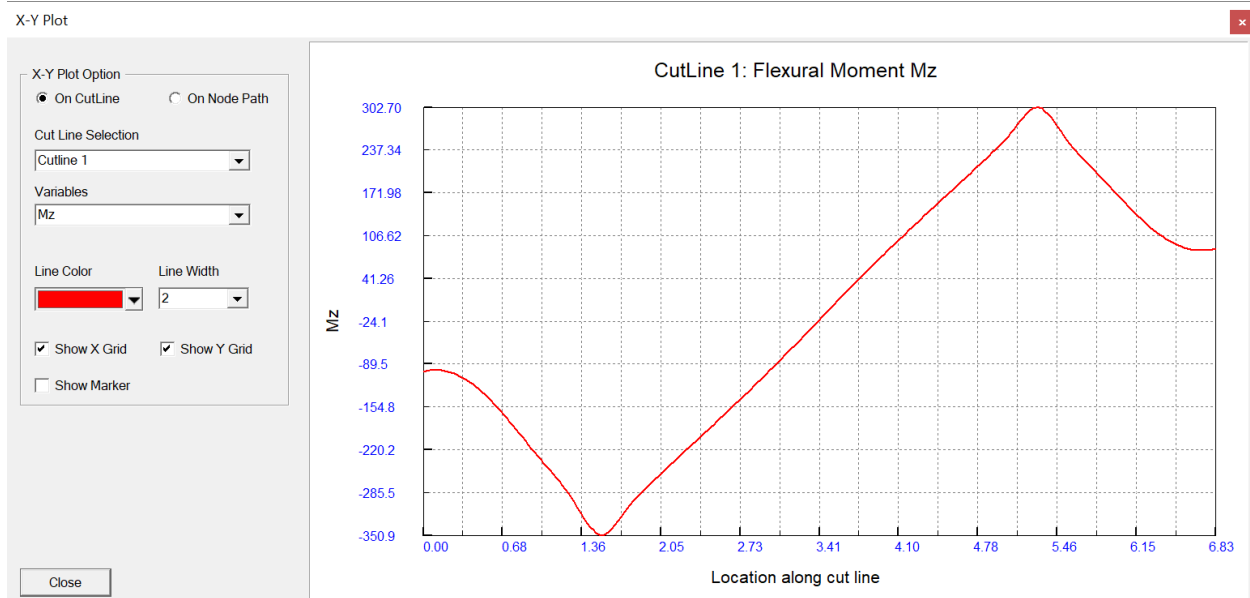


Figure 6.41 Bending moment (Mz) along a cutting line of Example 9

6.11 Example 10: Square slab on soil - Vlasov method

6.11.1 Problem Description

This example considers a square concrete slab resting on the soil. A concentrated force $P = 56 \text{ kip}$ is applied at the center of the slab (Figure 6.42). The slab is 33 ft by 33 ft, and is 0.851 ft thick. The Young's modulus and the Poisson's ratio of the slab and soil are listed in Table 6-1. Assume the compressible thickness of the soil is 30 ft. Bottom boundary of the soil is assumed to be fixed in x,y,z directions.

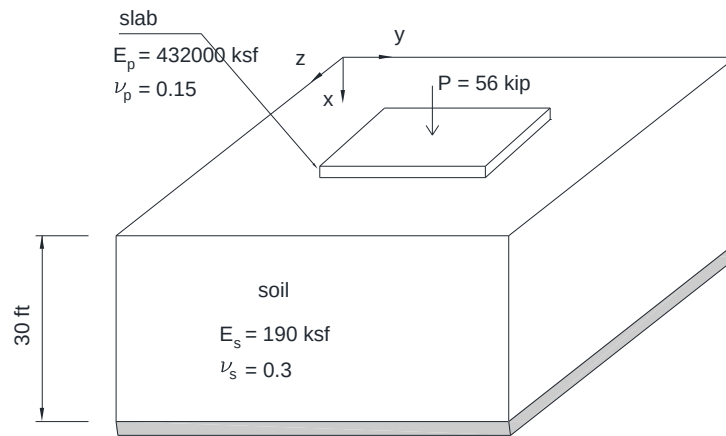


Figure 6.42 Configuration of slab resting on soil of example 10

Table 6-1: Parameters of slab and soil in example 10

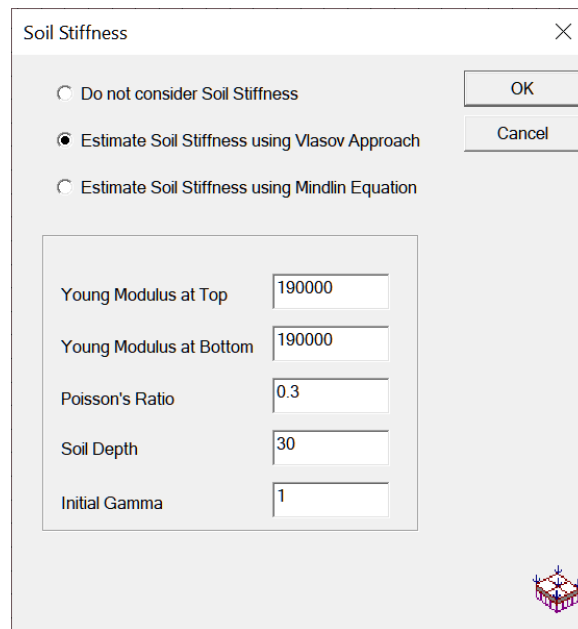
	Young's modulus E(psf)	Poisson's ratio ν	Thickness (ft)
slab	432000000	0.15	0.851
soil	190000	0.3	30

6.11.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the Template dialog box, choose "Rectangular – Equally Spaced, Quadratic 8 Noded" for the Shape and Element Types. Enter 33 for both "Length" and "Width", and the "Number of Divisions" as 24. Here a fine mesh is desired because coarse mesh can't capture the peak forces near the concentrated load. Click **OK** to draw the slab on the screen.

Go to **Input Data -Title** and enter a name for the current project; this one is *Example 10*. Click **OK** to save this as the project title. Next, return to the **Input Data** menu and proceed to **Material Properties**. Leave the default "Element Number/Set" value as ALL, and reset the remaining parameters to those given in the problem statement. Click **Apply** to save the changes, then click **OK** to close the dialog.

To set the soil stiffness, go to **Input Data → Soil Stiffness**. Select the second option “Estimate Soil Stiffness using Vlasov Approach”. Input soil properties as shown in Figure 6.43. Keep the initial Gamma as 1.0. This default value works well for most cases. Click **OK** to exit.



Soil Stiffness

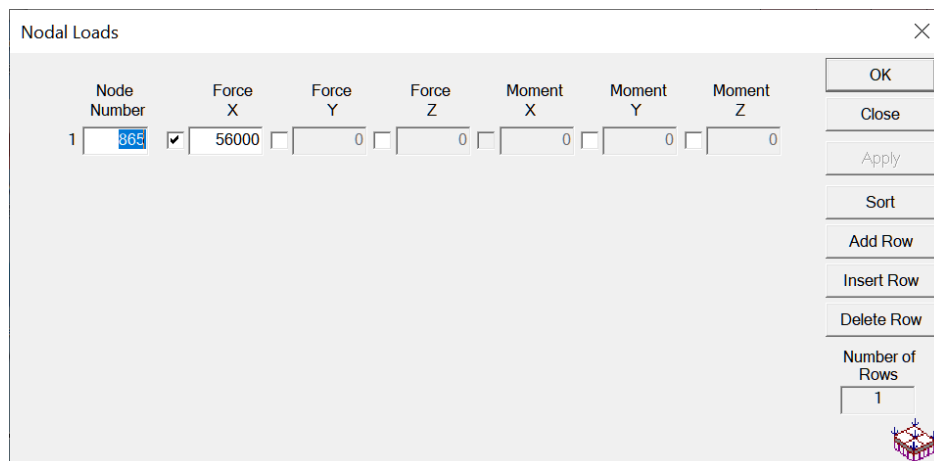
☐ Do not consider Soil Stiffness
☒ Estimate Soil Stiffness using Vlasov Approach
☐ Estimate Soil Stiffness using Mindlin Equation

Young Modulus at Top: 190000
 Young Modulus at Bottom: 190000
 Poisson's Ratio: 0.3
 Soil Depth: 30
 Initial Gamma: 1

OK Cancel

Figure 6.43 Input soil properties in Vlasov approach

Next, go to **Input Data → Nodal Loads**. The concentrated load is applied at the plate center (node 865). Click **Add Row** to add one new row, input 865 or click icon  on the toolbar, then select the node at the plate center. Check the option for Force X and input 56000 as shown in Figure 6.44. Display of node number can be controlled by menu **Options → Node Numbers**.



Nodal Loads

Node Number	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1 865	☒ 56000	☐ 0	☐ 0	☐ 0	☐ 0	☐ 0

OK
 Close
 Apply
 Sort
 Add Row
 Insert Row
 Delete Row
 Number of Rows: 1

Figure 6.44 Define concentrated load at slab center

Before executing the file, go to **File** → **Save As...** and save the file as example10.slb. Once this is completed, go to **Computations** → **Run Analysis** to launch the analysis.

6.11.3 Output Results Review

When the analysis is completed, go to **Computations** → **View Output File**. All deformation and internal forces are plotted into this file. Since node number in this example is large, it is not convenient to look for the output for one specific node. Go to **Results** → **Node Results**, then input node number 5521 which is loaded. Check the deflection at this node is 0.0118 ft (Figure 6.45).

Figure 6.45 View node output for one specific node

Go to **Results** → **Section Cut Plot**, select “My” as the field variables. The distribution of moment My along the center line of the plate is plotted as shown in Figure 6.46. Change field variable to “Qxy” to plot the shear force along the same cut line as in Figure 6.47. Since a fine mesh is defined, the peak moment and shear force can be observed in these cut line plots.

