EnFEM 2026 – Example Manual

A Finite Element Analysis Program

by

Daqing Xu
Shin-Tower Wang
Luis G. Vasquez
Matthew D. Price

for

ENSOFT, INC.
3003 W. Howard Lane
Austin, Texas 78728
United States of America

(Release Date: March 2026)

COPYRIGHT

SOFTWARE LICENSE AGREEMENT & DISCLAIMER

IMPORTANT NOTICE: *Please carefully read the terms of the license agreement and disclaimer at the end of this book. You signify acceptance with those terms by usage of the software.*

Table of Contents

List of Figures

List of Tables

CHAPTER 1. Introduction

1.1 Program Description

EnFEM is a versatile finite element analysis software designed to simulate the 2D mechanical behavior of solid materials under various loading conditions. It supports plane-strain, plane-stress, and axi-symmetric analyses, with a primary focus on addressing geotechnical engineering challenges. Equipped with an extensive range of linear and non-linear material models, boundary conditions, and element types, EnFEM offers users a comprehensive set of analytical tools to tackle both simple and complex geotechnical problems.

1.2 Manual Description

This manual is designed to help users familiarize themselves with the features of EnFEM through a series of step-by-step tutorial examples focused on common geotechnical problems. Each example introduces different model development skills, making it highly recommended to begin with the first example and progress sequentially through the entire series. Users will build and analyze models from scratch; however, all example problems, including input, mesh, and results data, are provided with the EnFEM installation and can be used as reference materials. The filenames referenced in the manual correspond to those files included with the installation and can be found in the following subdirectory under the root Windows drive (unless a different location was selected during installation):

- *(Root Drive)\Ensoft\EnFEM2025-Examples*

Users are advised to create a new working folder or directory for saving files while following the tutorial examples to avoid overwriting the original example files.

Additional supporting information on EnFEM, including program features and options, is available in the User Manual.

1.3 Font Convention

Different fonts are employed to distinguish specific operations as follows:

- `setWall` : Courier font , used for *key-in* name, coordinate, numbers etc.
- **Run Mesh** : Bold Calibri, used for menu command or button

CHAPTER 2. Examples

2.1 Example 1: Plasticity of Thick Cylinder Subjected to Internal Pressure

The physical configuration of the thick cylinder is shown in Figure 2.1-1. The internal radius is 100-mm and the external radius is 200-mm. Due to symmetry, a quarter-cylinder model is sufficient for analysis, as shown in the right image of Figure 2.1-1. Two radial boundaries are set at 45 degrees from the vertical axis. The units used in this example are "N, mm, second”.

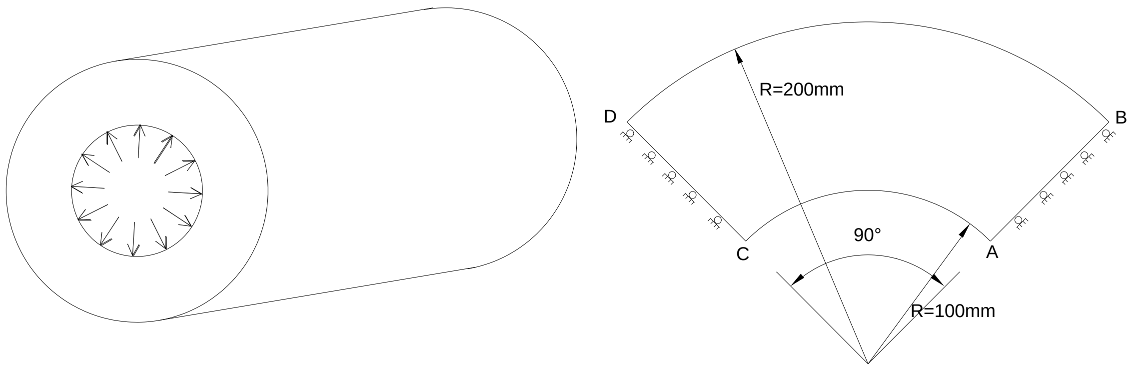


Figure 2.1-1 Configuration of thick cylinder subject to internal pressure

Material Properties:

- Young’s Modulus:..........$E = 2.1 \cdot 10^5\ MPa\ (N/mm^2)$
- Poisson’s ratio:..............$\nu = 0.3$
- Yielding stress:...............$\sigma_y = 240\ MPa\ (N/mm^2)$
- Material model:..............Von Mises plasticity

Boundary Conditions:

- displacement constraints: normal to edge in the local direction
- internal pressure: 200 *MPa*

Model development and analysis steps:

Step 1: Create geometric entities

The coordinates of four corners (seen in Figure 2.1-1) of the fan region are listed in Table 2.1-1:

	A	B	C	D
x	70.71	141.42	-70.71	-141.42
z	70.71	141.42	70.71	141.42

Table 2.1-1: Coordinates of controlling vertices in example 1

To begin using EnFEM, start the application from the Windows Start Menu or by double-clicking **"EnFEM.exe"** in the installation directory. Upon launching, a dialog will appear to define control data for the new project.

In this dialog, update the project title to ***"Thick cylinder subjected to internal pressure".*** For the analysis type, select "Plane Strain". Set the units to "N, mm, second" to define the system of measurement. With these settings, the corresponding stress unit will be N/mm² or MPa. This ensures the project's units are consistent and appropriate for the analysis.

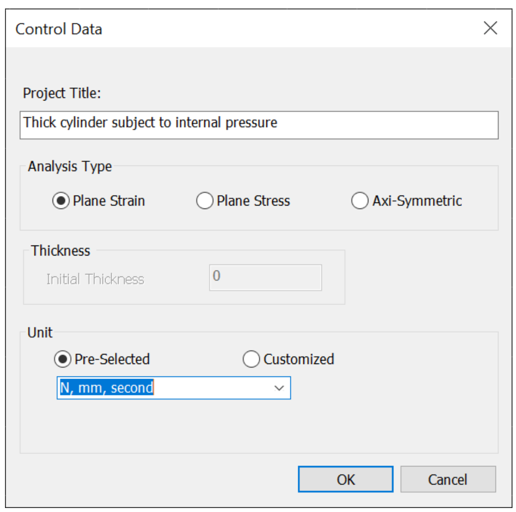


Figure 2.1-2: Define fundamental data in example 1

The default background color for the working zone is black, but users can customize it to their preferred color. To change the background color, go to the menu bar, click on "**Options**" and then select "**Background Color**." From the drop-down list, choose the desired color (see Figure 2.1-3). This background color will apply to both "CAD MODE" and "MESH MODE".

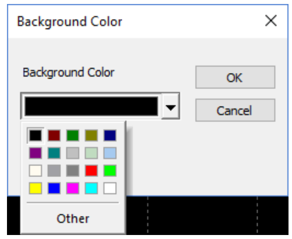


Figure 2.1-3: Change background color for the CAD and Mesh mode

The coordinates of the current cursor are displayed at the bottom of the EnFEM program window, allowing users to conveniently view vertex coordinates. By pressing and holding the **CTRL** key, the mouse cursor snaps to grid intersections or region vertices if the current position is near a snap point.

To check the left and right dimension extents, move the mouse. If the default drawing space is too large or small, users can adjust it via “**Options → Drawing Space”**. It is recommended to set the drawing space slightly larger than the model's dimension extents. In this example, the drawing space is configured as shown in Figure 2.1-4:

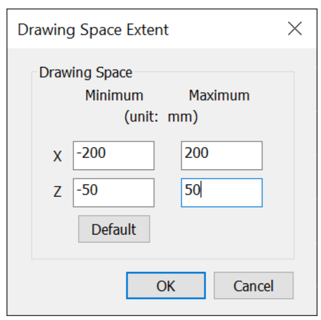


Figure 2.1-4: Setup drawing space

The default grid spacing is “100” which is too large in this example. From the menu bar, click “**Option->Grid Spacing”** to open the grid size dialog and change the grid spacing in both the X and Z directions to “*10*” (Figure 2.1-5).

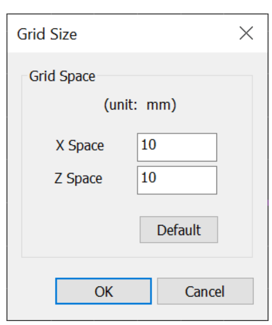


Figure 2.1-5: Set up grid space

Click the toolbar icon  to activate the arc-creation tool, or select "**Create → Arc"** from the menu bar to start the command.

<u>Note</u>: Most toolbar shortcut icons correspond to menu commands, and both methods achieve the same result.

To create an arc in EnFEM, follow these steps:

1. create the vertex of the arc center,
2. create the vertex of the arc start,
3. create the vertex of the arc end.

Arcs are always created in a counter-clockwise direction (Figure 2.1-6).

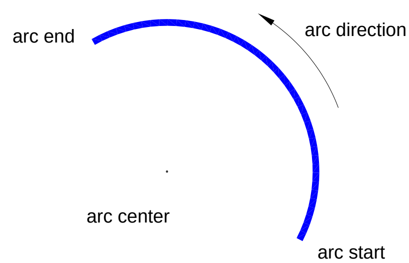


Figure 2.1-6: Create an arc in EnFEM

To input vertex coordinates, use the coordinate input box located below the working zone. Click the input box, enter "`0,0`" (Figure 2.1-7), and press **RETURN** to create the arc center.

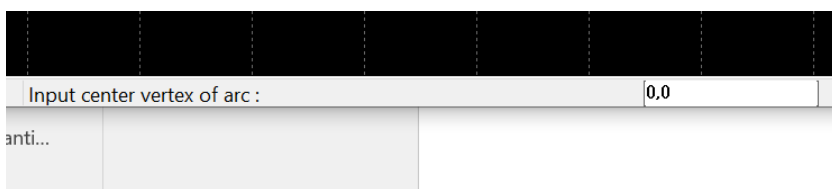


Figure 2.1-7: Create arc center by keying in coordinate

A dashed line from the arc center to the arc start will appear on the screen.

<u>Note:</u> The dashed line color depends on the working zone background:

- **Black background (default):** Dashed line appears cyan.
- **White background:** Dashed line appears red (Figure 2.1-8).

Figure 2.1-8: Dashed line of arc creation

The prompt in the coordinate input box now reads "*Input start vertex of arc*." Enter *"[70.71,70.71]"* and press **RETURN**. Square brackets "[ ]" denote relative coordinates (relative to the arc center), while absolute coordinates are entered without brackets (Figure 2.1-8).

A temporary dashed arc appears, waiting for the arc end vertex. Enter *"[100,90]"* and press **RETURN** to complete the arc creation (Figure 2.1-9). Here, the relative coordinate for the arc end is specified as [radius, arc angle], differing from the start vertex format.

The first arc of the cylinder's internal edge is now complete and displayed in blue.

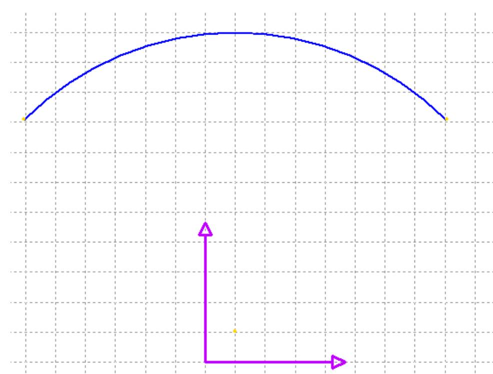


Figure 2.1-9: Create the internal arc by using the coordinate input box

To create the external arc, use the snapping method to place its center vertex, which shares the same center as the internal arc. By default, snap options are enabled in EnFEM.

1. Ensure the arc creation tool is activated .
2. Hold the **CTRL** key and move the mouse cursor near the center of the internal arc.
3. When the cursor changes to a small circle with a cross (the snap icon, Figure 2.1-10), **LEFT-CLICK** to create the arc center.

Note**:** This technique, referred to as the "snap method," will be used throughout this document. Alternatively, the instructions may say "snap to..." followed by a geometric reference, such as "snap to the bottom-right vertex." Use the same method: hold the **CTRL** key and move the cursor until the snap icon appears.

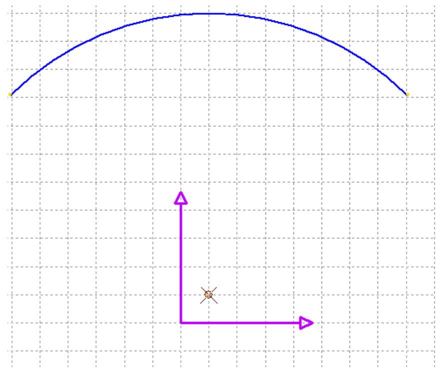


Figure 2.1-10: 'Snap' to existing center point to create a new arc

Enter the relative coordinate "`[141.42,141.42]`" in the coordinate input box and press **RETURN**. Then, enter "`[200,90]`" and press **RETURN** to complete the external arc (Figure 2.1-11).

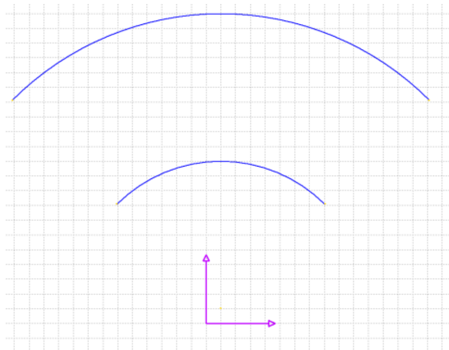


Figure 2.1-11: Create the external arc

Next, create two lines to close the fan shape:

1. Click the line creation tool from the toolbar .
2. Use the snap method (hold the **CTRL** key, move the cursor until it changes to a small crossed-circle, and then **LEFT-CLICK**) to create the start vertex of the new line (Figure 2.1-12)

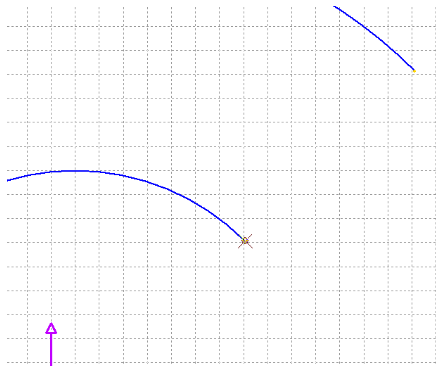


Figure 2.1-12: Snap to arc start to create a new line

While holding the **CTRL** key, move the cursor to the start vertex of the external arc. A dynamically displayed dashed line helps identify the pending line endpoint. When the snap icon (crossed circle ) appears, **LEFT-CLICK** creates the end vertex.

Since line creation in EnFEM is continuous, **RIGHT-CLICK** and select "**End Creation"** to terminate the line command. Repeat these steps to create the line on the left side of the fan shape (Figure 2.1-13)

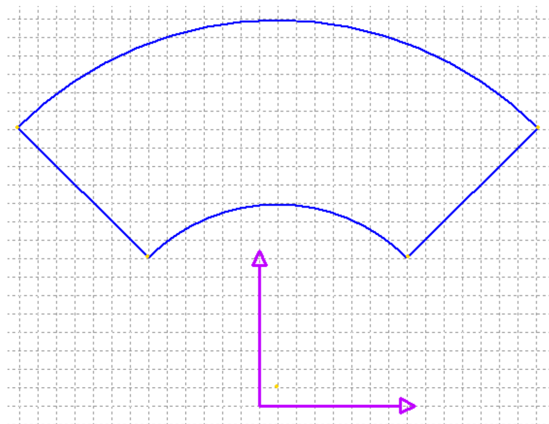


Figure 2.1-13: Independent lines and arcs of the fan region

The fan-shaped region, composed of arcs and lines, is classified as a general region. To convert these components into a region object:

1. From the menu bar, click "**Create → Polygon from Lines/Arcs"**.
2. The coordinate input box prompt changes to **"**Select independent lines/arcs**."**
3. Select the lines and arcs shown in Figure 2.1-13.
4. **RIGHT-CLICK** and select "**End Selection"** to complete the fan region creation

EnFEM provides seven selection filters. By default, the **"**Select All**"** filter is active , allowing selection of multiple entity types with priority given to **vertex > line/arc > region**. For instance, if the cursor is near a vertex, line, and region, only the vertex is selected. Other filters allow selection of specific entity types, such as regions or lines.

EnFEM offers two main methods for selecting geometric objects:

1. **SHIFT-Click Selection:** Hold the **SHIFT** key to select multiple entities individually.
2. **SHIFT-Selection Window:** use a selection window

The selection window is more convenient, which is available in two modes:

- **Contained Window: LEFT-CLICK** any point and drag the cursor to the top-right. The contained window, shown with solid edges (green if the background is black), selects entities fully inside the window.

- **Crossing Window: LEFT-CLICK** any point, hold the button, and drag the cursor to the bottom-left. The crossing window, displayed with dashed edges (yellow if the background is black), selects entities both fully inside and intersecting the window.

For more details on selection techniques, refer to the user manual.

EnFEM can automatically determine the appropriate filter for many operations, such as assigning materials to regions or applying displacement boundary conditions to edges. Selected entities are highlighted for clarity. In Figure 2.1-14, selected lines/arcs are shown in red.

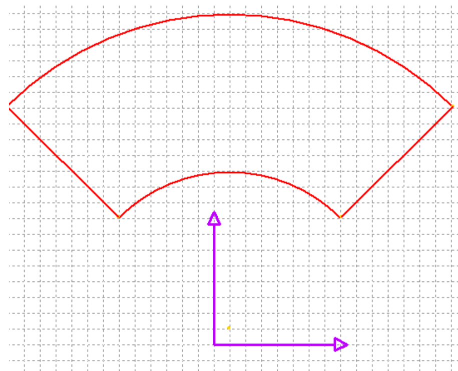


Figure 2.1-14: Selected lines/arcs by a selecting window

Independent lines/arcs are displayed in blue. Once they become part of a region, they lose independence and appear in gray, matching the color of their parent region (Figure 2.1-15).

<u>NOTE:</u> Only independent lines/arcs can be converted into a general polygon.

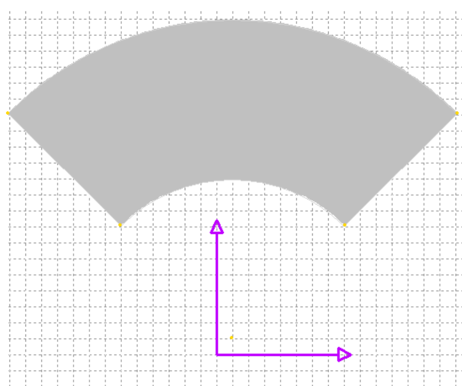


Figure 2.1-15: Convert lines/arcs to the fan region

Click to save the model file as "`Cylinder.sep`".

Step 2: Define stage data

From the menu bar, click "**Data->Stage Management"** to open the stage groups window (Figure 2.1-16).

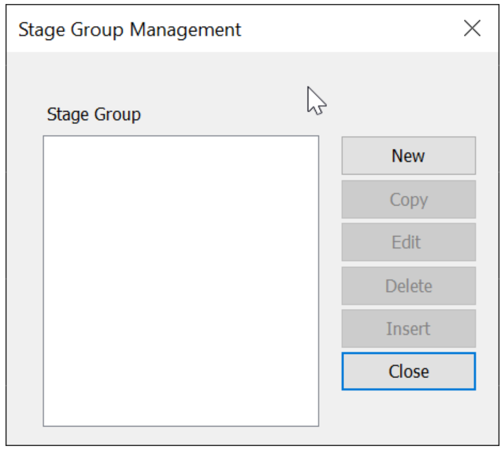


Figure 2.1-16: Stage group management

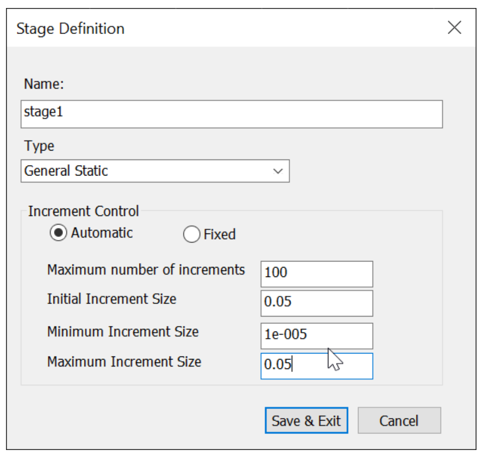


Figure 2.1-17: Define stage data for Example 1

Click "**New"** to define a new stage (Figure 2.1-17):

- Enter the stage name as "`stage1`".
- Select "General Static" as the stage type.
- Keep the increment option set to "Automatic", but update:
  - Initial increment size to "0.05"
  - Maximum increment size to "0.05"

Click "**Save & Exit"** to return to the stage group management dialog.

Click "**Close"** to exit.

Step 3: Define material properties

From the menu bar, click “**Data->Material Properties”** to open the material group management window (Figure 2.1-18).

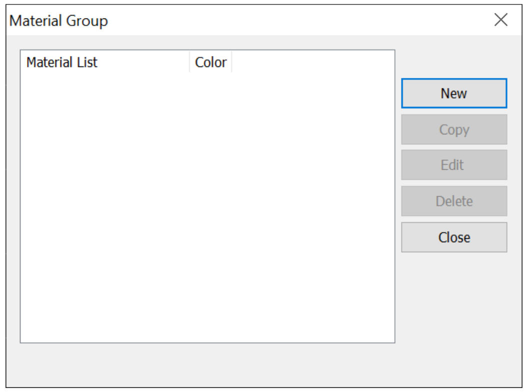


Figure 2.1-18: Dialog of material group management

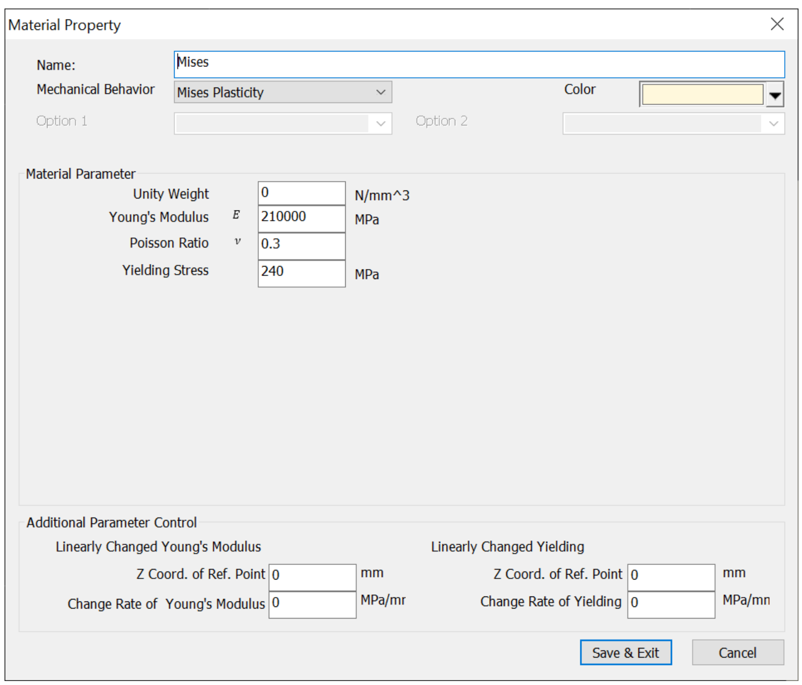


Figure 2.1-19: Define von-Mises plasticity in Example 1

Define new material:

1. Click “**New”** in the material group management window (Figure 2.1-19).
    - Enter the material name: "`Mises`".

- Select "Mises Plasticity" from the mechanical behavior options.
- Input values for Young's modulus, Poisson's ratio, and yielding stress.
- Leave unit weight undefined, as it is not required for this example.

