

Redefining simplicity for geotechnical engineers

GeoFEA[®]
VERSION 9.0

QUICK START MANUAL

This is a quick start manual for the introduction of basic features of GeoFEA[®].

Table of Contents

1	Introduction	4
2	Installation	4
3	Launching GeoFEA	4
4	Creating a new project	5
5	Elements of user interface	6
6	Creating a geometry	9
6.1	Edge tools	9
6.1.1	Creating a point	10
6.1.2	Creating a straight line	10
6.1.3	Creating a circle	11
6.1.4	Creating a freeform curve	11
6.2	Creation tools	12
6.3	Combination tools	13
6.4	Modification tools	14
6.5	Transform tools	16
6.6	Boolean tools	18
7	Generating a mesh	19
8	Assigning material properties	21
9	Assigning boundary conditions	24
9.1	Assignment of fixities	24
9.2	Assignment of pore-pressure boundary conditions	25
9.3	Assignment of loads	26
10	Staged analysis	27
10.1	Setting up the project	28
10.2	Creating the geometry	28
10.3	Generating the mesh	29
10.4	Assigning the material properties	29
10.5	Initial conditions	30
10.5.1	Assigning fixities	31
10.5.2	Assigning in-situ stresses	31
10.5.3	Exclusion/Inclusion of the geometry	31
10.6	Excavation of 1 st layer	32
10.7	Installation of 1st layer of strut	33
10.8	Excavation of other layers	33

11	Solution	34
11.1	Solver options	34
11.1.1	For symmetric stiffness matrix	35
11.1.2	For nonsymmetric stiffness matrix	35
11.1.3	Auto Select	36
11.2	Geometry update options	36
11.3	Out-of-balance load options	36
12	Post processing	37
12.1	Deformed shape	37
12.2	Contour plot	37
12.3	Vector plot	38
12.4	Bar/Beam plot	39
12.5	Line plot	40
12.6	Point plot	40
13	Known Issues	41

1 Introduction

This is the quick start manual of GeoFEA® version 9.0 for the introduction of basic features of the software to users. GeoFEA® is a finite element software for the two- and three-dimensional analysis of geotechnical problems with a wide range of linear and nonlinear constitutive models for the materials and numerous solver options for different types of the problems. This quick start manual mainly focuses the features for two-dimensional analysis only, although many operations are similar for both two- and three-dimensional analyses.

2 Installation

If you received the GeoFEA® product on DVD, insert the DVD in your computer's DVD drive; the installation should start automatically. If it does not, open the top-level folder of the DVD drive in Windows Explorer and double-click on the executable file (**GeoFEA_Setup_32.exe**). Or, you may download the executable file directly from the website www.geosoft.sg. Double-click on the executable file to begin the installation and follow the instructions on the screen.

3 Launching GeoFEA

After installation, go to **Start | All Programs** and select GeoFEA 9.0 to launch it. At the start of the software, you may be prompted to key in the serial number (for the first time only) and activate it (Figure 1) if you have bought the software or you may continue to evaluate the Trial version of the software free by hitting **Continue Trial** for a limited period of time. If you have decided to buy this software, you can proceed with **Buy License** button. This will direct you to to contact us on our website www.geosoft.sg.



Figure 1. Starting window.

4 Creating a new project

A **project** implies a finite element analysis session with a specific modeling and analysis data set. The data set for a project is stored completely in a single file. This file is termed here as project file. The project file has **.geoproj** extension. A project can be saved at any time and the work can be resumed later on opening the existing project file.

1. To create a new project, click on **Start | Create** command.
2. To open an existing project, click on **Start | Open** command.

A new window will pop-up to set up the dimension and type of analysis, and units to be used for the new project. The choice of the analysis dimension and type is dependent on the approaches to the finite element analysis of the problem to be carried out. Select the suitable parameters for the analysis from the drop-down lists.



Figure 2. New project set up.

GeoFEA® allows users to conduct total stress, effective stress, and consolidation analyses.

There are three categories in **Analysis Dimension**:

- Plain strain
- Axisymmetric
- 3 Dimensional

There are two categories in **Analysis Type**:

- Drained/Undrained
- Consolidation

For **Units**, the following four sets of units are available to be used.

	1	2	3	4
Length	m	mm	mm	ft
Force	kN	N	mN	lbf
Time	sec	sec	sec	hr
Pressure	kN/m ²	N/mm ²	mN/mm ²	lbf/ft ²
Unit weight	kN/m ³	N/mm ³	mN/mm ³	lbf/ft ³
Permeability	m/s	mm/s	mm/s	ft/hr

5 Elements of user interface

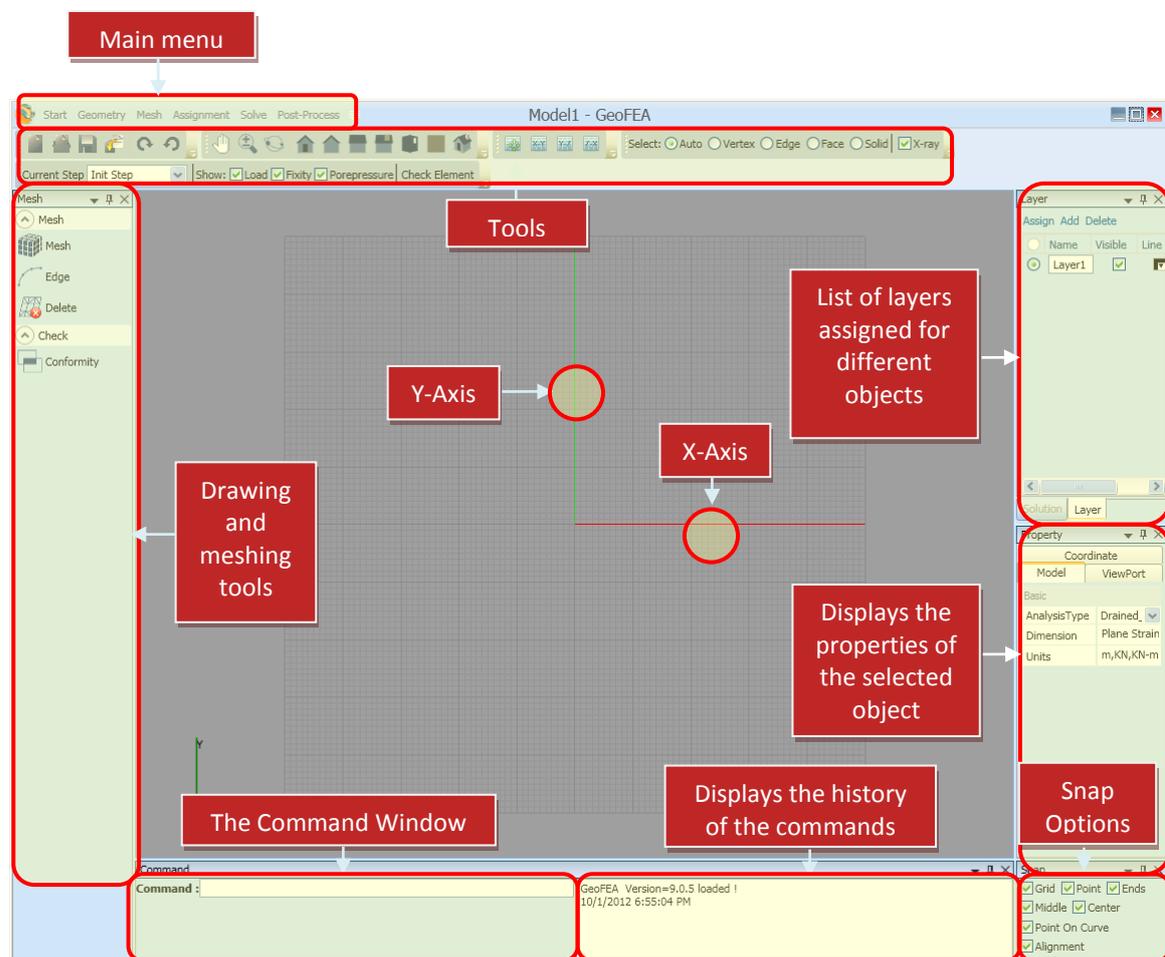


Figure 3. Working window.

The main menus are located on the top-left corner of the working window. The menu items (**Start**, **Geometry**, **Mesh**, **Assignment**, **Solve**, and **Post-Process**) are logically placed so as to sequentially follow the steps of an analysis. A drop-down button beside the main menus allows the users to turn on/off the component windows (e.g. **Geometry**, **Solution**, and **Property**) and to check the license or updates of the software. Usually the update is automatically checked when you launch the software. The other user interface tools are

listed below the main menu. Mouse-hovering on these icons shows their names. They are as shown in Figure 4.

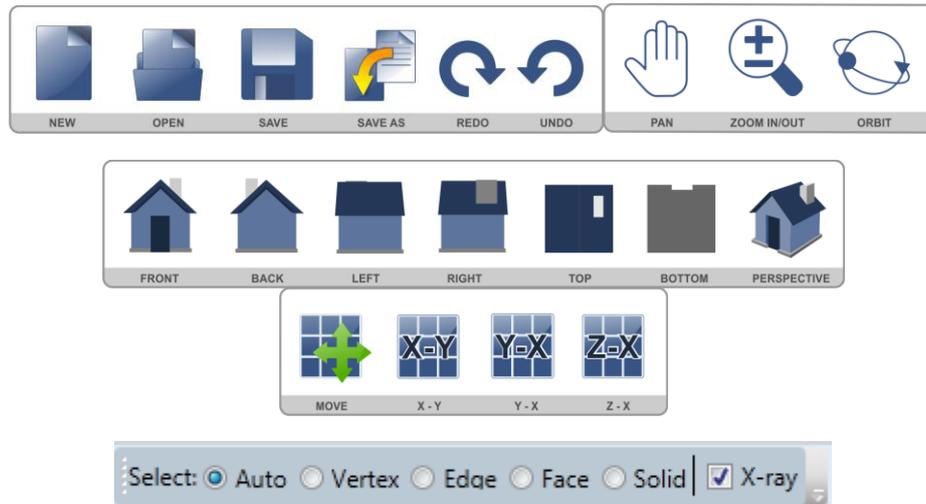


Figure 4. User interface tools.

The name for each icon is shown below it in Figure 4. A brief description of each icon is as follows:

1. **New:** To exit from the running project and open a new project.
2. **Open:** To open an existing project.
3. **Save:** To save the current project.
4. **Save As:** To save the project with a new/different name.
5. **Redo:** To do a command over again.
6. **Undo:** To reverse a command. The opposite of **Undo** is **Redo**.
7. **Pan:** To move the window view so as to follow the Pan icon.
8. **Zoom In/Out:** To enlarge/shrink the viewing. Zooming scale can be adjusted on command window.
9. **Orbit:** To rotate the viewing of objects in different angles with reference to origin. See the command window for hints.
10. **Front:** To show the front view of the objects (or finite element model).
11. **Back:** To show the back view of the objects.
12. **Left:** To show the left view of the objects.
13. **Right:** To show the right view of the objects.
14. **Top:** To show the top view of the objects.
15. **Bottom:** To show the bottom view of the objects.
16. **Perspective:** To view the objects in a perspective angle. [Note: The coordinate axes at the bottom-left corner will also rotate according to different views/rotation angle.]
17. **Move:** To move the coordinate axes to a different location.
18. **X-Y, Y-Z, Z-X:** To turn on/off the grid planes, respectively.
19. **Select tools:** Select tools are for selecting different types of elements of a geometry/mesh. These elements are also called objects. For example, when the select mode is in **Auto**, it will automatically decide the selection of an object when the cursor is placed near the object. Similarly, **Vertex** is for the selection of points/nodes, **Edge** is for the selection of lines, **Face** is for the selection of a face (a plane) of an object, and **Solid** is for the selection of an object in 3D space.

GeoFEA has a docking mechanism feature that allows for easy docking of windows. The windows (e.g. **Geometry**, **Solution**, and **Property**) display in a docked view by default; however, they can also be displayed as floating windows. The visibility, size, and position of the windows can be adjusted for user's convenience and visibility of the model. The docking mechanism displays arrow indicators, as shown in Figure 5, when the window is dragged to a new location by placing the cursor on title bar of the window and moving with holding down the left mouse button. When the cursor touches one of the docking indicators, a blue rectangle shows the windows docking position. Release the left mouse button to allow the window to be docked in the specified location. The windows can also be minimized, pinned/unpinned, or closed using these  icons.

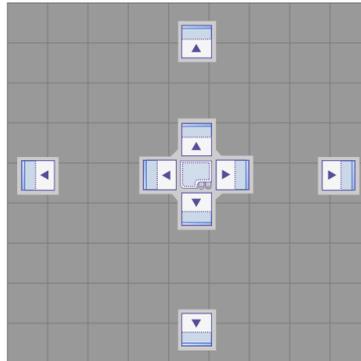


Figure 5. Windows docking indicators.