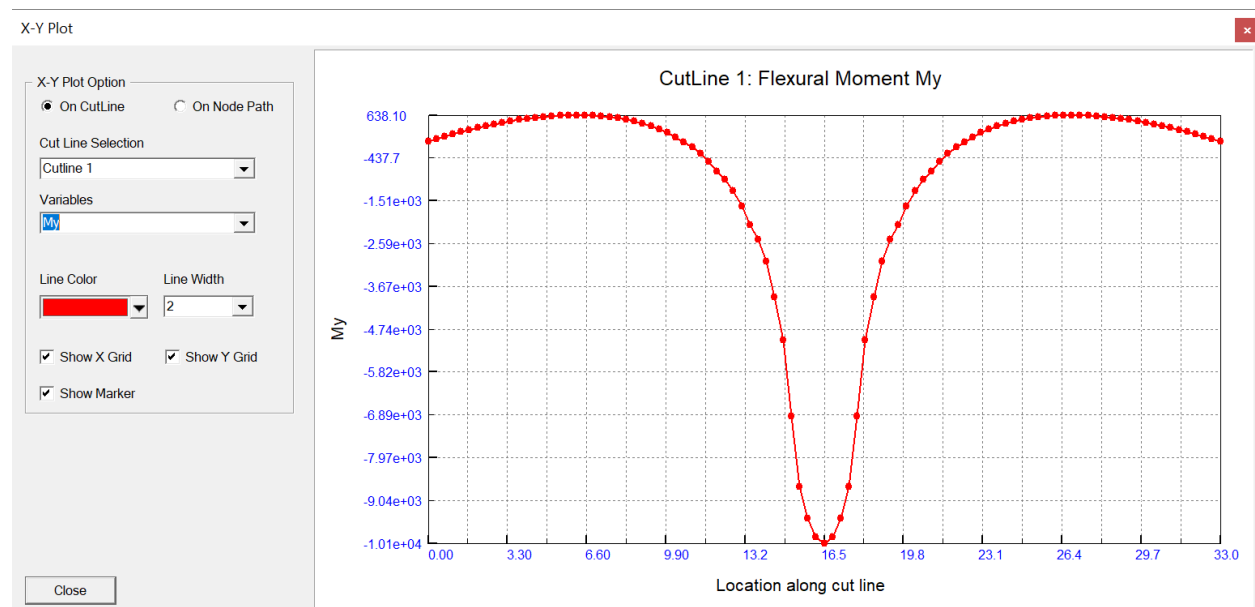


Figure 6.46 Distribution of moment My along cut line 1 in example 10

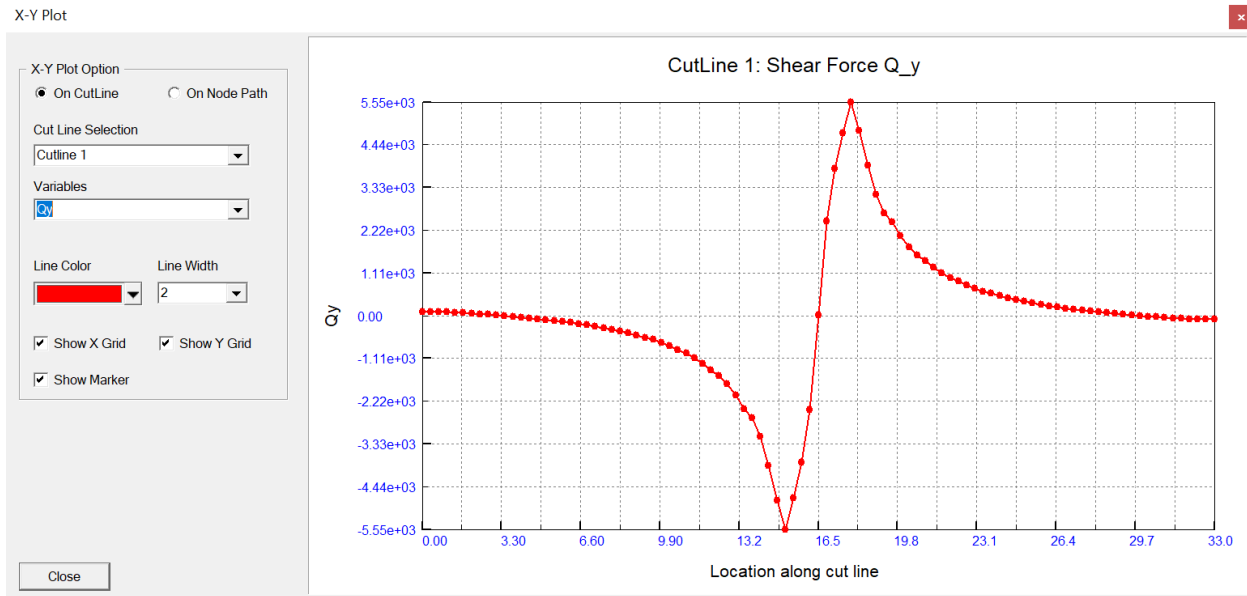


Figure 6.47 Distribution of shear force Q_y along cut line 1 in example 10

In the Vlasov approach, parameters γ , k and $2t$ are iterated until convergence of displacements is reached. The initial value of γ is set to 1 and the final γ increases to 1.81 as shown in Figure 6.48.

```

step    1   tcur   =   1.0000   dt =   1.0000

gamma =   1.000000, k =   0.8684E+04, 2t =   0.6456E+06

iter =   1   err1 = 0.1000E+01 d =   0.1999E+00 u =   0.1999E+00 rhs =   0.5600E+05
        err2 = 0.1000E+01 r =   0.2104E-01 ru = 0.2104E-01 rrhs=   0.0000E+00

gamma =   1.730552, k =   0.9564E+04, 2t =   0.5278E+06

iter =   2   err1 = 0.2800E-01 d =   0.5605E-02 u =   0.2002E+00 rhs =   0.4982E+03
        err2 = 0.4621E-01 r =   0.1018E-02 ru = 0.2204E-01 rrhs=   0.2091E+03

gamma =   1.800223, k =   0.9700E+04, 2t =   0.5165E+06

iter =   3   err1 = 0.4607E-02 d =   0.9196E-03 u =   0.1996E+00 rhs =   0.4885E+02
        err2 = 0.3094E-02 r =   0.6839E-04 ru = 0.2210E-01 rrhs=   0.1998E+02

gamma =   1.809304, k =   0.9719E+04, 2t =   0.5151E+06

iter =   4   err1 = 0.6331E-03 d =   0.1263E-03 u =   0.1995E+00 rhs =   0.6346E+01
        err2 = 0.3817E-03 r =   0.8439E-05 ru = 0.2211E-01 rrhs=   0.2586E+01

gamma =   1.810514, k =   0.9721E+04, 2t =   0.5149E+06

iter =   5   err1 = 0.8493E-04 d =   0.1694E-04 u =   0.1995E+00 rhs =   0.8452E+00
        err2 = 0.5049E-04 r =   0.1116E-05 ru = 0.2211E-01 rrhs=   0.3442E+00

```

Figure 6.48 Output of γ , k and $2t$ in example 10

The concentrated load of 56 k is applied at Node 5521, which is approximately at the center of the slab. The shear force at the cutting line clearly indicates the change of shear force from negative to positive before and after Node 5521 with the concentrate load as shown in Figure 6.47. The total change of shear force should yield the theoretical value of 56 k. However, the accuracy of numerical solution will

depend mostly on the element sizes used near the concentrated load as indicated in Figure 6.49. When the element sizes are finer enough, the change of shear force gets closer to the theoretical value. The result from finer meshes is also in agreement with those computed based a three-dimensional model using ABAQUS from Dassault Systems.

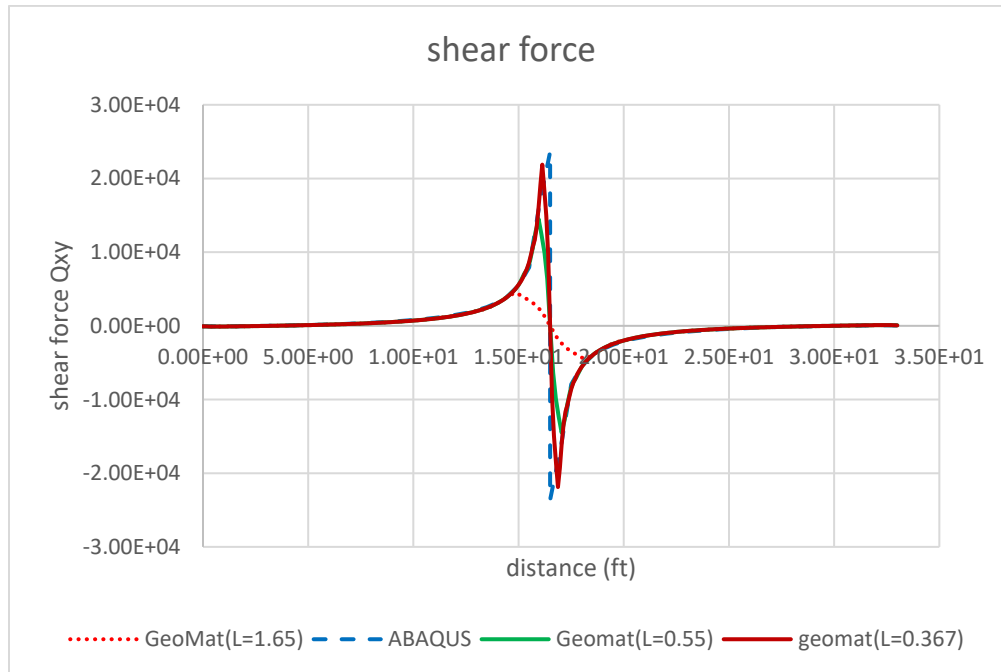


Figure 6.49 Influence of mesh size to shear stress in example 10

6.12 Example 11: Circular wind turbine foundation - Vlasov method

6.12.1 Problem Description

This example considers a wind turbine foundation with circular shape (Figure 6.50). The parameters of soil are listed in Table 6-2. The depth of compressible soil is 30 ft. Dimensions of the concrete foundation are also listed in Table 6-2. Applied loads include vertical thrust force $P = 696 \text{ kip}$, $V = 81.6 \text{ kip}$, and $M = 25250 \text{ kip} \cdot \text{ft}$. The loads are applied in xy plane in *GeoMat*. Units used in this example are *kip*, *ft*.

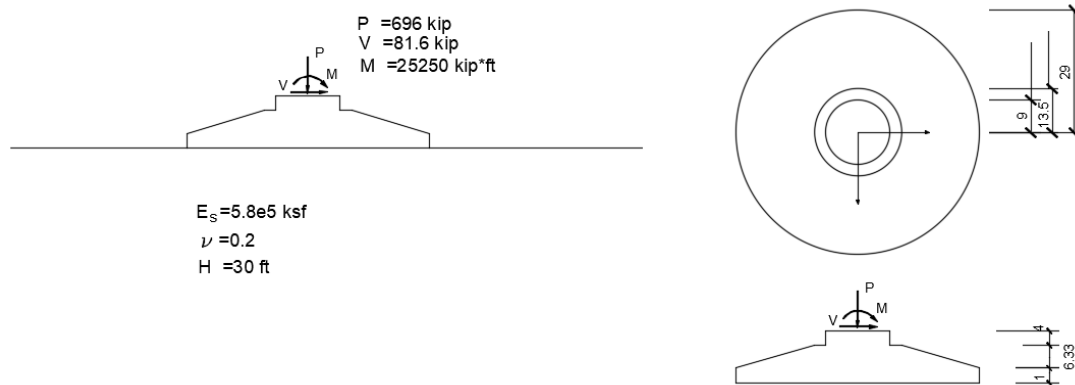


Figure 6.50 WTB foundation resting on soil of example 11

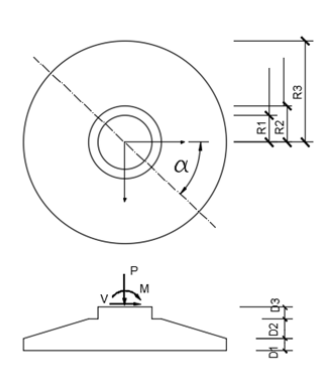
Table 6-2: Parameters of slab and soil in example 11

	Young's modulus $E(\text{ksf})$	Poisson's ratio ν	Thickness (ft)	R1 (ft)	R2 (ft)	R3 (ft)	D1 (ft)	D2 (ft)	D3 (ft)
foundation	5.8e5	0.15		9	13.5	29	1	6.33	4
soil	1320	0.2	30						

6.12.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the foundation type selection dialog, select "Circular wind turbine foundation". In the following template, input foundation dimensions and loads as shown in Figure 6.51. Keep the default division of 40. Keep the default "Quadratic 8-noded" element type and "2x2" integration rule. Click **OK** to generate the mesh.

Template of Circular Wind Turbine Foundation



Plane Dimension

R1 = 9
 R2 = 13.5
 R3 = 29
 Angle(deg) = 0

Section Dimension

D1 = 1
 D2 = 6.33
 D3 = 4

Load

P = 696
 V = 81.6
 M = 25250

Finite Element Option

☐ Linear 4 noded
☒ Quadratic 8 noded
☐ Quadratic 9 noded
 Division of $2 \cdot \pi \cdot R3$ = 40
 Typical FE Size = 2.28

Integration Rule

☐ 1x1
☒ 2X2
☐ 3X3

OK Cancel

Figure 6.51 Template for circular wind turbine foundation in example 11

Now define material parameters for the concrete foundation. Click menu "Input Data->Material Properties". The foundation is divided into four sets based on different thicknesses. Click the name under "Element Number/Set" will highlight selected element set in cyan color (

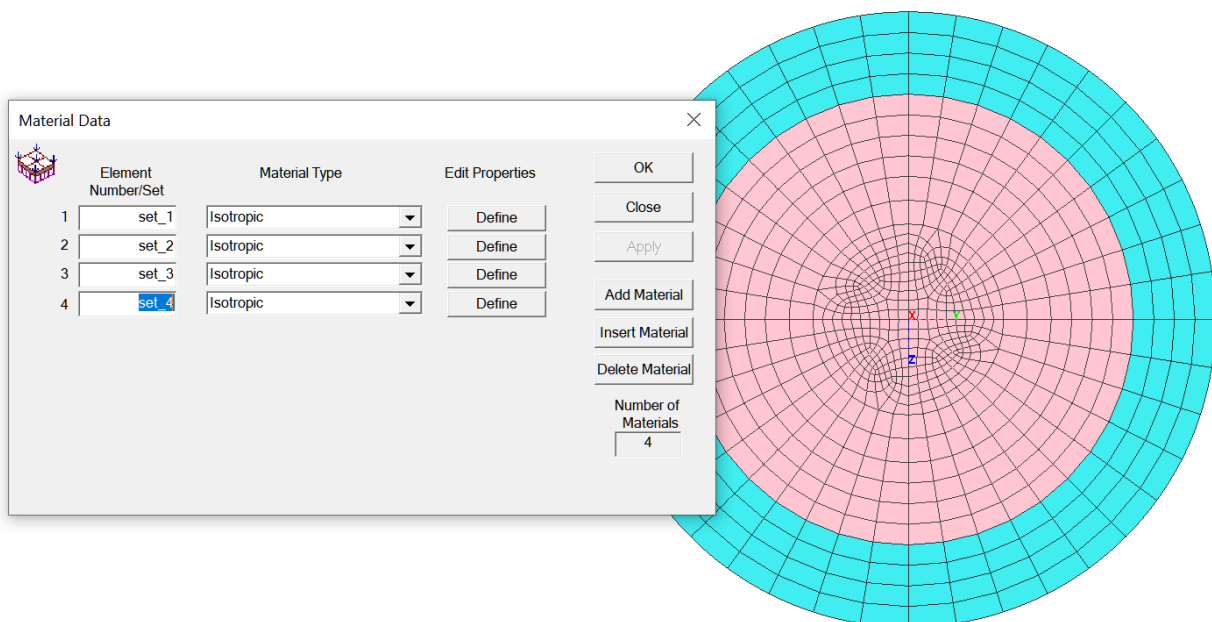


Figure 6.52 main window for material definition in example 11

). Click button **Define** to change Young's modulus E and Poisson's ratio to 5.8×10^5 and 0.15 respectively for each element set. Click **OK** to exit from the material property window.

The next step is to define soil parameters. Soil in the Vlasov approach is assumed to have one layer. Click menu **"Input Data->Soil Stiffness"** and select the second option "Estimate Soil Stiffness using Vlasov Approach". Change Young's both modulus at top and bottom to be 1.8×10^6 as shown in Figure 6.54. Change Poisson ratio to 0.2 . Click **OK** to exit.

Click menu **"Input Data->Nodal Loads"** to check equivalent nodal loads calculated by *GeoMat* (Figure 6.55). Nodal loads are concentrated point forces in green color. Click **OK** to exit.