2. Click "**Save & Exit"** to return to the material group management dialog.
3. Click "**Close"** to exit

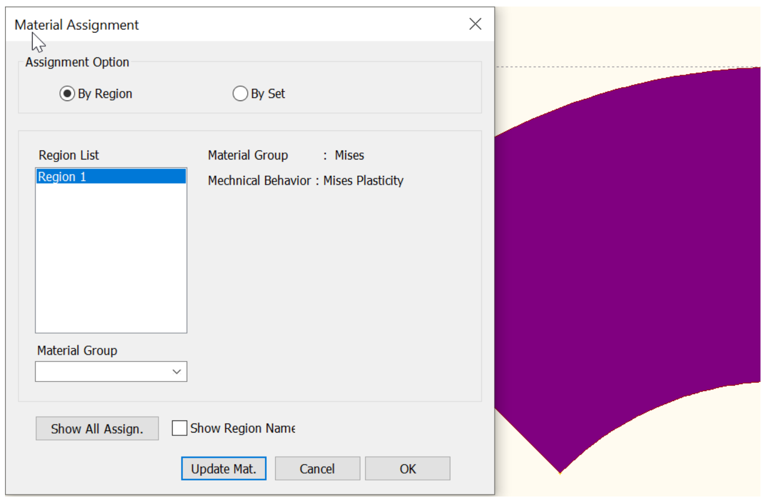


Figure 2.1-20: Assign material property to region object

Assign the material to a region:

1. From the menu bar, click "**Data → Material Assignment"** to open the material assignment dialog.
   - The Region List window displays regions automatically named as "Region + number" by EnFEM.
   - Available materials are listed in the Material Group drop-down list.
2. Select the option "By Individual Region", as this example contains only one region.
3. Click "Region 1" in the region list. The region will be highlighted in the cyan color.
4. Choose the material **"**Mises**"** from the material group drop-down list.
5. Click "**Assign"** to complete the material assignment. The region's color will change to purple (Figure 2.1-20).
6. Click "**Close"** to exit the dialog

Step 4: Define boundary conditions

From the menu bar, click “**Data->Boundary conditions”** to open the dialog for boundary condition management (Figure 2.1-21).

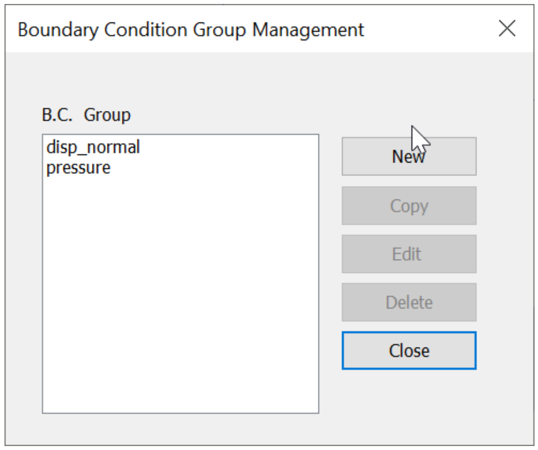


Figure 2.1-21: Dialog for boundary condition management

This example includes two boundary conditions: displacement constraints applied normal to the two straight edges and a uniform pressure of 200 MPa applied to the internal curved edge.

To define the displacement constraint, click “**New”** to open the BC definition dialog. Enter the name *"disp_normal"* and select "Edge Displacement" as the boundary condition type from the drop-down list, as shown in Figure 2.1-22. Choose "In Local Direction" and check the option "Normal to Edge". Keep the value set to **zero**, representing no movement in the normal direction. Click “**Save & Exit”** to return to the group management dialog.

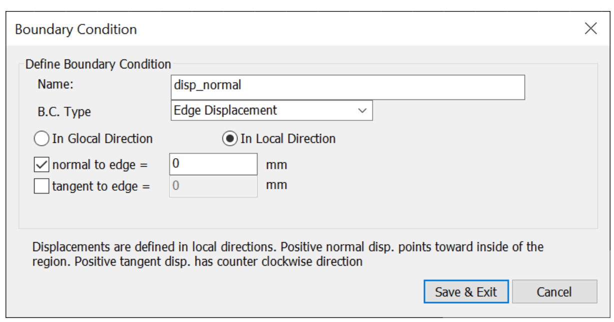


Figure 2.1-22: Define displacement constraint on inclined edges

To define the second boundary condition for the applied pressure, click “**New”** in the BC group management dialog (Figure 2.1-23). Enter the name *"pressure"* and select "Surface Load" as the boundary condition type. Keep the option set to "In Local Direction" and enable the "pres Start" checkbox. Since the pressure is uniform, input *"200"* for both "pres start" and "pres end".

Click “**Save & Exit”** to return to the BC group management dialog, then click “**Close”** to exit.

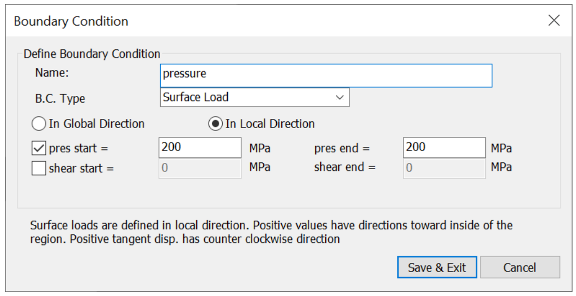


Figure 2.1-23: Define boundary condition of pressure

To assign boundary conditions, click “**Data → Boundary Assignment”** from the menu bar to open the boundary assignment window. Available boundary conditions are listed in the B.C. Group box. By default, the start stage is set to the first stage.

To assign the displacement constraint **"**disp_normal**"**, select it from the list and click “**Assign”**. The selection filter will automatically switch to the line/arc filter . If the filter is disabled, click to enable it again. Hold the **SHIFT** key and click to select both straight edges of the fan region. Once selected, RIGHT-CLICK and choose “**End Selection”** to complete the assignment. Green roller icons will appear next to these edges (Figure 2.1-24).

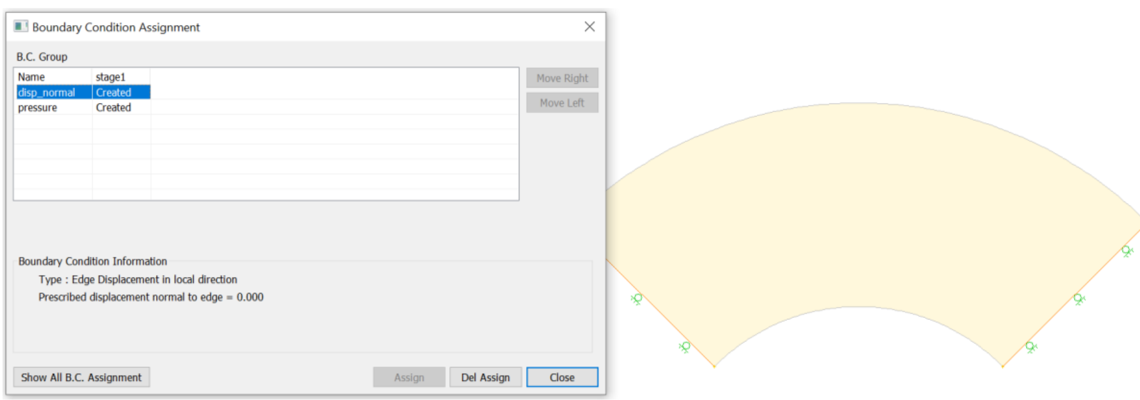


Figure 2.1-24: Normal displacement constraint applied to inclined edges

Next, assign the boundary condition **"**pressure**"**. Select it from the group list and click “**Assign”**. Select the internal arc, RIGHT-CLICK, and choose “**End Selection”**. Pressure symbols will be displayed outside this arc (Figure 2.1-25). The yellow and blue circles represent the start and end points of the applied pressure. These circles are particularly helpful when the pressure is non-uniform.

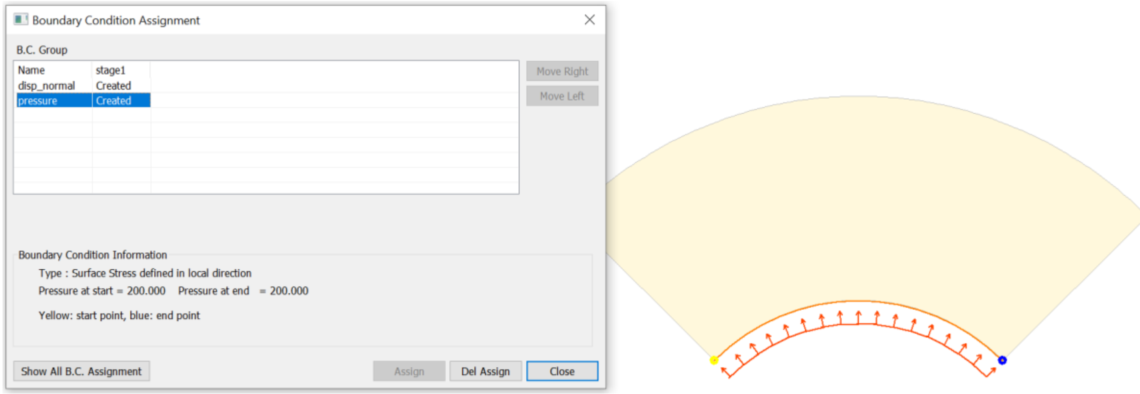


Figure 2.1-25: Applied pressure to the internal arc

NOTE: If a boundary condition is assigned to the wrong geometric entities, select the boundary from the B.C. Group box and click “**Del. Assign**” to remove the assignment. Once deleted, the “**Assign**” button will be re-enabled, allowing users to reselect the correct geometric entities for the boundary.

After completing the assignments, click “**Close**” to exit the dialog and click the “**Save**” icon to save the file.

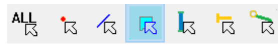
The “**Show All Assignment**” button Figure 2.1-25 displays all applied boundaries. Alternatively, users can click “**View → Show Boundary Assign**”. to achieve the same result.

Step 5: Mesh generation

EnFEM provides two methods for mesh generation:

- **Automatic Mesh Generation:** By clicking “**Pre-Mesh → Auto Mesh Generation**”, EnFEM determines the mesh type, element shape, and density automatically. While this method is user-friendly, it may not produce an optimized mesh.
- **Manual Mesh Generation:** This method gives users complete control over the mesh creation process and involves four steps:
  - Define Mesh Parameters for Solid Regions:
    - Element shape (quadrilateral or triangle)
    - Mesh type (structured or unstructured for solid elements)
    - Element type (Q4, Q8, T3, T6). Beam and geogrid elements always have three nodes, while strut elements have two nodes.
    - Integration rule for quadrilateral solid elements (reduced or full)
  - Define Global Element Size
  - Define Local Mesh Refinement (optional)
  - Run Mesh Generation

This example demonstrates the manual method to generate a structured Q8 mesh:

1. In "CAD-MODE**"**, click "**Pre-Mesh → Advanced Mesh Control"** from the menu bar. The region selection icon will activate automatically .
2. Select the fan region, then RIGHT-CLICK and choose "**End Selection"**. The mesh control dialog opens (Figure 2.1-26).
3. In the dialog:
   - Set "Quadrilateral" as the element shape.
   - Choose "Structured" as the mesh type.
   - Select "Quadratic Q8" as the element option.
   - Check the "Reduced" integration rule (Figure 2.1-26).
4. Click "**Save & Exit"** to complete the setup.

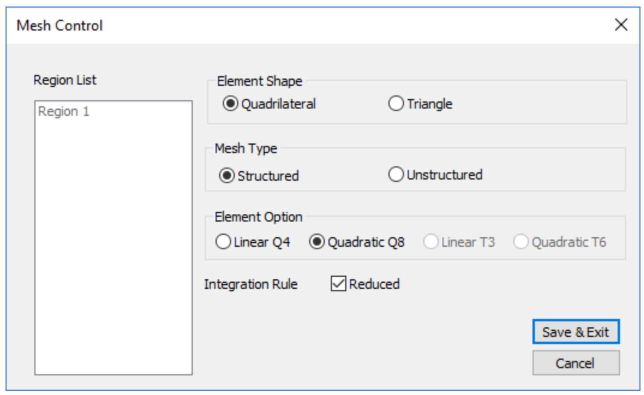


Figure 2.1-26: Dialog for global mesh control

From the menu bar, click **"pre-Mesh->Global Seed Control"** to open the global element size dialog. Change the default size to "*5*" and click **"Assign"**. Red circles will appear along the region edges, with each circle indicating the position of a finite element node (Figure 2.1-27). Click **"Close"** to exit the dialog.

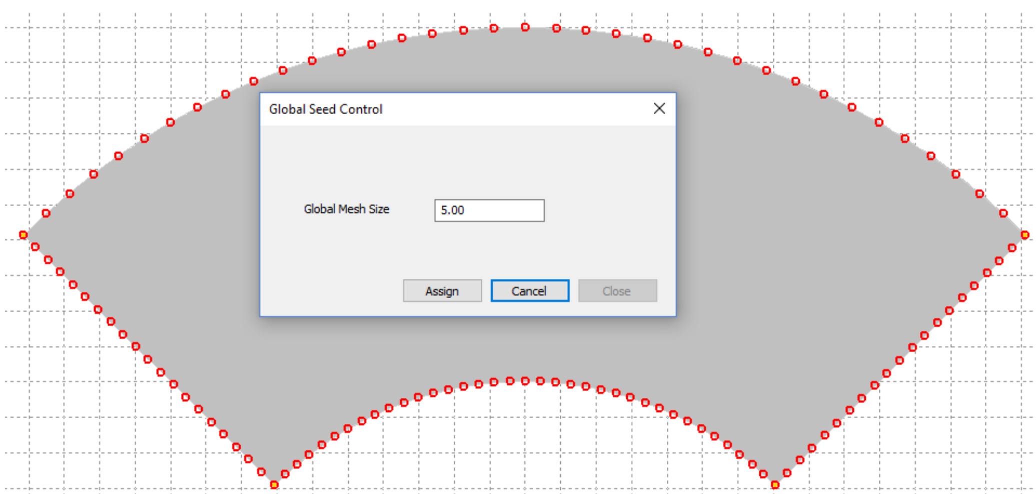


Figure 2.1-27: Global mesh size control

A structured mesh requires equal divisions on opposite edges. To verify and adjust edge divisions in EnFEM:

- **Check Division Numbers**

  1. From the menu bar, click "**Pre-Mesh → Show Edge Division"** to display division numbers.
  2. Division numbers are shown in green for solid elements and yellow for structural elements (e.g., beams, geogrids, struts). Numbers for internal edges are displayed inside the parent region, while shared edges show numbers for both adjoining regions.

- **Resolve Unequal Divisions**
  In Figure 2.1-28, the two arcs have unequal division numbers due to differing arc lengths, violating the structured mesh requirement.

  1. From the menu bar, click "**Pre-Mesh → Local Seed Control"**.
  2. The line selection icon will activate automatically 

. Select the external arc, then RIGHT-CLICK and select "**End Selection"**.
  3. In the local mesh refinement dialog (Figure 2.1-29), keep all settings at their default but update the division number to *36*.
  4. Click "**Assign"** and then "**Close"** to finalize the adjustment.

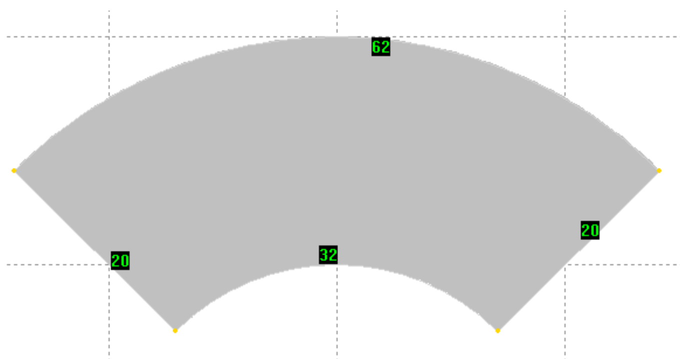


Figure 2.1-28: Show division number on edges

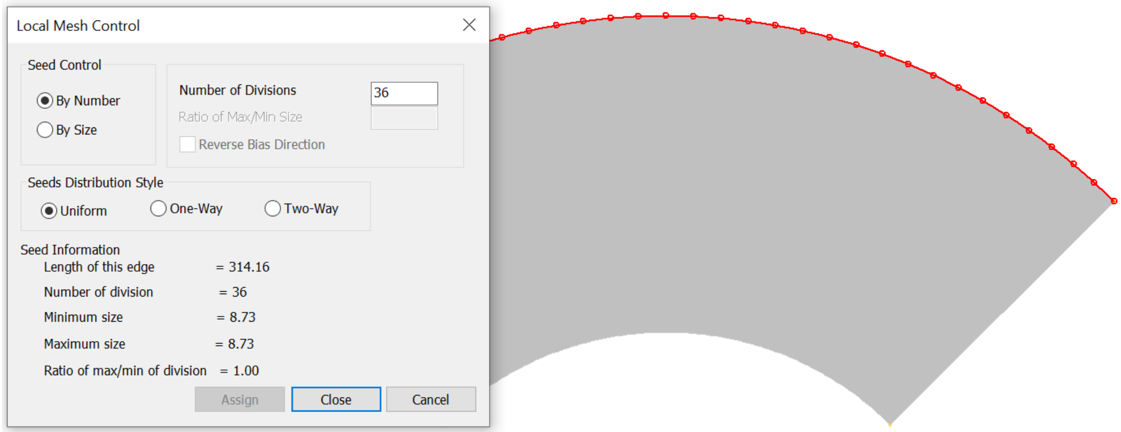


Figure 2.1-29: Dialog for local mesh refinement

To adjust the division numbers, first select the external arc in the "Local Seed Control**"** dialog and set its division number to *36*. Click "**Assign"** to confirm the changes and verify the discretization details displayed at the bottom of the dialog. Afterward, repeat the same steps for the internal arc edge, ensuring its division number is also set to *36*. Once completed, click "**Close"** to exit the dialog.

EnFEM offers several tools to verify the mesh setup. Using "**Pre-Mesh → Show Edge Division"** will display division numbers for each edge. Selecting "**Pre-Mesh → Show Seeds"** will visualize the seed distribution across the edges. Finally, "**Pre-Mesh → Show Mesh Assignment"** allows inspection of the assigned element types, such as tetrahedrons or quadrilaterals.

When satisfied with the setup, proceed to generate the mesh by clicking "**Pre-Mesh → Run Mesh Generation"**. After the process is completed, the mesh will load automatically into the working zone for review, as shown in Figure 2.1-30.

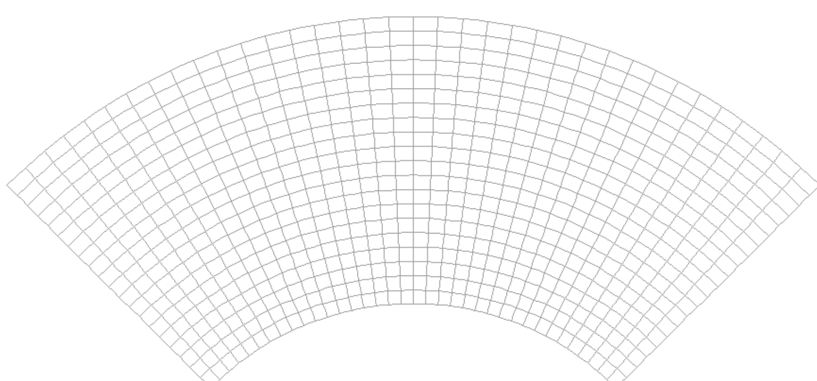


Figure 2.1-30: Generated quadrilateral quadratic mesh in Example 1

NOTE: Once the mesh is generated, EnFEM automatically switches into the "MESH-MODE". Uses may click to go back to "CAD MODE" and click to go back to "MESH MODE" again.

For quadrilateral element shapes, achieving square-like elements is critical for optimal performance. Square elements provide better numerical accuracy and stability during simulations. A well-designed mesh should strive to maximize the proportion of square-like elements while minimizing distortion or irregularity in element shapes.

This principle ensures better convergence and more reliable results in finite element analyses. Adjusting edge divisions, element types, and refining local mesh parameters can help achieve this objective in EnFEM.

Click to save the file.

Step 6: Analysis

To configure the analysis settings for the model, click **"Analysis -> Analysis Setting"** in the menu bar. This action opens the analysis setting dialog, as shown in Figure 2.1-31.

For this example, the default settings are appropriate and do not require any modification. Simply click **"OK"** to accept the default configuration and close the dialog. This completes the analysis setup step.

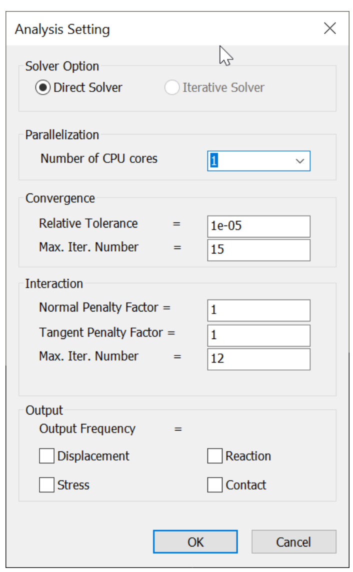


Figure 2.1-31: Dialog of analysis setting

To run the analysis, click **"Analysis -> Run Analysis"** from the menu bar. This opens the job control window, as shown in Figure 2.1-32. Initially, no stages are selected for a new project.

Select **"stage1"** by marking the checkbox next to it and then click **"Run"** to submit the analysis job. A command window will open, displaying convergence details for each load increment.

Once the analysis is completed or terminated due to errors, the log file (e.g., "Example1-cylinder.log") will open automatically. Reviewing the log file is strongly recommended to assess the convergence history. In the event of errors, the log file often contains valuable insights for troubleshooting and resolving issues.