Model tree-view under **Solution** window lists out all the entities of the project or model. Similarly, the **Property** window displays the properties of a selected object. The background colour of the working space, extent of the grid, grid density, axis colour, etc. can be adjusted by modifying the properties of **Front** in **Solution** window.

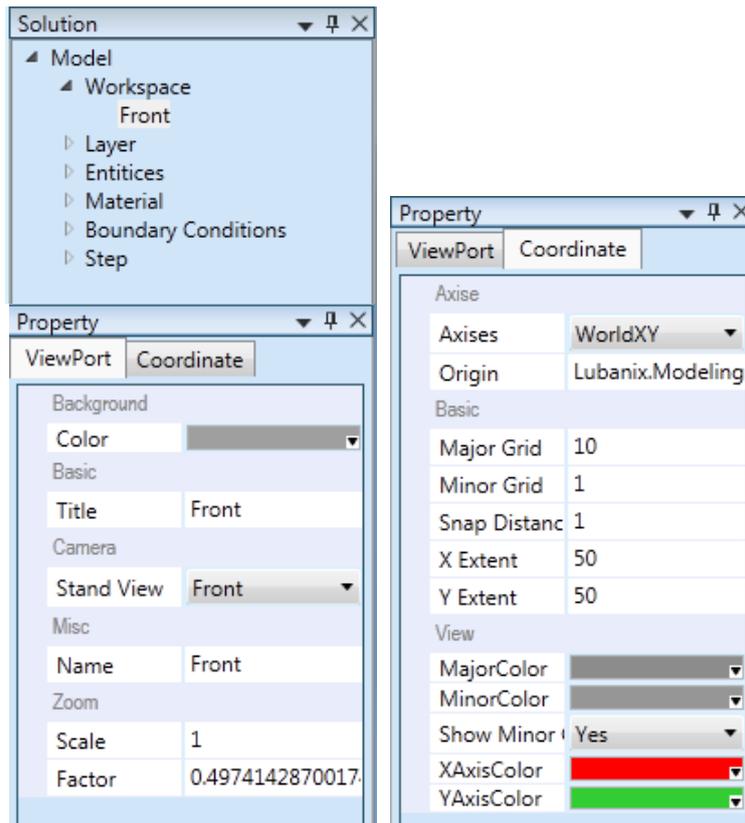


Figure 6. Adjustment of grids and working space.

- In **Solution** window, go to **Model | Workspace** and select **Front**.
- Adjust the properties of the **Front** by editing the values shown in **ViewPort** and **Coordinate** tabs of **Property** window (see Figure 6).

These property windows are also accessible when you click on the screen beyond the grid extents.

6 Creating a geometry

6.1 Edge tools

The **Edge** tools under **Geometry** are for creating a model. The tool palette contains seven tools to create a point, line, circle, ellipse, parabola, hyperbola, and freeform curve, as shown in Figure 7. Once a tool is selected, the **Command** window displays hints for what to do next and other relevant parameters. Press **Esc** to exit from its active state.

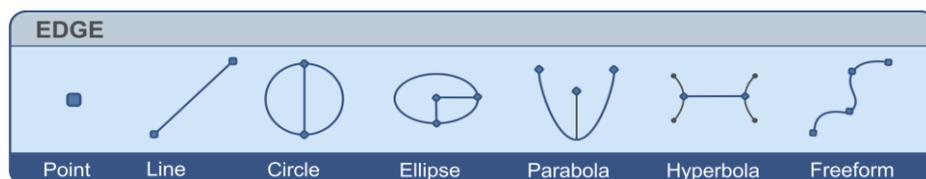


Figure 7. Edge tools.

6.1.1 Creating a point

A point is required to be created on a model for the application of a point load or point boundary conditions.

1. Select the **Point** tool and click on the desired location or enter the coordinates of the point directly on the editable box following the cursor.

6.1.2 Creating a straight line

1. Select the **Line** tool.
2. Command window displays location of start/end of the line according to cursor position and other options, as shown in Figure 8.
 - **Start/End of line:** Either click on the starting point of the line or enter the coordinates directly on the editable box following the cursor.
 - **Input Parameter:** Select the radio button depending on whether you want to draw multiple lines connecting one another or a single line. The default is multiple lines.
 - **Options:** The length and angle options provide more control over creating a line. After the starting point of a line, you can create a line with a combination of several options. Use **Tab** key to navigate and lock/unlock the input boxes after keying in the values. Note that only one input box (**Coordinate**, **Length**, or **Angle**) is editable at one time.

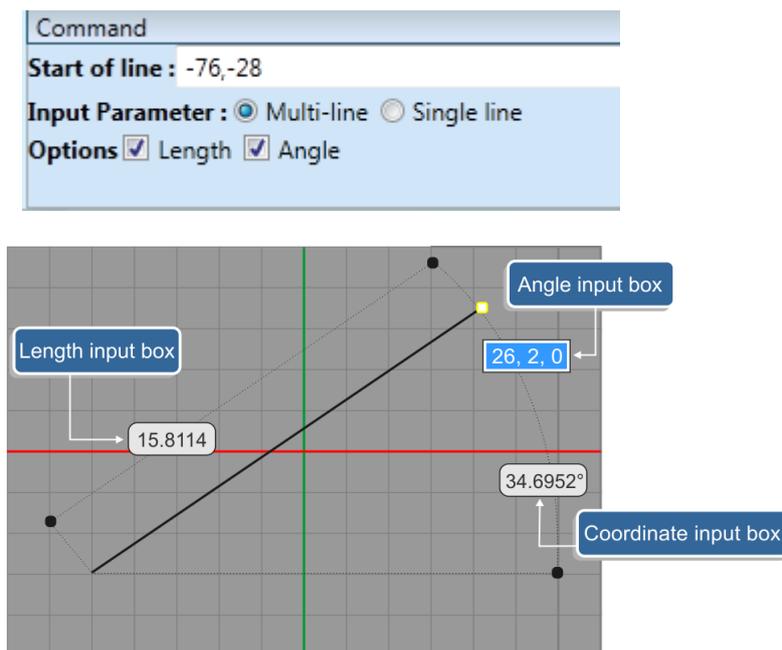


Figure 8. Creating a line.

3. Based on above options, a line can be created in many ways:
 - a. Click on the first end point and the second end point of a line.
 - b. Enter the coordinate of the first and second end points.
 - c. Combination of (a) and (b).
 - d. Combination of length and coordinate, angle and coordinate, or length and angle.
 - e. Press **Enter** to create a line after editing the values. Press **Esc** to end the command.

- The construction of a line or other geometries can further be enhanced by using the snapping facilities. Figure 9 shows the various snapping options available in GeoFEA®. These snapping options can be accessed from bottom-right corner of the window.

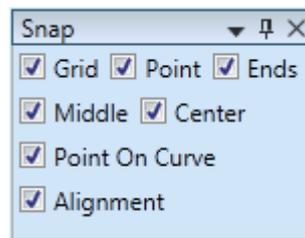


Figure 9. Snapping options.

6.1.3 Creating a circle

- Select the **Circle** tool.
- Command** window displays that the circle can be created in three ways – From Centre and Radius, Two-point circle, and Three-point circle.
- From centre** and radius circle:
 - Click on (or enter the coordinates of) a point for the centre of a circle.
 - Click on a point (or enter coordinates) lying on the circumference or input the radius as shown in Figure 10.
 - Use **Tab** to navigate the input boxes, similar to creating a line.
 - Press **Enter** to create a circle after editing the values. Press **Esc** to end the command.
- Two-point** circle:
 - The two input points form the diameter of the circle.
- Three-point** circle:
 - A circle will be created passing through the entered three points. That is, these points lie on the circumference of the circle.

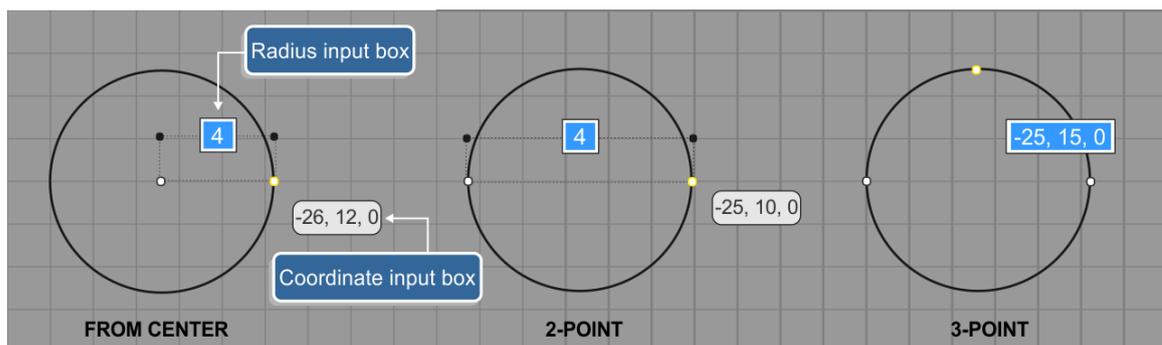


Figure 10. Creating a circle.

Similarly, other objects such as **Ellipse**, **Parabola**, and **Hyperbola** can be drawn following the hints and options in the **Command** window.

6.1.4 Creating a freeform curve

A free style curve may not be a straight line. There are two ways to construct a freeform curve.

- Through points:** The curve passes through the entered points as shown in Figure 11. This is the default option. The dot points are the entered points.

2. **Control points:** The curve does not pass through the entered points, but its alignment is defined by each control points entered as shown in Figure 11.

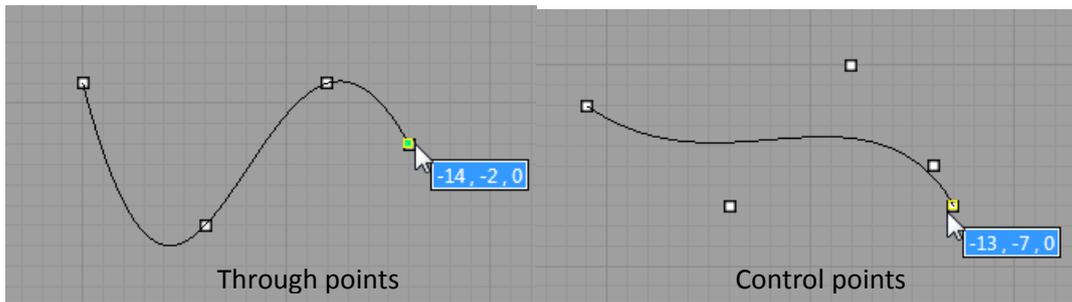


Figure 11. Creating freeform curves.

3. **Is closed:** If this option is checked, the curve will form a closed boundary. The option is unchecked by default.

6.2 Creation tools

In GeoFEA®, a mesh cannot be generated without a face. After drawing the base geometry, creation tools are used to find a face for meshing. However, to create or to find a face, the geometry must have a closed boundary. The tools are:

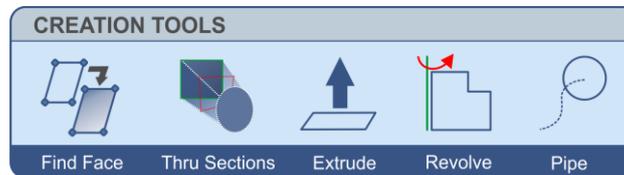


Figure 12. Creation tools.

1. Find Face:

- Create a geometry such that it forms a closed boundary in a plane.
- Select the **Find Face** tool.
- Select the objects (closed boundaries). The colour of selected boundary lines turns out to be yellow to indicate their selection.
- There are two options for face: **Surface only** and **Split** as shown in the command line window. The latter one is by default. That means, if there is another object within the outer boundary area or multiple areas are selected at once, each closed area will have its own face. If this option is unchecked, the face for the outer boundary area will only be found. The **Surface only** option is for a complicated 3D meshing and will not be discussed here.
- Press **Enter** to generate a face. The area is filled with a slight dark colour indicating a face is found for that area, as shown in Figure 13.

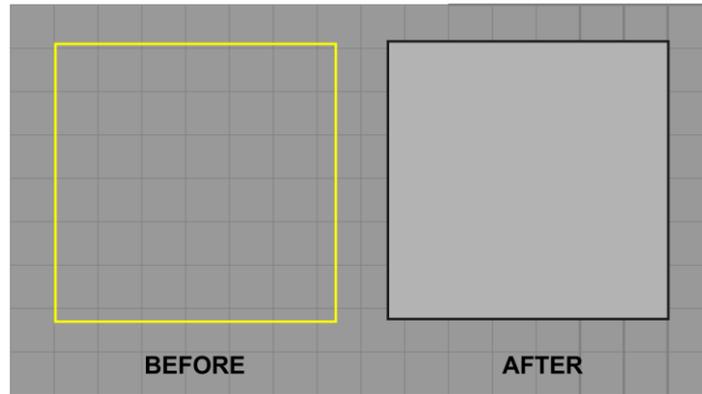


Figure 13. Finding a face.