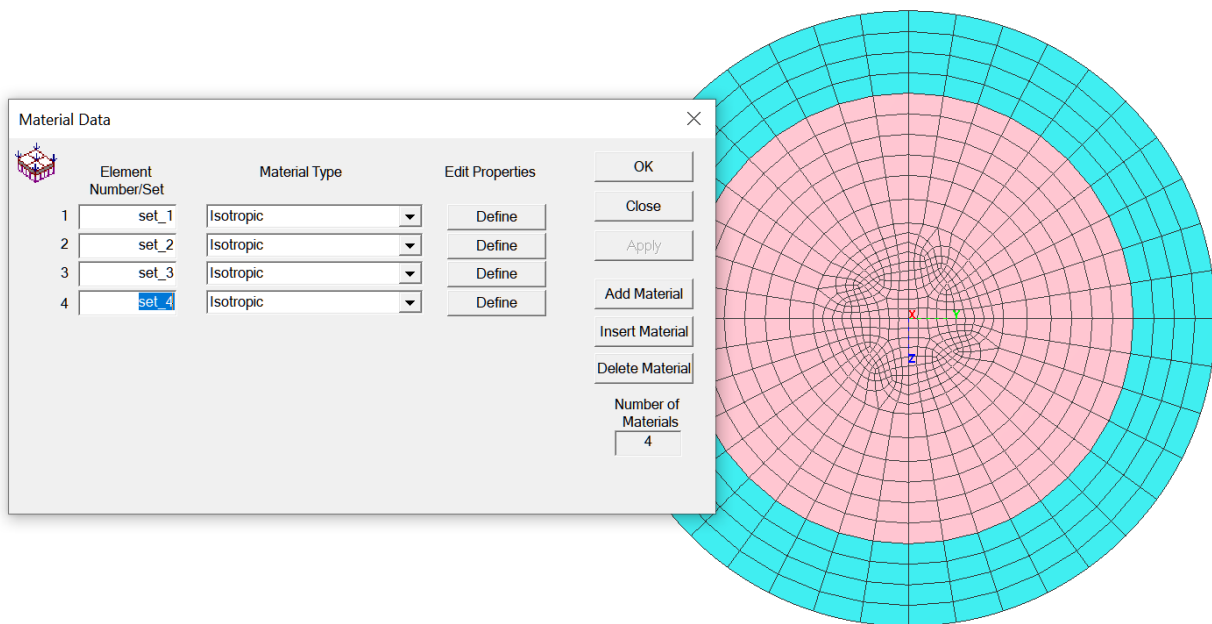


Figure 6.52 main window for material definition in example 11

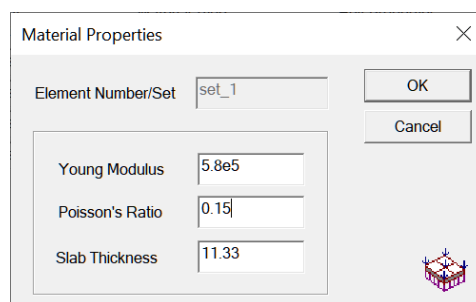
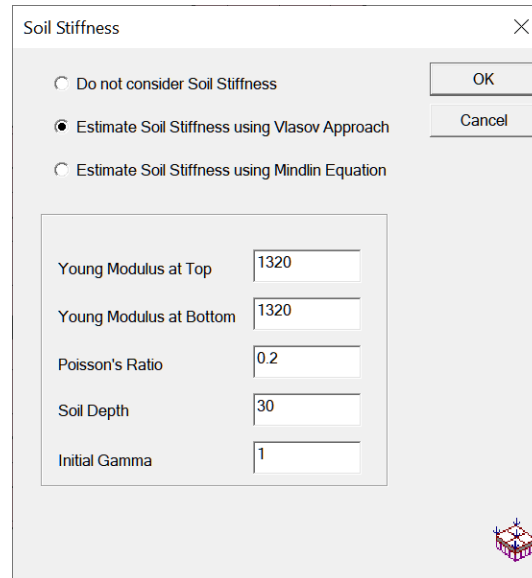


Figure 6.53 Define Young's modulus and Poisson ratio for each slab sets in example 11



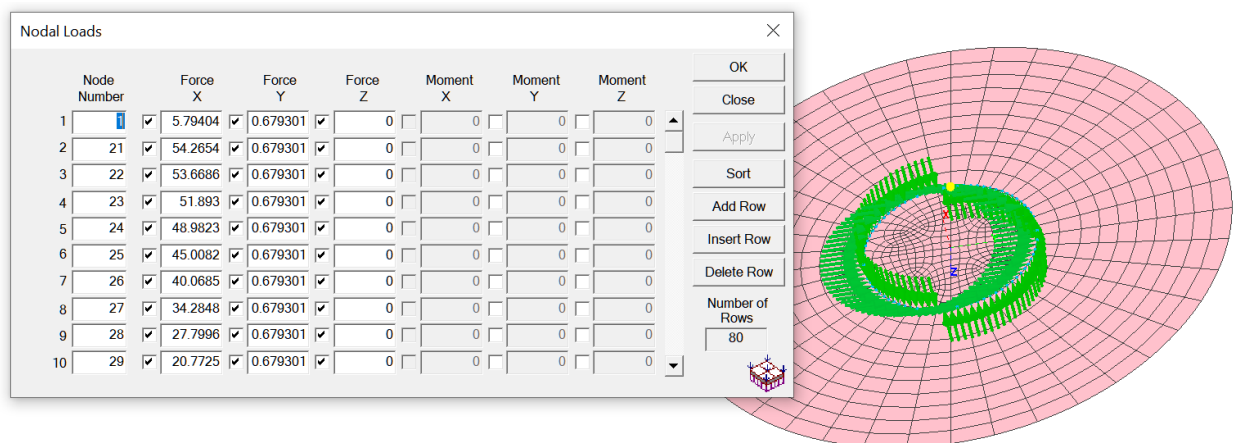
Soil Stiffness

☐ Do not consider Soil Stiffness
☒ Estimate Soil Stiffness using Vlasov Approach
☐ Estimate Soil Stiffness using Mindlin Equation

Young Modulus at Top: 1320
 Young Modulus at Bottom: 1320
 Poisson's Ratio: 0.2
 Soil Depth: 30
 Initial Gamma: 1

OK Cancel

Figure 6.54 Define soil property using Vlasov approach in example 11



Nodal Loads

	Node Number	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	1	5.79404	0.679301	0	0	0	0
2	21	54.2654	0.679301	0	0	0	0
3	22	53.6686	0.679301	0	0	0	0
4	23	51.893	0.679301	0	0	0	0
5	24	48.9823	0.679301	0	0	0	0
6	25	45.0082	0.679301	0	0	0	0
7	26	40.0685	0.679301	0	0	0	0
8	27	34.2848	0.679301	0	0	0	0
9	28	27.7996	0.679301	0	0	0	0
10	29	20.7725	0.679301	0	0	0	0

OK Close Apply Sort Add Row Insert Row Delete Row Number of Rows 80

Figure 6.55 Check equivalent nodal loads in example 11

Click icon  on the toolbar to reset view to y-z plane. Click menu “**Input Data->Nodal Constraints**” to define displacement boundary to remove rigid body motion. Click **Add Row** to add one new line. Click icon  to activate *pick and select by mouse*. Pick the most right node (node 682) and check “Displac. Y” and “Displac. Z”. Keep displacement boundaries to be 0. Similarly add other two rows to define fixed point at node 686 and 678. Constrained nodes are displayed in Figure 6.56. Click **OK** to exit.

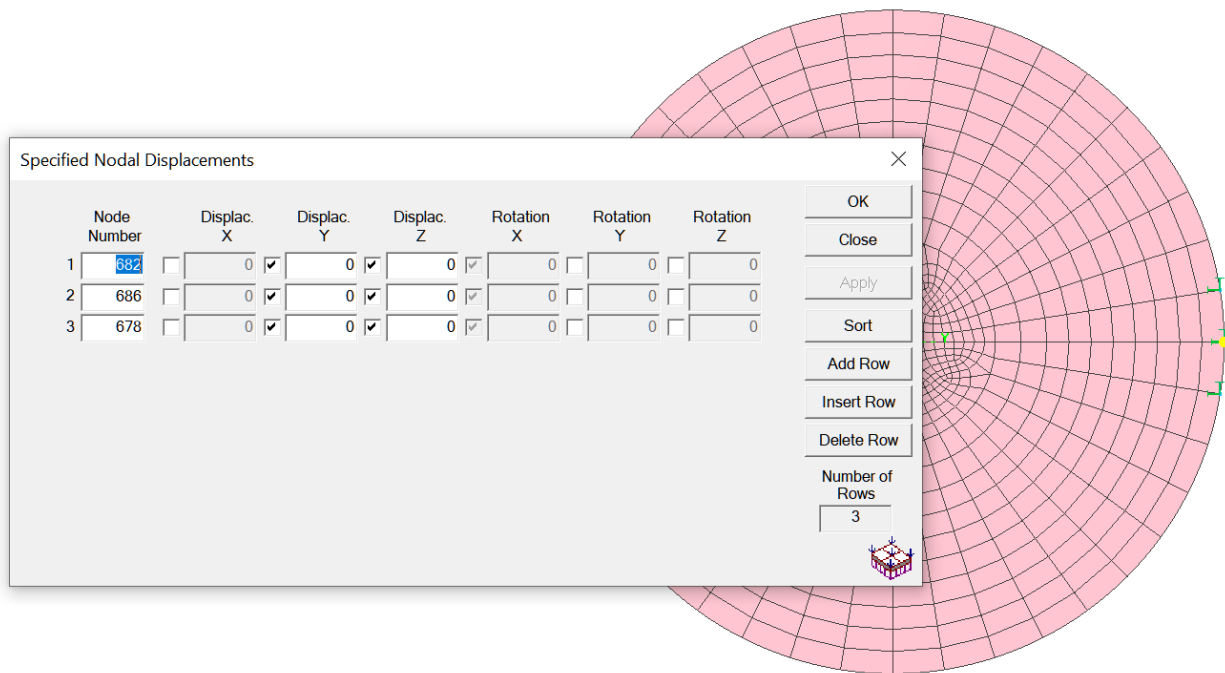


Figure 6.56 Apply nodal constraints to remove rigid body motion in example 11

6.12.3 Output Results Review

Click menu “**Computations->Run Analysis**”. When analysis is done, open message file to review iterations of γ , k and $2t$. Click menu “**Results->Section Cut Plot**” to plot deflection along the first cut line as shown in Figure 6.57. The maximum is 0.0106 ft (0.127in).

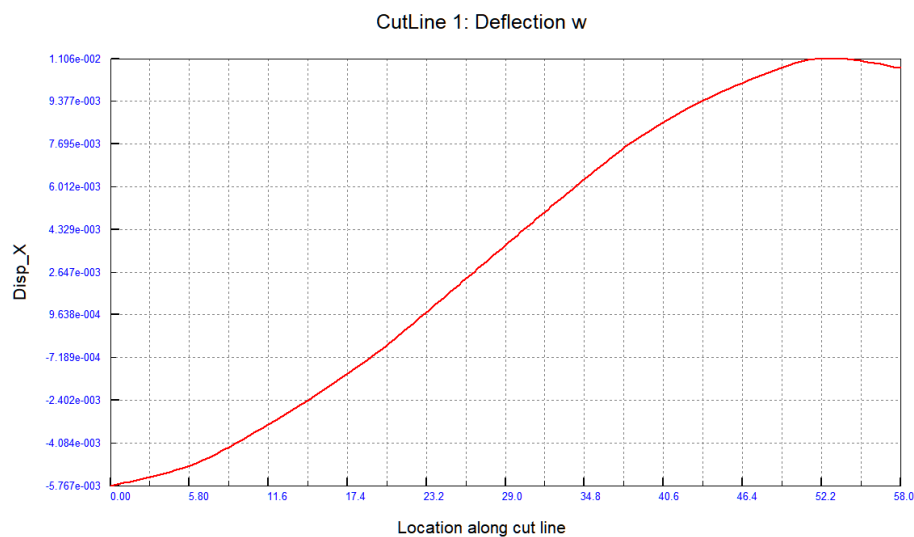


Figure 6.57 Deflection along cut line 1 in example 11

6.13 Example 12: Rectangle slab supported by nonlinear spring

6.13.1 Problem Description

This example presents the procedure to analyze a rectangle slab supported by four piles. These piles are simulated as nonlinear springs. One corner pile has weaker axial stiffness than other three piles. The 3D and side views are shown in Figure 6.58.

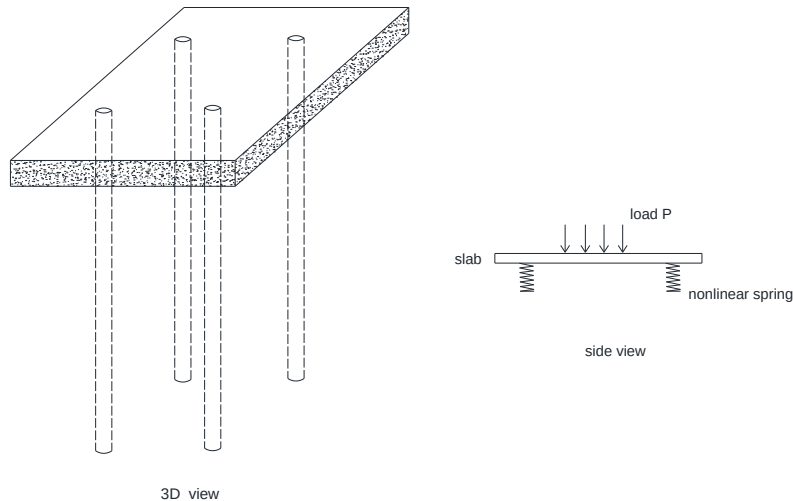


Figure 6.58 3D and side view of pile-supported slab in example 12

6.13.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the foundation type selection dialog, select “Circular or Rectangular Conventional” to create a 4 by 4 slab with eight divisions in each direction. Keep the default “Quadratic 8-noded” element type and “2x2” integration rule. Click **OK** to generate the mesh using constant units (kips and inches in this example).

Click the menu “**Input Data->Material Properties**” to define the slab material parameters: Young’s modulus $E = 2 \times 10^6$ and Poisson’s ratio $\nu = 0.2$. Click **OK** to exit from the material property window.

The next step is to define nonlinear springs. Click menu “**Input Data->Nonlinear Spring**” to define two new spring groups (Figure 6.59). The stiffness of the weak spring is about half of the normal spring. The dialogs to define two nonlinear springs is shown in Figure 6.60. Click the button “**New**” to create a new spring data group. Users can input data manually or click “**Open File**” to import the spring data from the file “nonlinear_spring_example12.txt”. Click **OK** to exit.