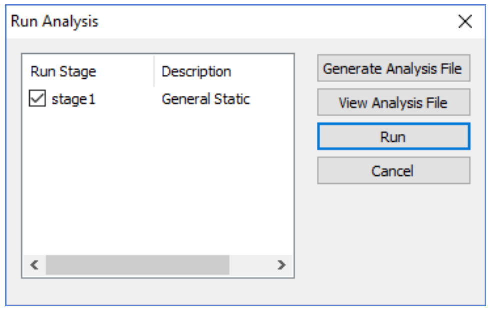


Figure 2.1-32: Run analysis dialog

Step 7: Post-processing of result

To visualize the analysis results, click the “RESULT-MODE” icon , represented by a specific icon (Figure 2.1-33). By default, the finite element mesh is displayed in a grey color.

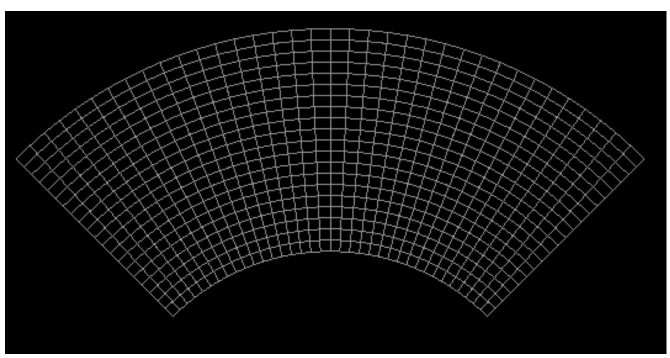


Figure 2.1-33: Initial mesh is displayed after switching to result mode

Visualization Tools in EnFEM:

- Contour plot: Displays distribution of variables across the model.
- Displacement vector image: Highlights displacements in solid elements.
- X-Y curve plot: Shows data for a specific node or along a node path (solid elements only).
- Cutline plot: Provides a sectional view of results for solid elements.
- Beam, geogrid, and strut results: Summarizes results for these structural components.

Click the contour plot icon  to open the contour plot dialog, as shown in Figure 2.1-34.

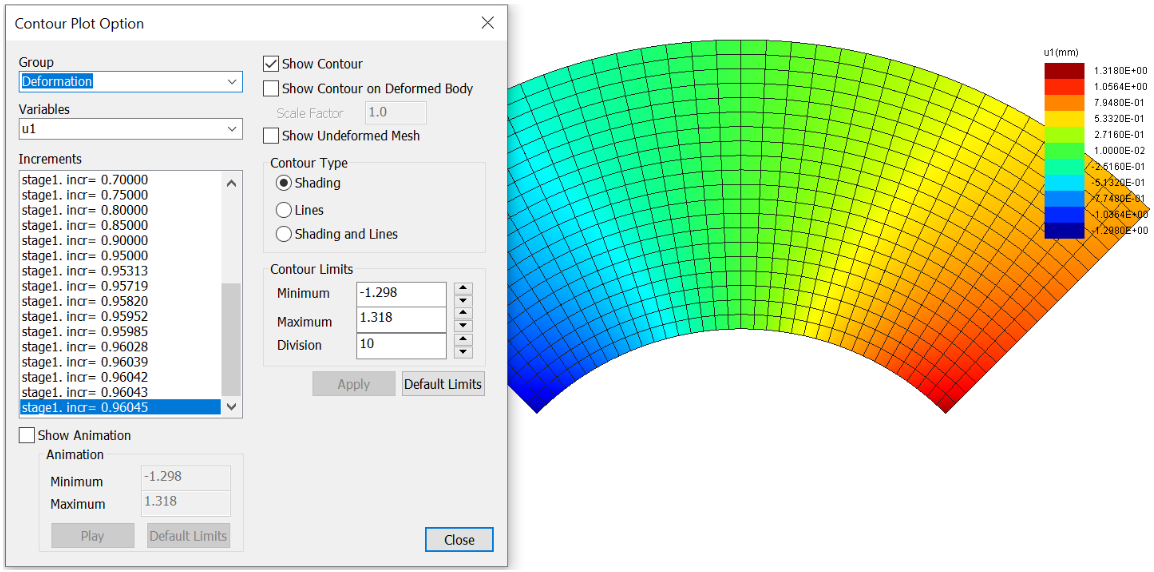


Figure 2.1-34: Contour plot in Example 1

EnFEM provides four main categories of output: displacement, stress, strain, and soil variables. Each category contains several variables with detailed explanations available in the user manual. The "increment**"** displays all successfully completed increments, and the contour plot in the working zone updates automatically when a variable is selected or the increment is changed.

Key Features in the Contour Plot Dialog:

- Deformed Body Display: Setting the checkmark for "Show Contour on Deformed Body" overlays the contour on the deformed mesh. Users can adjust scale factors to emphasize deformations.
- Undeformed Mesh: Enabling the "Show Undeformed Mesh" option overlays the original undeformed mesh in dark green.
- Contour Types: Choose between "Shading," "Lines," or "Shading and Lines" for contour visualization. The default setting is shading.
- Customizable Contour Limits: Users can manually set contour limits to highlight specific ranges. Changes are applied by clicking "**Apply**."

For this example, the Mises stress contour is selected for investigation. Navigate to the "Stress" group and select "Equivalent Shear q**"**. At increment 0.8, the contour plot shows that the plastic zone extends to roughly 40% of the cylinder's thickness (Figure 2.1-35).

Animation Feature**:** Enable the "Show Animation" checkbox and click the animation button to play the contour animation. Users can refine animation by setting custom contour limits using "Minimum/Maximum" values.

Click “**Close”** to exit the contour plot dialog when the visualization is complete.

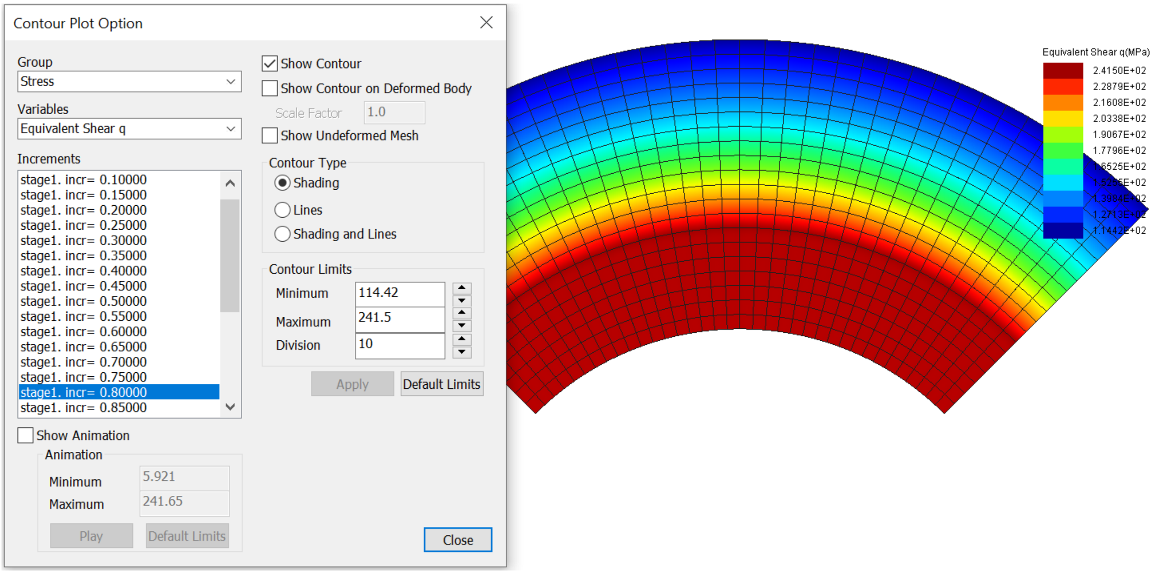


Figure 2.1-35: Contour of Equivalent Shear stress at increment=0.8

To visualize the animation of the contour plot, set the checkmark for “Show Animation” and then click the animation button 

to play it. Contour limits for the animation can be customized by specifying values in the “Minimum/Maximum” fields. After reviewing the animation, click “**Close”** to exit the contour plot dialog.

Next, click the displacement vector plot icon  to open its dialog (Figure 2.1-36). The displacement field is displayed as small red arrows, where the size of each arrow is proportional to the displacement magnitude.

Options for Displacement Vector Plot:

- Users can select to plot displacement vectors for the overall magnitude, or for the **X** or **Z** component individually.
- Adjusting the **multiplier** value changes the overall scale of the displayed vectors, making it easier to observe small or large displacements.

This tool provides a clear visualization of the displacement field and its components, offering valuable insights into the model's deformation behavior.

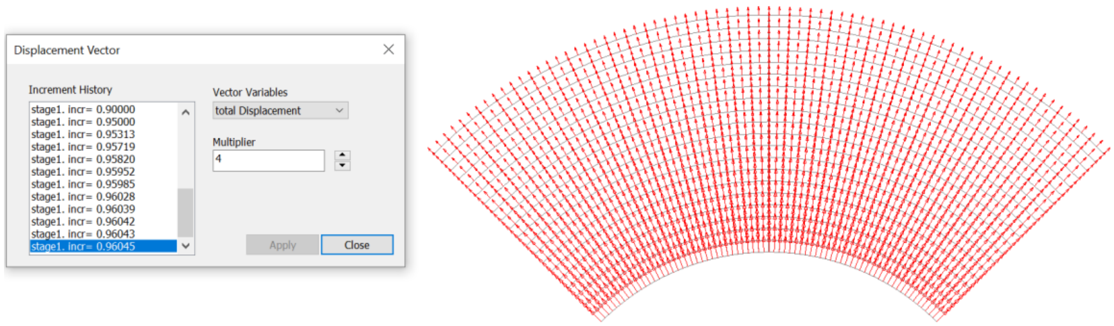


Figure 2.1-36: Plot displacement vector of Example 1

To create an X-Y plot, click the **X-Y Plot** icon  to open the dialog. X-Y plots in EnFEM operate in two modes: Historic Result or Field Results. Historic Result is used to plot the curve of one element node over time or increments, such as displacement versus increments. Field Results are used to plot the curve along a user-defined path, such as stress distribution along a segment of the model.

To create a Historic Result plot, first enable node numbers by clicking "**Mesh → Show Node Numbers"** from the menu bar. Finite element node numbers will be displayed in yellow on the model. Use the zoom tool to focus on the midpoint of the inner arc and identify the node number of the lowest point on the symmetric axis, which is labeled as 399 in this example.

In the X-Y plot dialog, ensure that Historic Result is selected. Enter *399* in the "Label of Single Node" field. From the Category list, select "Primary Variable", then choose "u2" (Vertical Displacement) from the Variables list. Click "**Plot"** to display the curve of Vertical Displacement (u2) versus Load Increments in the graph region of the dialog, as shown in Figure 2.1-37.

This method allows tracking the performance of specific nodes through the simulation stages and increments effectively.

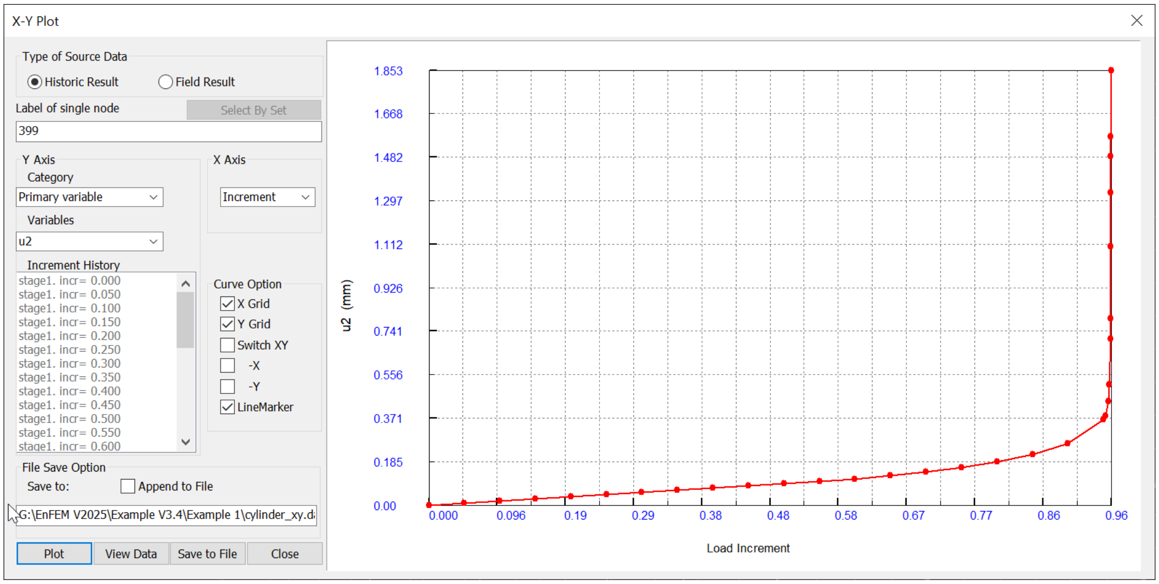


Figure 2.1-37: X-Y plot of the vertical displacement on the internal boundary

Now switch the plot type selection to **"Field Result"**. Change the content of "Label of node series" to "*399:379:-1*" and select **"S11"** as the plot variable. Select **"increment=0.9"** from the list of increment. Click "**Plot**" to display the distribution of circumferential stress along the symmetric axis (Figure 2.1-38).

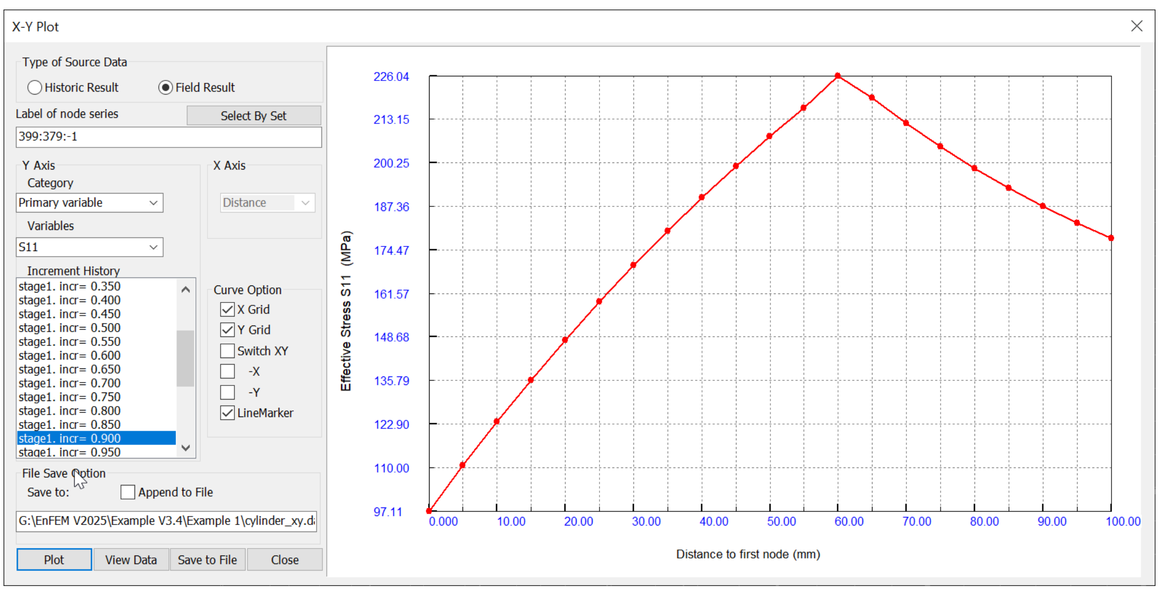


Figure 2.1-38: X-Y plot of circumferential stress of Example 1

The "**Save to File"** button allows users to save X-Y plot data in ASCII format for further analysis or documentation. By default, the saved file is named "***_xy.dat," where *** corresponds to the project file's name. For this example, the default filename for the X-Y plot is "Example 1 cylinder_xy.dat", and it is stored in the same directory as the project file.

The filename and save location can be manually adjusted if needed. Additionally, enabling the "Append to File" option ensures that new data is appended to the end of the existing file, preserving earlier results. If this option is unchecked, clicking "**Save to File"** will overwrite the file's contents with the new plot data. This functionality provides flexibility for managing and recording analysis results efficiently.

2.2 Example 2: Analysis of shallow foundation subjected to vertical settlement

The configuration of this example is illustrated in Figure 2.2-1. Because of the symmetry about the Z-axis, only the right-half of the foundation region is modeled and simulated to reduce computational effort while maintaining accuracy.

For this example, English units are used:

- Force: lbf
- Length: inches (in)
- Time: seconds (s)

These units ensure that derived quantities, such as stress, are expressed in lbf/in² (psi).

The modeled region includes the soil and the foundation structure, with appropriate boundary conditions and material properties assigned. Symmetry conditions along the Z-axis are incorporated by fixing displacement in the X-direction along this boundary.

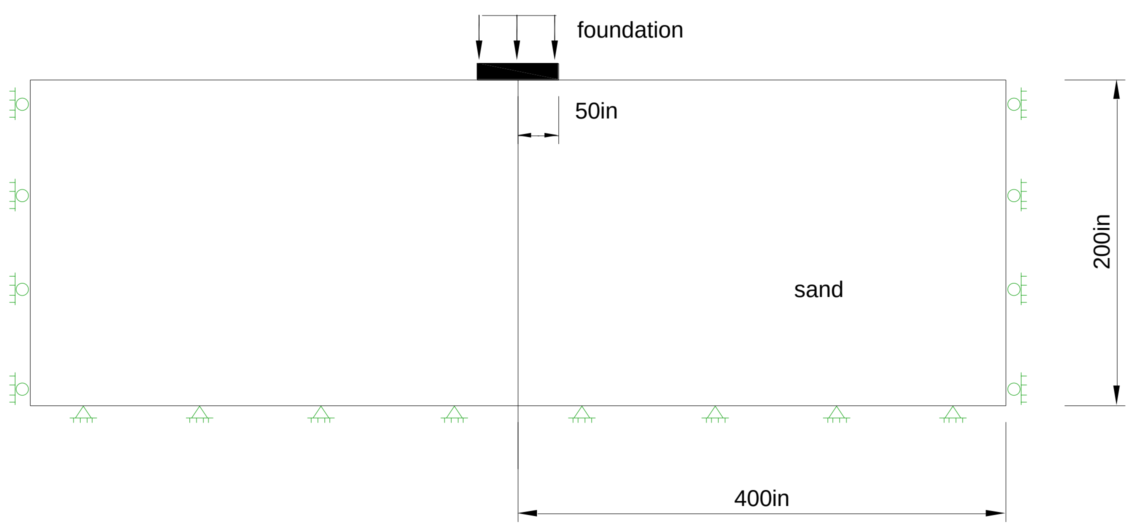


Figure 2.2-1: Configuration of shallow foundation analysis

Material Properties:

- Unit weight:.. $\gamma = 0.089\ pci$
- Young's Modulus:................................ $E = 20000\ psi$
- Poisson's ratio:.................................... $\nu = 0.3$
- Cohesive strength:............................. $c = 1\ psi$
- Friction angle:..................................... $\phi = 30°$
- Dilation angle:..................................... $\psi = 20°$
- Material model:.................................. Mohr-Coulomb plasticity

Boundary Conditions:

- Displacement constraints in X-direction
- Displacement constraints in X and Z-directions
- Gravity force on soil
- 2-inch settlement of rigid foundation

In this example, the foundation (not simulated) is assumed to be rigid. No contact interface is simulated. The interface between soil and foundation is frictionless.

Model development and analysis steps:

Step1: Create geometric entities

Start EnFEM, click **"File->New"** to start a new project. Input project title as "`Analysis of Shallow foundation subject to vertical displacement`", keep analysis type as "Plane-Strain", and select "lbf, in, second" from the unit option (Figure 2.2-2).

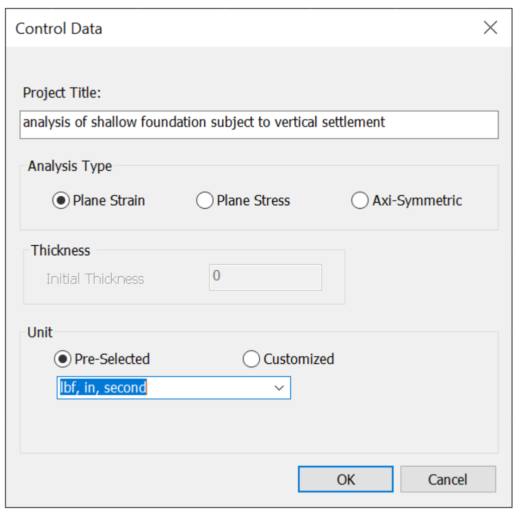


Figure 2.2-2: Initial data for shallow foundation example

Click **"Options->Drawing Space"** to update the drawing space extents (Figure 2.2-3). Click **"OK"** to exit.

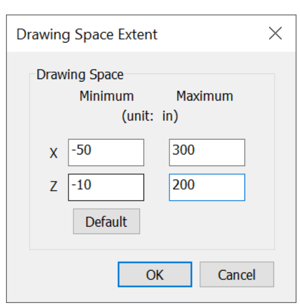


Figure 2.2-3: Setup drawing space for shallow foundation example

Click **“Options->Grid Spacing”** to open the grid spacing dialog. Change the grid space to *“10”* for both the X and Z-directions (Figure 2.2-4).

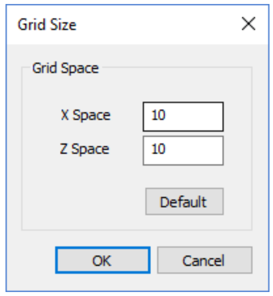


Figure 2.2-4: Setup grid space for shallow foundation example

To define the foundation region, start by clicking the polygon creation icon  from the toolbar. In the drawing space, use the snap-method for precise placement. Begin at the working zone origin with coordinates (x=0, z=0). Hold the **CTRL** key and create the next point at (x=0, z=200). Continue adding points at (x=50, z=200), (x=400, z=200), and (x=400, z=0). Once all vertices are placed, right-click and select "**Close Polygon**" to finalize the shape. The polygon representing the foundation region is now complete, as depicted in Figure 2.2-5.

Figure 2.2-5: Create polygon region for shallow foundation example

Click  to save the model file as "*Shallow foundation.sep*".

Step 2: Define stage data

In “CAD MODE”, click **"Data->Stage Management"** to open the stage group management window. Click **"New"** to add one new stage. Input *"Initial Stress"* as the stage name. Select **"**Initial Soil Stress**"** as the stage type (Figure 2.2-6).