2. **Thru Sections, Extrude, Resolve and Pipe:** These tools are for 3D meshing and will not be discussed here.

6.3 Combination tools

Combination tools are used to group and ungroup of the objects. These are:

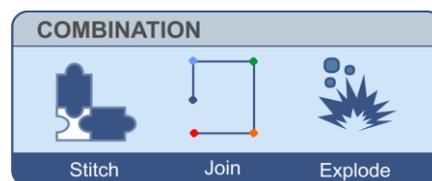


Figure 14. Combination tools.

3. **Stitch:** Let's take an example shown in Figure 15. Initially the lines AB, BC, CD, DA, and circle E are all separate. Separate means they can be selected and edited individually. However, **Stitch** will group all of them to be one so that all can be selected at once. This function is useful when the user wants to edit or assign the same property and/or boundary conditions to all the objects as once.
 - Select **Stitch** tool.
 - Select the objects to be joined. Use rubber band for easy selection, if applicable.
 - Press **Enter** to stitch them together. Now all the lines and circle are grouped together.

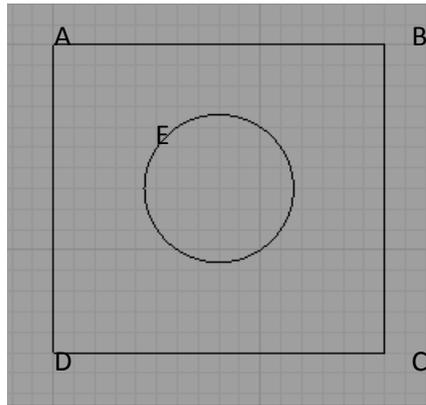


Figure 15. Combining objects.

4. **Join**: This tool functions similar to **Stitch** except that it only groups the connected lines. For the example shown in Figure 15, **Join** can group the lines AB, BC, CD, and DA to one, but not the circle.
5. **Explode**: This tool is to ungroup all the grouped objects. **Explode** is opposite of **Stitch** or **Join**.

6.4 Modification tools

Modification tools are used to modify the created objects. There are four available tools (Figure 16) that can be used to modify objects.

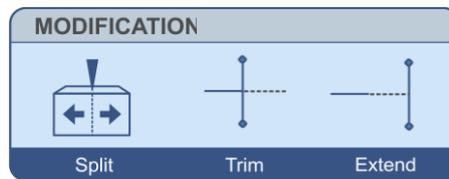


Figure 16. Modification tools.

1. **Split**: Split tool is used to break the lines or objects at intersection points. Let's take an example shown in Figure 17. Lines AB and CD are intersecting at point E, but they are not broken at E. In order to split these lines at E:
 - a. Select **Split** tool.
 - b. Select the lines that are to be split (e.g. both lines AB and CD in this case), and press **Enter**. If only AB or CD is selected, only that line will be split.
 - c. Select the lines that are used to split/cut. Selecting both lines again will split all of them so that lines AE, EB, CE, and ED are generated. On the other hand, if only AB or CD is selected, that line cuts the other line at E.

Split tool is able to operate line on surface, surface on line, surface on surface, depending on which object is used as the splitting tool. The region that is common to both the splitting tool and target will be removed from the target object. Figure 18 shows the operation of split tool for surface on surface. Let's say we need a geometry which has a rectangular shape with a circular cut at one corner. How can we create such geometries?

- a. Create a rectangle ABCD using the line tool (Section 6.1.2) and find a face (Section 6.2) as shown in Figure 18.
- b. Create a circle XYZ centered at C (Section 6.1.3) and find a face (Section 6.2).
- c. Select the **Split** tool.
- d. Select the rectangle ABCD (the object to be split) and press **Enter**.

- e. Select the circle XYZ (the splitting object which acts as a knife) and press **Enter**.

This splits the rectangle such that it becomes ABXYD while the circle XYZ remains intact. The circle can be deleted, if not required. Conversely, we can split the circle XYZ with the rectangle ABCD such that the circle becomes a 3 quarter circle XCYZ.

Split tool can also operate on objects in 3D space, but will not be elaborated here.

2. **Trim:** This tool is used to trim the offset lines. For example, if AB and CD are two lines and you do not want the BE or AE part of the line AB in Figure 17, you can use **Trim** tool.
 - a. Select **Trim** tool.
 - b. Select the cutting line CD.
 - c. Click on BE side to delete BE and vice versa.

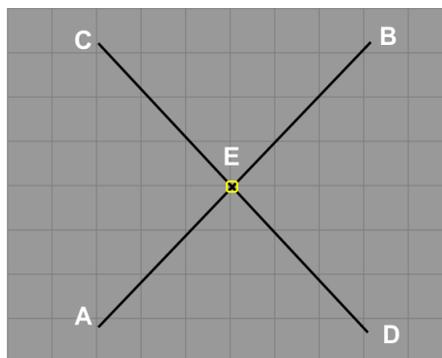


Figure 17. Modification of lines.

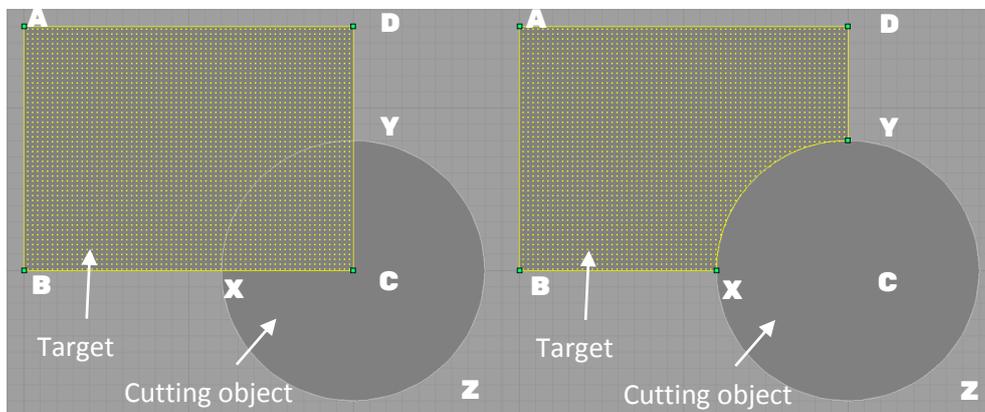


Figure 18. Using one surface to split another surface.

3. **Extend:** This tool is useful when a shorter line needs to be extended up to the boundary. For example, in Figure 19, the line AD needs to be extended up the boundary line BC.
 - a. Select the **Extend** tool.
 - b. Select the boundary line BC and press **Enter**.
 - c. Selecting the line AD will automatically extend it up to intersection point E.

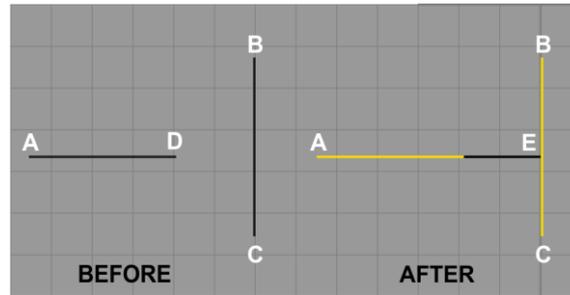


Figure 19. Extending a line.

6.5 Transform tools

Transform tools are used when an object needs to be moved or copied at a different location than its current location. There are four tools (Figure 20) that can help an object transformed.

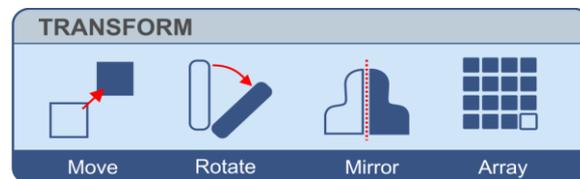
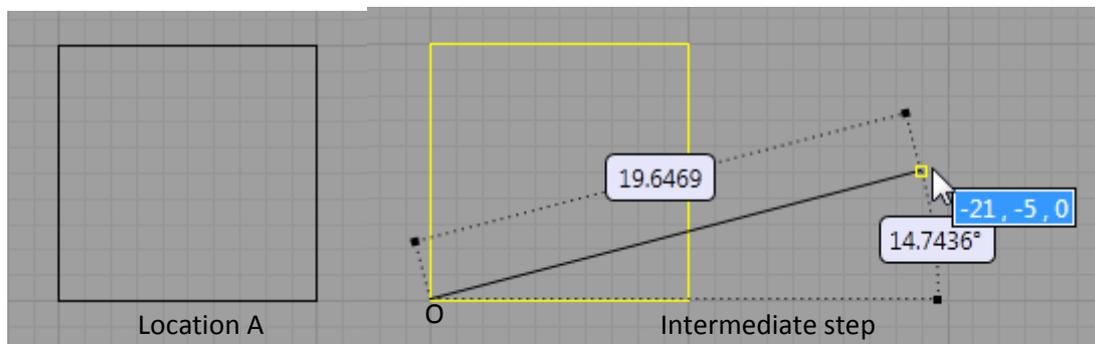


Figure 20. Transform tools.

1. **Move:** This tool is used for moving an object from one location to another. Let's take an example as shown in Figure 21. A rectangle in location A needs to be moved to location B.



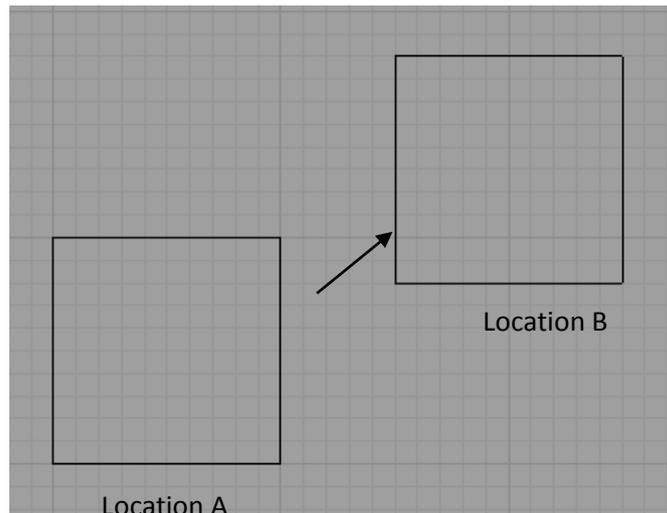


Figure 21. Moving an object.

- a. Select **Move** tool.
 - b. Select the object(s) to be moved; rectangle in this case and press **Enter**.
 - c. Now the movement direction and details are to be entered. Click on any reference point, e.g. point O. The length or angle of the new position with respect to point O or an absolute co-ordinate of the new position can be entered in a way similar to creating a line in section 6.1.2.
 - d. Press **Enter** to move the object to new location B.
 - e. If the **Copy: Yes** is checked in the **Command** window, the object will be copied to the new location B without deleting it at location A. However, if this option is not selected, the object at location A will disappear.
2. **Rotate**: Rotate tool is used to rotate the object with respect to its original position.
 - a. Select the **Rotate** tool.
 - b. Select the object to be rotated and press **Enter**.
 - c. Input the **axis** of rotation.
 - d. For angle of rotation, either enter angle in the angle input box or move the mouse so that it forms a required angle with respect to the axis of rotation and press **Enter**.
 3. **Mirror**: Mirror tool can be used when the same object needs to be duplicated reflecting symmetrical to a plane/axis.
 - a. Select the **Mirror** tool.
 - b. Check **Yes to Copy** in the **Command** window if you do not want to delete the original object.
 - c. There are two options for creating a mirror – by **Point** or **Axis**. **Point** only needs a single point to be entered (see section 6.1.1). The program will automatically create a mirror passing through the point and normal to the object. **Axis** requires a line to be created as described in section 6.1.2, which acts as a mirror. The default option is by **Axis**.
 - d. Select the object that needs to be duplicated (rectangle in Location A of Figure 22) and press **Enter**.
 - e. Create an axis along X-Y as described in (c) above.
 - f. The object will be mirrored at Location B.

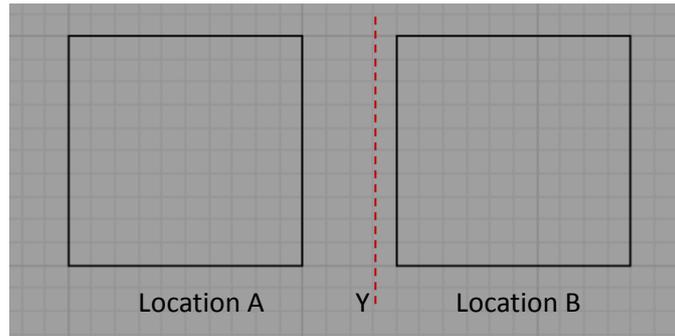


Figure 22. Mirroring an object.