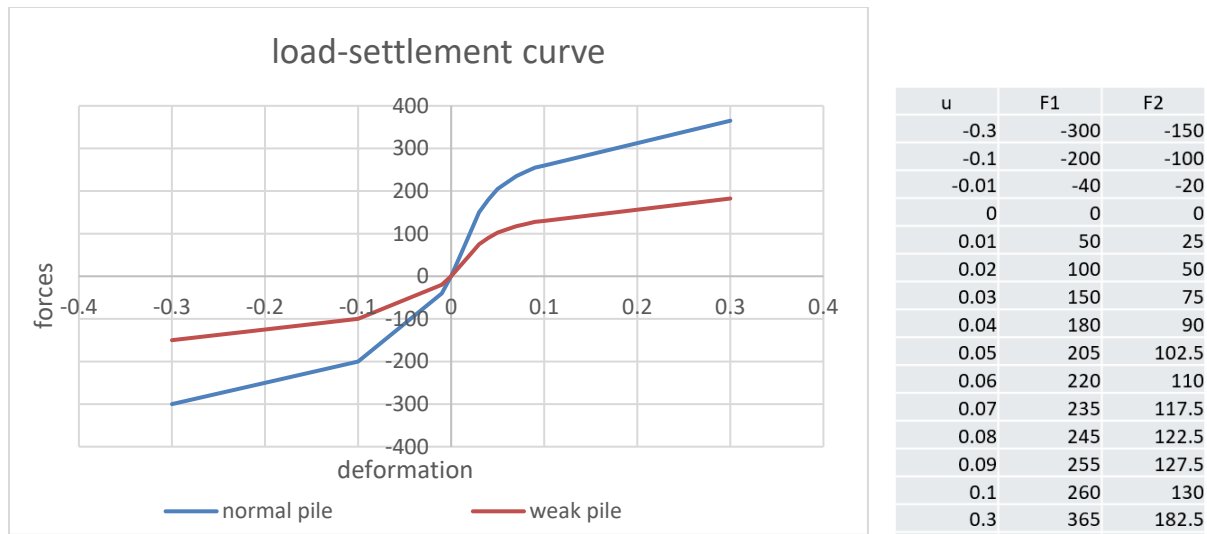


Figure 6.59 Load-settlement curves for nonlinear springs in example 12

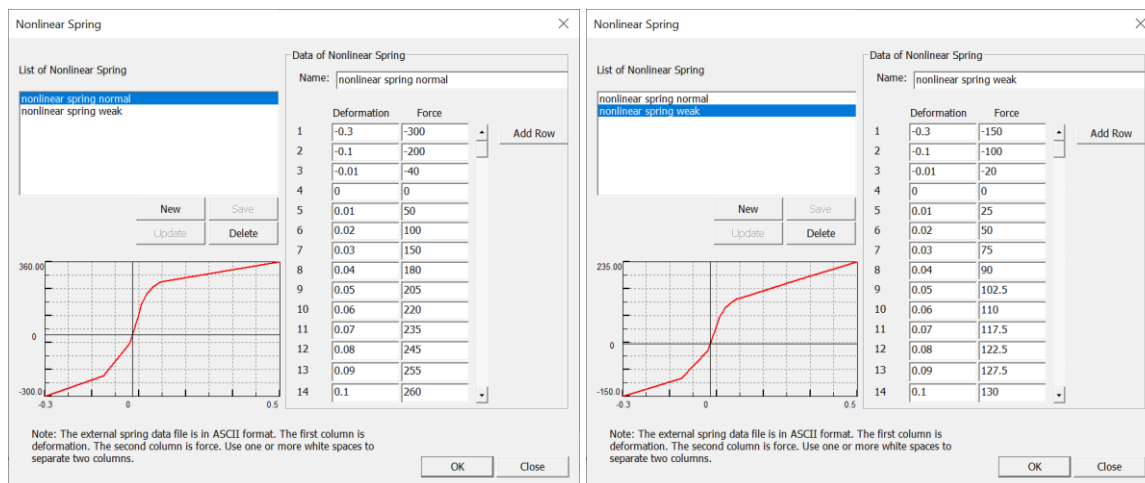


Figure 6.60 Define two nonlinear spring groups in example 12

Click the menu “**Input Data->Node Sets**” to define the first node set as shown in Figure 6.61. This node set includes node 21,137 and 53. Then define the second node set which has the right top spring at node 161. Click **OK** to exit.

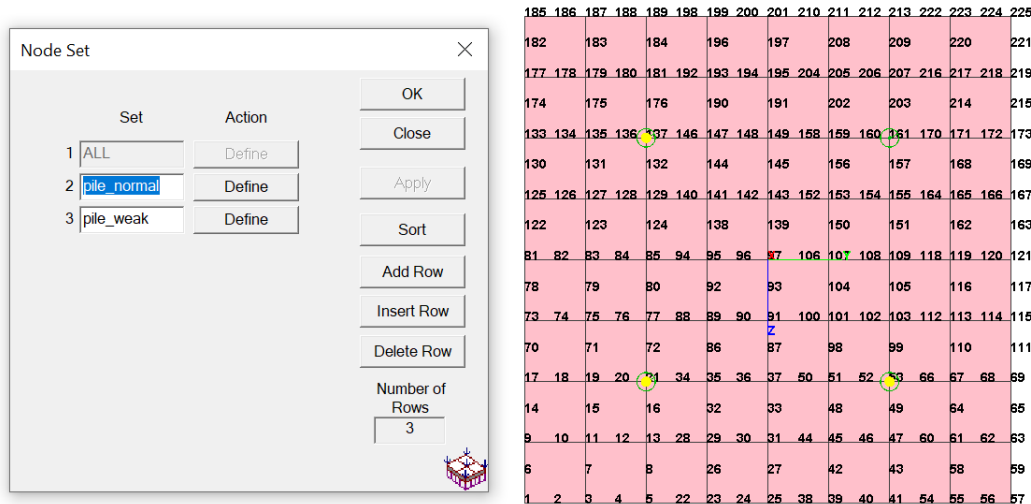


Figure 6.61 Define node set for normal piles with normal stiffness in example 12

Click menu **“Input Data->Nodal Stiffness->Use Nonlinear Spring”** to define two nonlinear stiffness groups. (Figure 6.62). Check the option **“Use NodeSet”**, then click the button **“Select Node Set”**. In the set selection dialog, select the node set **“pile_normal”**. Click **OK** to exit from the set selection dialog. Select **“nonlinear spring normal”** from the spring list. Click **Save** to finish the first nonlinear stiffness group. Similarly assign the node set **“pile_weak”** and **“nonlinear spring weak”** to the second nonlinear stiffness group. Note that all these stiffnesses will be applied to settlement direction along axis X. Click **Close** to exit.

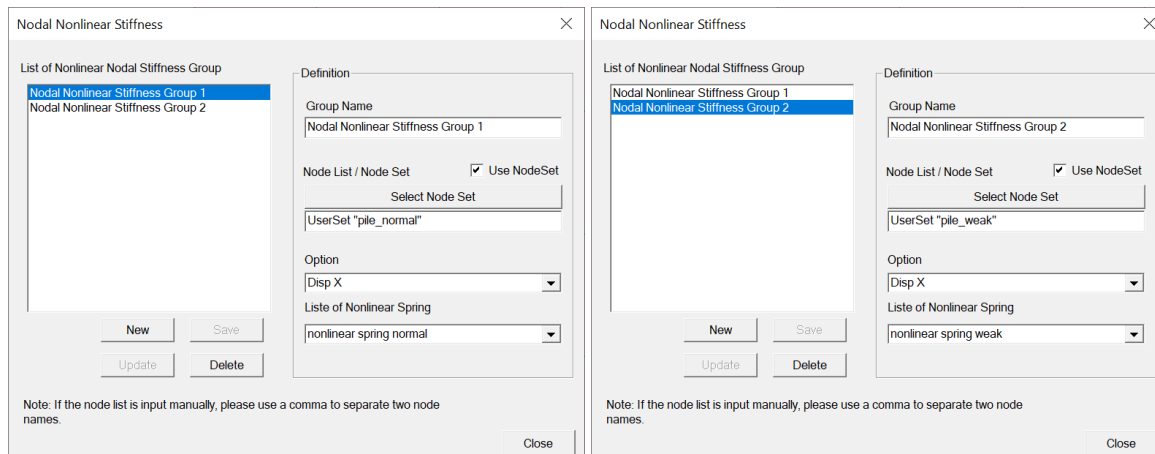


Figure 6.62 Define two nonlinear stiffness groups in example 12

Click the menu **“Input Data->Distributed Load”** to define uniform 250 distributed loads on elements 21~28,37~44 as own in Figure 6.63.

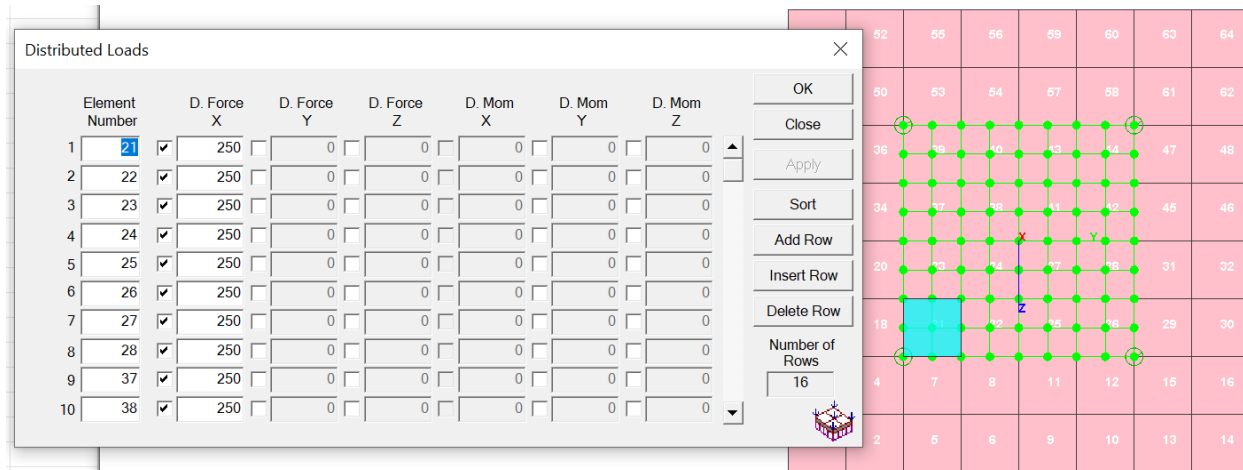


Figure 6.63 Define distributed element loads in example 12

6.13.3 Output Results Review

Click the menu “**Computations->Run Analysis**”. The message file will be opened automatically after calculation is finished. Since the data provided to the weak spring is limited to the settlement ≤ 0.3 , the calculation cannot converge as shown in Figure 6.64.

```

step      1   tcur   =    1.0000  dt =    1.0000

Current increment =    0.1000E+01
iter =      1 err1 = 0.1000E+01 d =  0.1175E+01 u = 0.1175E+01 rhs =  0.2322E+03
          err2 = 0.1000E+01 r =  0.2777E+00 ru= 0.2777E+00 rrhs=  0.0000E+00

Current increment =    0.1000E+01
iter =      2 err1 = 0.6299E+00 d =  0.1826E+01 u = 0.2899E+01 rhs =  0.1092E+03
          err2 = 0.7984E+00 r =  0.1077E+01 ru= 0.1348E+01 rrhs=  0.2666E-10

Error: Can not find spring force from given nonlinear profile for plate displacement =  0.15602E+00 at node   53
***** Error found during calculation. Analysis is terminated. *****

```

Figure 6.64 Error message if nonlinear spring does not have adequate range in example 12

Go back to the nonlinear spring definition dialog. Select the weak spring group. Click “**Add Row**” to add one more data line to the weak spring (Figure 6.65).

Click the menu “**Computations->Run Analysis**” again. This time the analysis is completed without issues.

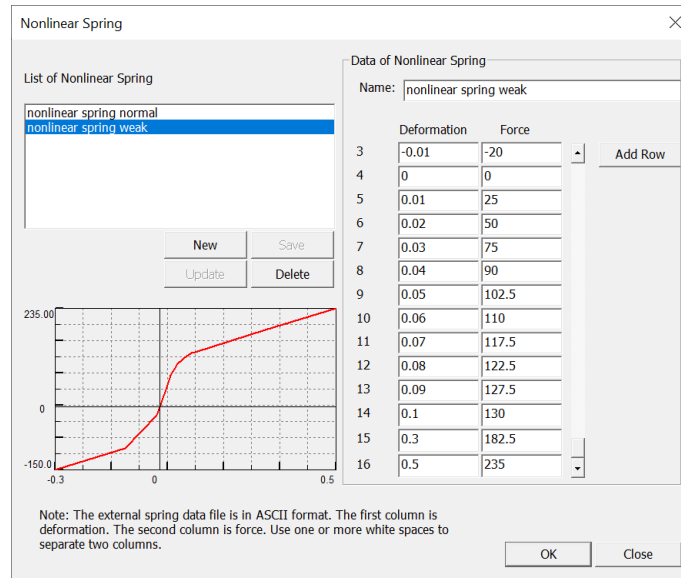


Figure 6.65 Extend data range to cover larger settlement in example 12

The contour of slab deflection is shown in Figure 6.66. Since the spring at node 161 is weaker than other springs, the whole slab inclines from node 1 to node 225 as shown in Figure 6.67.