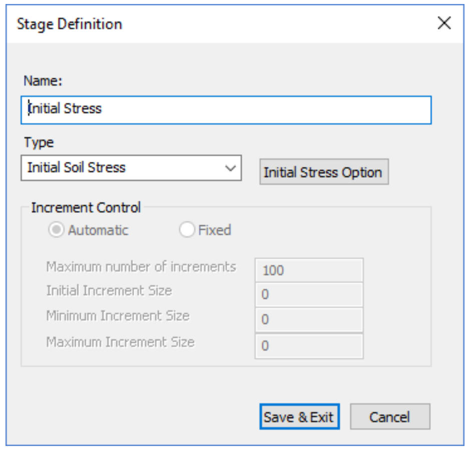


Figure 2.2-6: Define initial stress stage for shallow foundation example

To define the initial stress, click “**Initial Stress Option”** to open the initial stress dialog. Choose the “K0 Method**”** option. Click “**Add”**, then select the soil region in the display window. Once the region is selected, right-click and select “**End Selection”** to add it to the Region List, as shown in Figure 2.2-7.

Next, click “**Input Data”** to open a new dialog, as shown in Figure 2.2-8. Click “**Add Row”** to begin entering data. Specify the z-coordinates for the layer's top and bottom, and input the vertical stresses at these positions. When finished, click “**Save & Exit”** to close the input data dialog. Return to the main window by clicking “**OK”**, followed by “**Save & Exit”**. You will then be back in the stage group management dialog.

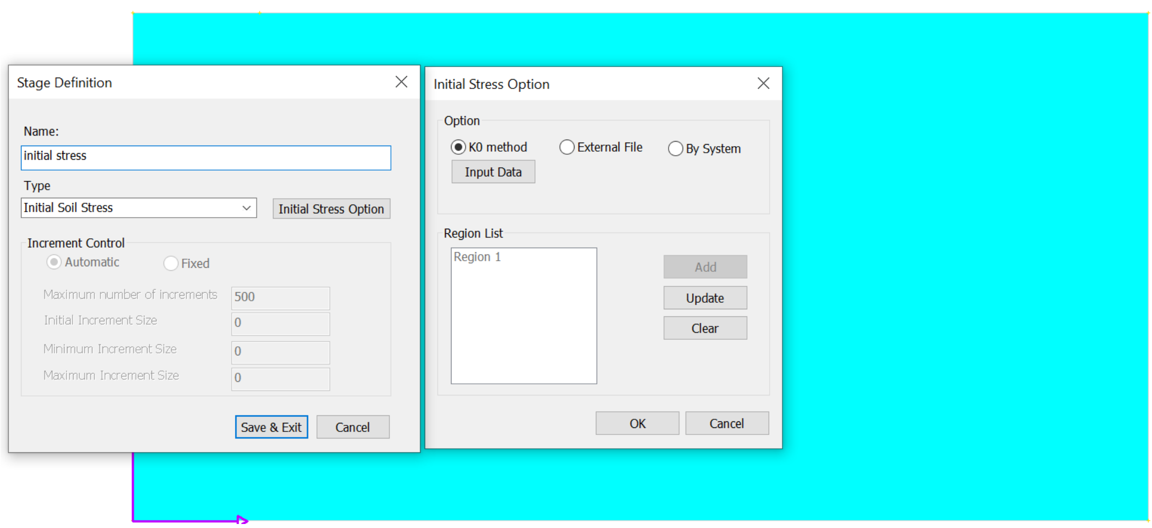


Figure 2.2-7: Define initial stress options for Example 2

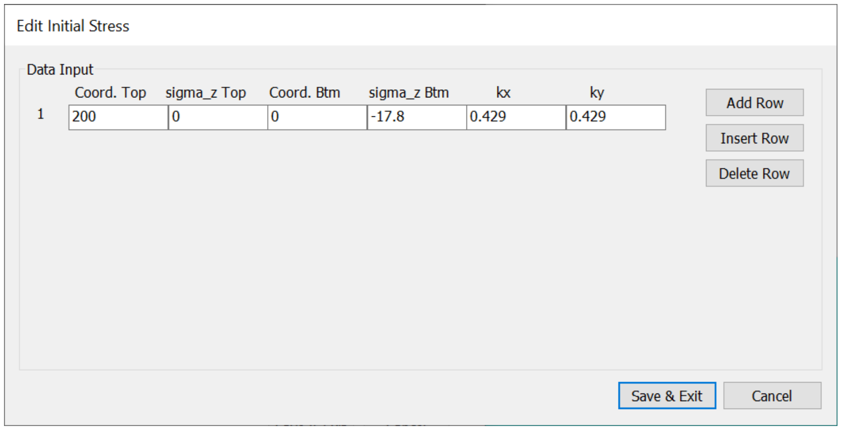


Figure 2.2-8: Define initial soil stress at controlling point

Back at the stage group management window, click **"New"** to define the second stage. Input `"displacement"` as the stage name. Select "General Static" as the stage type. Keep the increment control as "Automatic". Change the initial increment size and the maximum increment size to `"0.05"` (Figure 2.2-9). Click **"Save & Exit"** to go back to the stage group management dialog. Click **"Close"** to exit.

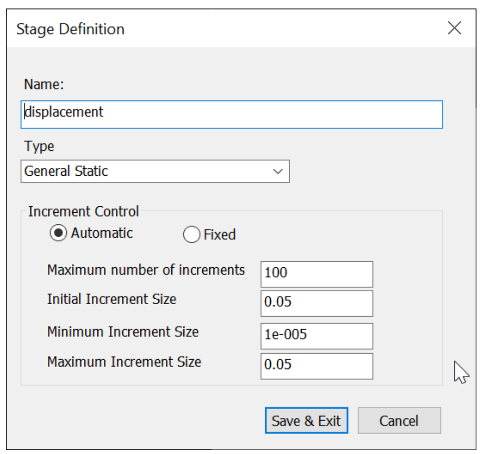


Figure 2.2-9: Define loading stage for shallow foundation example

Step 3: Define material properties

To define the material properties, navigate to the menu bar and click "**Data -> Material Properties"** to open the material group management window. Click "**New"** to create a new material. Select the "Mohr-Coulomb" material type and input the material properties as illustrated in Figure 2.2-10. Note that the dilation angle differs from the friction angle, reflecting the application of a non-associated flow rule to control soil expansion.

After entering the necessary values, click "**Save & Exit"** to return to the material group management dialog. Finally, click "**Close"** to exit the material properties window.

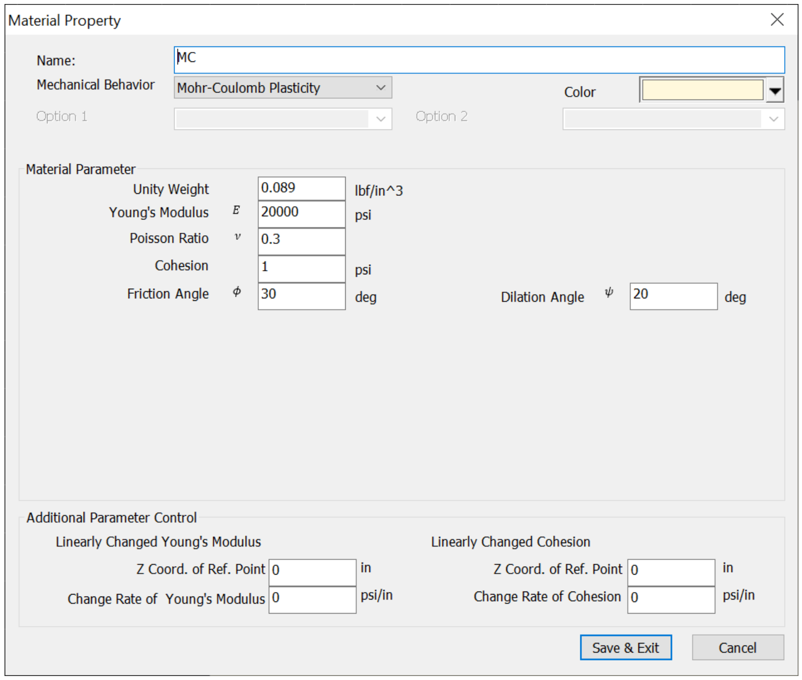


Figure 2.2-10: Define Mohr-Coulomb material for shallow foundation example

From the menu bar, click **"Data->Material Assignment"**. Pick the region name "Region 1" from region list, select **"MC"** from the drop down list for the material group, then click **"Assign"** (Figure 2.2-11).

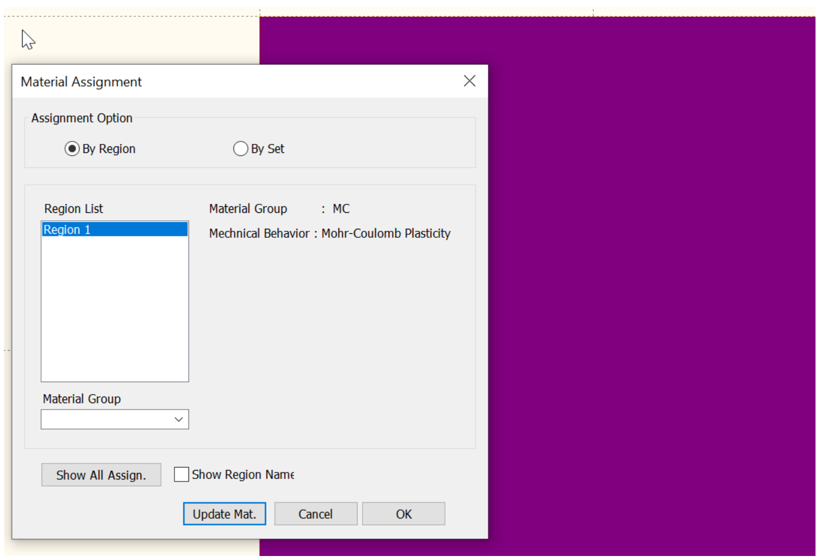


Figure 2.2-11: Assign material to soil region for shallow foundation example

Step 4: Define boundary conditions

From the menu bar, click **"Data->Boundary Conditions"** to open the dialog for the boundary condition group management. Click **"New"** to define a new displacement boundary condition "*disp_x*" (Figure 2.2-12). Select **"Edge Displacement "** as the BC type. Keep "In Global Direction", then set the check mark before "x direction". Keep the default displacement value as zero. This boundary condition will be applied to the vertical edges to prevent motion in the X-direction. Click **"Save & Exit"** to return to the BC group management dialog.

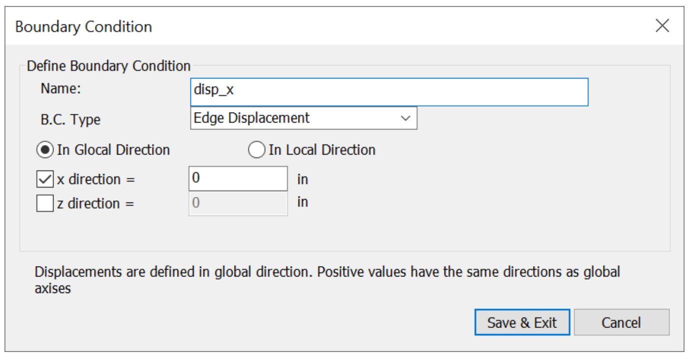


Figure 2.2-12: Define displacement boundary in x direction

Similarly, create the second boundary condition for the soil bottom edge as shown in Figure 2.2-13. This boundary condition is 'pin' type to prevent soil movement in both X and Z-directions. Click **"Save & Exit"**.

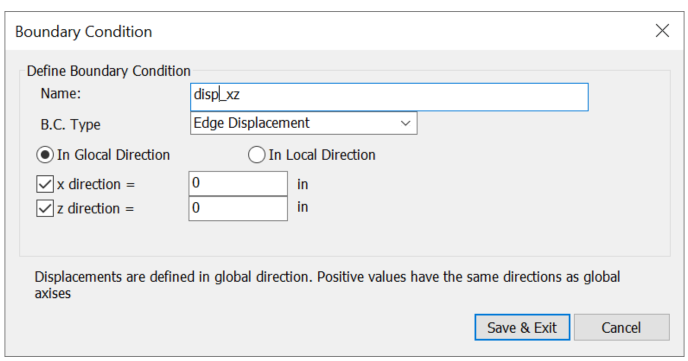


Figure 2.2-13: Define displacement boundary in both x and z-directions

Usually, Mohr-Coulomb model needs the initial soil stress to be estimated in the first stage. The gravity load should be defined to keep equilibrium with these initial soil stresses. Click **"New"** to add the third BC like that seen in Figure 2.2-14. The value for the gravity load is "*-1*" because the unit weight has been defined in the material property. A negative quantity represents the direction in which the load will be applied in the negative direction of the Z-axis. Click **"Save & Exit"** to return to the BC group management dialog.

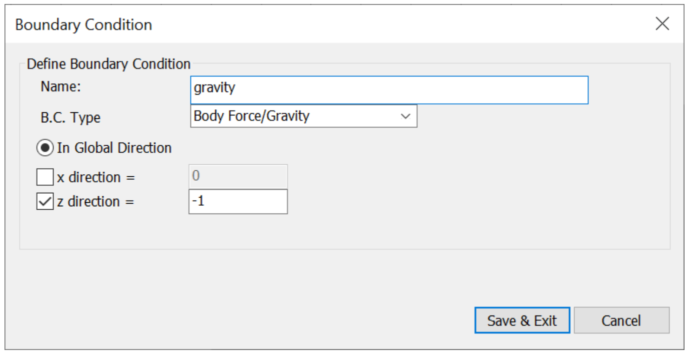


Figure 2.2-14: Define gravity load B.C in Example 2

The last boundary condition is the 2-inch downward displacement of the rigid foundation. Click **“New”** and define this BC as shown in Figure 2.2-15. Click **"Save & Exit"** to exit to the BC group management dialog. Click **"Close"** to exit.

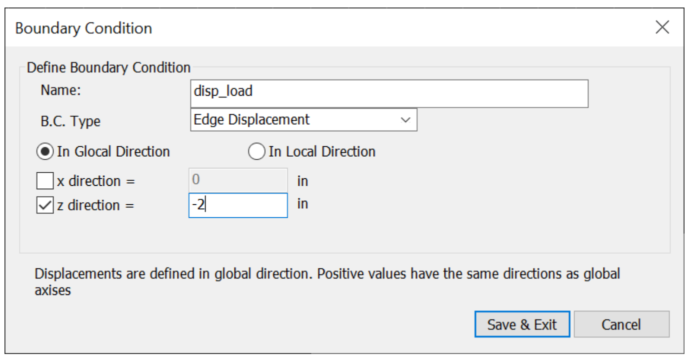


Figure 2.2-15: Define displacement load of the rigid foundation

From the menu bar, click **“Data->Boundary Assignment”** to open the BC assignment dialog. By default, all boundary conditions start from the first stage “initial stress”(Figure 2.2-16). Select the boundary condition **"**disp_x**"**, click **“Assign”**, then select both vertical edges of the soil region. Make sure to press and hold the SHIFT for multiple selections. RIGHT-CLICK and select **“End Selection”**. The assigned boundary condition looks like that shown in Figure 2.2-17. The green roller icons adjacent to these edges represent displacement constraints of "disp_x".

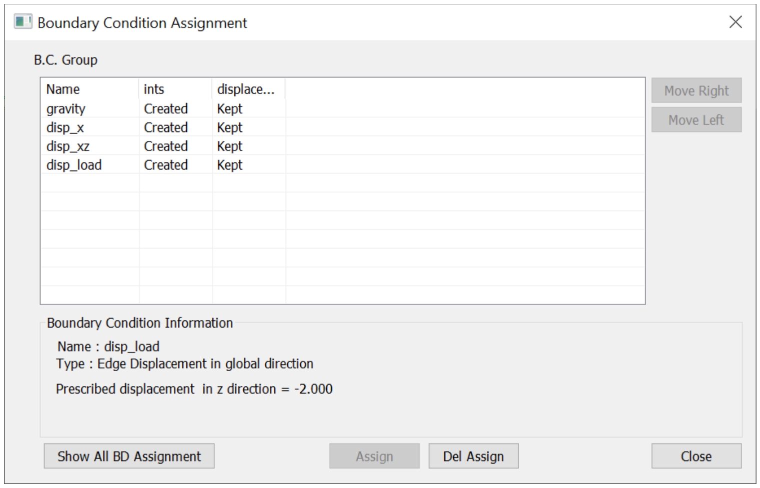


Figure 2.2-16: Assign boundary condition to soil region

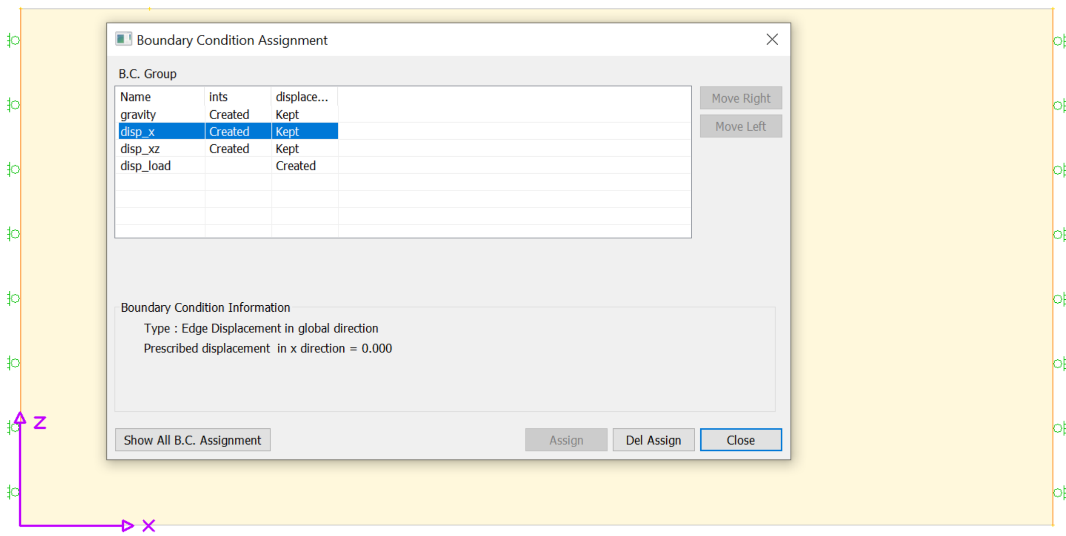


Figure 2.2-17: Roller displacement B.C. of the soil region

Similarly, assign the second boundary condition "disp_xz" to the bottom edge of the soil region (Figure 2.2-18).

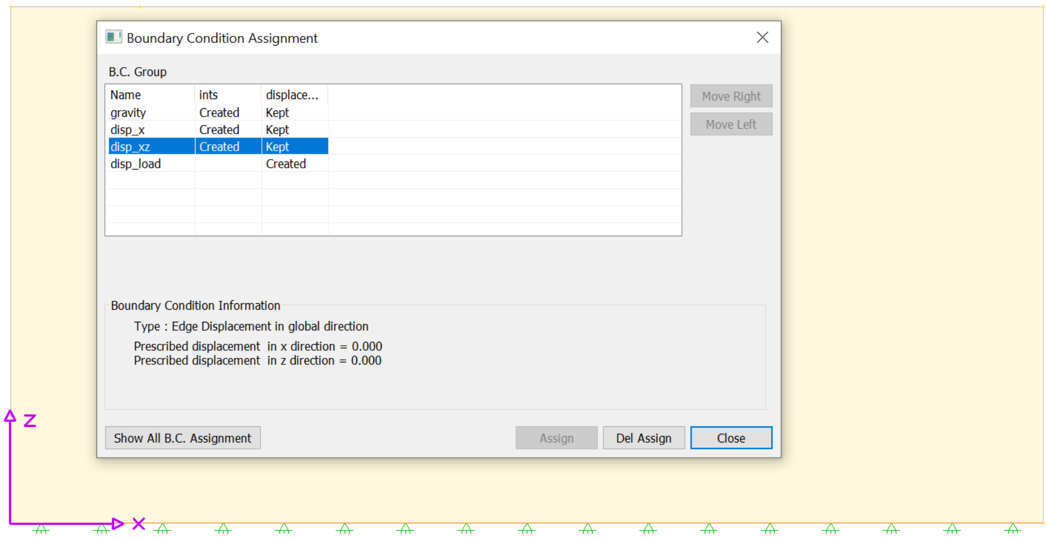
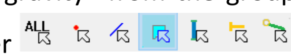


Figure 2.2-18: Assign pinned B.C. to the bottom of soil region

Now assign the gravity load to the soil regions Select “gravity” from the group list, then click “**Assign**”. EnFEM automatically activates the region filter . Click to select the soil region. RIGHT-CLICK and select **“End Selection”**. The region is now shaded with vertical lines (Figure 2.2-19).

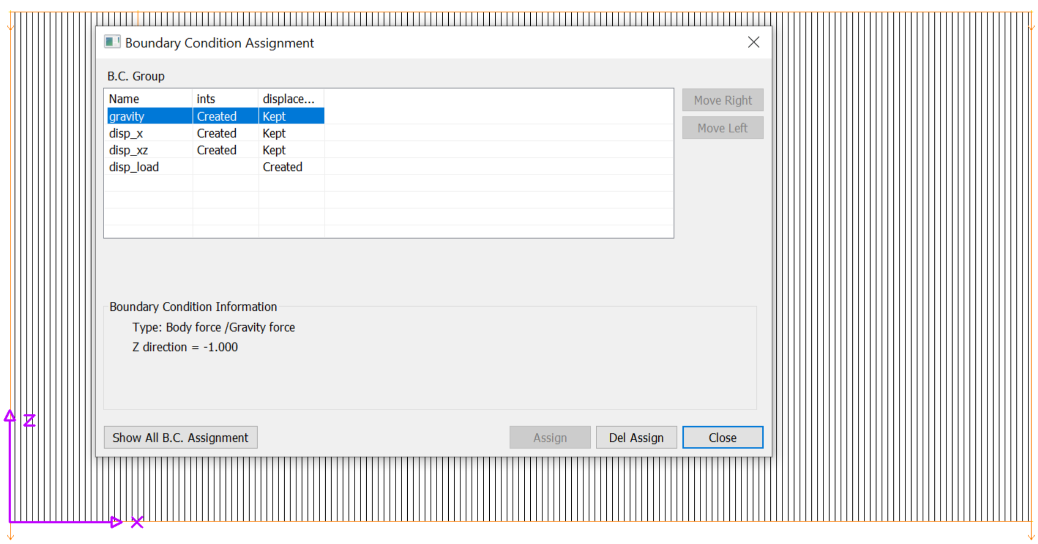


Figure 2.2-19: Assign gravity B.C. to soil region

Next, select the second column “created” of the last boundary condition **“**disp_load**”.** Click “**Move Right**” to change the start stage to “displacement”. Click **“Assign”** and then select the upper-left edge representing the contact zone with the rigid foundation. RIGHT-CLICK and select **“End Selection”**. The assigned BC looks like that in Figure 2.2-20.