4. **Array:** Array tool is to create a multiple copies of an object in one shot. For example, let's say we want to create six circles C1-C6 as shown in Figure 23. We need to create only one circle (either C1 or C6) using the method described in section 6.1.3 and use **Array** tool to generate other circles.
 - a. Let's say, circle C1 is created with centre at origin (0, 0).
 - b. Select **Array** tool.
 - c. Select the circle C1, and press Enter.
 - d. Command window will change for **input direction**. Input direction is the direction that you want the objects (circles) to propagate.
 - e. Set the **number of copy** to five in Command window. This gives the total number of circles to be six.
 - f. For input direction, click on centre of the circle C1 (0, 0) and click on centre of the circle C6 (40, 0). This generates five circles C2-C6 as shown in Figure 23.
 - g. The **Array** function is very useful when there are several objects in the mesh with identical geometries.

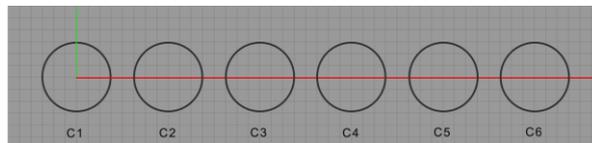


Figure 23. Array of circles.

6.6 Boolean tools

Boolean operation is used when an object overlaps the other and you may want to work only a part of them or whole. There are three tools for the Boolean operation as shown in Figure 24.

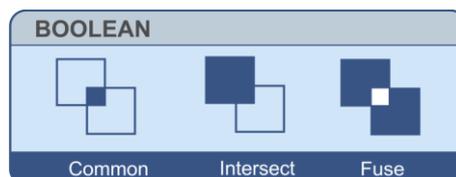


Figure 24. Boolean tools.

1. **Common:** This tool is used to find common area of two objects. Let's take an example as shown in Figure 25. ABCD and XYZ are two objects. Faces for both objects have already been found. See section 6.2 for more details for finding a face.

Both objects have a common area XMN. If you only require a common area of both objects, **Common** is the tool to be used.

- a. Select **Common** tool.
 - b. Select the objects ABCD and XYZ.
 - c. Press **Enter** to remove all parts except the common area XMN (see Figure 25).
2. **Intersect**: This tool is used when overlapping objects are used to intersect each other. For the same example in Figure 25. The intersect operation creates three regions: ABCMXND, XMN, and MYZN. For this,
- a. Select **Intersect** tool.
 - b. Select the objects ABCD and XYZ.
 - c. Press **Enter** to get ABCMXND, XMN, and MYZN zones.
3. **Fuse**: This tool is used to combine individual objects into one.
- d. Select **Fuse** tool.
 - e. Select the objects ABCD and XYZ.
 - f. Press **Enter** to get an object ABCMYZND.

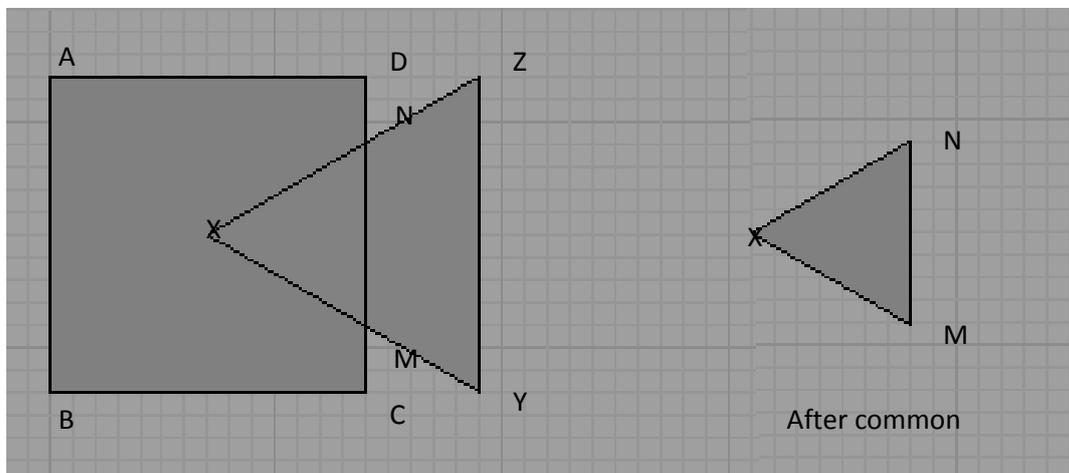
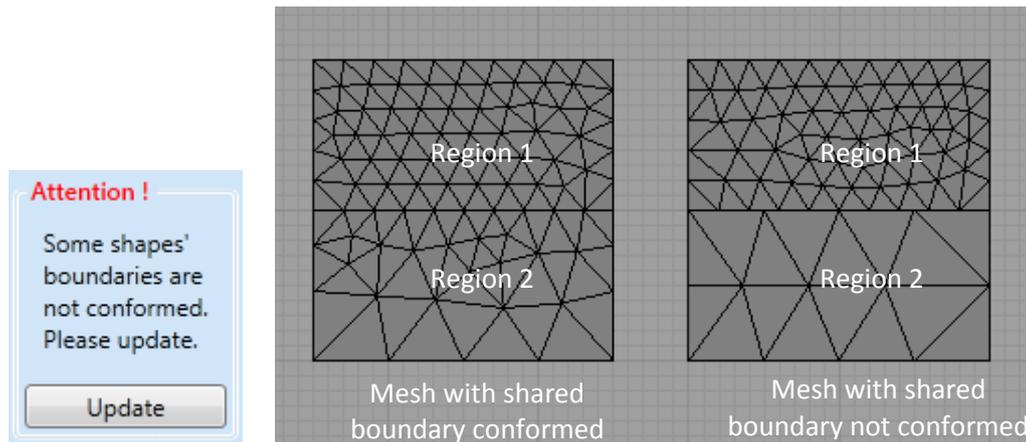


Figure 25. Boolean operation on two overlapping objects.

7 Generating a mesh

Mesh generation is the next step after creation of the geometry and finding faces. When you go to **Mesh** menu, you will be asked to update the new geometries (Figure 26a) before processing for mesh generation. This allows the program to automatically conform all the geometries at shared boundaries so that the generated elements at different regions are compatible at those boundaries. Alternatively, the **Conformity** tool can be used for checking the conformation of the shared boundaries of adjacent regions. If the boundaries are not conformed, the meshes of adjacent regions will not be talking to each other (Figure 26b) and can lead to spurious results. Hence, it is important that they are conformed well before meshing.



(a)

(b)

Figure 26. (a) Updating new changes in the geometry, (b) Effect of conformation of shared boundaries on meshing.

There are three tools, as shown in Figure 27, for the mesh generation.



Figure 27. Meshing tools.

1. **Solid (Mesh): Solid (Mesh)** is used for generating 6-noded linear strain triangular elements for 2D analysis and 10-noded linear strain tetrahedral elements for 3D analysis. The steps are similar for both.

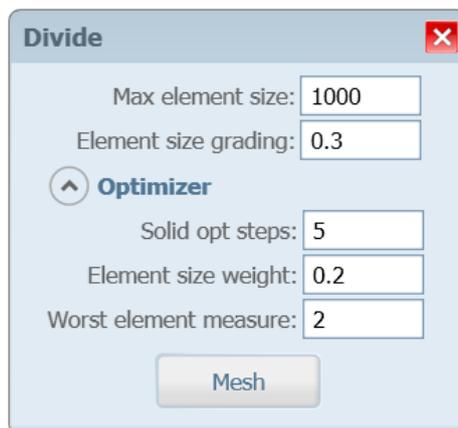


Figure 28. Mesh size and optimization.

For mesh generation, select the regions and click on **Solid** tool. As shown in Figure 28, you can adjust the **Max element-size**, which controls the density of the mesh. The number of generated elements can be viewed from the property window for the selected regions. If the generated mesh is too dense or too sparse, the mesh can be regenerated for the selected region(s) and adjusting the element size without deleting the previous mesh. The **Element-size grading** controls the gradation of

element sizes near the interface boundaries if the two adjacent regions have different sizes of elements.

The optimizer parameters are for smoothing and optimization of generated elements. **Solid opt steps** controls the number of iteration of optimization. Its default value is set to 5, which is sufficient in most cases. Larger this number, more time it takes for the mesh generation. Similarly, **Element size weight** and **Worst element measure** are used to measure whether optimization is required or not.

2. **Edge:** Edge tool is used to mesh individual line elements such as for bar/beam elements. This tool is also used when a boundary of the mesh needs to be defined to contain a definite number of elements and in a defined pattern of the element sizes (e.g. 10 elements along an edge with 1:2 ratio for the minimum and maximum size of the elements).
 - a. Select the line and select **Edge** tool.
 - b. A window will pop up as shown in Figure 29.
 - c. Select **Free Edge** if the edge has not been meshed yet. Or, select **Mesh Boundary** if it is a boundary of the meshed area.
 - d. Select the Nodal compatibility to **Yes** or **No** depending on whether you want the mesh nodes to be compatible with the shared boundary or not.
 - e. Edit the **Number** and **Weights** of the Divisions to control the size and pattern of the generated elements.
 - f. **Click on Divide** to **generate** the elements.

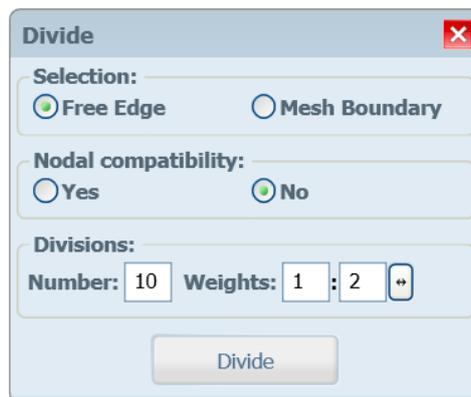


Figure 29. Meshing by Edge.

3. **Delete:** Delete is for deleting the generated mesh. Select the unwanted mesh and click on Delete for deleting.

8 Assigning material properties

Assignment of materials option is under **Assignment** menu. Materials are categorized into two groups: **Soil** and **Structure**, as shown in Figure 30. **Soil** material is for solid continuum elements (e.g. triangular elements in 2D and tetrahedral elements in 3D analysis) and **Structure** material is for bar/beam type of elements. The steps to create the material database are:

1. Select **Soil** (for solid elements) and **Structure** for (bar/beam type of elements).

2. Right click and select **Add/Delete** to add materials to the list or to delete them from the database, as shown in Figure 30.
3. Or, simply use **Add/Delete/Assign** button on the top or bottom of the window to add or to delete the materials in the database, and to assign the selected material to the mesh, respectively.

GeoFEA® offers a wide range of linear and nonlinear material models which can be effectively used for modeling the soil and structural behavior. The following are the material models available in GeoFEA®:

1. Transverse Isotropic Elastic
2. Linear Isotropic Elastic
3. Elastic Perfectly Plastic (Associated/Non-associated Flow Rule) with
 - a. von Mises Yield Criterion
 - b. Tresca Yield Criterion
 - c. Mohr-Coulomb Yield Criterion
 - d. Drucker-Prager (Inscribing) Yield Criterion
 - e. Drucker-Prager (Outscribing) Yield Criterion
4. Original Cam Clay
5. Modified Cam Clay
6. Hyperbolic Cam Clay
7. Schofield

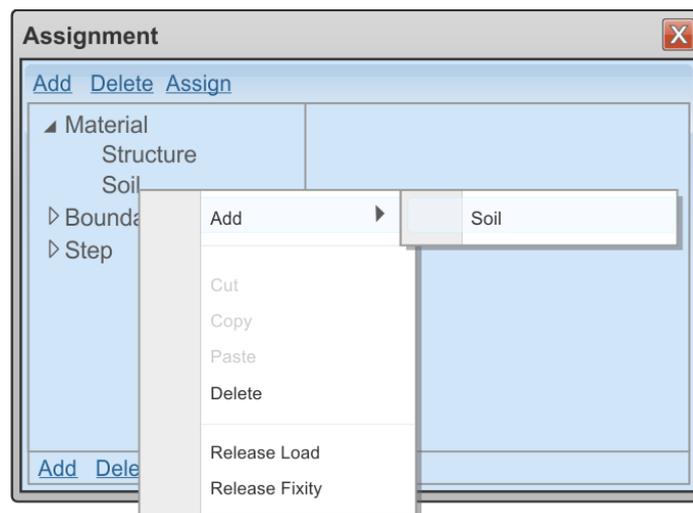


Figure 30. Assignment of materials.