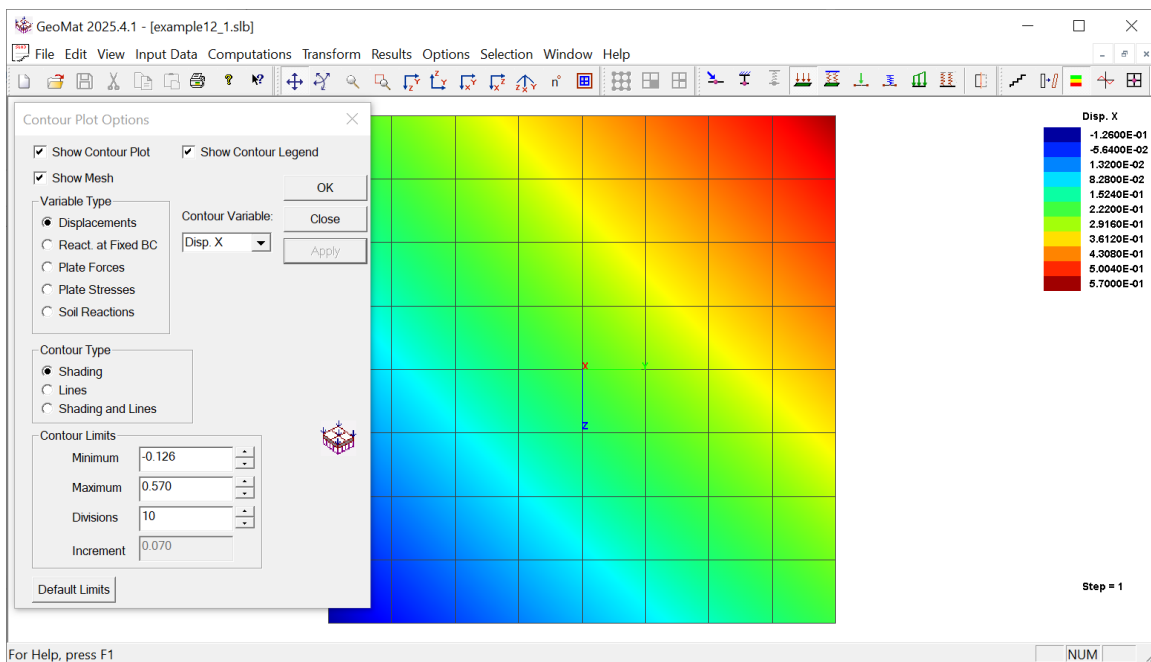


Figure 6.66 Contour of slab deflection in example 12

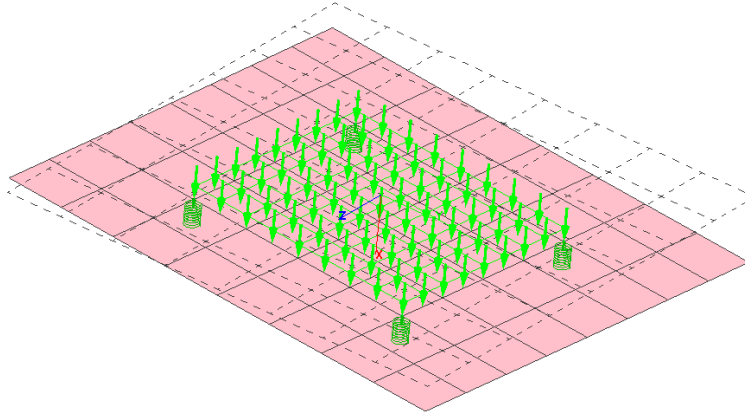


Figure 6.67 Deformed slab mesh in example 12

Click the menu “**Results->Node Results**”. Change the node number to 21 which locates at the left-bottom pile. Select variable type to “Soil Reaction”. The reaction provided by the nonlinear spring is 209. Its direction points upward as the negative direction of X axis.

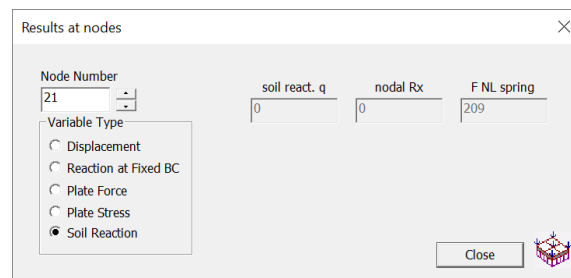


Figure 6.68 Access nodal reaction force from nonlinear spring in example 12

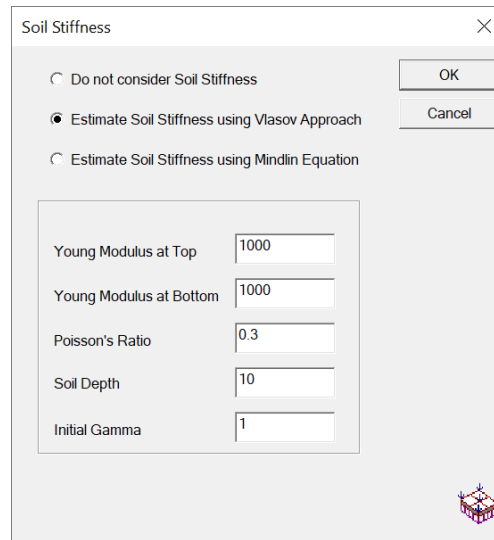
Settlements and reactions from nonlinear springs are listed in Table 6-3. The total load is $250 \times 2 \times 2 = 1000$ which agrees with the sum of all spring reactions.

Table 6-3: Settlements, Reactions and load verification in example 12

	Node 21	Node 53	Node 45	Node 49		Total load
Settlement	0.0526	0.224	0.224	0.401		
Reaction Force	-209	-291	-291	-209		
Sum of Reaction					1000	1000

6.13.4 Combination of Nonlinear Spring with Other Soil Stiffness

The nonlinear spring can be combined with others soil stiffness method like constant nodal stiffness or soil stiffness using Vlasov approach. Figure 6.69 shows parameters of the Vlasov soil stiffness which will be combined with nonlinear springs.



Soil Stiffness

☐ Do not consider Soil Stiffness
☒ Estimate Soil Stiffness using Vlasov Approach
☐ Estimate Soil Stiffness using Mindlin Equation

Young Modulus at Top: 1000
 Young Modulus at Bottom: 1000
 Poisson's Ratio: 0.3
 Soil Depth: 10
 Initial Gamma: 1

OK Cancel

Figure 6.69 Combine Vlasov soil stiffness with nonlinear spring in example 12

This combination can be applicable to an analysis when the pile cap contacts with soil bed. Loads from outstanding structure are carried by the soil bed and piles together. Three cases are studied here:

- model A: nonlinear springs to simulate piles only
- model B: Vlasov soil stiffness only
- model C: combination of nonlinear springs and Vlasov soil stiffness

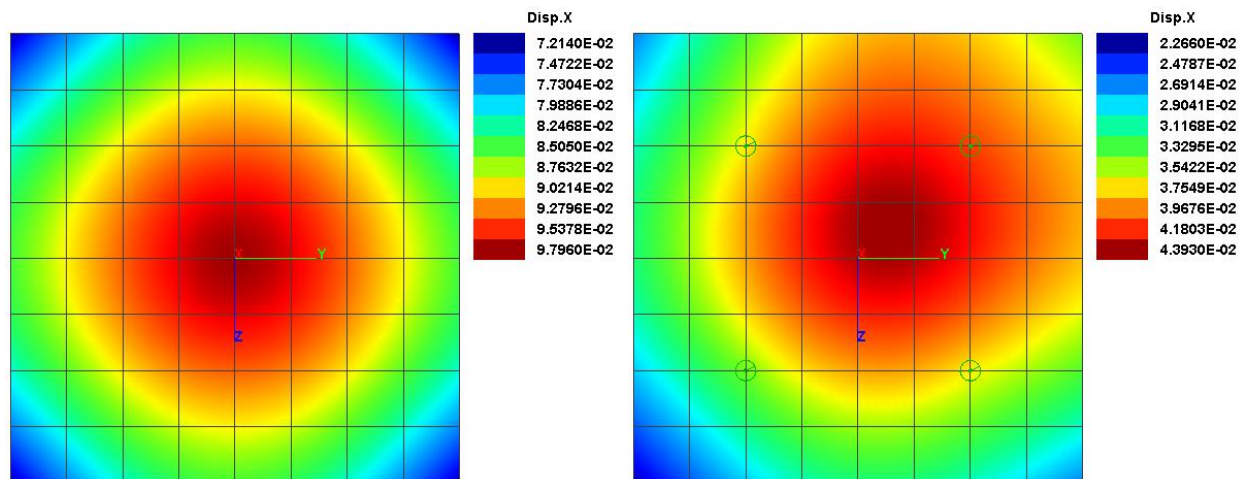


Figure 6.70 Contour of slab deflection from model B and C in example 12

Contours of deflection from model B and C are displayed in Figure 6.70. After four nonlinear springs are added into the model, the deflection at the slab center reduces from 0.0979 to 0.0439. Compared with Figure 6.67, the maximum deflection in model B and C occurs at the slab center instead of the right-top corner in the model A.

6.14 Example 13: Case Study – Pile-Raft Analysis

6.14.1 Problem Description

Yamashita and Yamada (1998) have reported a performance of a piled-raft foundation in stiff clay. The building is a five-story reinforced-concrete structure. The subsurface soils consist of a surface layer of alluvial Kanto Loam with a thickness of 5m, which is underlain by a layer of loose sand (between depths of 5 m and 14 m) and a thin layer of stiff clay between depths of 14 m and 16 m below the ground surface. A medium to dense sand layer was encountered at depths of 16m to 19m, which is underlain by stiff to hard clay with varying thickness. The soil profile and soil properties are summarized in Table below.

Layer/Material	Depth (m)	Young Modulus (MPa)	Cohesion, c (kPa)	Friction Angle, ϕ (degrees)
Kanto Loam	0.0 - 5.0	19.1	93	10
Loose Sand	5.0 - 14.0	48.4	14	20
Stiff Clay	14.0 - 16.0	87.9	48	10
Medium Sand	16.0 - 19.0	123.0	101	10
Stiff clay	19.0 - 29.0	71.4	350	10
Hard Clay	29.0 - 39.0	47.2	350	10
Hard Silt	39.0 - 42.0	87.9	350	10
Dense Sand	40.0 - 50.0	218.6	350	10

The pile length was approximately 15.8 m and the tips of the piles are seated in the medium to dense sand layer. The piles were constructed by inserting a H-shape of steel member into a pre-drilled borehole filled with mixed-in-place soil cement. The predicted ultimate bearing capacity of the composite pile is 2.85 MN for a diameter of 0.8m and 2.52 MN for a diameter of 0.7m. The reinforced-concrete raft was 28 m x 27 m in plan and 2.4 m thick. A total of 20 piles were placed at each column position, 10 of which were 0.7 m in diameter and the other 10 were 0.8 m in diameter. The layout of the piles is shown in Figure 6.71.

It should be noted that the pile types of P1 and P2 have the same length (15.8 m) and diameter (0.8 m), but different steel H-shape inserts (414x405x18x28 for P1 and 400x400x13x21 for P2). The pile types of P3 and P4 also have the same length (15.8 m) and diameter (0.7 m), but different steel H-shape inserts (350x350x12x19 for P3 and 300x300x10x15 for P4). The ultimate axial capacities reported by Yamashita and Yamada (1998) are 2.85 MN for the piles (P1 and P2) of 0.8 m in diameter and 2.52MN for the piles (P3 and P4) of 0.7 m in diameter. The axial stiffness at the pile top also depends on the structural stiffness of the pile material. The H-shape inserts are the main structural elements. The axial stiffness (AE) of H-shape inserts is 5,908,000 kN for P1, 4,374,000 kN for P2, 3,478,000 kN for P3, and 2,396,000 kN for P4. The load-vs.-settlement curves for all four types of piles are computed by Program SHAFT and the settlement curves under compressive loads are presented in Figure 6.72. The load-vs.-movement curves under compressive and tensile loads will be used to represent the nonlinear axial stiffness of the pile at the raft.

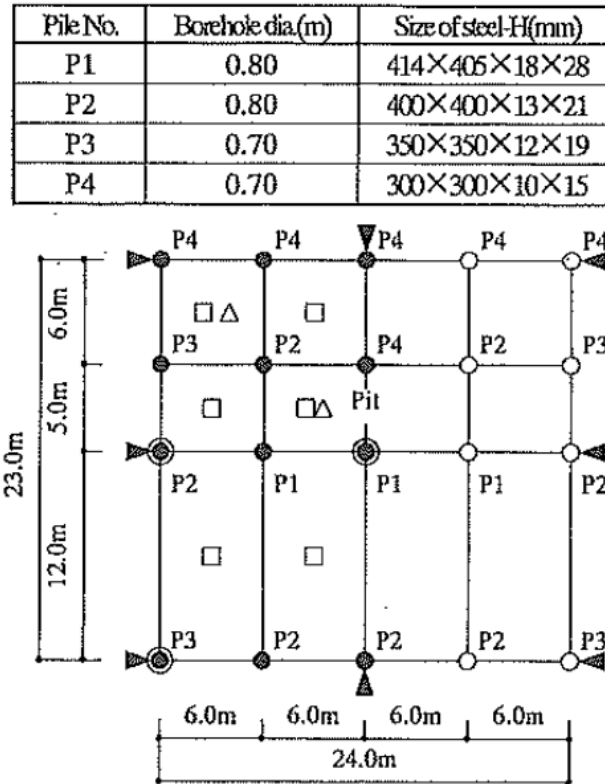


Figure 6.71 Foundation plan for the piled raft at Urawa City site in example 13a

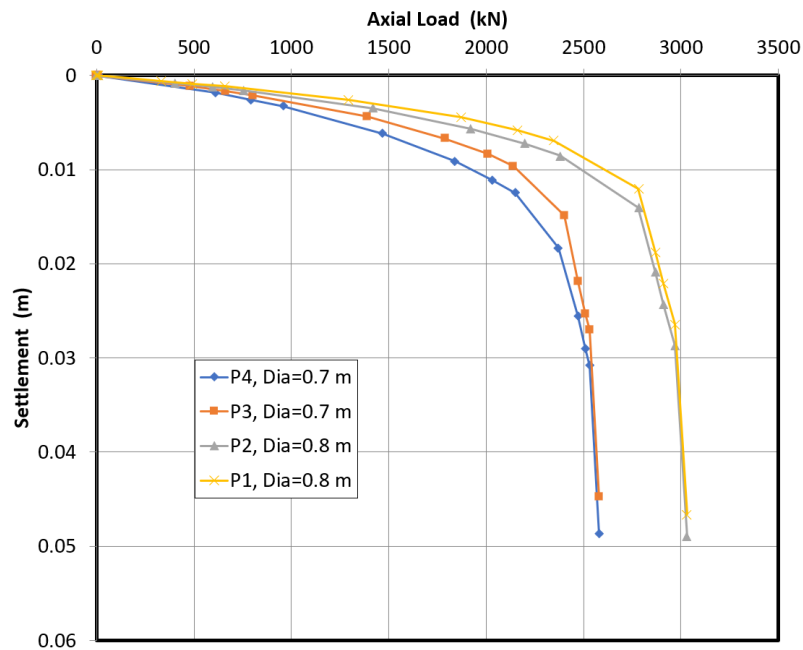


Figure 6.72 The nonlinear axial stiffness of piles at the piled raft at Urawa City site in example 13a

The final building load on the foundation is reported as 47.5 MN. The load from each column is presented in Figure 6.73 and the total of all column loads is 46.3 MN, which is close to 47.5 MN.