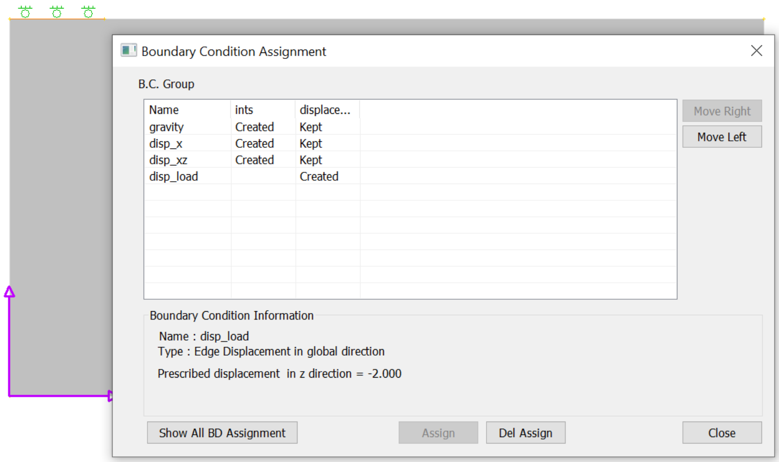


Figure 2.2-20: Assign displacement load to edge of soil region

Click **"Show All Assignment"** to verify all boundary conditions were assigned correctly, as seen in Figure 2.2-21.

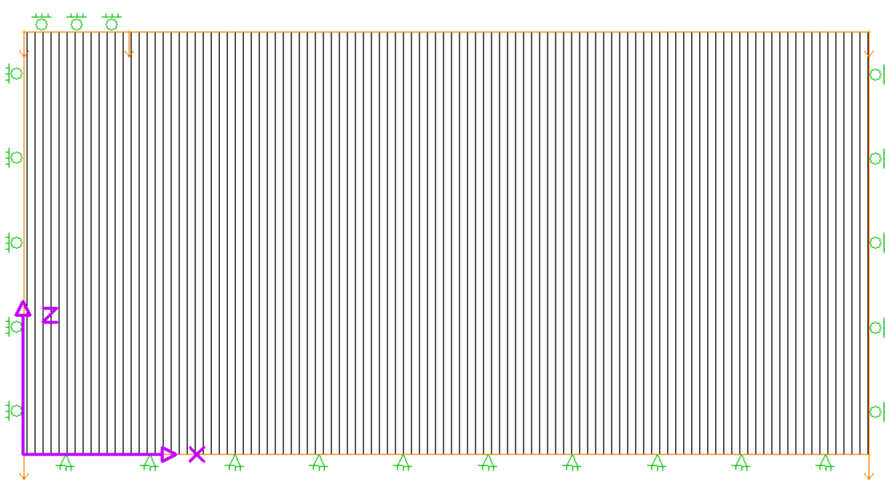
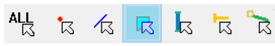


Figure 2.2-21: Show all assigned B.C. for assignment verification

Click **"Close"** to exit from the dialog and save the file.

Step 5: Mesh generation

In this example, the point at the edge of the rigid foundation (x=50,z=200) is a singular point where finite elements suffer from significant deformation and stress. Refining the mesh locally around the foundation edge can capture stress peak more accurately. For this purpose, an unstructured mesh is applied to the soil region.

From the menu bar, click **"pre-Mesh->Advanced Mesh Control".** EnFEM will activate the region selection icon  automatically. Click to select the soil region, then RIGHT-CLICK and select **"End Selection"**. The global mesh control dialog will open, as shown in Figure 2.2-22. In general, if contact (contact constraint method) is not simulated in the models, quadratic quadrilateral Q8 elements with reduced integration rule are highly recommended. Make the appropriate selections for "Quadrilateral", "Unstructured" and "Quadratic Q8", and check the box in front of "Reduced" (Figure 2.2-22). Click **"Save & Exit"** to close the dialog.

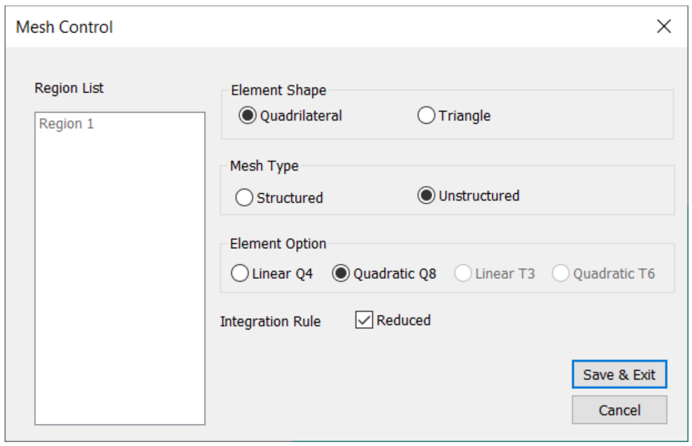


Figure 2.2-22: Advanced mesh control for shallow foundation example

From the menu bar, click **"pre-Mesh->Global Seed Control"** to open the global element size control dialog. Change the default size to "12" and click **"Assign"**. Red circles are marked at each seed position on the region edges, as seen in Figure 2.2-23.

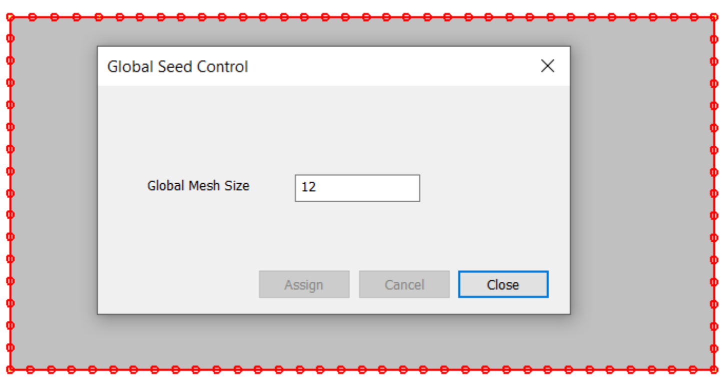
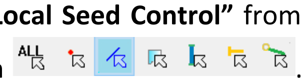


Figure 2.2-23: Apply global discretization seeds uniformly

Click **"Close"** to exit from this dialog.

To refine the mesh around the foundation edge and capture the stress peak, apply local seed control near the singular point. Begin by selecting "**pre-Mesh -> Local Seed Control"** from the menu bar. This action automatically activates the line selection icon .

Click to select the long, top-right edge of the foundation. Once selected, RIGHT-CLICK and choose "**End Selection"** to open the local mesh control dialog. The selected edge will be highlighted in red. Change the "Seed Control Type" to "By Size**"** and select "One Way**"** from the "Seed Distribution Style" options. A red arrow will appear along the edge to indicate the direction of refinement (coarse to fine). If needed, check "Reverse Bias Direction" to reverse the refinement direction.

Set the minimum size to `1` and the maximum size to `12`, as shown in Figure 2.2-24. Click "**Assign"** to apply these settings to the edge. Seed icons, represented by red circles, will appear along the edge, reflecting the refinement. The recalculated maximum and minimum element sizes will be displayed at the bottom of the dialog, which may slightly vary from the original input values.

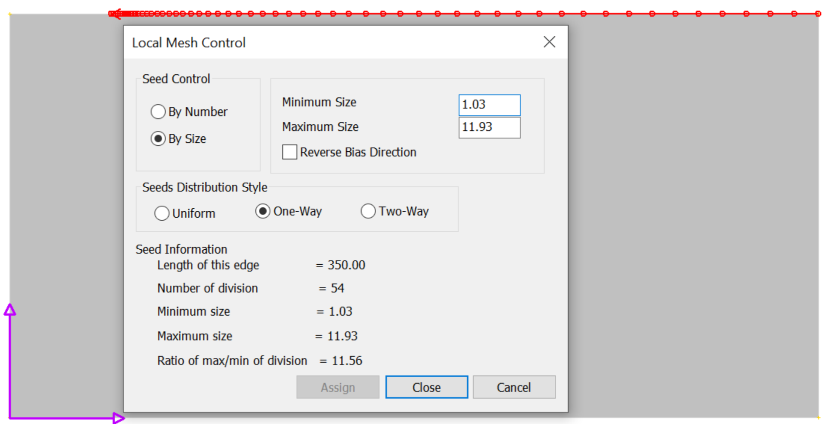


Figure 2.2-24: Mesh refinement around foundation edge

After clicking "**Close"** to exit the local mesh control dialog, note that the local mesh control command remains active. Proceed to assign local seed control to the top-left (short) edge.

Select the top-left edge by clicking on it. Once selected, RIGHT-CLICK and choose "**End Selection"** to open the local mesh control dialog for this edge. Follow a similar process as before to apply the local seed control. Adjust the parameters as needed and ensure proper refinement settings are applied to this edge, as illustrated in Figure 2.2-25. Finally, click "**Assign"** to confirm the settings and observe the displayed seed icons for verification.

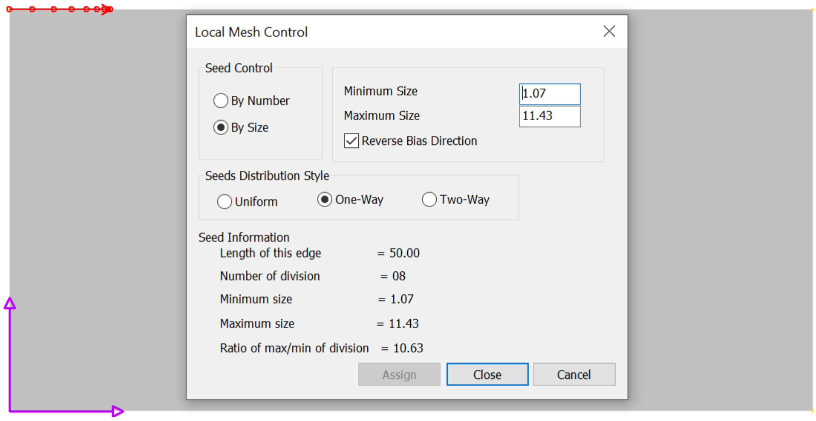


Figure 2.2-25: Local mesh control on edge below foundation

From the menu bar, click "**pre-Mesh->Show Edge Division"** and "**pre-Mesh->Show Seeds"** to visualize the applied local seed distribution patterns and edge divisions, as demonstrated in Figure 2.2-26. This will display green division numbers along edges and red seed icons, making it easy to review the applied settings. Once the review is complete, click them again to turn off the display. This ensures a clutter-free view in the working zone for further operations.

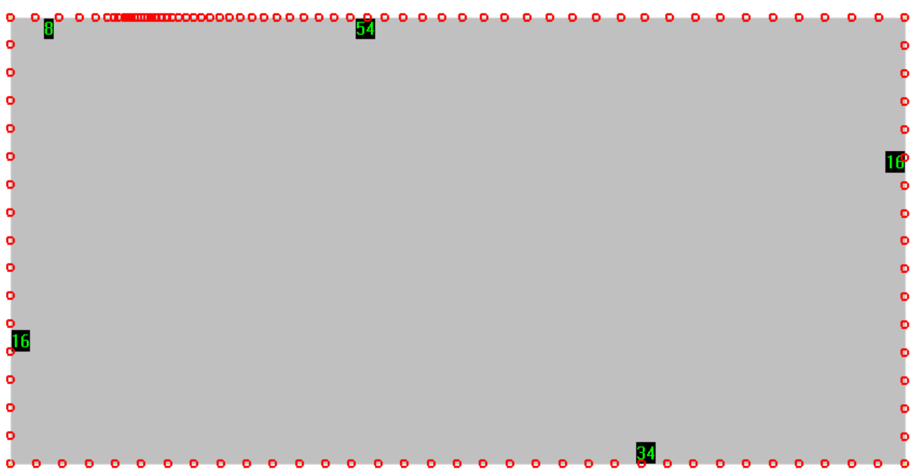


Figure 2.2-26: Show seeds and division number to check local refinement

After assigning the global mesh size and applying local mesh control, the seed distribution satisfies these general rules:

- It addresses the requirements for capturing local stress concentration effectively.
- Larger element sizes on external edges help reduce the total number of elements, optimizing computational efficiency.
- A smooth transition from refined mesh areas to coarser mesh areas ensures a balanced and accurate representation.

From the menu bar, click "**pre-Mesh->Run Mesh Generation"** to generate the mesh. Upon successful completion, the generated mesh will load automatically, as illustrated in Figure 2.2-27.

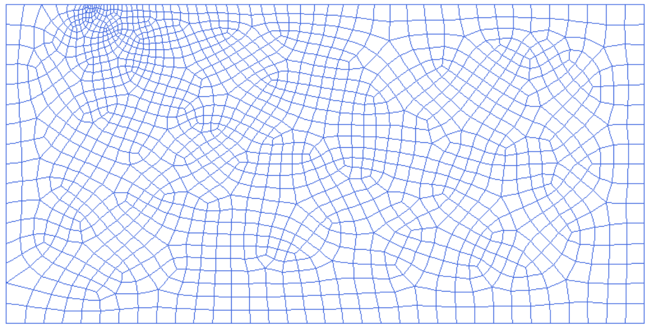


Figure 2.2-27: Unstructured quadrilateral mesh for shallow foundation example

After the mesh is generated, the working mode switches from the "CAD-MODE" to "MESH-MODE" automatically. If desired, the user can show/hide the yellow FE-node symbols on the mesh by clicking the  icon.

Click  to save file.

Step 6: Run Analysis

From the menu bar, click "**Analysis->Analysis Setting"** to open the dialog for analysis options (Figure 2.2-28).

Note**:** The convergence group includes two key default settings:

- **Relative Tolerance**: This parameter represents the ratio of the residual force to the total applied external loads. Residual force is the difference between internal nodal forces and equivalent nodal forces generated by external loads. Smaller relative tolerance values improve accuracy but may require more iterations and longer computation time.
- **Maximum Iteration Number**: This parameter limits the number of iterations per increment. If the iteration count exceeds this limit without achieving convergence, the increment size is automatically reduced.

In most cases, these default settings are sufficient, and adjustments are unnecessary.

The "Output Frequency" parameter controls how often solver results are saved to the hard drive. The default setting is "1," meaning results are written for each successfully completed increment. However, small increment sizes may lead to large result files. Increasing the output frequency (e.g., to "5") can reduce file size. Regardless of the frequency setting, results for the first and last increment of each stage are always saved.

EnFEM also provides options to write displacements, reactions, stresses, or contact results into a .out file. However, these can produce very large files. Using the post-processor described in this manual is generally more efficient for analyzing results than directly examining the text output.

For this example, set the output frequency to "5" and proceed with the default settings for other parameters.

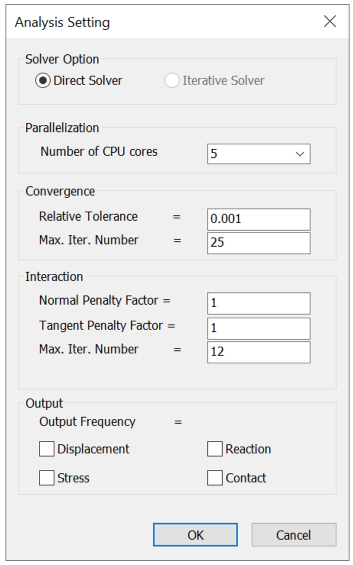


Figure 2.2-28: Setup analysis parameters for shallow foundation example

From the menu bar, select "**Analysis-> Run Analysis**." Click "**Run**" in the dialog to submit the job to the solver. A command window will display convergence data for each load increment. Once the analysis completes or terminates due to errors, the log file will open automatically. Review the convergence history and check the ".log file" for error details if any are detected by EnFEM.

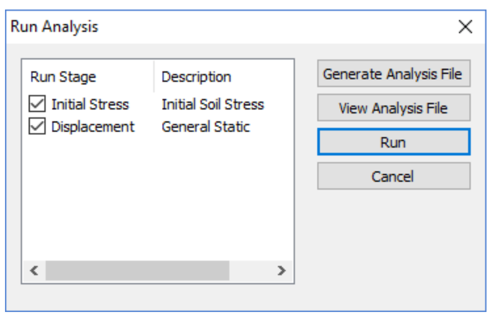


Figure 2.2-29: Run Analysis parameters for shallow foundation example

Step 7: Post-processing of result

Click  to switch to "RESULT MODE," then click  to open the contour plot dialog. The default contour shows displacement in the X-direction (u1) at the last load increment. From the Group drop-down list, select "Stress," then choose "S22" from the Variables list. Select the increment "Initial Stress Incr=1.000." The contour of vertical stresses (S22, representing initial soil stress) will be plotted, as shown in Figure 2.2-30.

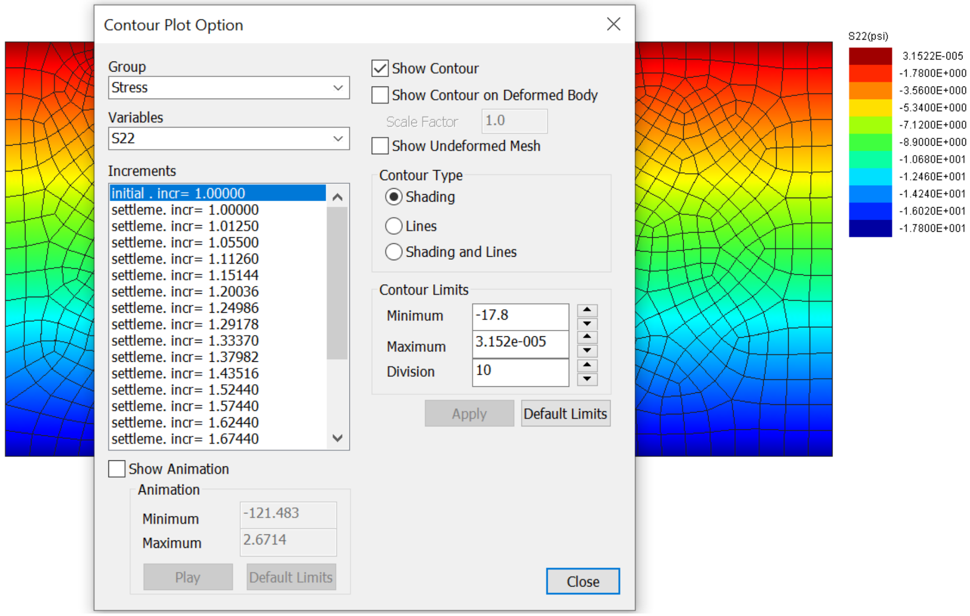


Figure 2.2-30: Contour of initial soil stress for shallow foundation example

Initial soil stresses result from the soil's self-weight. Since the ground is flat and the model simulates only one soil layer, vertical stress increases uniformly with depth. The minimum stress (S22) at a depth of 200 inches is -17.8 psi, which matches the unit weight (0.089 pci) multiplied by the depth. Next, select "Strain" from the Group drop-down list and "PEEQ" from the Variables list. Click the last increment to display the contour of equivalent plastic strain after the 2-inch settlement of the rigid foundation is applied.

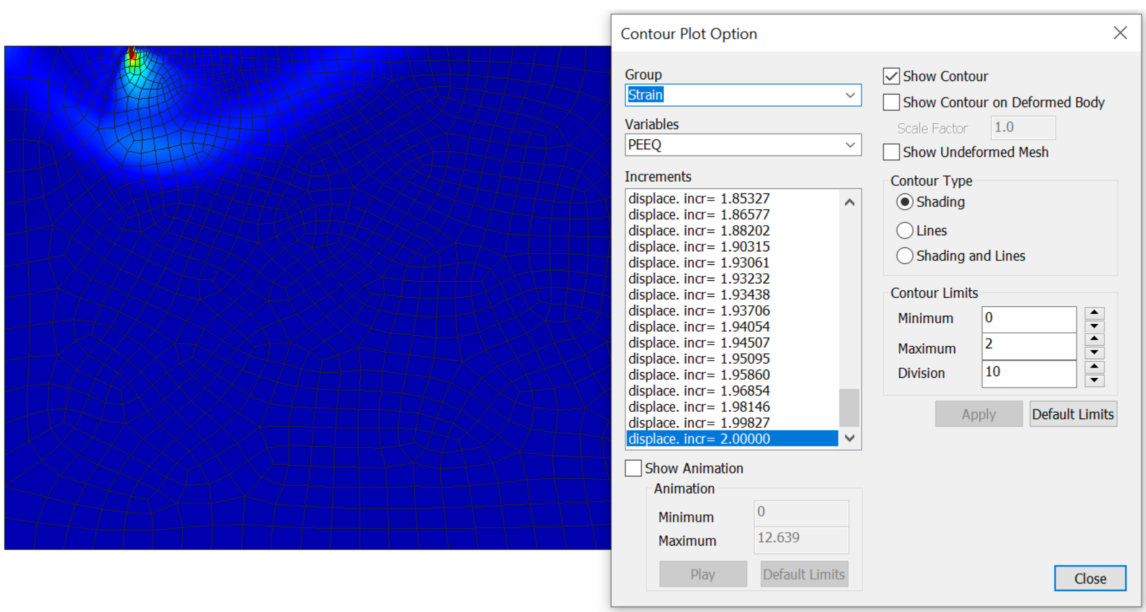


Figure 2.2-31: Sliding zone of shallow foundation example

Figure 2.2-31 shows the equivalent plastic strain contour, where the sliding zone can be clearly identified. Reduce the maximum "Contour Limit" if necessary (to approximately 2.0), to make the contour result clearer. Click **"Close"** to close contour plot dialog.

Next click  to open up the cutline plot dialog. Input start point of cutline at ***(x=-1, z=199)*** and end point at ***(x=401, z=199)***. Click **"Add"** to add it to the "cutline list" window. A red cutline will be displayed on screen. Select **"u2"** from the Field Variables drop-down list, and select the last increment from increment history list. Click **"Plot"** to plot the vertical displacement on "*cutline 1*" – 1-inch below the ground surface, as shown in Figure 2.2-32.