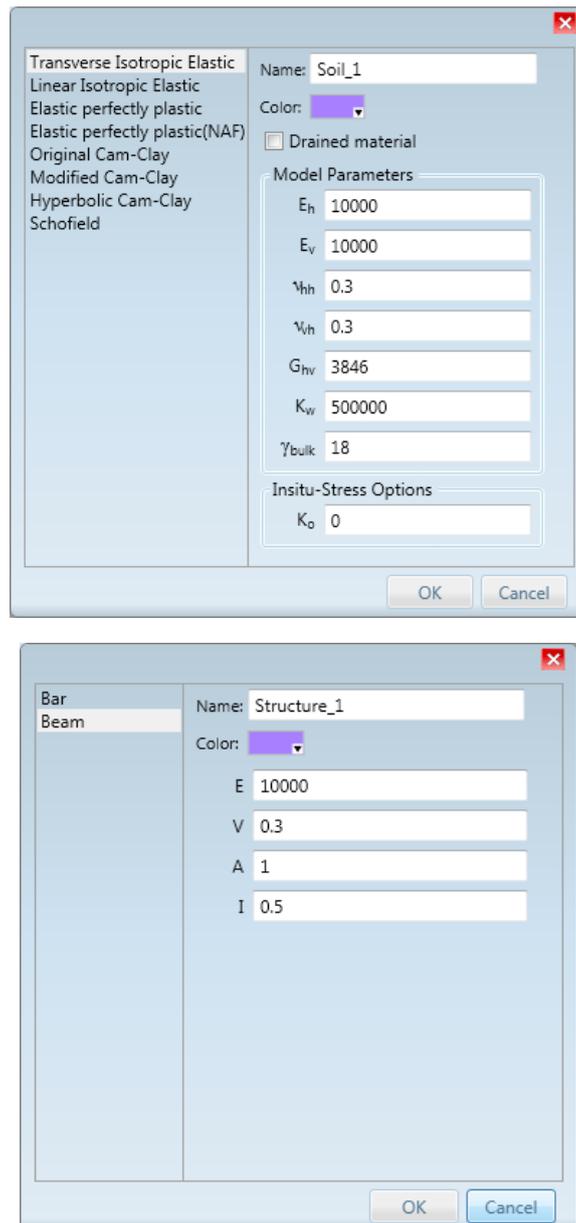


Figure 31. Defining material properties.

Available material models will be listed on the left column of the window, as shown in Figure 31.

1. Select a suitable material model and edit the parameters shown on the right column of the window.
2. Users have the flexibility to input the in situ earth pressure coefficient (K_0). If this field is left unspecified (i.e. $K_0 \leq 0$), the program will compute the K_0 value using an empirical relationship. If K_0 value is specified, this value will be used for in situ stress computation.
3. Customize the name and color of the material to more meaningful names, as appropriate, and press **OK** to add to the database.
4. Initially, when the material is added to the database, a zero term in the bracket will be displayed together with the name of the material. This term is for the indication of the number of regions, where the material is assigned for. A zero term means the

- material has not been applied to any zones yet. The number will change as you assign the materials to various regions.
5. For assignment, select the zone(s) where the materials to be assigned. Select the material from the list and click on **Assign** button on the top/bottom of the assignment window.
 6. The parameters can be edited after adding to the database and after the assignment as well.

9 Assigning boundary conditions

Assignment of boundary conditions option is also under **Assignment** menu. There are three categories for boundary conditions: **Fixity**, **Pore pressure**, and **Load**. Note that these boundary conditions cannot be defined for the in situ stage, if the in situ stress is set to zero at **in situ Step**.

9.1 Assignment of fixities

1. To assign the fixity, right click on **Fixity** and go to **Add | Fixity**, as shown in Figure 32. Or, select the appropriate **Step**, right click on it and click **Add | Fixity**. A window will pop up to set up the type of fixity and its values (see Figure 32).
2. Edit the name, type, and values as desired and press **OK** to list in the database. You can also edit the values after it is listed in the database. The description of parameters are:
 - a. Name: Name given to fixity.
 - b. Step: The stage at which the fixity to be applied.
 - c. Fix X-dir: Set it to **Yes** for specifying x-directional fixity. Set **No** if this fixity is not required.
 - d. Fix Y-dir: Set it to **Yes** for specifying y-directional fixity. Set **No** if this fixity is not required.
 - e. X Offset: Specify the x-direction fixity value if **Fix X-dir** is set to **Yes**.
 - f. Y Offset: Specify the y-direction fixity value if **Fix Y-dir** is set to **Yes**.
 - g. Reverse X/Y: This is for the reflection of fixity symbols in the mesh after its application. It has no effect on magnitude and or direction.
 - h. For assignment, select the fixity and click on or near the boundary edge.
 - i. The selected curve's colour changes to yellow (see Figure 33).
 - j. Press **Assign** button of the **Assignment** window. Three filled triangles (symbol) on the edge indicate the assignment of fixity. Modify **View | Reverse X** and/or **View | Reverse Y** of the fixity to **Yes/No** if the symbol is not displayed as per intention.
 - k. If you select the assigned fixity, its color changes to yellow (see Figure 33) to indicate its selection.

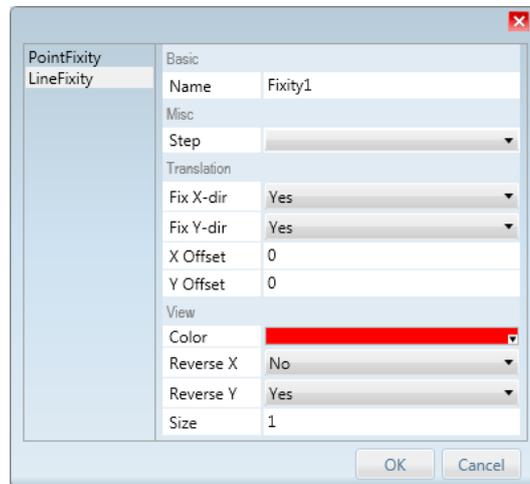
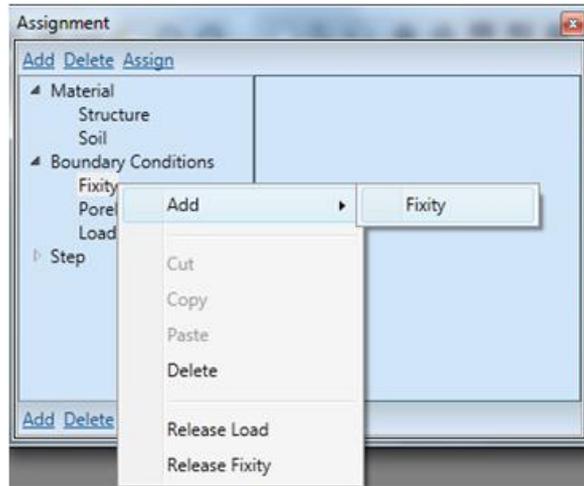


Figure 32. Defining a fixity.

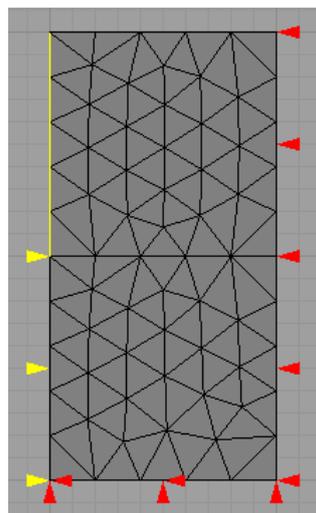


Figure 33. Assigning a fixity.

9.2 Assignment of pore-pressure boundary conditions

The assignment of pore pressure boundary condition is very similar to the assignment of displacement boundary conditions described in section 9.1. A pore pressure database has to

be created before assigning them. The pore pressure variation can be **uniform** or **linearly varying** with depth (Figure 34). Similarly, there are three modes to define the pore pressures – **incremental**, **absolute**, and **total**. The **incremental** and **absolute** value mode works on excess pore pressure (excess means excess over hydrostatic). The pore-pressure fixity using incremental value operates in the same manner as the displacement fixity. The excess pore pressure is given an incremental change by the user. The **absolute** pore-pressure fixity is only used where one cannot use an incremental fixity value, e.g. the top boundary surface. The **total** pore pressure mode can be used when the user knows beforehand the value of total pore pressure.

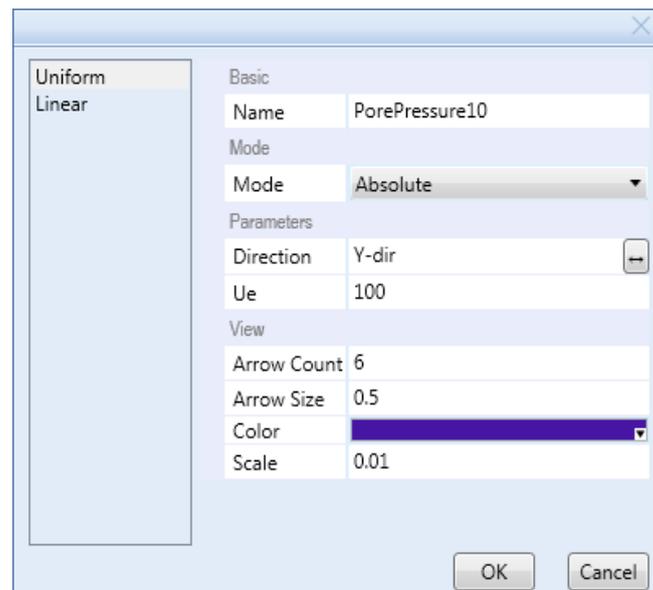


Figure 34. Defining pore pressure boundary conditions.

9.3 Assignment of loads

The assignment of loads is also similar to the assignment of fixities. There are a number of different types of load, as shown in Figure 35, which can be used depending on the type of analysis.

1. **Point load:** Point loads are applied at nodes (or created points). However, nodes has to be created (see section 6.1.1) before applying point loads.
2. **Linear load:** These are distributed forces with linear variation along a curve. **XLinear** and **YLinear** loads are the special cases of linear load with load varying along x- and y-directions, respectively. The difference between Linear load and XLinear/YLinear load is that the former load interpolates the linear variation of a load for the applied curve only, while the latter load needs the coordinate of starting and end point of the loads so that any intermediate curve within this range of coordinates will have an interpolated linear load.
3. **Parabolic load:** These are distributed forces with quadratic variation along a curve.
4. **Uniform load:** These are forces distributed uniformly over a curve or a surface.

Note that a load can only be applied to one object. Hence, the load database has to be equal to the number of intended objects where load is to be applied. The **Arrow count** and **Arrow size** controls the density of arrows and their sizes. The **Color** defines the color of arrows and **Scale** defines the length of the arrows.

In order to delete the load, fixity, or pore pressure boundary conditions, select the particular item from assignment window and click on **Delete** button.

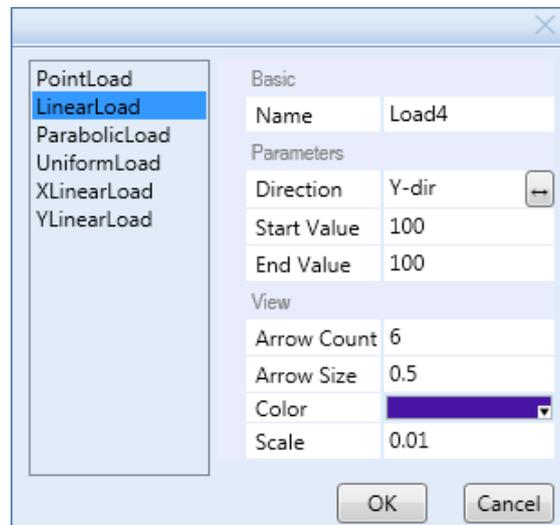


Figure 35. Assignment of loads.

10 Staged analysis

Staged construction is an important aspect in many geotechnical engineering applications such as in excavation, tunneling, and pile foundations. Hence, the boundary conditions such as fixities and loads are also assigned step-by-step. Unless otherwise changed, the fixity conditions usually do not change over steps. Hence, they do not need to specify for each **Step**/stage. Similarly, the

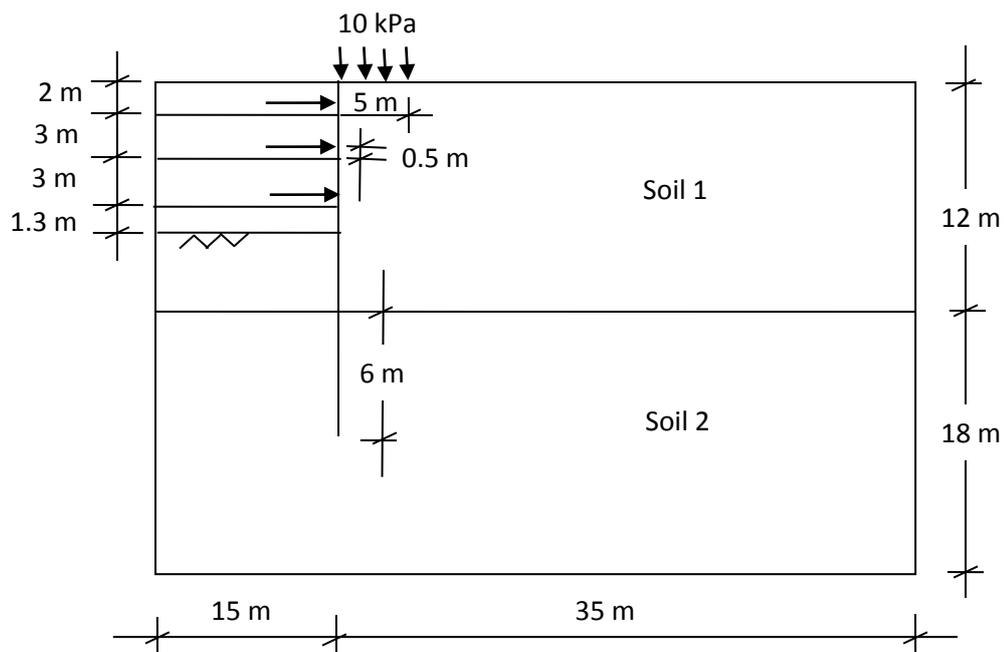


Figure 36. An excavation problem.

elements exclude/include capability in GeoFEA® offers an easy way of modeling the real construction sequences. This is demonstrated with the help of an excavation example. This example is to introduce basic GeoFEA® features and does not include all the features of GeoFEA®.