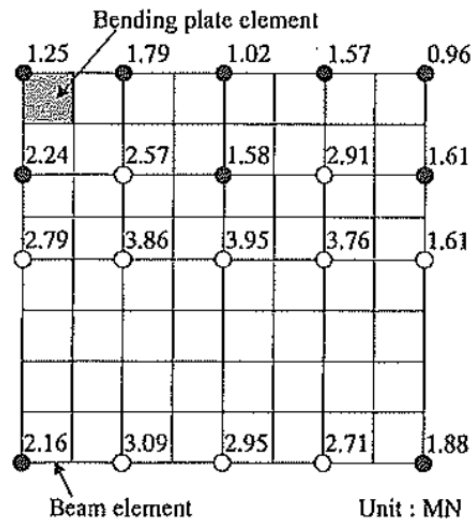


Figure 6.73 The load from each column at the piled raft at Urawa City site in example 13a

6.14.2 Modeling Instruction

To compute this problem using *GeoMat*, use the **New from template** option in the **File** menu. In the foundation type selection dialog, select “Circular or Rectangular Conventional” to create a 23 by 24 slab with 1 m x 1 m for element size in each direction. Keep the default “Quadratic 8-noded” element type and “2x2” integration rule. Click **OK** to generate the mesh using constant units (kN and meters in this example).

Click the menu “**Input Data->Material Properties**” to define the slab material parameters: Young’s modulus $E = 2.15 \times 10^7$, Poisson’s ratio $\nu = 0.3$ and the slab thickness = 2.4 m. Click **OK** to exit from the material property window.

The next step is to define nonlinear springs. Click menu “**Input Data->Nonlinear Spring**” to define four new spring groups. The dialogs to define four nonlinear springs are shown in Figure 6.78. Click the button “**New**” to create a new spring data group. Users can input data manually or click “**Open File**” to import the spring data file such as “P1.txt” for each type of piles. Click **OK** to exit.

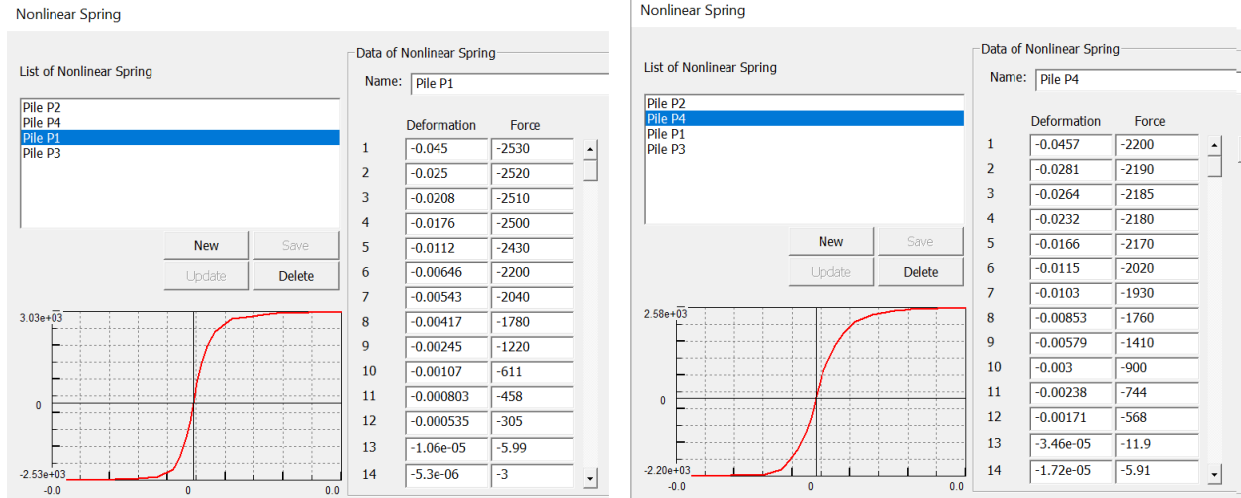


Figure 6.74 Extend data range to cover the nonlinear axial springs (examples for Piles P1 and P4)

Click menu **“Input Data->Nodal Stiffness->Use Nonlinear Spring”** to define four groups of nonlinear stiffness for Piles P1, P2, P3, and P4 (Figure 6.79). Click **“New”** to add **“Nodal Nonlinear Stiffness Group 1”** in the left dialogue window. Check the option **“Use Node List”**, then click the button **“Select Node List”** and enter the nodal numbers separated by a comma. Select **“Pile P1”** from the List Nonlinear Spring list. Click **Save** to finish the first nonlinear stiffness group. Similarly, assign the node List **“Nodal Nonlinear Stiffness Group 2”** and **“Pile P2”** to the second nonlinear stiffness group and repeated for P3 and P4. Note that all these stiffnesses will be applied to movement direction along axis X. Click **Close** to exit.

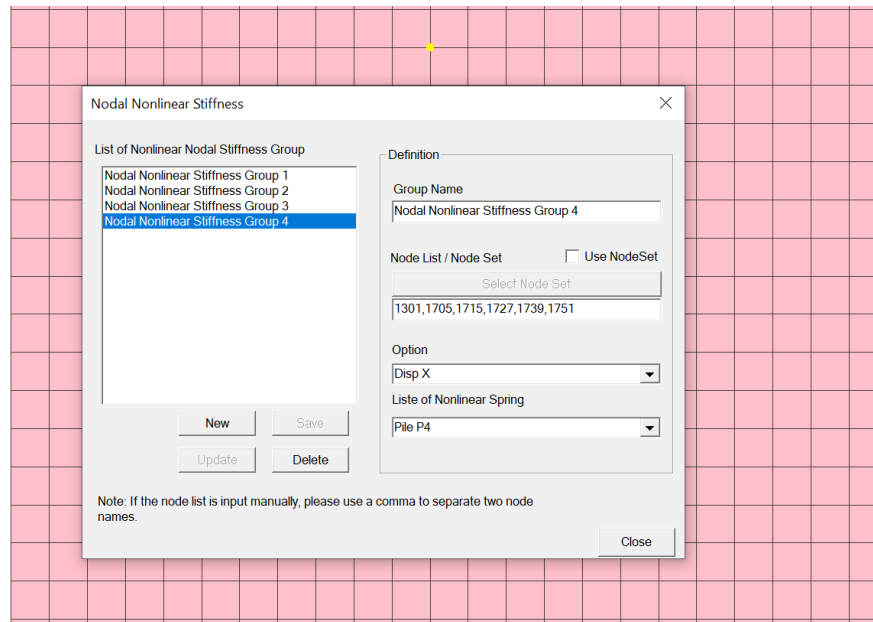


Figure 6.75 Define four nonlinear stiffness groups in example 13a

Click the menu “**Input Data->Nodal Load**” to define column loads as shown in Figure 6.80. Click “New” and enter the column load in units of kN for Force X at the associated nodal point entry. Repeat the same steps until all the column loads are entered as shown in Fig. 6.76.

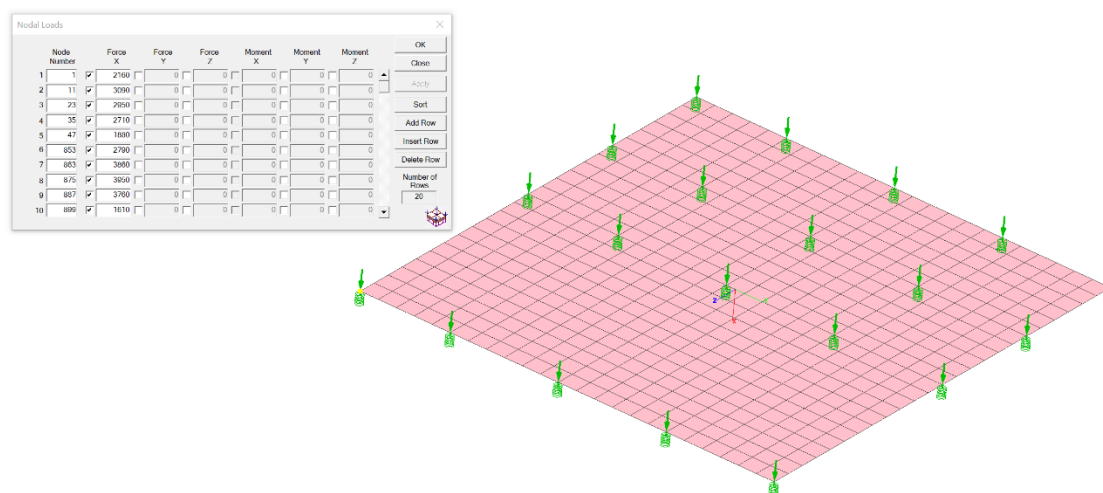


Figure 6.76 Enter column loads in example 13a

6.14.3 Combination of Nonlinear Spring with Other Soil Stiffness

The nonlinear spring for piles can be combined with the subgrade support from the soil beneath the slab. The soil stiffness calculated by using Vlasov approach was assigned. shows parameters of the Vlasov soil stiffness which will be combined with nonlinear springs for piles.

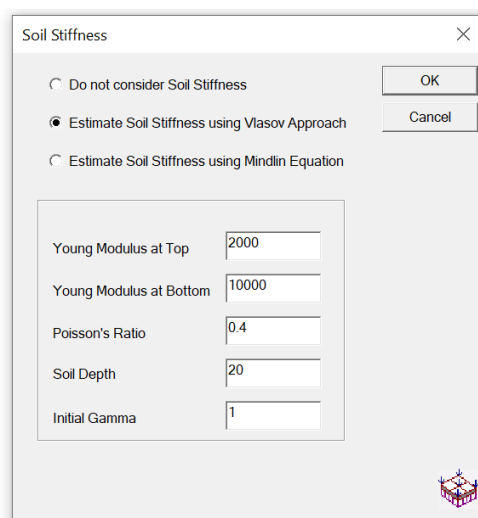


Figure 6.77 Combine Vlasov soil stiffness with nonlinear spring in example 13a

6.14.4 Output Results Review

Click the menu “**Computations->Run Analysis**”. The message file will be opened automatically after calculation is finished. The contour of slab vertical settlement (or movement) is shown in Figure 6.78. Since the column loads were unevenly distributed the vertical settlement at the lower-left zone is greater than those at the upper-right zone of the foundation. The maximum vertical settlement is approximately 1.48 cm which is in a close agreement with the measured settlement by Yamashita and Yamada (1998) as shown in Figure 6.79.

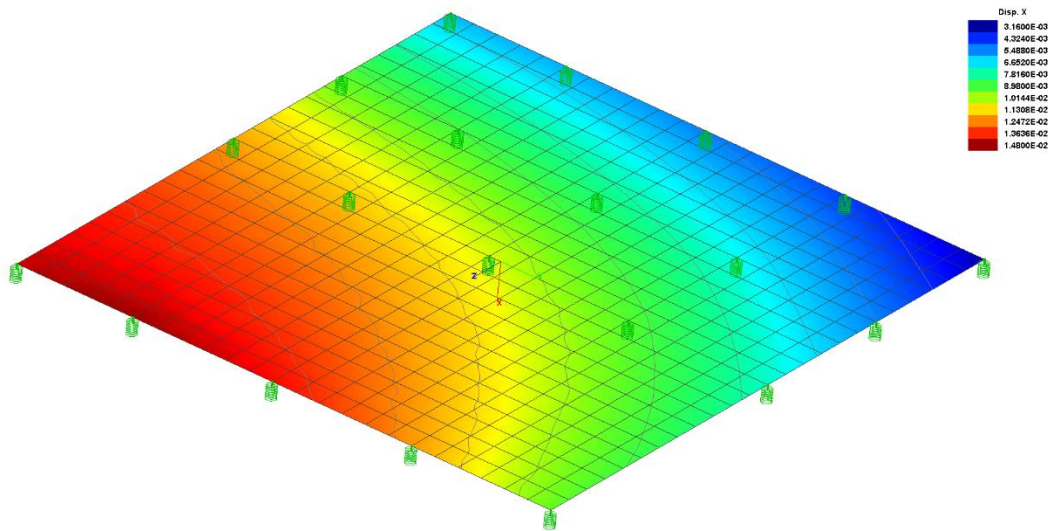


Figure 6.78 Contour of slab settlement using column loads in example 13a

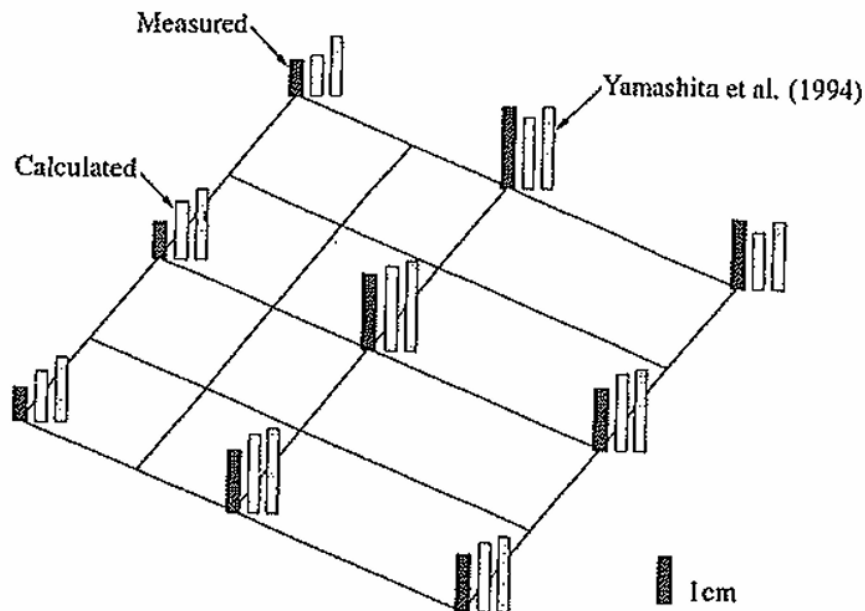


Figure 6.79 Slab settlement reported by Yamashita and Yamada (1998)

Alternatively, a uniform distributed load of $86.05 \text{ kPa} = 47500 \text{ kN}/(24 \text{ m} \times 23 \text{ m})$ can be applied uniformly on the slab instead of using the column loads (Example 13b). The contour of slab vertical settlement (or movement) and undeformed meshes are shown together in Figure 6.80, which has a better agreement with the measured data.