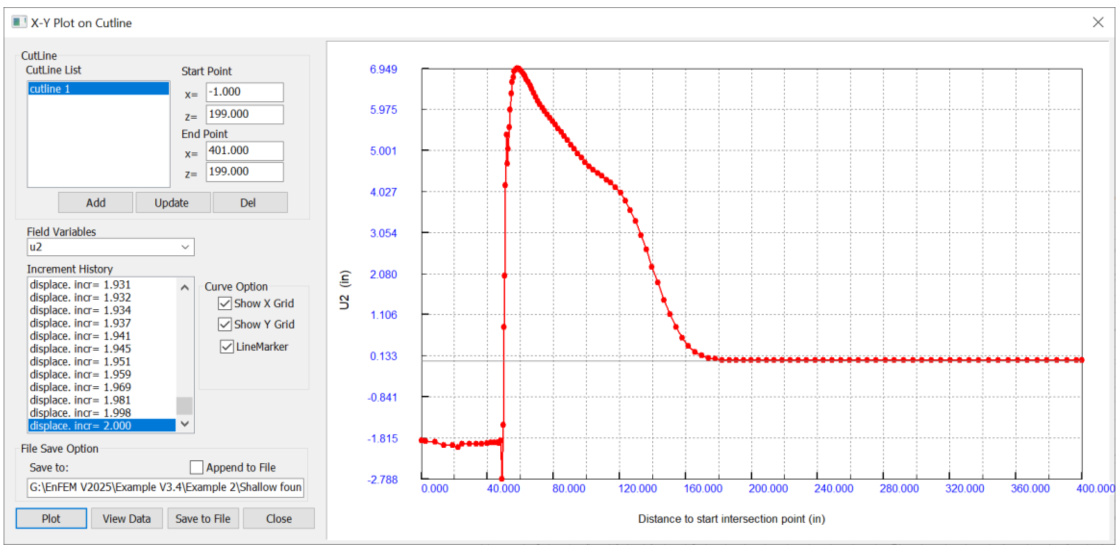


Figure 2.2-32: Vertical displacement along cutline 1, 1-inch below ground

Looking at the plot, two 'peaks' can be found around x=50-in which indicate the singular point around the foundation edge. Another finding is significant soil dilation at range x=50-in to x=120-in. Dilation of sand is inevitable when sand is disturbed. The dilation angle can be used to control excessive volume change. Ordinarily the dilation angle used in the simulation is determined by lab tests where the same volume expansion is obtained from numerical simulation and the lab test.

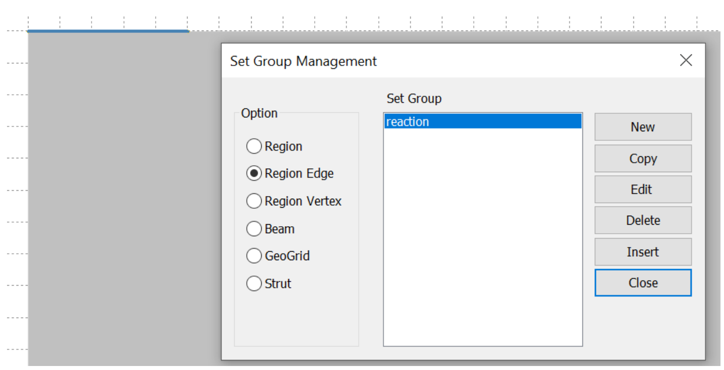


Figure 2.2-33: Define set for reaction output

Reactions VS Settlement is another interesting plot for this type of analysis. Click to go back to the CAD mode. Click menu "**Tool->Set Management**" to open the set group window, define a "Region Edge" set for the 2 in displacement boundary as shown in Figure 2.2-33.

Click to open the X-Y plot dialog as seen in Figure 2.2-34. Keep the source data type as **"**Historic Result**"**. Click "**Select by Set**" and select previously defined edge set "reaction". Set "Reaction Force" as in the category and "Z Reaction" as the variable. Click "**Plot**" to show the curve between the total reaction force and settlements (Figure 2.2-34).

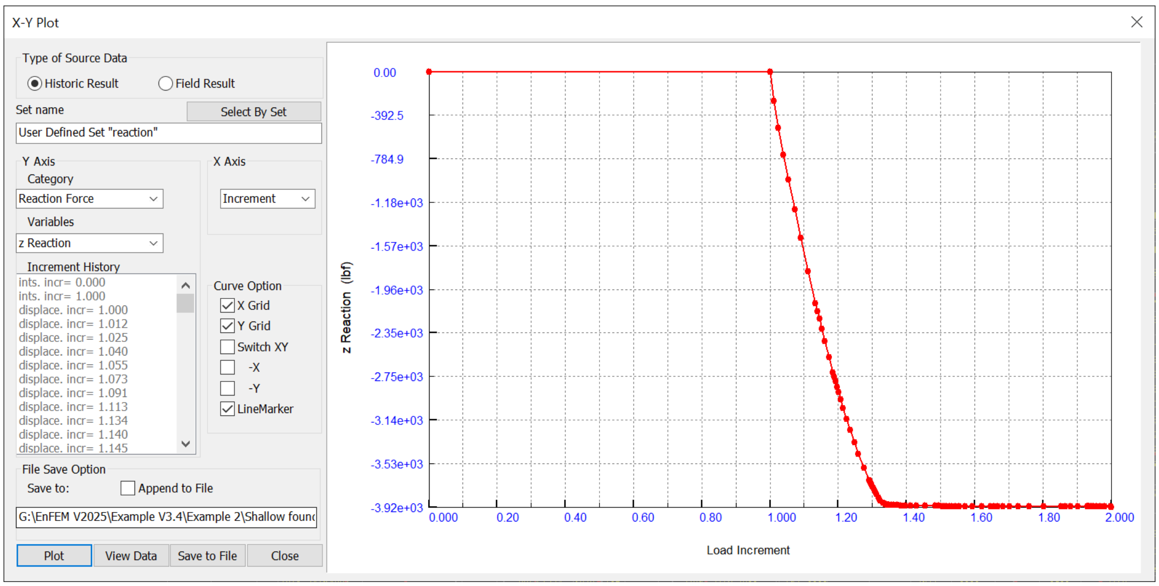


Figure 2.2-34: X-Y curve plot of Reactions VS Settlement

The reaction from increment=0 to increment=1, is zero because it is the initial soil stress stage. After that, reactions will increase nonlinearly until 3,924-lbf. The soil reaches the ultimate capacity and starts to slide at settlement=0.67-inch (increment=1.32).

2.3 Example 3: Stability analysis of slope using strength reduction method (SRM)

This example presents procedures to do a slope stability analysis using strength reduction method (SRM). The physical configuration of the slope is shown in Figure 2.3-1. Units are selected as “kN, m, second”.

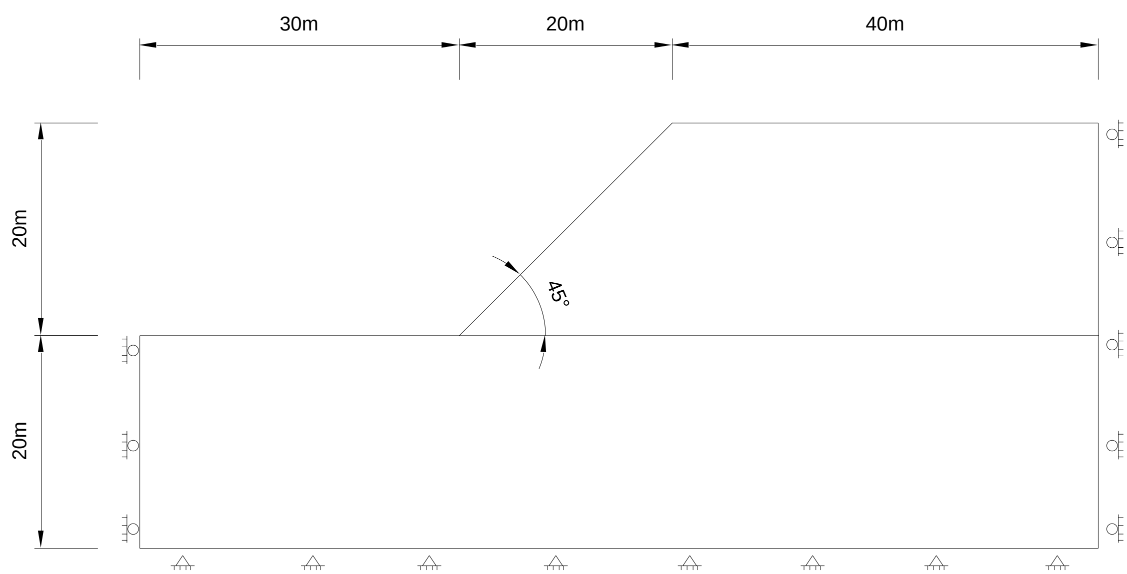


Figure 2.3-1: Configuration of slope stability analysis example

Material Properties:

Top soil layer:

- Unit weight:.. $\gamma = 20\ kN/m^3$
- Young’s Modulus:................................ $E = 8 \cdot 10^4\ kPa$
- Poisson’s ratio:................................ $\nu = 0.4$
- Cohesive strength:.............................. $c = 35\ kPa$
- Friction angle:................................. $\phi = 25°$
- Dilation angle:................................. $\psi = 1°$

Bottom soil layer:

- Unit weight.. $\gamma = 20\ kN/m^3$
- Young’s Modulus:................................ $E = 1.0 \cdot 10^5\ kPa$
- Poisson’s ratio:................................ $\nu = 0.3$
- Cohesive strength:.............................. $c = 40\ kPa$
- Friction angle:................................. $\phi = 35°$
- Dilation angle:................................. $\psi = 1°$

The Mohr-Coulomb plasticity model will be used in this example.

Boundary Conditions:

- Displacement constraints on vertical soil edges
- Displacement constraint on bottom edge of bottom soil layer
- Gravity load
- Strength reduction factor applied through material property field variable

Model development and analysis steps:

Step 1: Create geometric entities

Launch EnFEM, or from the menu bar, click "**File->New**" to create a new project. Update the title to "`slope stability using strength reduction method`" and keep the problem type as **"**Plane strain**"** (see Figure 2.3-2)**Error! Reference source not found.**. Choose "kN, m, second" unit option. Click **"OK"** to confirm.

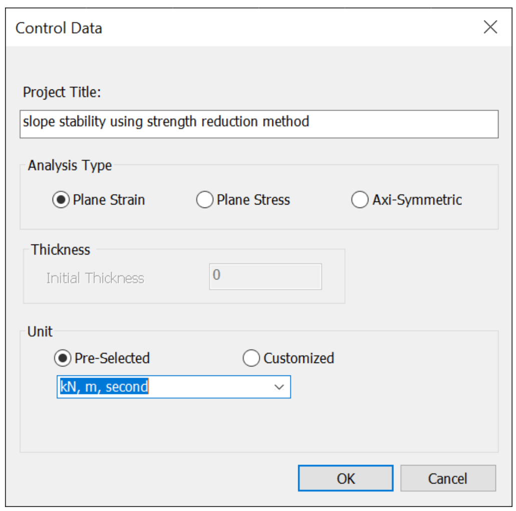


Figure 2.3-2: Define control data for slope stability analysis

Click "**Options->Drawing Space**" to open the drawing space dialog. Change the default settings to those seen in Figure 2.3-3. Click **"OK"** to exit.

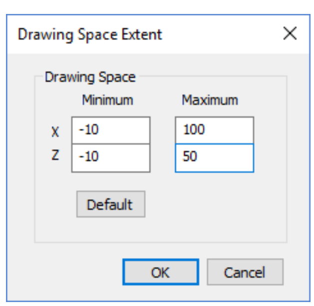


Figure 2.3-3: Set up drawing space for slope stability example

Navigate to “**Options → Grid Spacing**” to open the grid spacing dialog. Set the grid spacing to "10" for both the X and Z directions, then click “**OK**”.

From the toolbar, select the Rectangle Creation Tool . Hold **CTRL**, snap the cursor to the origin (x=0, z=0), and **LEFT-CLICK** to define the first point. Continue holding **CTRL** and use snapping to set the diagonal vertex at (x=90, z=20).

Next, activate the Polygon Creation Tool . Hold **CTRL**, snap to (x=30, z=20), and **LEFT-CLICK** to set the first vertex. Similarly, create vertices at (x=50, z=40), (x=90, z=40), and (x=90, z=20). **RIGHT-CLICK** and select “**Close Polygon**” to complete the shape. Two soil layers should now appear, as shown in Figure 2.3-4.

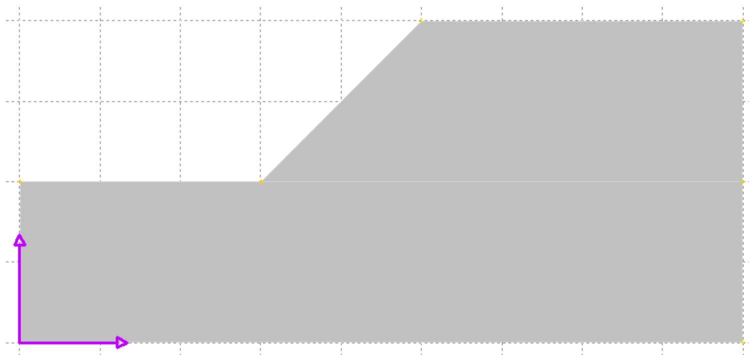


Figure 2.3-4: Create two soil regions for slope stability example

Click the save icon to save the model as "`Slope Stability Using SRM.sep`". Upon creating the top polygon, EnFEM will automatically ensure geometric compatibility by adding a vertex at (x=30, y=20) along the bottom layer's top edge.

Step 2: Define stage data

From the menu bar, click **"Data->Stage Management"** to open the stage group management window. Click **"New"** to create a new stage. Name the stage *"Ints"* and set the type to **"**Initial Soil Stress**"**. Click **“Initial Stress Option”** and select the **“**By System**”** option. Then, click **“Add”** to select both soil regions. RIGHT-CLICK and click “**End Selection**” to add them to the *‘Region List’* (Figure 2.3-6). Click **"Save & Exit"** to return to the stage group management window.

Next, click **“New”** to create the second stage with the name *"SRM"* with the type **"**Stability (Strength Reduction Method)**"**. Retain **"**Automatic**"** increment control but adjust the maximum increments to *“200”*. Set the initial and the maximum increment size to *“0.02”*. Click “**Define SRM**” and adjust the safety factor range from 0.5 ~ 2.0 to 0.5 ~ 1.6. Click “**OK**” to exit the SRM dialog. Click **“Save and Exit”** and then **“Close”**. The two stages are ready as shown in Figure 2.3-5.

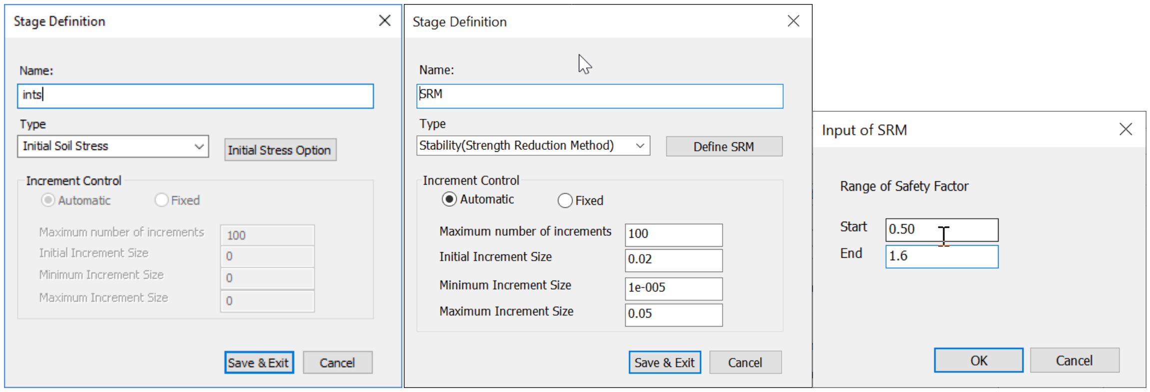


Figure 2.3-5: Stage definition for slope stability analysis

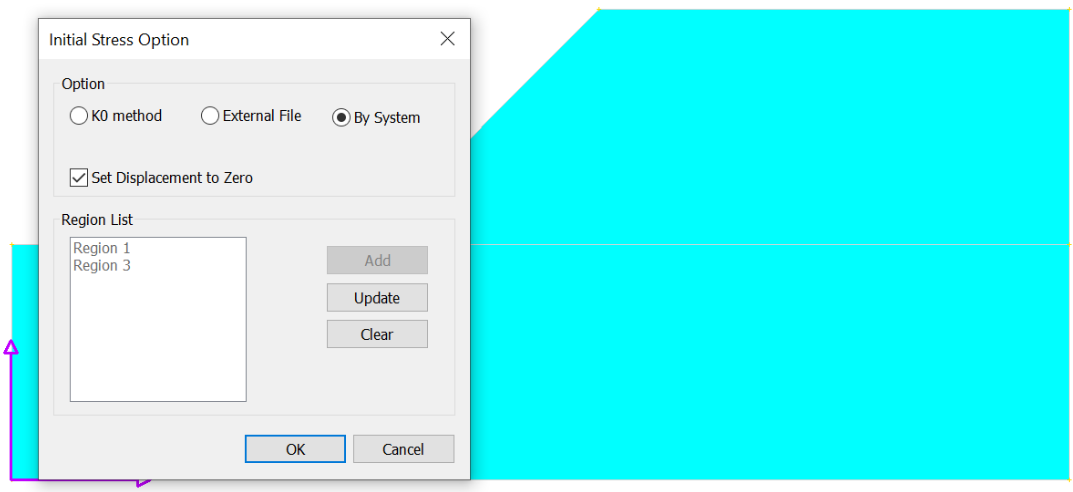


Figure 2.3-6: Initial stress option

Step 3: Define materials

From the menu bar, click **"Data->Material Properties"** to define two Mohr-Coulomb type soils *"MC_top"* and *"MC_bot"*. The soil is assumed to have minimum volume expansion. Since the dilation angle in EnFEM cannot be zero, 1 ° is input as shown in Figure 2.3-7 and Figure 2.3-8.

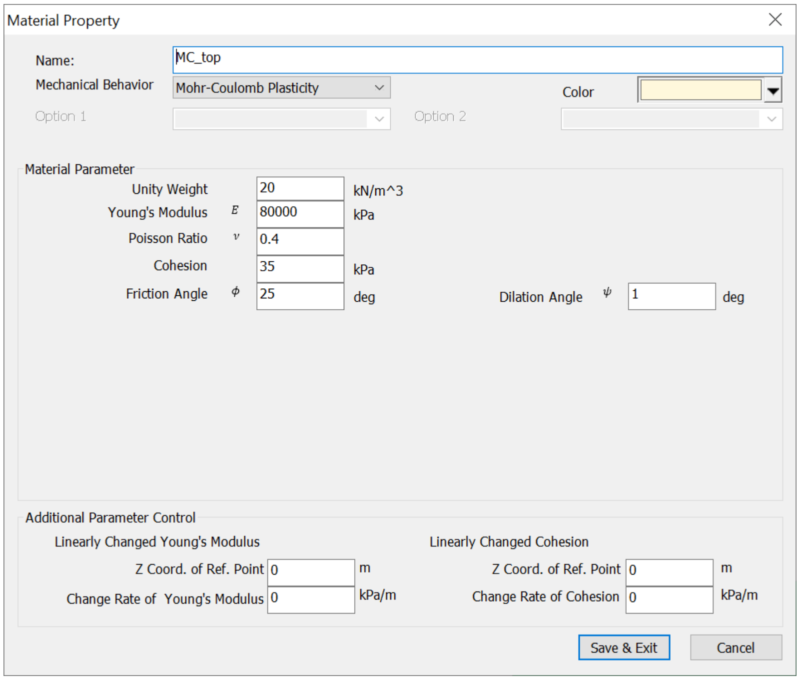


Figure 2.3-7: Material property of top soil layer

Similarly, create the second material "*MC_bot*" for the bottom soil layer as shown in Figure 2.3-8.

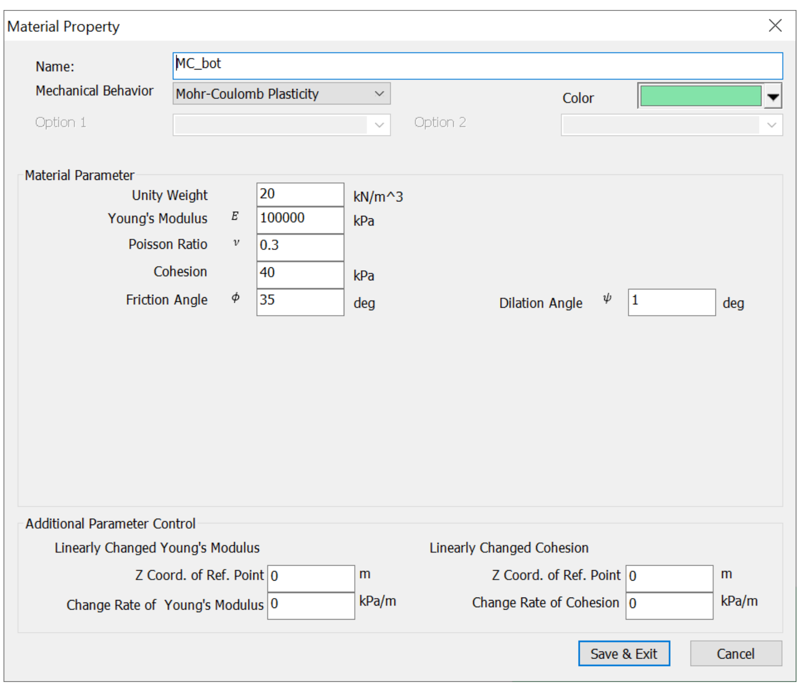


Figure 2.3-8: Material property for bottom soil layer

Click **"Data->Material Assignment"**, and assign material **"*MC_top*"** to the top soil layer, and **"*MC_bot*"** to the bottom soil layer (Figure 2.3-9).

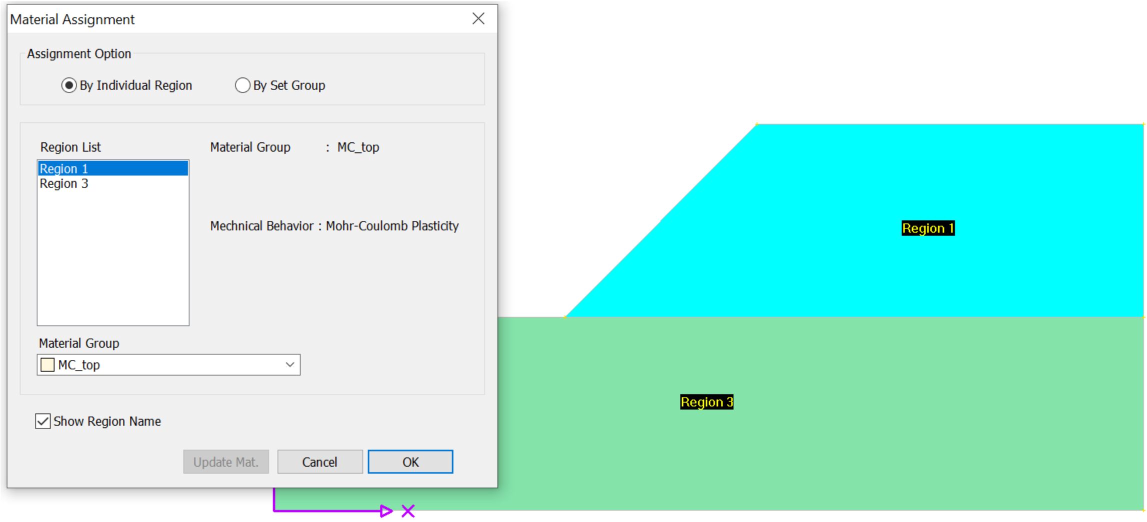


Figure 2.3-9: Assign material property to regions for example of slope stability analysis

Step 4: Define boundary conditions

From the menu bar, click **"Data->Boundary Conditions"** to open the boundary condition management window. Click **"New"** to create a new displacement boundary "`disp_x`" as shown in Figure 2.3-10. Ensure to the option "In Global Direction" is selected. Click **"Save & Exit"** to return to the BC group management window. This boundary condition will be applied to the vertical soil edges.