Let's take an excavation example as shown in Figure 36. The problem involves 9.3 m excavation supported by three layers of struts. The ground consists of two soil layers and the water table is at ground surface. The depth of diaphragm wall is 8.7 m below the dredge line.

10.1 Setting up the project

1. Follow the steps in section 4 to create a new project.
2. Set the analysis dimension as **plain strain**, analysis type as **consolidation**, and units as **m/kN,kN-m**, etc. and press **OK**.
3. Go to **Model | Workspace | Front** of the **Solution** window on the right side of the screen.
4. Go to **Coordinate** tab of the **Property** window and adjust **X Extent** to 100.
5. Use **Zoom in/out** tool to bring the x-y axes to the centre of the screen.

Now the project and workspace are set up.

10.2 Creating the geometry

1. Select the **Line** tool from **Geometry | Edge** window on the left of the screen.
2. Click on (0, 0), (50, 0), (50, 30), (20, 30), (15, 30), (0, 30), (0, 0), and press **Esc**. This creates the outer boundary of the mesh.
3. For two layers of soil, select the line tool again. Click on (0, 18), (50, 18), and press **Esc**.
4. For diaphragm wall, select the line tool again. Click on (15, 30), (15, 12), and press **Esc**.
5. For the sake of meshing, draw a horizontal layer passing through the base of the diaphragm wall. Select the line tool again. Click on (15, 12), (0, 12), and press **Esc**.
6. For excavation layers, select the line tool again. Click on (0, 28), (15, 28), and press **Esc** for the first layer. The same procedure can be repeated for other excavation layers.
7. For dredge level, it is a little tricky to draw the lines with exact coordinates because the default **Snap distance** is set to 1 for **Front's Coordinate** property in section 10.1. Hence, either input the exact coordinates or adjust the snapping to draw the dredge line.
8. To adjust snapping, go to **Coordinate** property of **Front** as described in section 10.1 and adjust **Snap distance** to 0.1.
 - a. Select the line tool, click on (0, 20.7). Use **Zoom in/out** tool for a clearer view.
 - b. Use **Tab** key to navigate to angle input box. Set the angle to zero.
 - c. Use **Tab** key to navigate to length input box. Set the length to 15 and press Enter.
9. Similarly, use the line tool to draw lines for three struts which are 0.5m above the excavation levels. For the sake of drawing, the snap distance can be adjusted to 0.5. Later, these lines will be assigned to structural properties for them to be struts and wall, etc.

This completes the geometry of the problem and the completed geometry will look as shown in Figure 37.



Figure 37. Geometry of the excavation problem.

10.3 Generating the mesh

Before generating the mesh, it is required to find the faces for each region. Follow the steps described in section 6.2 to find the faces. It is possible to select all the regions at once using either **Shift** key or rubber band and find the faces at once.

For meshing, since excavation area has small regions, a single size of element for the entire region may not be suitable. Because excavation area is more sensitive, it is good to have a dense mesh in the excavation and nearby area and sparse mesh in the rest of the areas. The steps for mesh generation are described in section 7. The generated finite element mesh with element sizes of 0.5 in the excavation area, 1 for area from 3m below dredge line up to toe of the wall, and 2 for the rest of the areas is as shown in Figure 38.

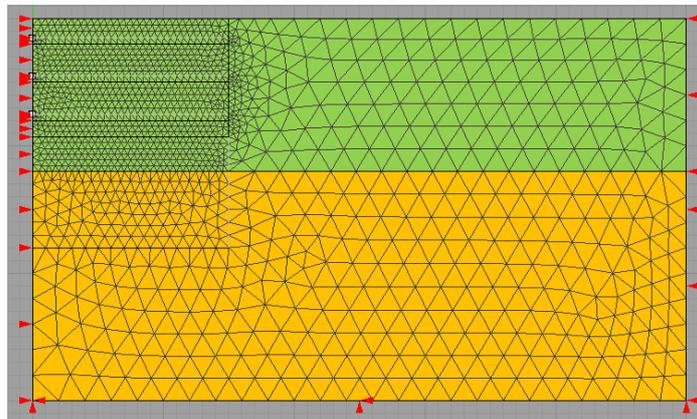


Figure 38. Finite element mesh of the excavation problem.

10.4 Assigning the material properties

The following are the material properties used for the analysis.

Table 1. Material properties of top and bottom layer of soils

Parameter	Name	Top layer	Bottom layer	Unit
Material model	Elastic perfectly plastic	Mohr-Coulomb	Mohr-Coulomb	-
Type of material behavior	Type	Consolidation	Consolidation	-
Young's modulus (constant)	E_0	5000	20000	kN/m ²
Cohesion (constant)	c	0	0	kN/m ²
Friction angle	ϕ	28	32	degree
Poisson's ratio	ν	0.3	0.3	
Bulk unit weight of soil	γ_{bulk}	18	18	kN/m ³
Permeability in hor. direction	k_x	1×10^{-9}	1×10^{-7}	m/s
Permeability in ver. direction	k_y	1×10^{-9}	1×10^{-7}	m/s
Unit weight of water	γ_w	10	10	kN/m ³
Over consolidation ratio	OCR	1	1	-
Reference level for varying E_0 , c , and OCR	y_0	30	18	m
Gradient for variation of E_0	m_E	500	3000	-
Gradient for variation of c	m_c	0	0	-
Gradient for variation of OCR	m_{OCR}	0	0	-
In situ earth pressure coefficient *	K_0	0	0	-

* $K_0 \leq 0$ allows program to compute its value.

Table 2. Material properties of strut and wall

Parameter	Name	Strut	Wall	Unit
Material model	Linear elastic	Beam	Beam	-
Cross-sectional area	A	0.0437	0.8	m ²
Young's modulus of elasticity	E	2×10^8	2.5×10^7	kN/m ²
Moment of inertial	I	0.000333	0.0426667	m ⁴
Poisson's ratio	ν	0.2	0.2	-

In Table 1, the Young's modulus of material is assumed to be increasing linearly with depth. The Young's modulus at any depth y is given by the equation $E = E_0 + m_E (y_0 - y)$, where y_0 is the reference elevation and m_E is the gradient for variation of Young's modulus. The other parameters such as cohesion or OCR can also vary linearly with respect to the same reference elevation y_0 . Similarly, linear elastic beam element is used for wall and struts. The wall is considered as 0.8m thick concrete diaphragm wall. The strut is taken as two H400×400×172 kg/m laced strut and the spacing between struts is assumed to be 4m. The creation of material database and assignment are as explained in section 8. The finite element model after assignment of materials is shown in Figure 38.

10.5 Initial conditions

Boundary conditions and in-situ stresses must be set before processing for any calculation stage. This is done at the initial stage – **Assignment | Step | Init Step**. However, the default set up for in situ stress is zero. If in situ stress is set to zero, fixities or any other boundary conditions cannot be applied at the in situ stage. Hence, change the in situ stress option by clicking on the **Insitu stress** button on the right of **Assignment** window before assigning boundary conditions. More details about in situ stress will be discussed later.

10.5.1 Assigning fixities

The left and right boundaries are constrained to move laterally (x-direction) and bottom of the mesh is fixed in both horizontal and vertical movement (x- and y-directions). Right click on **Init Step** and add fixity. Details of assigning fixities are discussed in section 9.1. The assigned fixities/boundary conditions will be listed under **Init Step** tree view.

10.5.2 Assigning in-situ stresses

GeoFEA® provides two ways to specify the in situ stresses – interpolate from the given reference values or calculate automatically from assigned material properties. For **interpolation from the reference points**, users need to specify the in situ stresses for different layers (depths) in a tabular form and program interpolates the stresses for each integration point. However, this option is only applicable when the soil layers are horizontal. In **automatic calculation** approach, the user needs to specify the location of ground water table and the unit weight of water only. Only the latter approach is discussed here.

1. The ground water table is assumed at ground surface. Specify the location of ground water table to 30 m.
2. Set the unit weight of water to 10 kN/m³ and press OK.
3. Set the **Gravity factor** to 1.

10.5.3 Exclusion/Inclusion of the geometry

For the staged construction of geotechnical problems, some geometry may be added or removed from the model on different stages. GeoFEA® provides Include/Exclude feature to model such construction sequences. Select **Init step**, you will notice that **Exclude** and **Include** buttons be added to the top and bottom of the Assignment window. For the given excavation problem (Figure 38), the struts and wall have not been installed at the in situ stage, and they should be excluded.

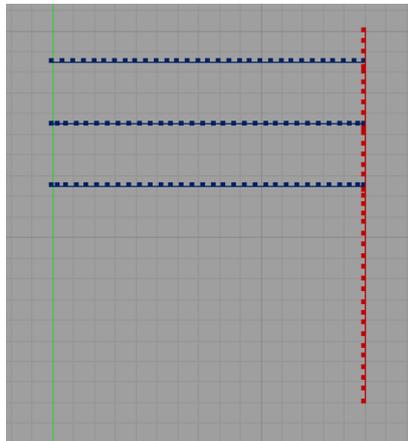


Figure 39. Excluded elements from the initial stage.

1. Select the struts and wall (lines) and click on **Exclude** to exclude them from the in situ stage.
2. Excluded items will be listed in the left column of assignment window in tree-view under Init step.
3. To see the excluded elements from the mesh, change the **View mode** from Current to **Exclude** using drop-down list. The excluded elements are as shown in Figure 39.

10.6 Excavation of 1st layer

After setting up the initial conditions, the excavation process can be started. The diaphragm wall is assumed to be installed before starting any excavation. The following are the steps for the excavation of 1st 2m.

1. To add a new step, right click on **Assignment | step** and add a new step.
2. Select the newly added step (Step_1).
3. Set the **Gravity factor** to zero, **Increment number** to 10, and **Time increment** to 172800 (2 days). The number of increment subdivides the step into smaller load steps so that the equilibrium error is within the acceptable limits. A larger value of increment number contributes a longer computational time. It is assumed that the 2m excavation is completed in two days. The time is input as seconds based on the selected **Unit** for the project.
4. For installing the wall, go to **Exclude** view mode, select the wall, and click on **Include** button on Assignment window.
5. For the excavation, change the view mode to **Current**. Select the mesh of 2m excavation depth, and click on **Exclude**. This simulates the excavation.
6. For the 5m backfill load behind the wall, right click on Step_1 and add load. Define the uniform load of 10 kPa and press OK.
7. Select the edge behind the wall, select the load, and click on Assign. This completes the load assignment.
8. For the pore pressure boundary condition to the exposed surface, right click on Step_1 and add pore pressure. Select **Uniform** pore pressure in **Total** mode of application. Set $u_e = 0$ leaving other values as default. This is to ensure that the total pore pressure is zero at exposed surface(s).
9. Select the edges of the exposed regions. If the exposed boundary edges are not selectable with **Auto select** mode then changing the select mode to **Edge** will help. Assignment of pore pressure boundary condition is discussed in detail in section 9.2.
10. For pore pressure boundary condition at distant edge (boundary), it is assumed that the pore pressure at the distant boundary does not change because this boundary is far away from the excavation so that the effect in pore pressure is minimal at this boundary. Hence, the pore pressure boundary of the far edge of the model is set as $\Delta u_e = 0$.
11. For zero change in pore pressure at distant boundary, use **Uniform | Incremental** value mode of pore pressure.

This completes the simulation of excavation of 1st layer and the model at the end of 1st stage will look as shown in Figure 40.