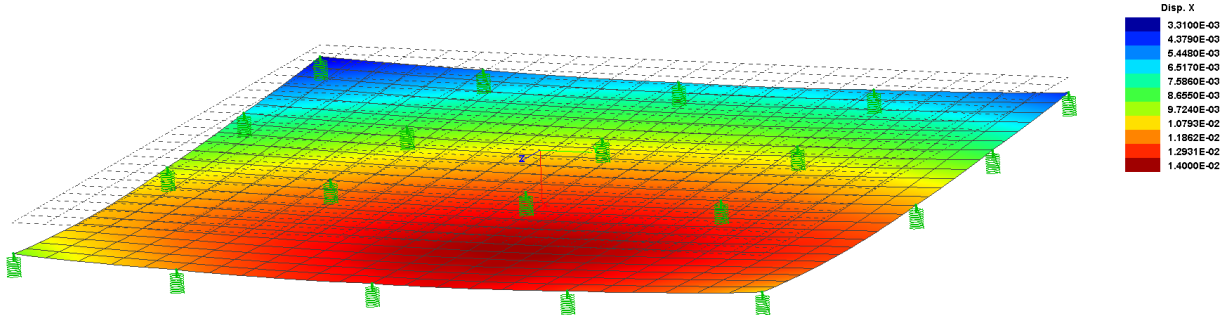


Figure 6.80 Contour of slab settlement using averaged distributed loads in example 13b

The user can define the nodal path for easy view and comparison of calculated results at different locations. Click “Nodal Path” under the input data menu to define each desired nodal path as shown in Figure 6.81.

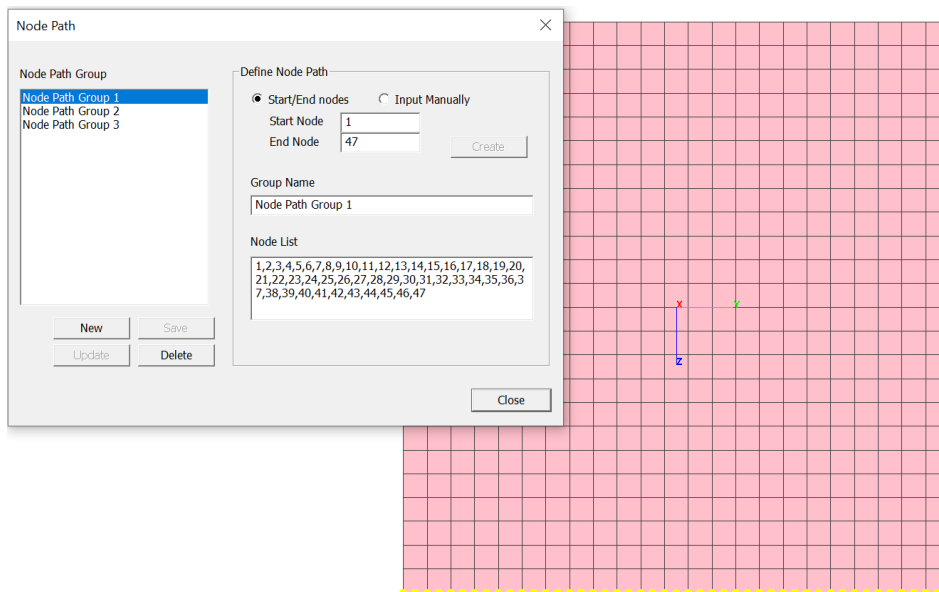


Figure 6.81 Define node path for X-Y plots in example 13a

The Sub menu “X-Y Curve Plot” under Results Menu can be used to review calculated forces from the piles at different location (as specified by the nodal path) as shown in Figure 6.82. user can define the nodal path for easy view and comparison of calculated results at different locations. The user can also

view the deflection, bending moment, and shear along the cutting line on the slab as shown in Figure 6.83.

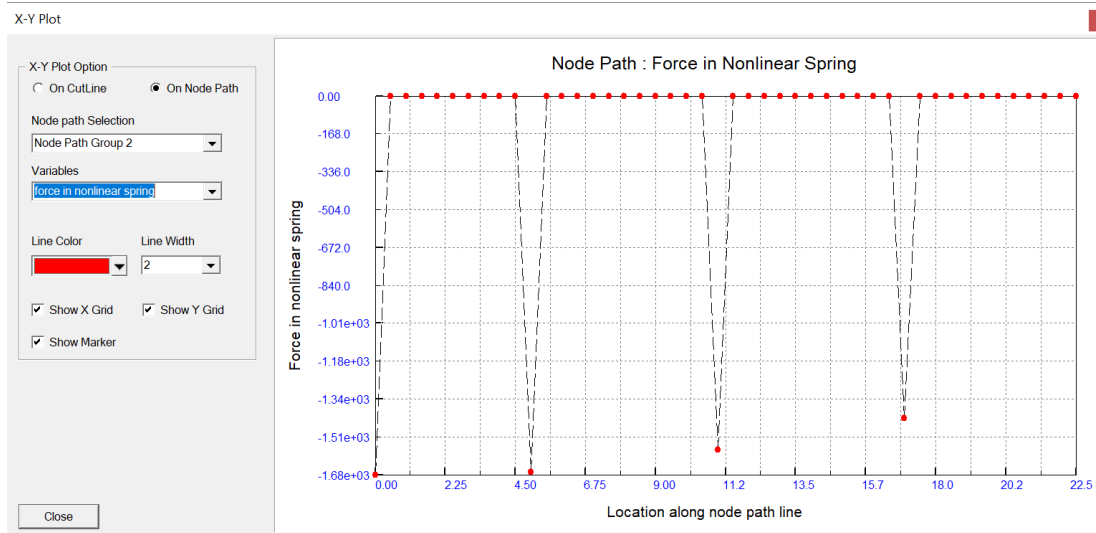


Figure 6.82 Force in nonlinear springs at the specified nodal path in example 13a

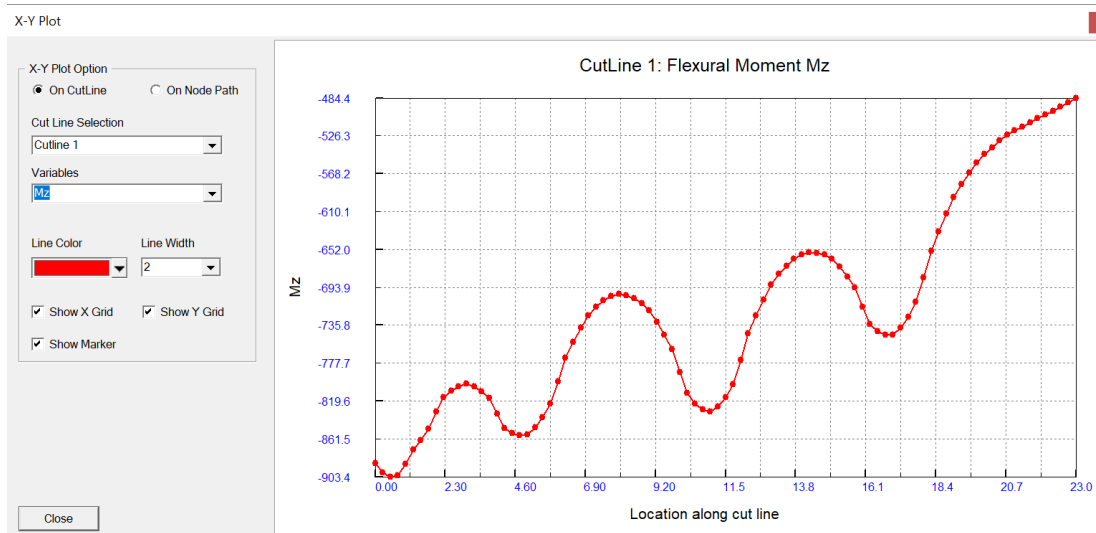


Figure 6.83 Bending moment of Mz in the slab at the specified cutting line in example 13a

6.15 Example 14: L-shape raft on soil

This example is modified from example 7 to demonstrate the ability to create irregular slab shape in *GeoMat*.

6.15.1 Modify slab from example 7

Open the model in example 7 and save it as “Example14.slb”. Click the menu “**Input Data->Element Connectivity**” to open the element connectivity window as shown in Figure 6.84. Click “**Option->Turn on Element Number**” to show element numbers. Click any cell for element 8. This element is highlighted in cyan. Click “**Delete Element**”. The color of element 8 will change to dark grey which represents this element will be deleted later. Similarly, delete elements 9~16, 26~32, 40~48, 56~64, 72~80 and 88~96. Click “**Apply**” to remove selected elements in the view and update model database. After applying element deletion, element numbers are reordered so that they are different from the original model. Element number starts from 1 and is continuous to the last element. Users may turn on the node numbers to compare node numbers before and after deleting elements. Node numbers are also renamed to keep continuity.

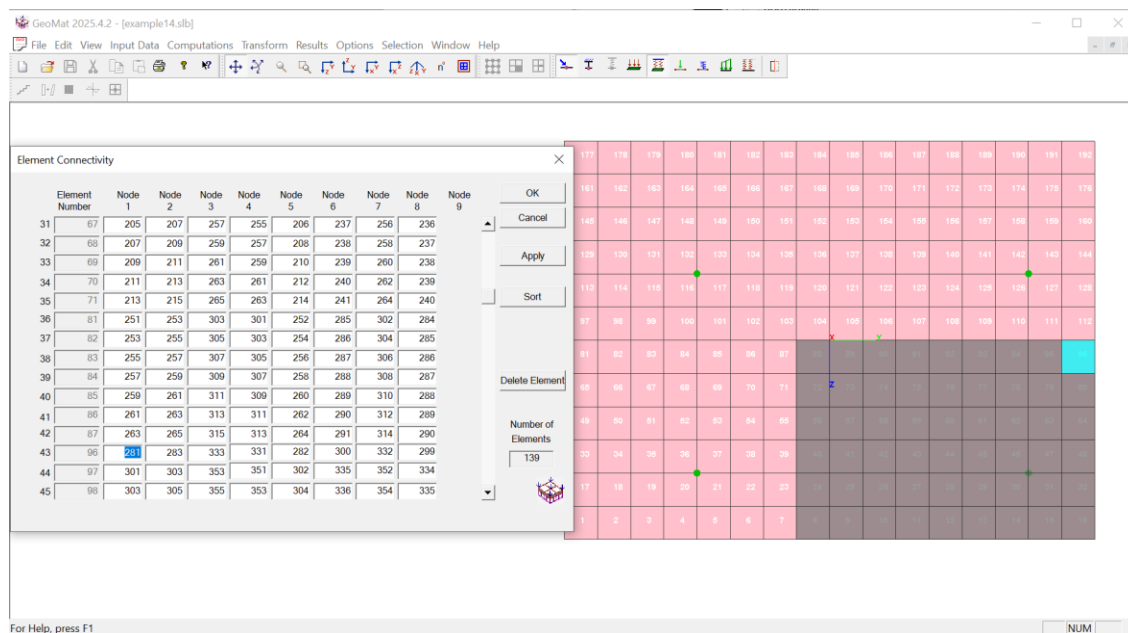


Figure 6.84 Delete elements to change slab shape

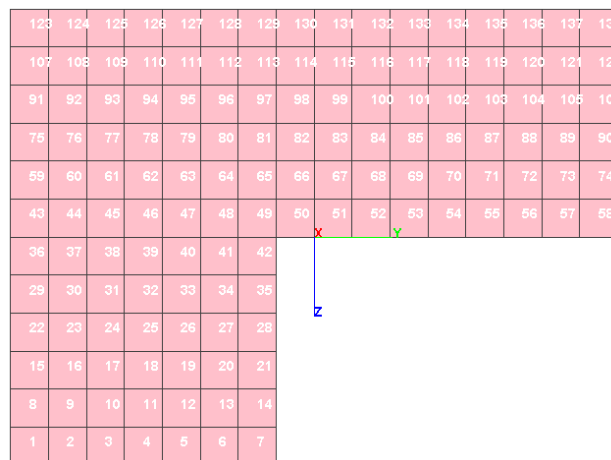


Figure 6.85 Irregular L-shape slab and renamed element number

When elements are deleted, nodes which don't belong to any element will be removed automatically. Distributed stiffness and point loads on deleted elements are also removed to keep consistency with the new mesh.