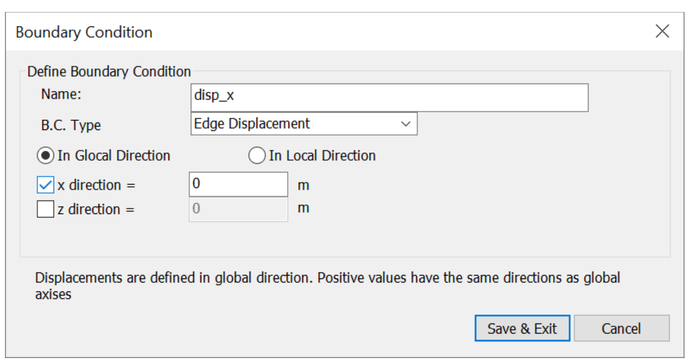


Figure 2.3-10: Define displacement BC for vertical edges in Example 3

Next, create the second displacement boundary, "`disp_xz`", as shown in Figure 2.3-11. This boundary condition will be applied to the soil bottom to prevent soil movement.

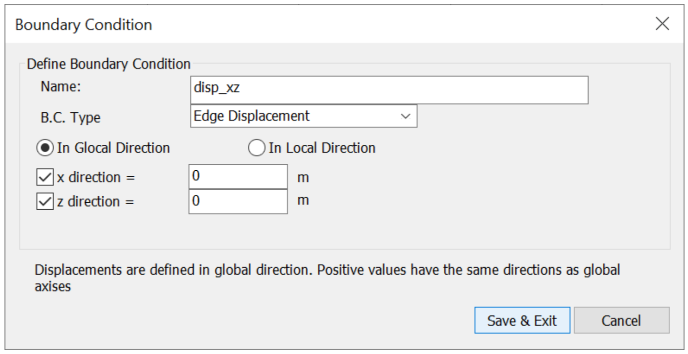


Figure 2.3-11: Define displacement BC for bottom edge in Example 3

Click **"Save & Exit"** to return to the boundary management window. Then click **"New"** to create the third boundary condition, "gravity", as seen in Figure 2.3-12. This boundary condition will be used to apply gravity load to the soil layers.

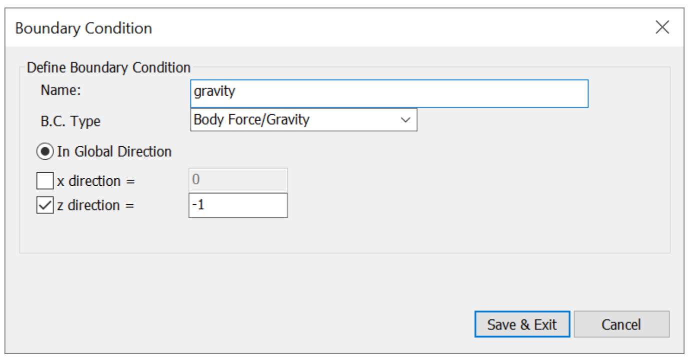


Figure 2.3-12: Define force BC in Example 3

Click **"Save & Exit"** to return.

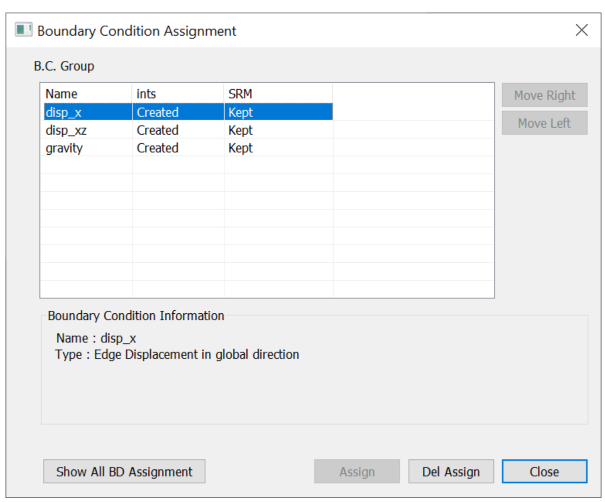


Figure 2.3-13: B.C. groups for slope stability analysis

From the menu bar, click **“Data->Boundary Assignment”** to open the BC assignment window. All boundary conditions are listed in the B.C. group window (Figure 2.3-13). By default, boundary conditions start from the first stage. Select the first boundary condition, **“**disp_x”, and click **“Assign**”. The line selection icon will be automatically activated . Press and hold the SHIFT key, then select the vertical edges of the soil region (three in total). RIGHT-CLICK and select **“End Selection**”**.** If the assignment is successful, green ‘roller’ icons will be displayed along the vertical edges, as shown in Figure 2.3-14.

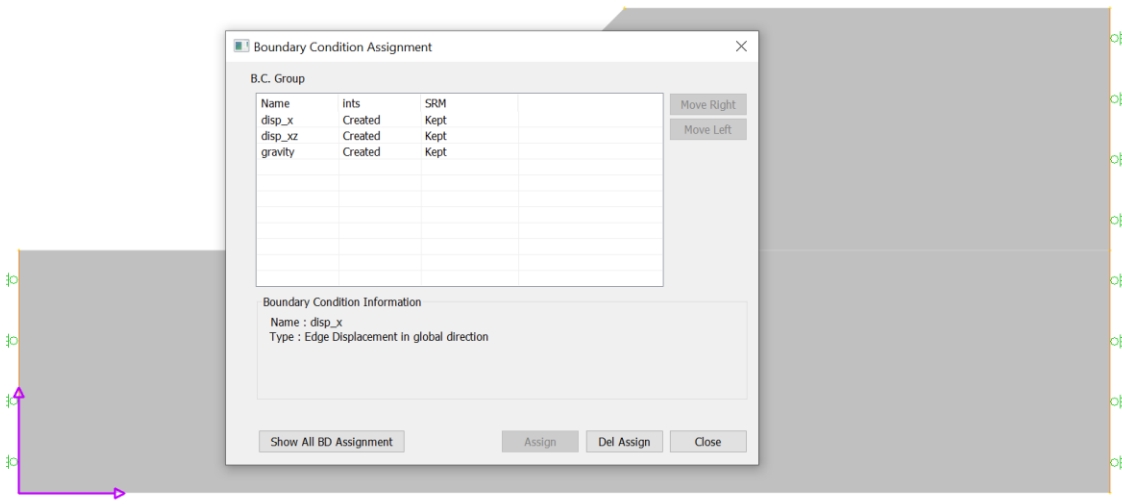


Figure 2.3-14: Assigned disp. B.C. for vertical edges in slope stability analysis

Next, select the boundary condition "disp_xz"**.** Click **“Assign”** and then select the bottom edge of the lower soil layer. RIGHT-CLICK and select **“End Selection”**. Green ‘pin’ icons will appear along the bottom edge, as seen in Figure 2.3-15.

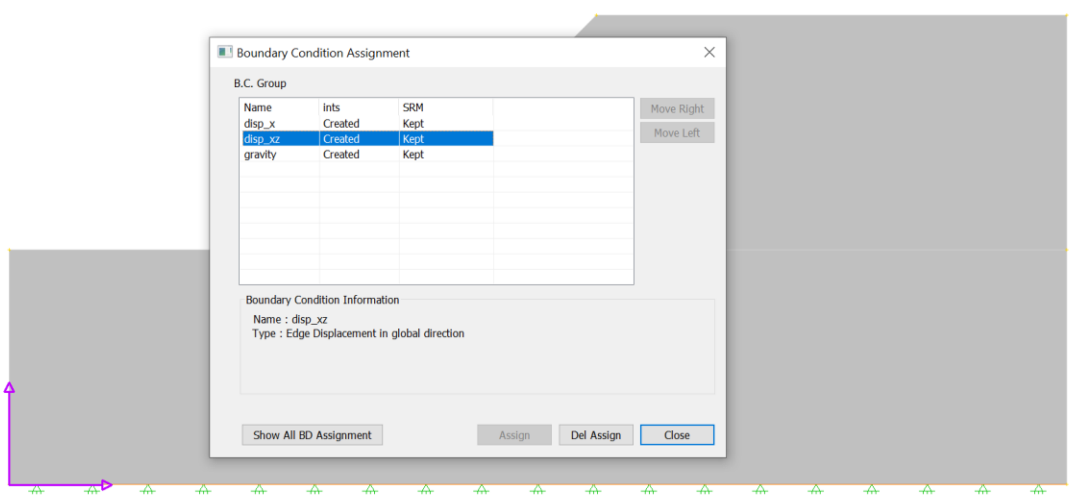
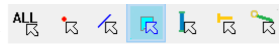


Figure 2.3-15: Assigned disp. B.C. for bottom edge in example of slope stability analysis

Finally, select the boundary condition "gravity". Click **“Assign”**. Since gravity is applied to region objects only, the region filter will be activated automatically . Select both soil regions. RIGHT-CLICK and select **“End Selection”**. The selected regions are shaded in vertical lines, and directional arrows will be plotted at each region vertex, as shown in Figure 2.3-16.

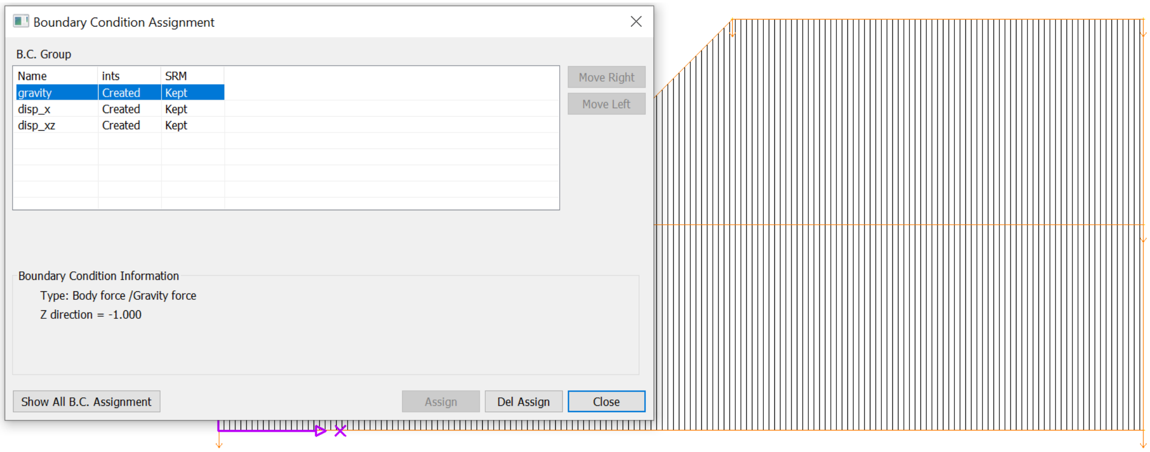


Figure 2.3-16: Assigned gravity B.C. for all soil layers in example of slope stability analysis

Users can click **"Show All BD Assignment"** to verify that all boundary conditions have been assigned correctly. Click **“Close”** to exit the dialog and save the model.

Step 5: Mesh generation

Quadrilateral quadratic Q8 elements are used to discretize the soil layer. The bottom layer, which is a rectangle, will be assigned the structured mesh type. The top layer, a quadrilateral, can be assigned a structured or unstructured mesh. Here the unstructured mesh is selected.

Click on the menu “**pre-Mesh->Advanced Mesh Control**”, select the top layer, RIGHT CLICK and select “**End Selection**” (Figure 2.3-17). Similarly, assign quadrilateral shape and structured mesh type to the bottom layer. Next, click “**pre-Mesh->Global Seed**” and define seed size of 2 on both soil layers. Finally, click “**pre-Mesh->Run Mesh Generation**” to generate the mesh, as shown in Figure 2.3-18. For more information about mesh generation, refer to *chapter 8: Finite Elements & Mesh Generation* in the user manual.

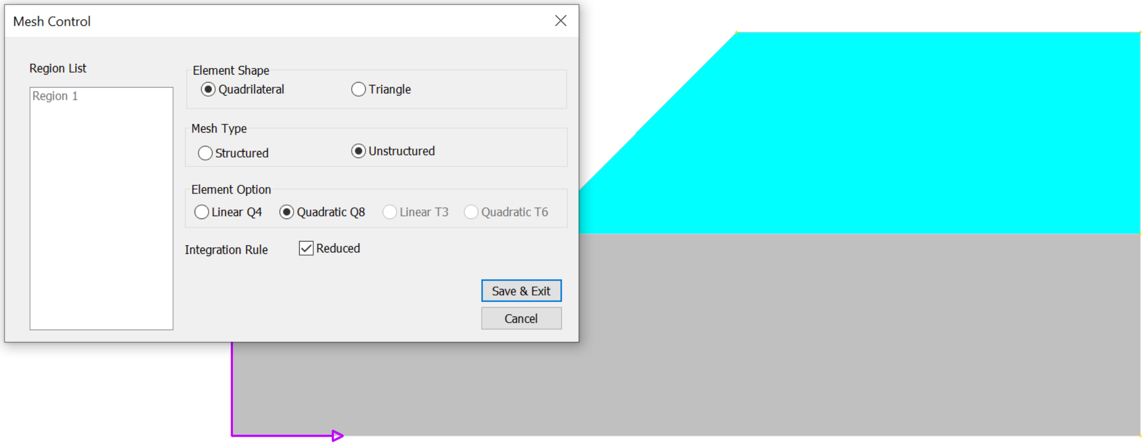


Figure 2.3-17: Assign unstructured quadrilateral mesh to top soil layer

In general, slope stability is represented by the overall safety factor. Local failure of finite elements near the base of the slope may cause calculation to be terminated (numerical failure), potentially occurring before slide of significant amount of soil mass. Therefore, using an excessively fine mesh may not be beneficial for the analysis.

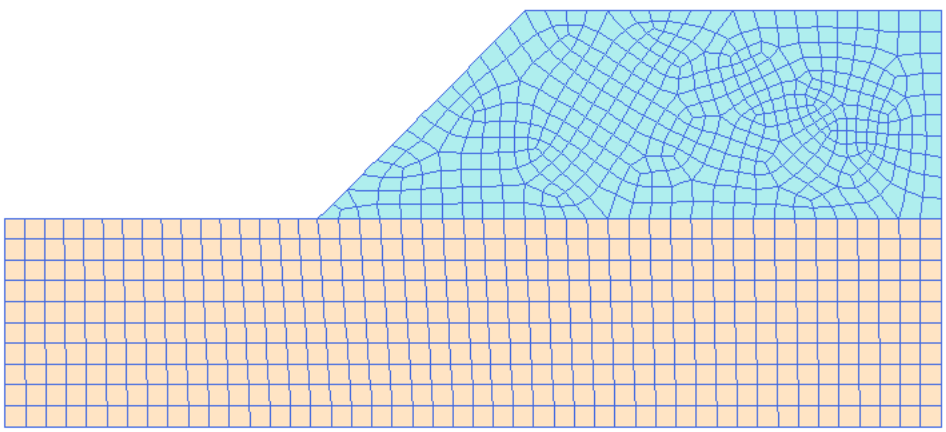


Figure 2.3-18: Generated quadrilateral, quadratic-Q8 mesh in slope stability analysis example

Step 6: Run Analysis

From the menu bar, click **"Analysis->Analysis Setting"** to open the analysis option dialog. For this example, there is no need to change default settings. Click **"OK"** to exit.

Next, from the menu bar, click **"Analysis->Run Analysis"**. Click **"Run"** to submit the job to the solver. EnFEM will first check the model first and generate an input file. Once the calculation is finished or if EnFEM encounters a model error, the log-file will be opened automatically. Users can review the log file to trace any errors.

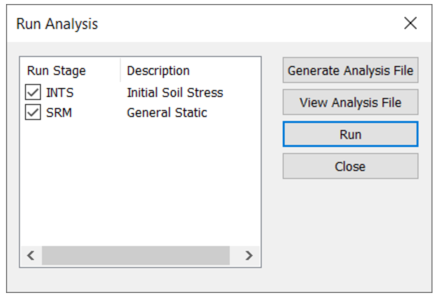


Figure 2.3-19: Window to Run Analysis in Example 3

Step 7: Post-processing of result

Click to switch to the “RESULT-MODE”, then click to open the contour plot dialog. The default contour shows the horizontal displacement u1 at the last loading increment, as seen in Figure 2.3-20. A significant amount of soil mass is sliding to the left, which indicates slope failure. The calculation fails to converge at increment = 1.74232, which corresponds to a safety factor of 1.317.

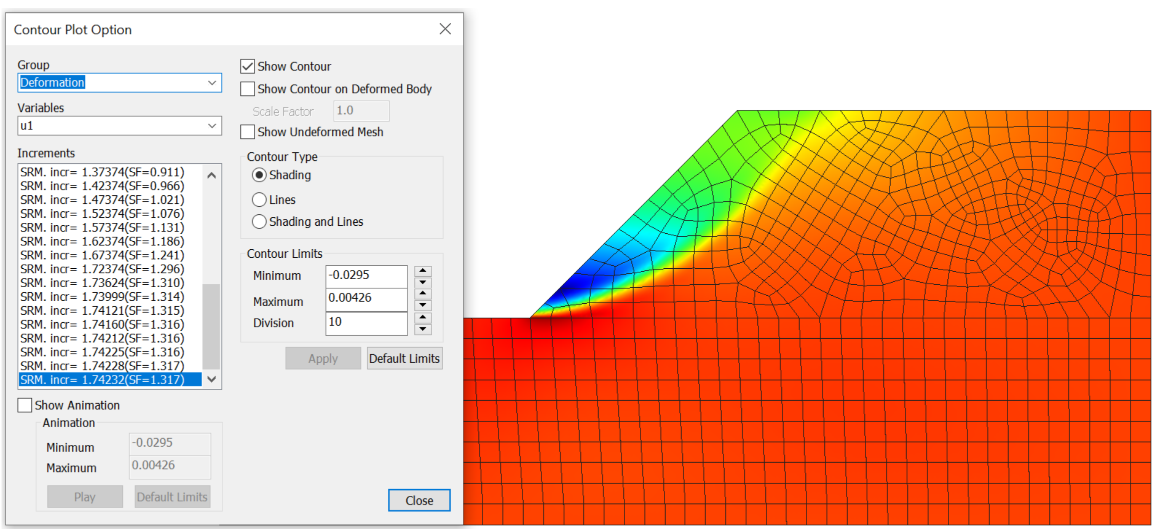


Figure 2.3-20: Contour of horizontal displacement in Example 3

Select **"Strain"** from the drop down under “**Group**”, then select **“PEEQ”** from the Variables drop-down. The contour of the equivalent plastic strain is plotted in Figure 2.3-21. The sliding zone extends almost to the top of the slope.

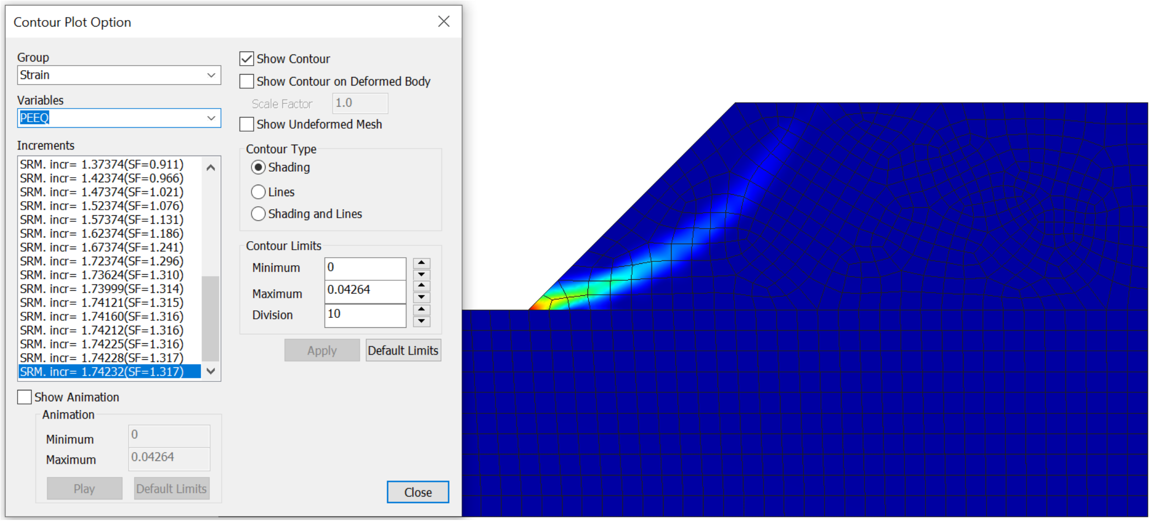


Figure 2.3-21: Contour of equivalent plastic strain in Example 3

The sliding soil body can be easily identified using the incremental contour of displacement. Click the icon  to open an incremental contour plot, as shown in Figure 2.3-22. Select the displacement magnitude as the plot variable. Since the sliding soil mass experiences significant displacement, the sliding surface is clearly visible in Figure 2.3-22.