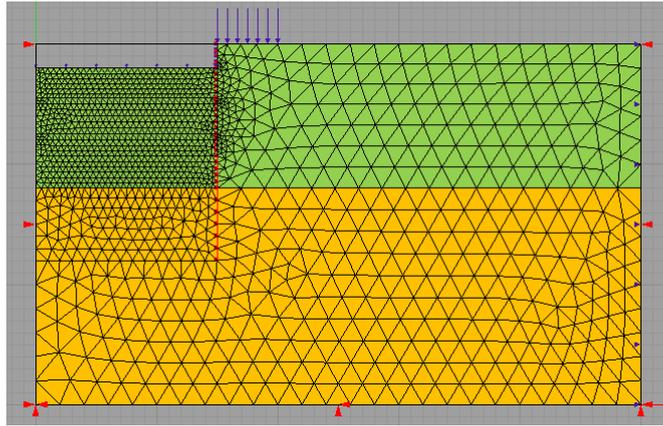


Figure 40. Finite element model at the end of 1st excavation.

10.7 Installation of 1st layer of strut

1. To add a new step, right click on Step and add a new step (Step_2).
2. Set the time increment to 86400 (1 day). Adjust the increment number to 10.
3. For the strut at 1.5m depth, select Step_2, go to **Exclude** view mode, select the first layer of strut, and click on **Include** of Assignment window.
4. To apply rotational and displacement fixity at the end node of the strut, go to **Geometry** and add a **Point** to the end of strut. Point fixity at vertex nodes can also be assigned without adding a point.
5. Right click on Step_2, add Point Fixity. Set the z-rotation and x-displacement to zero. Assign the fixity to the point (end of strut).
6. This completes the simulation of the installation of strut. The model at the end of step 2 is as shown in Figure 41.

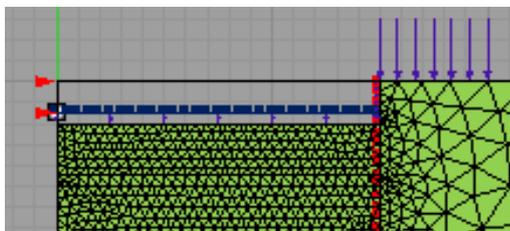


Figure 41. Part of the finite element model at the end of step 2.

10.8 Excavation of other layers

Repeat the procedures described in sections 10.6 and 10.7 for other layer of excavations and installation of struts.

1. Excavation of next 3m depth of soil and installation of 2nd strut at 4.5 m depth.
2. Excavation of next 3m depth of soil and installation of 3rd strut at 7.5 m depth.
3. Excavation of final 1.3m depth of soil to the dredge level.

Figure 42 shows the finite element model at the end of excavation.

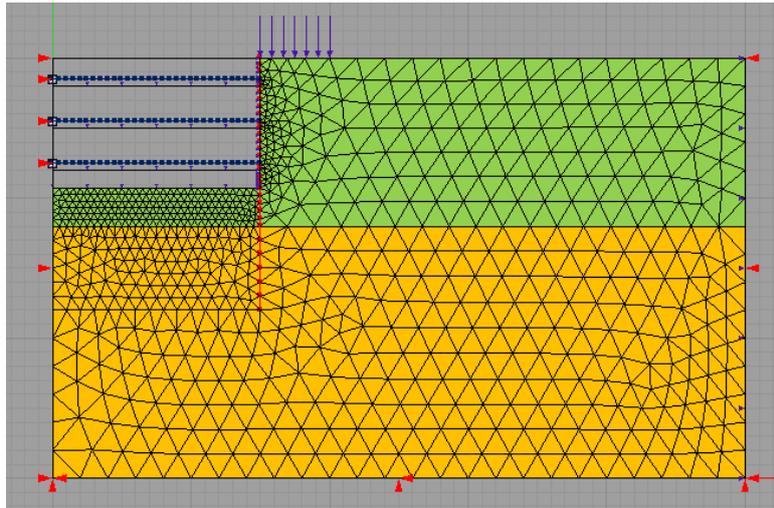


Figure 42. Finite element model at the end of excavation stages.

11 Solution

To solve the problem, click on **Solve** menu at the top of the window. A window will pop up, as shown in Figure 43, for the selection of a solver, options for geometry updates, and options for dealing with the out of balance loads.

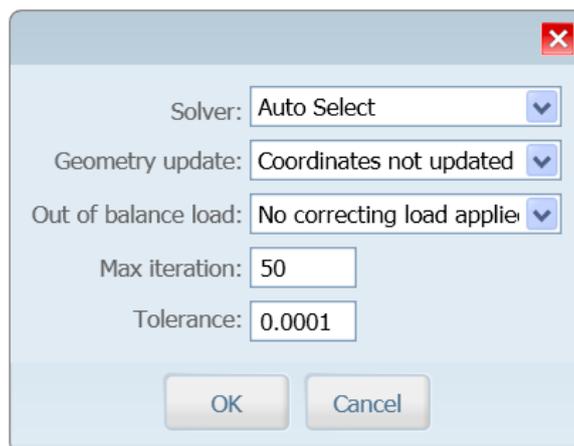


Figure 43. Solution settings.

11.1 Solver options

GeoFEA[®] provides a number of solvers that can suit for a variety of geotechnical problems. Besides a direct solver (Frontal), several iterative solvers are available for large-scale finite element analysis. Note that, not all solvers are equally suitable for every problem. Users may need to select a solver that is suitable depending on the problem and type of analysis. If the user is unsure about the characteristics of different solvers, **Auto Select** is there to help. More details about auto select will be discussed shortly. Only a few solvers are explained here for the illustration purpose.

1. Frontal
2. SBD2-PCG
3. GJ-SQMR

4. GJ-SQMR-SP
5. SBD4-SQMR
6. GJ-QMR
7. SBD5-QMR

The “**Frontal**” technique is an efficient direct solution process while others are iterative solvers. The main idea of frontal solution is to assemble the equations and eliminate the variables at the same time. The Frontal solver is usually robust for the 2D analysis. This is because the size of the linear system from the 2D finite element analysis is usually not very large. Iterative solvers are preferred for large-scale finite element analysis. For iterative solvers, the maximum number of iteration is automatically adjusted by the software and the solution is said to be converged when the relative and true residual norms satisfies the tolerance criteria of 1×10^{-4} and 1×10^{-3} , respectively.

11.1.1 For symmetric stiffness matrix

The stiffness matrix is symmetric for linear elastic, Cam Clay family, and elastic perfectly plastic materials with associated flow rule. If the problem involves material models that generate symmetric stiffness matrices, the preconditioned conjugate gradient (PCG) and symmetric quasi-minimal residual (SQMR) solvers are preferred.

Total stress analysis

SBD2-PCG: The simplified block diagonal preconditioned conjugate gradient method (SBD2-PCG) is an iterative solver using sparse matrix strategy. This solver is preferred for the soil-structure interaction problems with significant stiffness contrasts.

Effective stress analysis

GJ-SQMR: The generalized Jacobi symmetric quasi-minimal residual method (GJ-SQMR) is an iterative solver using element-by-element strategy tailored to solve consolidation problems. This solver is preferred when the available memory (RAM) is not enough for sparse matrix strategy.

GJ-SQMR-SP: The generalized Jacobi symmetric quasi-minimal residual method (GJ-SQMR-SP) is an iterative solver using sparse matrix strategy. This method has a slight edge over GJ-SQMR in terms of speed of the solution, but it will require a larger memory space.

SBD4-SQMR: The simplified block diagonal preconditioned quasi-minimal residual method (SBD4-SQMR) is an iterative solver using sparse matrix strategy. This solver is preferred for the consolidation analysis of soil-structure interaction problems with significant stiffness contrasts.

11.1.2 For nonsymmetric stiffness matrix

If the problem involves material models that generate nonsymmetric stiffness matrices, the quasi-minimal residual (QMR) solver is preferred.

GJ-QMR: The generalized Jacobi preconditioned quasi-minimal residual method (GJ-QMR) is an iterative solver using element-by-element strategy tailored to solve problems with non-symmetric linear system.

SBD5-QMR: The simplified block diagonal preconditioned quasi-minimal residual method (SBD5-QMR) is an iterative solver using sparse matrix strategy. This solver is preferred for

the problems with material models following non-associated flow rule and having significant difference in stiffnesses of the materials.

11.1.3 Auto Select

The **Auto Select** solver option is for elementary users who have no prior knowledge about the solvers. For 2D analysis, GeoFEA® will always select the Frontal solver. For 3D analysis, it selects a suitable iterative solver depending on the type of analysis and material properties.

11.2 Geometry update options

There are three options for “Geometry Update”:

Option 1: Coordinates not updated

Option 2: Apply Updated Lagrangian scheme

Option 1 corresponds to the normal assumption that external loads and internal stresses are assumed to be in equilibrium in relation to the original geometry of the finite element mesh, known as the “small displacement” assumption.

Option 2 includes large strain effects leading to a stiffer load-deformation response near failure and some load enhancement of the load carrying capacity of the soil. The Updated Lagrangian scheme is based on small-strain and large-rotation framework through the incremental procedure.

[**Hints:** If the user is mainly interested in the estimation of a collapse load using elasto-plastic soil models, then it is probably best to use the small displacement approach (**option 1**). Collapse loads can then be compared and should correspond with those obtained from a classical theory of plasticity approach.]

11.3 Out-of-balance load options

There are three options for **Out-of-balance Loads**:

Option 1: No correcting load applied

Option 2: Apply correcting load in next increment

Option 3: Newton-Raphson’s scheme

Option 1 is recommended when elastic models are used and with no “Updated Lagrangian scheme” for the geometry updates.

Options 2 and 3 are recommended for elasto-plastic models. With **option 2**, the “unbalanced” loads (if any) that arises from this procedure are added to the next increment of loading. This scaling back to the yield surface is arbitrary, but ensures that the yield criterion is never violated by an excessive amount.

Under **Option 3**, the Newton-Raphson’s scheme attempts to rectify the errors that arise when constitutive behaviour is based on illegal stress states by evaluating the behaviour in or very near to legal stress space. The Newton-Raphson’s scheme is recommended whenever “**Updated Lagrangian scheme**” is applied to ensure that the calculation is continued until the nonlinear equilibrium equations are satisfied. The maximum number Newton-Raphson iteration is set to 15 and the default convergence tolerance is 0.0001, i.e. 0.01%, of out of balance load. The tolerance will systematically be relaxed by the program if the Newton-Raphson is not effective in redistributing the stresses. Users can adjust the initial value of tolerance and the maximum number of iterations to be allowed.

12 Post processing

Post-processing is the stage to visualize the analysis results graphically so that their contents can easily be interpreted or understood. There are a number of functions related to visualizing the data. One or more functions can be active at a time. They are:

- Deformed shape
- Contour
- Vector
- Bar/Beam
- Line plot
- Point plot

12.1 Deformed shape

Upon successful solution, the post-process window will automatically pop up. Select **Deformed shape** on the left panel and check the box beside **Show deform shape** to view the deformed shape of the model. Adjust the displacement direction and zoom factor for visibility. The default direction is Normal. The results for a particular step can be selected from the drop down list beside **Step** or sliding bar beside it can be used for the same. There is also an option to view **All** or **Mesh only**. The latter option turns off the display of boundary conditions such as load and fixity. Similarly, the mesh can be hidden if you check the box for **Mesh outline only**. **Close** is for exiting from the post-processing stage.

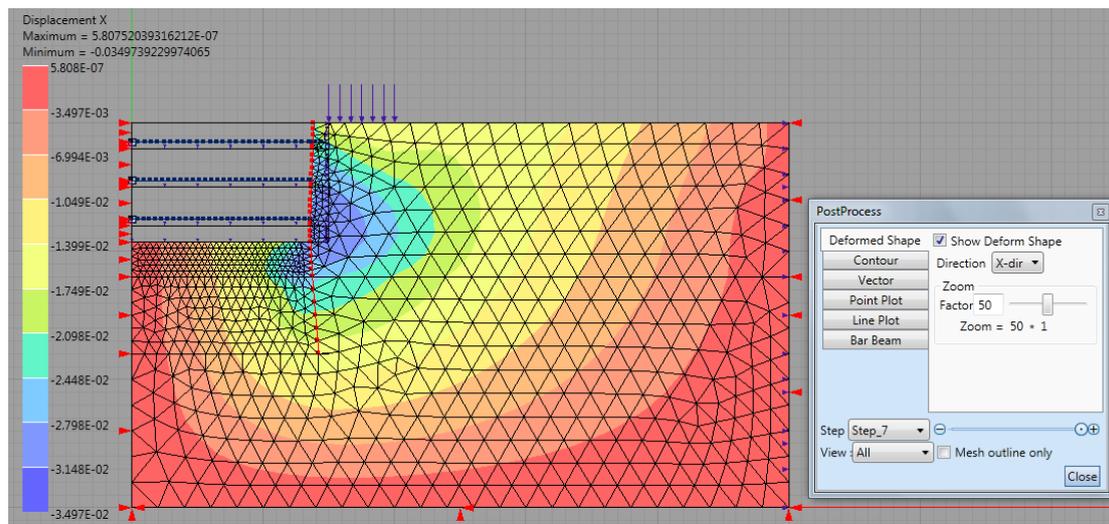


Figure 44. Deformed shape.