Click the menu “**Computation->Run Analysis**”. The contour of deflection is shown in Figure 6.86.

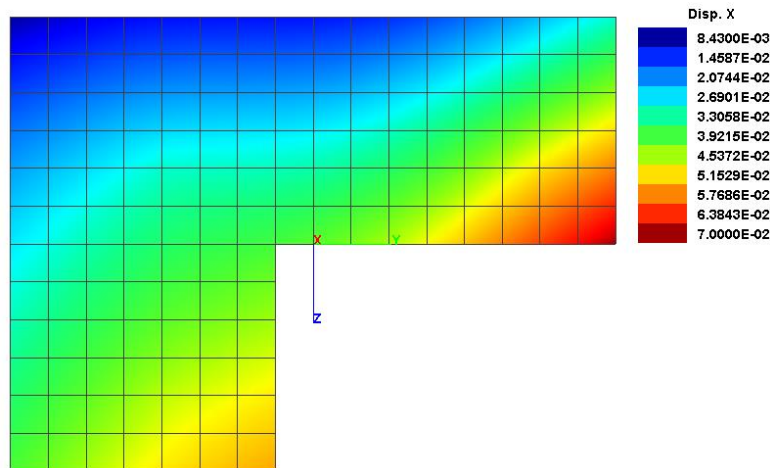


Figure 6.86 deflection contour of L-shape slab

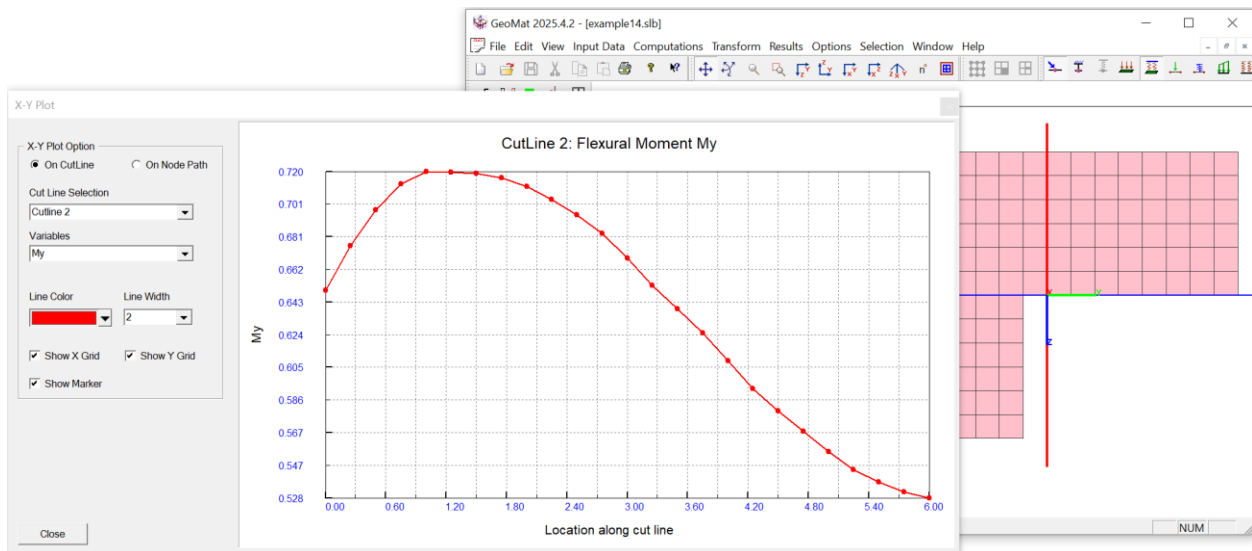


Figure 6.87 My on cutline 2 in example 14

The moment M_y along the cutline 2 is plotted in Figure 6.87. Remaining slab has 6 m vertically on the right side which is consistent with the range of the horizontal axis.

6.16 Example 15: Import external mesh

6.16.1 Problem Description

This example presents how to import an external mesh for the analysis on a slab with irregular shape. Concentrated loads are applied to simulate loads transferred from columns. The geometry is shown in Figure 6.88. Concentrated load $P=500$ kN. The distributed soil stiffness is assumed to be 400 kPa.

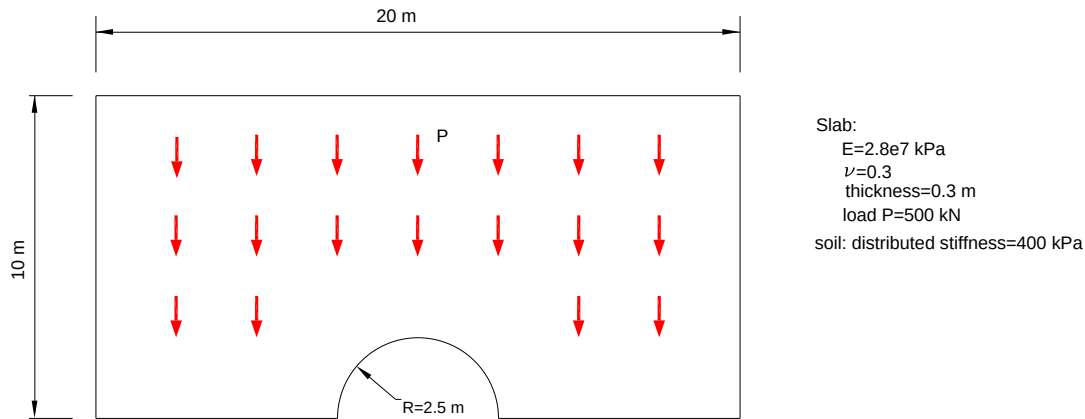


Figure 6.88 Geometry and parameters of slab in example 15

6.16.2 Import mesh from an external mesh file

Click the menu “**File->Import External Mesh**”. In the file search window, select the mesh file “example_15_ext.mesh”. The slab is shown as in Figure 6.89.

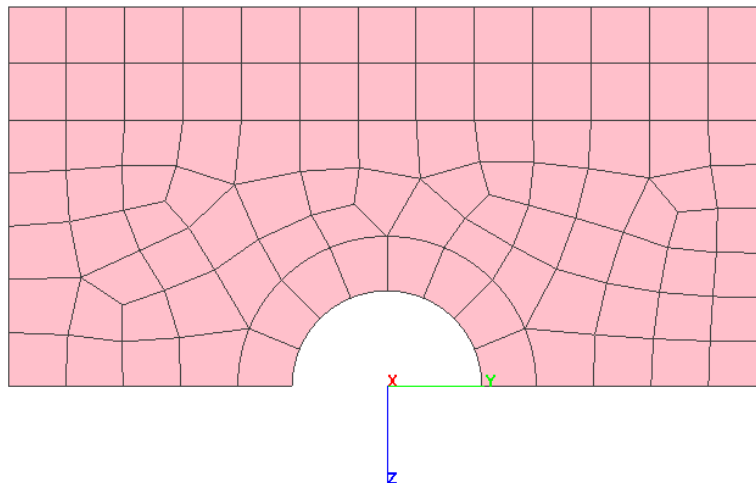


Figure 6.89 Mesh of slab mesh imported from an external mesh file

Click the menu “**Input Data->Material Properties**” to define the slab material as shown in Figure 6.90.

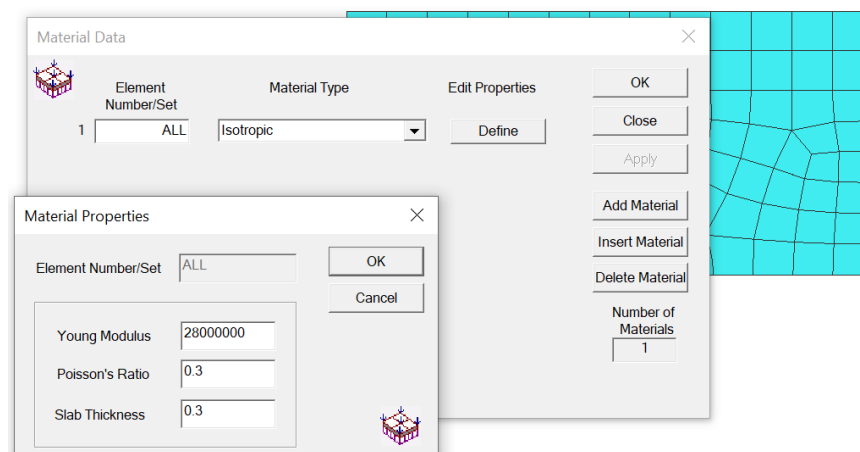


Figure 6.90 Define slab material in example 15

6.16.3 Model Instruction

Click the menu “**Input Data->Concentrated Load**” to input point loads at specified coordinates as shown in Figure 6.91. Load $P=500$ kN. Spacing between these column loads P is 2.5 m in both horizontal and vertical directions.

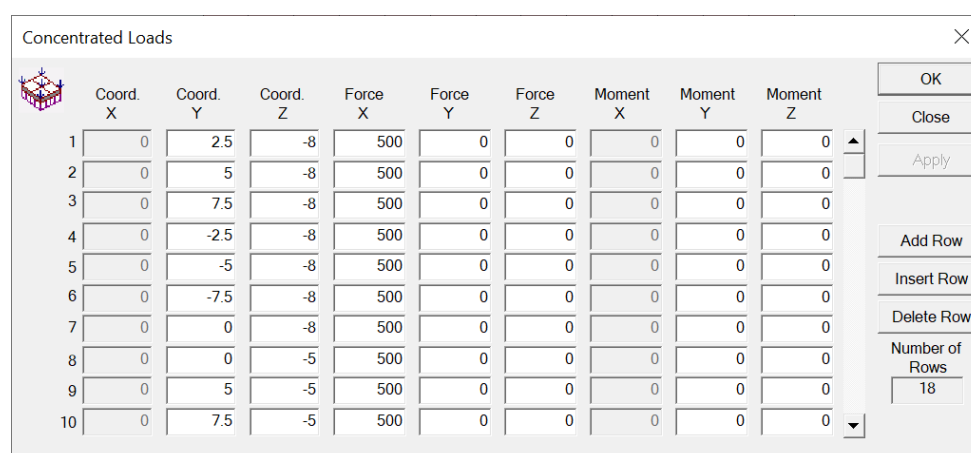


Figure 6.91 Define concentrated load on slab in example 15

Click the menu “**Options->Concentrated Loads**” to turn on display of concentrated loads as shown in Figure 6.92. Green spots represent user defined concentrated loads.

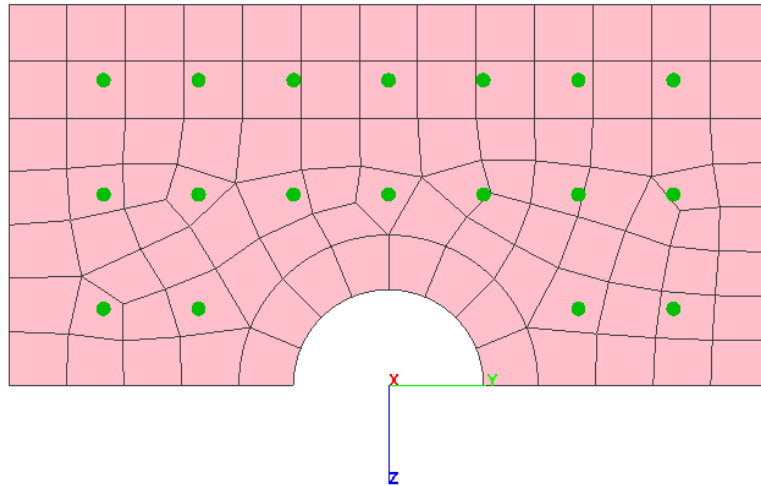


Figure 6.92 Display concentrated loads on slab

6.16.4 Out Results

Click the menu “**Input Data->Distributed Stiffness**” and define distributed soil stiffness $k=400$ kPa. Click “**Computations->Run Analysis**”. The contour of deflection is shown in Figure 6.93. The center of all concentrated loads moves towards the top slab edge. The slab has more deflection near the top edge.

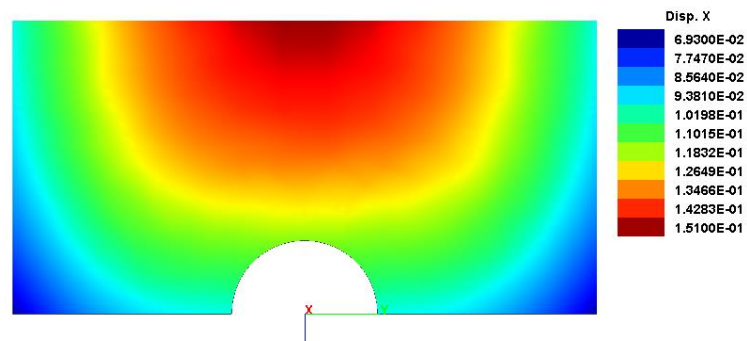


Figure 6.93 Contour of deflection in example 15

7 References

Weaver, William Jr and Johnston, Paul R. "Finite Elements for Structural Analysis", Prentice-Hall, Inc. Engelwood Cliffs, New Jersey, 1984

Bowles, J.E., "Foundation Analysis and Design", McGraw-Hill Book Co., New York, 1982.

Hudson, W. R. and Matlock, H. "Discontinuous Orthotropic Plates and Pavement Slabs", Center for highway research, university of Texas, Austin, 1966

Blaauwendraad, Johan, "Plates and FEM – Surprises and Pitfalls", Springer, 2010

Yamashita, K. and Yamada, T., "Simplified method for analyzing piled raft foundations," *Deep Foundations on Bored and Auger Piles*, ed. Van Impe & Haegeman, Balkema, Rotterdam, 1998.