12.2 Contour plot

Contour plot is a two-dimensional representation of three-dimensional data. Contouring data is the most frequently used method of visualizing finite element analysis results. The distribution is shown in a band of colors filled between the contour lines. This method is appropriate for visualizing the distribution of scalar data, such as stress, strain or displacement, on a plane or surface. Each contour represents a continuous curve of the same value. Thus, a gradual variation of data on planes or surfaces is scaled by a number of contours. GeoFEA® renders colored contour bands instead of contour lines. The entire span of the scalar value is divided into a number of sub-spans. Each sub-span is represented by a

contour band. The numerical values represented by border lines of contour bands are indicated in the contour scale bar which

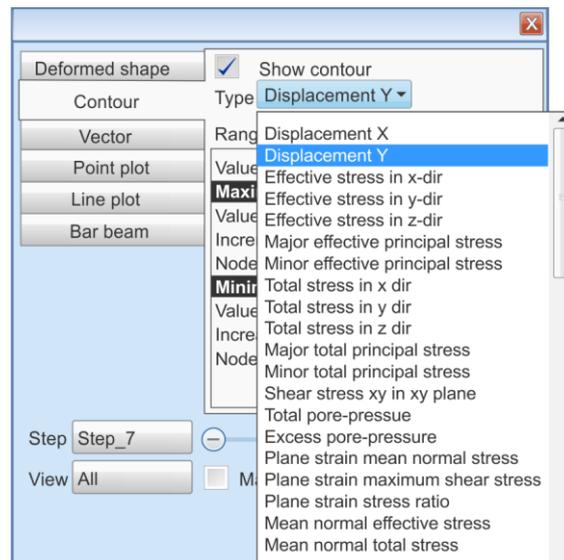


Figure 45. Contour setting.

appears together with a contour image. The ten contour bands are as shown in the left part of Figure 44.

In order to get the contour image of a data set, first select **Contour** item from the left panel of post process window and check the box beside **Show contour**. GeoFEA® provides a number of data types to be plotted in contour image. The type of data can be selected from the dropdown list as shown in Figure 45. The range of data for the contour plot can be adjusted either for the **entire analysis** or for the **current step** or for the **current view** only. Similarly, the range and format of data in the contour bar can also be adjusted from the **Setting** tab. The **Value info** provides the details of maximum and minimum contour values.

12.3 Vector plot

Vector data is defined as a quantity with both magnitude and direction. Displacement and principal stresses are typical examples of vector data. Vector data is usually visualized in arrow form. In order to plot the data in vector plot, its contour plot is required to be plotted beforehand. Select **Vector** item from the left panel of post process window and check the box beside **Show vector** as shown in Figure 46. Select **Mesh only** in view and check the box beside **Mesh outline only** so that the arrows can be seen clearly. GeoFEA® plots an arrow at each vertex node points. The length, width, and color of the arrow as well as the starting position of the arrow can be adjusted. The length of the arrow is proportional to the resultant displacement at each node. The length of the arrow can be adjusted using zoom slide bar.

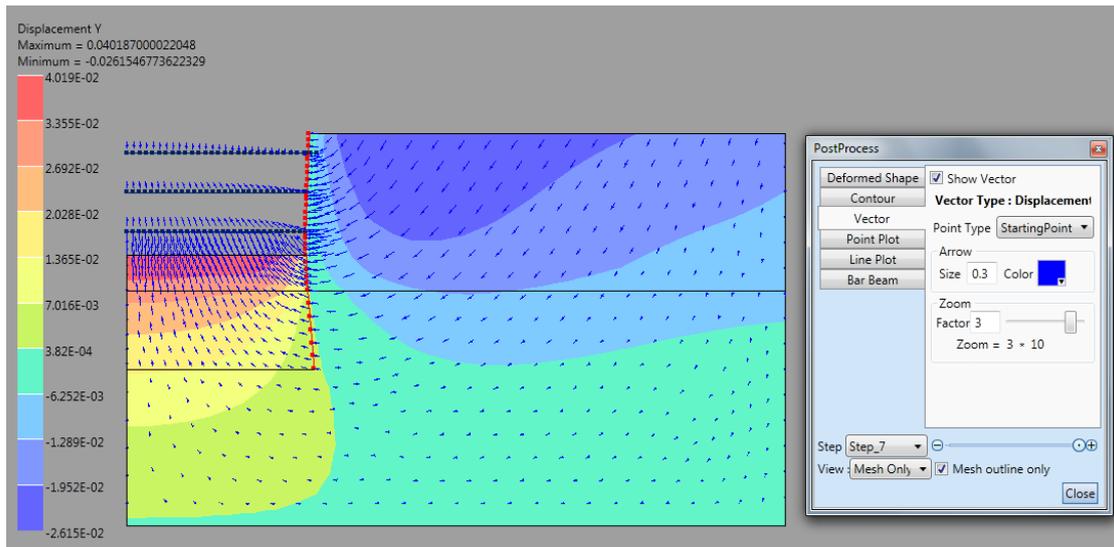


Figure 46. Vector plot.

12.4 Bar/Beam plot

Bar/Beam plots are for plotting the axial forces, shear forces, bending moments, etc. for the structural elements such as bar or beam elements. For a bar/beam plot, users must have to exit from the contour plotting. For a bar/beam plot, select **Bar Beam** from the left panel of post process window, and select the interested Value type from the dropdown list, as shown in Figure 47. Select the interested structural element for plotting and click on **Show** in post process window. It will plot a diagram for that element as shown in Figure 47. The scale of the plot can be adjusted by manually entering the **Graph scale** value or using **Auto set** button. The plotted data can also be exported in a text file by using **Export** option or can be cleared for other plots by using **Clear** option. Click on the displayed diagram, a transparent small window will be visible around the plot to select a different **Type** of plot. The **Scale** and **Step** view for these plots can also be adjusted independently from other plots (if there are many). The diagram can also be dragged to another place, if needed. Turn off the display of boundary conditions by selecting **View | Mesh only** from post process window for a clear visibility of the diagrams.

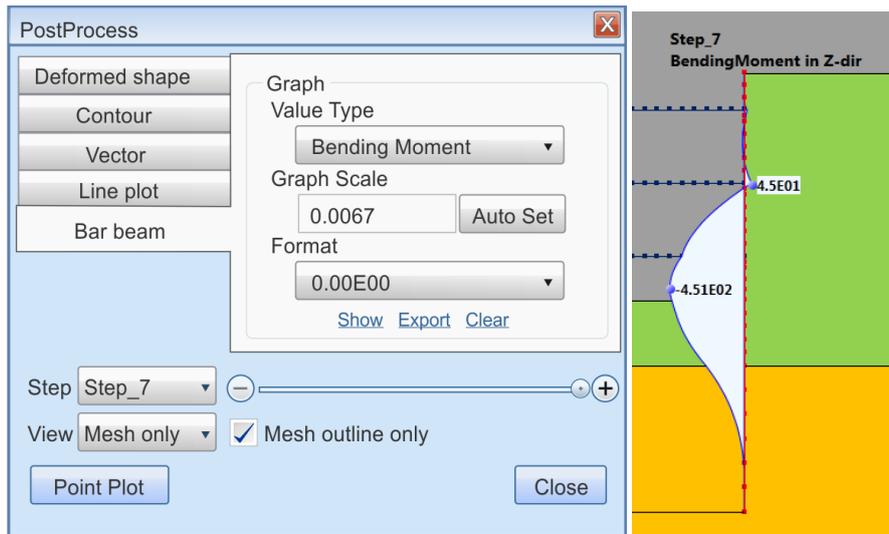


Figure 47. Bar/Beam plot.

12.5 Line plot

Line plot is to assist in viewing the data along a specified line/curve in the form of a graph. It is useful for plotting the data along a cross-sectional line. For line plotting, select **Line plot** from the post process window and click on **Add** to add a line. Either a new line can be drawn or an existing line can be selected for plotting the data. Figure 48 shows a horizontal displacement plot along a line drawn just behind the wall. Note that, depending on the size of the model, the plotting of data for the specified line may take a few or several minutes. Many lines can be added for the plot. These lines can be **copied** for different types of plot, the plot data can be **exported** or they can be **deleted** if not required. The plots can also be hidden by removing check on the **Visible** option.

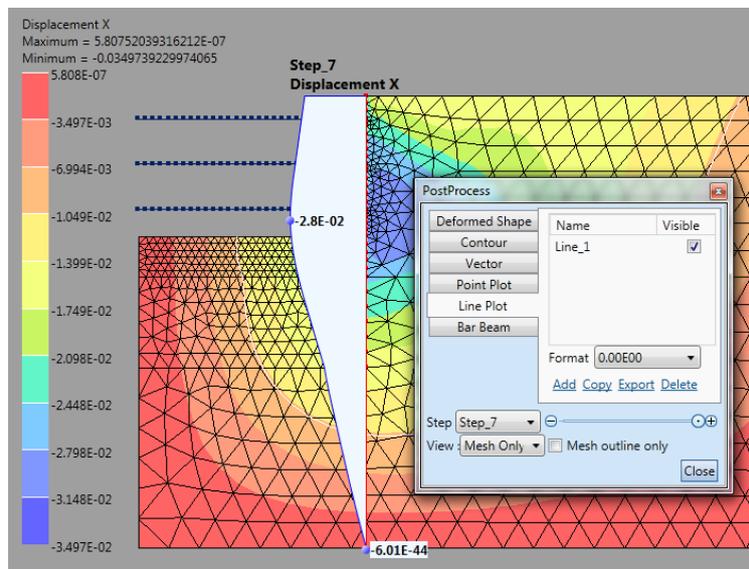


Figure 48. Line plot.

12.6 Point plot

Point plot is very useful to visualize the changes in stresses or displacements at a particular point or node at different stages/time. A point plot graph can be added using **Point plot** |

Add function. Several graphs can be added for different purposes. There is no limit for the number of graphs. In the point plot graph, click on **Add Graph** to add the points for plotting and then click on the desired point in the mesh. The point is not necessarily being the nodal point of the mesh. It can be any location within the mesh. Figure 49a shows the horizontal displacement of the corner

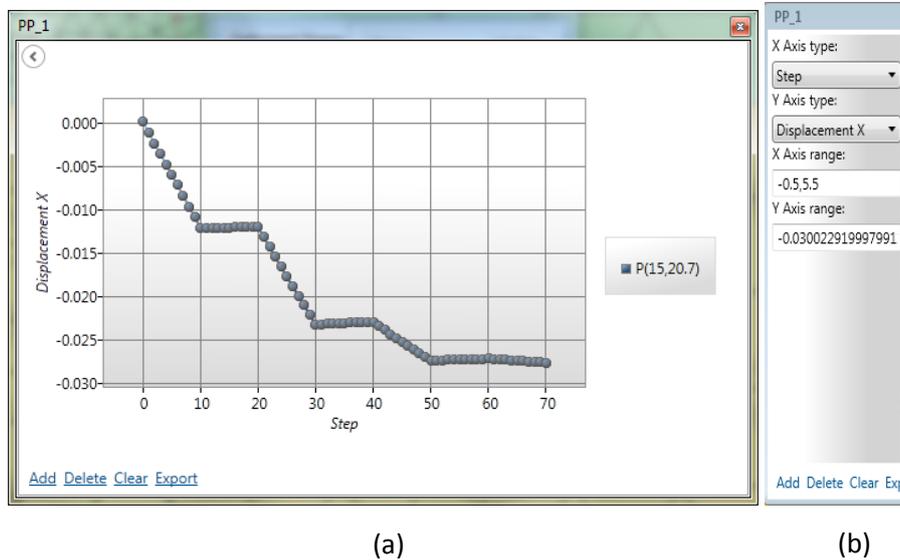


Figure 49. (a) Point plot, (b) Axes selection for point plot.

point at dredge level at different stages. Bringing the cursor near the left side of the plotting window (near y-axis) shows a dialog box for adjusting the type of data for x-axis and y-axis, type of data, and axes ranges as shown in Figure 49b. The plotted data can be **exported** in a text format for other applications such as Excel, etc. Results for several points can be added in the same graph. Use **Delete** button to delete the unnecessary points after selecting it. The **Clear** button cleans up the entire plot. The legend of the plot includes the coordinate of the selected point(s).

13 Known Issues

The following features are known not to work in this version:

- Stop/restart facility
- Plate elements
- Slip elements
- User defined material model

We will be delighted to receive feedbacks or suggestions for the further improvement of the software through the following URL:

<http://www.geosoft.sg/contactUs/contactUs.php>



Redefining simplicity for geotechnical engineers

geofea[®]



GEOSOFT

Website: www.geosoft.sg