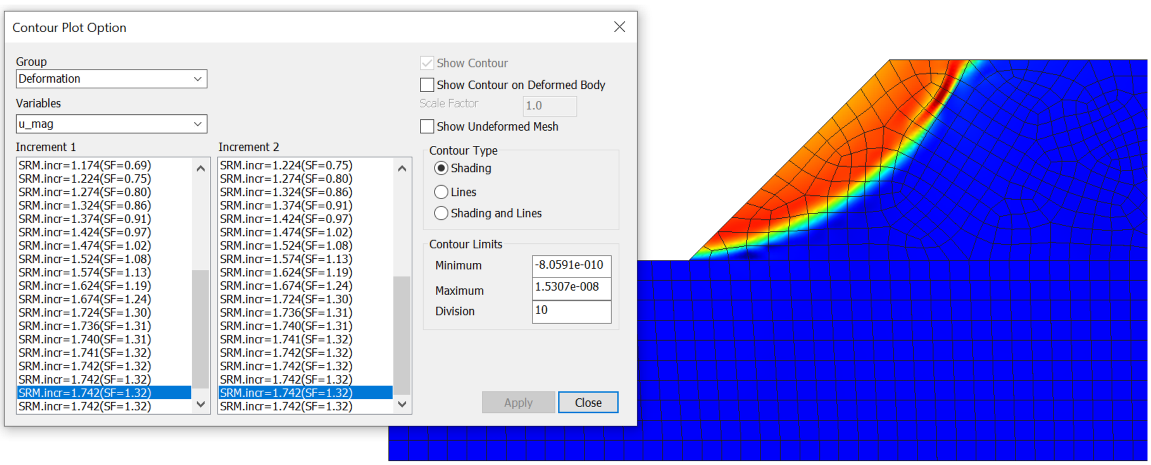


Figure 2.3-22: Incremental Contour of displacement u1 in Example 3

The overall safety factor is determined through interpolation (2.3-1):

$$SF = y_a + \frac{y_b - y_a}{x_b - x_a} \cdot (x_c - x_a) \tag{2.3-1}$$

where,

y_a = First *defined* factor (field variable) of analysis

y_b = Last *defined* factor (field variable) of analysis

x_a = First increment of strength reduction stage (in this example 1)

x_b = Last increment of strength reduction stage (in this example 2)

x_c = Last *converged* increment of the analysis result

SF = Calculated Factor of Safety

For this example and result this equation yields:

$$SF := 0.5 + \frac{1.6 - 0.5}{2 - 1} * (1.747 - 1) = 1.32$$

(2.3-2)

To verify the failure of soil, you can analyze the curve of displacement in the X-direction versus load increments. Start by enabling the display of node numbers by clicking "**Mesh -> Show Node Numbers**" from the menu bar. This will allow you to identify specific nodes. The maximum lateral translation at the slope toe occurs at node "74". For better visibility, zoom into the area around the slope toe.

Next, navigate to "**Results -> X-Y Plot**" from the menu bar. Set the plot type to "Historic result" and input "*74*" as the node set to focus on. From the Category list, select "Primary Variable," and then choose "u1" from the Variables list, which corresponds to the horizontal displacement. Click "**Plot**" to generate the graph of horizontal displacement (u1) versus load increments.

Observe the curve (Figure 2.3-23) for any abrupt and significant increase in displacement. Such behavior signifies slope failure, as soil particles near the slope toe undergo very large movements during this phase. This method provides a clear visual representation of soil failure under increasing loads.

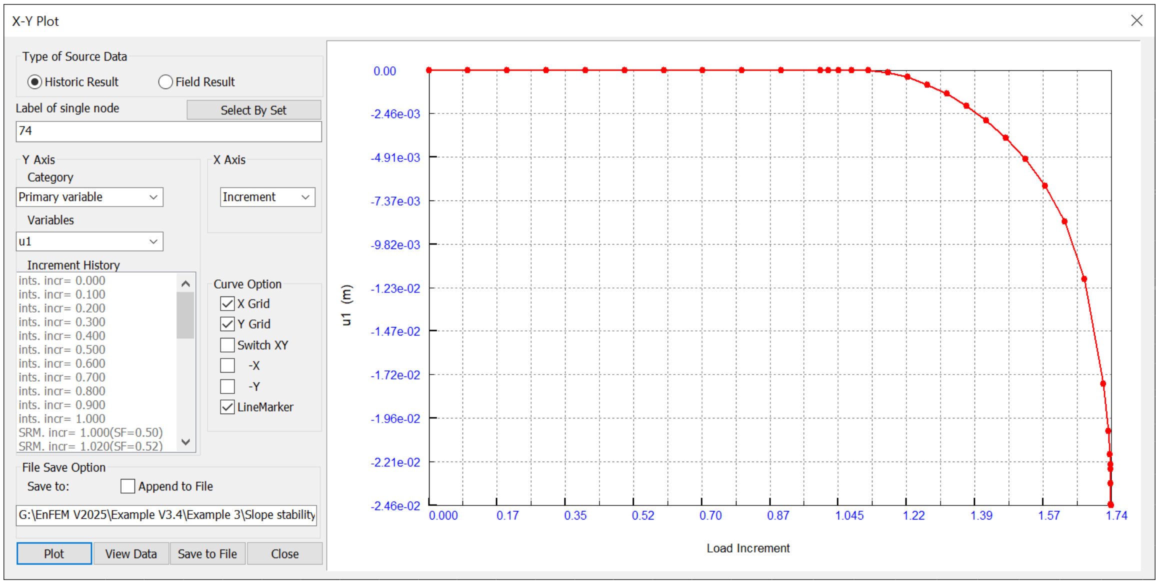


Figure 2.3-23: X-Y plot of horizontal Displacement VS Load Increment

2.4 Example 4: Soil-Structure-Interaction (contact) analysis of buried culvert

Buried culverts are common in geotechnical engineering, with their soil-structure interaction challenging to analyze due to high nonlinearity. Figure 2.4-1 illustrates a concrete culvert buried in soil, subjected to soil weight and ground surcharge. This example demonstrates solving such interactions in EnFEM using a computational contact algorithm. The culvert and soil are discretized into solid elements, with units set as "lbf, in, second".

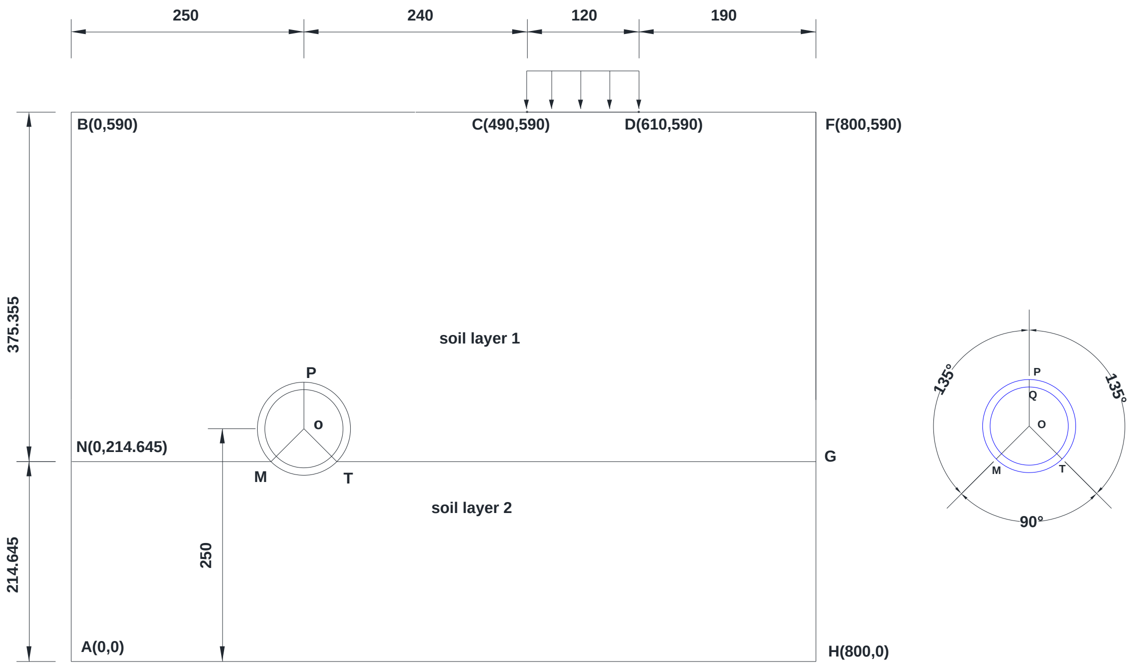


Figure 2.4-1: Configuration of buried culvert example

The concrete culvert has an external radius of 50-inches and its thickness is 8-inches. Material properties and boundary conditions are listed below.

Material Properties:

Concrete (Culvert):

- Unit weight:.................................... $\gamma = 0.09\ lbf/in^3$
- Young's Modulus:............................ $E = 4031000\ psi$
- Poisson's ratio:................................ $\nu = 0.3$

Top soil layer:

- Unit weight:.................................... $\gamma = 0.074\ lbf/in^3$
- Young's Modulus:............................ $E = 2000\ psi$
- Poisson's ratio:................................ $\nu = 0.45$
- Cohesive strength:.......................... $c = 5\ psi$

- Friction angle:.. $\phi = 30°$
- Dilation angle:.. $\psi = 30°$

Bottom soil layer:

- Unit weight:.. $\gamma = 0.074\ lbf/in^3$
- Young's Modulus:.. $E = 2000\ psi$
- Poisson's ratio:.. $\nu = 0.35$
- Cohesive strength:.. $c = 20\ psi$
- Friction angle:.. $\phi = 35°$
- Dilation angle:.. $\psi = 35°$

Boundary Conditions:

- Displacement constraints on vertical external soil edges
- Displacement constraints on bottom edge of lower soil layer
- Gravity load
- Ground surcharge of 35 psi
- Contact interface between soil and culvert.
  - The culvert is the master object and the soil is the slave object
  - Coefficient of friction on the contact interface is 0.36

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM, or from the menu bar, click "**File->New**" to start a new project. Update the title to "`Interaction analysis of buried culvert`". Keep the problem type as **"**Plane strain**"** as seen in Figure 2.4-2. Set the unit to be "lbf, in, second". Click **"OK"** to exit.

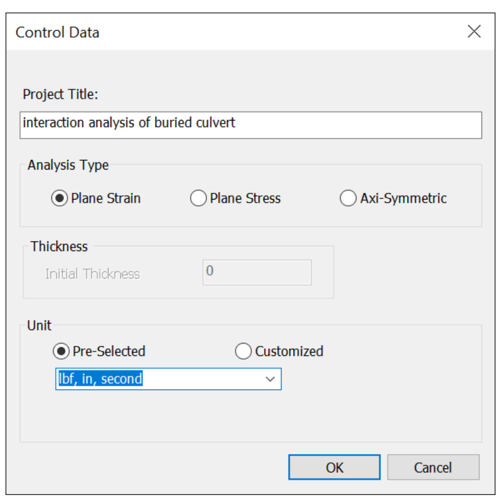


Figure 2.4-2: Control data for buried culvert

The culvert has a ring shape, constructed in EnFEM using three "fan-shaped" regions. While two regions are sufficient, three are used to include an extra vertex at the culvert's top, simplifying result extraction. Each fan-shaped region has four sides to enable a structured mesh

Controlling vertexes, A through T (as seen in Figure 2.4-1) have coordinates listed in Table 2.4-1:

	A	B	C	D	F	G	H	N
x	0.0	0	490	610	800	800	800	0
z	0.0	590	590	590	590	214.645	0	214.645
	O	**M**	**P**	**T**	**Q**			
x	250	214	250	284.7	250			
z	250	214.645	300	214.645	292			

Table 2.4-1: Vertices for buried culvert analysis

In "CAD MODE", click **"Options->Drawing Space"** and setup the drawing space as shown in Figure 2.4-3:

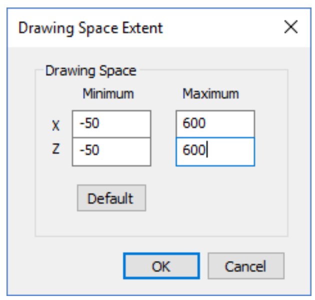


Figure 2.4-3: Setup drawing space for soil-culvert interaction analysis

In this example, the culvert will be created first. Afterwards, use the snapping-method to create the soil layers.

The three external arcs of the culvert are created in the following sequence:

1. arc O-P-M
2. arc O-M-T
3. arc O-T-P

To create the arc, click  to activate the arc creation tool. In the coordinate input box (located beneath the working zone), enter the culvert center coordinate "`250,250`" (point O in Table 2.4-1 and Figure 2.4-4) and press RETURN. Then, key in coordinates for the start vertex of arc O-P-M as "`[0,50]`", and press RETURN.

<u>NOTE:</u> Square brackets indicate relative coordinates, where dx = 0 and dz = 50 specify the vertex's shift from the arc center.

Next, enter "`[50,135]`" and press RETURN to finish arc O-P-M (shown in Figure 2.4-5).

NOTE: The relative coordinate of the arc end vertex has a different meaning than the arc-start. Here *'50'* represents the arc radius, and *'135'* is the desired arc angle.

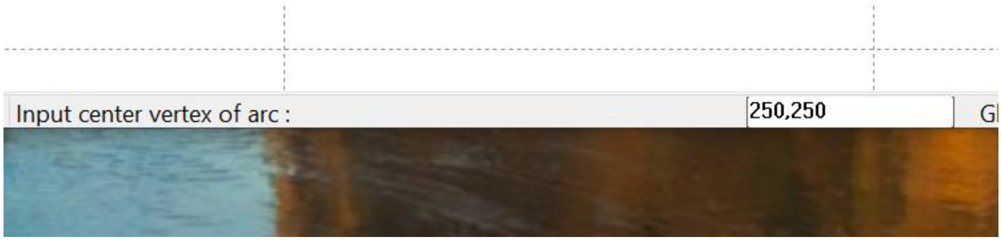


Figure 2.4-4: Input coordinate of center of culvert

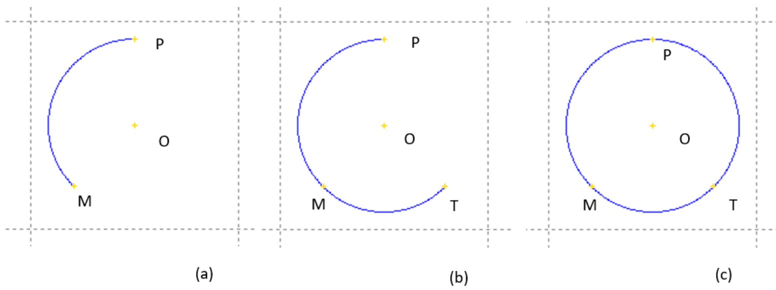


Figure 2.4-5: Create external arcs in Example 4

Hold the CTRL key and move the mouse cursor to the culvert center (vertex *O*). When the snapping mark  appears, LEFT-CLICK to create the center of arc O-M-T. Press CTRL again, move the cursor to snap to vertex *M*, and LEFT-CLICK to create the arc start. Enter the relative coordinate "*[50,90]*", and press RETURN to define vertex *L*. Here, *'50'* refers to the culvert radius and *'90'* the desired arc angle. The second arc O-M-T should now be visible (image (b) of Figure 2.4-5).

For the third arc, use the snap method entirely. Hold the CTRL key, snap to vertex *O*, and LEFT-CLICK to create the arc center. Then snap to vertex *T*, and finally to vertex *P* to complete the external circle of the culvert (image (c) in Figure 2.4-5).

To create the internal circle of the culvert, repeat the process. Hold the CTRL key, snap to Vertex *O* to create the arc center. Key in "*[0,42]*" to create the arc start, and key in "*[42,135]*" to finish the first internal arc. Next, snap to vertex *O* and the previous arc end-point (below the center), then enter "*[42,90]*" for the second arc. Finally, snap to the center point *O* again and to vertices on existing arcs to complete the last internal arc (Figure 2.4-6).

This completes the construction of the culvert's full circular geometry.

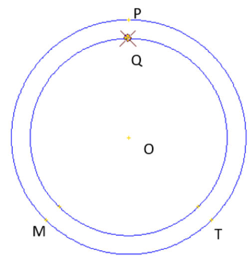


Figure 2.4-6: Create internal arcs of culvert

To create the radial lines of each fan region, click  to activate the line creation tool. CTRL+ LEFT-CLICK to 'snap' and create points at Vertex *P* (line start), then Vertex *Q* (line end), and then RIGHT-CLICK and select **"End Creation"** to terminate the line command. Similarly, create the second and third lines starting at vertex *P* and vertex *T*, and ending with their corresponding points on the internal arcs.

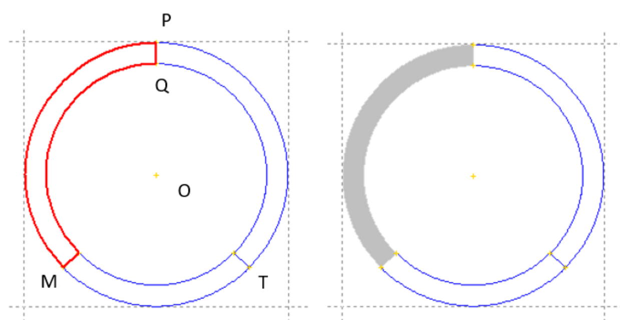


Figure 2.4-7: Select lines and arcs and convert to a general polygon

Next, three general polygons will be created from these lines/arcs. From the toolbar, click **"Create->Polygon from Lines/Arcs"**. Use a 'contained' window to select lines and arcs of the upper-left 'fan region' as seen in the left image of Figure 2.4-7. RIGHT-CLICK and select **"End Selection"**. A new general polygon is created from selected lines and arcs (right image in Figure 2.4-7).

NOTE: Lines/arcs of a region are grey in color as they are no longer independent entities (blue color). If an independent line has been used to create a general polygon, the line must be re-created for use in the new polygon. Such line/arc recreation can be found on adjoining regions (as seen in Figure 2.4-7),

In this case, recreate line P-Q, and then create the upper-right 'fan-region'. Next, recreate radial lines at points P and T before generating the bottom fan-region. After generation of all three fans, the culvert should look like those seen in Figure 2.4-8.

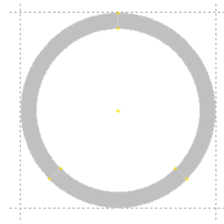


Figure 2.4-8: Completed three fan-regions of culvert

Next, create the top and bottom soil layers. These soil layers are created as general polygons because both regions have arc edges. The following procedures highlight some useful tools to create edges preciously. First click to activate the line creation tool. Next, press and hold the CTRL key, move the mouse to vertex *M*, and LEFT-CLICK once 'snapped' to this point. Then key in the relative coordinate, "[-350,0]", RIGHT-CLICK and select **"End Creation"**. Now create a new vertical line from vertex A (x=0, z=0) to (x=0, z=400). The new vertical should intersect with the horizontal line (shown in the left image of Figure 2.4-9). From the toolbar, click **"Modify->Divide Line/Arc by Intersection"** and select both lines. Then RIGHT-CLICK and select **"End Selection"**. Selected lines will be divided at their intersection point, vertex N. From the toolbar, click **"Edit->Delete"**, and then select the leftover line segments - above, and to the left, of vertex N. RIGHT-CLICK and select **"End Selection"** to finish this deleting operation. The final geometry will look like the right image in Figure 2.4-9, where the line segment N-M has been precisely created.

Still using the line creation tool ( ), create a horizontal line A-H. Then create a vertical line from vertex H (x=800, z=0) to vertex F (x=800, z=590). RIGHT-CLICK and select **"End Creation"**. Start a new line from vertex T, using 'snap' method, enter the relative coordinate "[600,0]" as the next line vertex. Repeat the steps above to divide the last two lines and delete the leftover line segment outside of the lower soil boundary. Finally, recreate the arc O-M-T by using the 'snapping method'. Now all lines/arcs of the lower soil layer are ready to be converted into a general polygon.

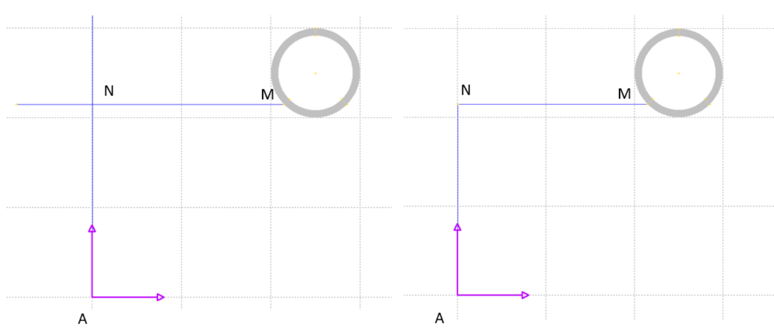


Figure 2.4-9: Use dividing lines method to create vertex with accurate coordinates

From the toolbar, click **"Create->Polygon from Lines/Arcs"**, and use a 'contained' window to select all line segments of the bottom soil layer, as well as the arc O-M-T. RIGHT-CLICK and select **"End Selection"** to finish generating the bottom soil region as seen in Figure 2.4-10.

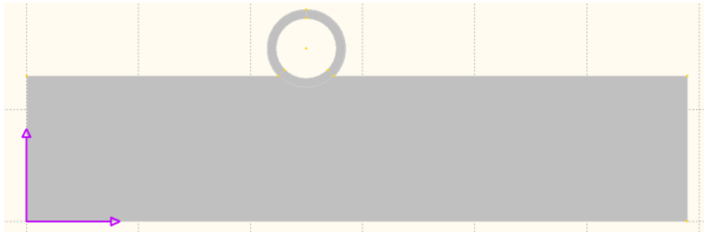


Figure 2.4-10: Generate bottom soil region from independent lines and arcs segment

<u>NOTE</u>: Edges of the existing culvert (in grey color) may be inadvertently selected. EnFEM will ignore these dependent lines/arcs.

The top soil layer can be created using the same procedure as to create the bottom soil layer. Note that independent line/arc segments N-M, T-G, O-P-M and O-T-P should be created again. Finally, convert all of the new lines/arcs to a general polygon. The final configuration looks like that in Figure 2.4-11.

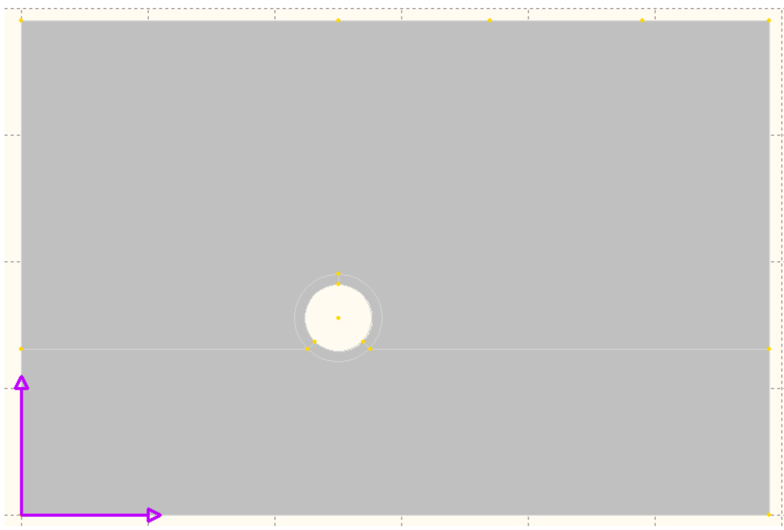


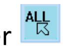
Figure 2.4-11: Generate the top soil region to finish creation of geometric entities

Click to save the model file as "`Analysis of buried culvert.sep`".

Step 2: Define region sets

This example has two independent configurations: one is the soil regions around the culvert; another one is the culvert composed of the three 'fan' regions. Contact definition and post-processing of results often require to operate on the overlapped edges. Model development can benefit from region sets.

In "CAD MODE", click **"Tools->Set Tool"** from the menu. In the set group management window, select **"Region"**. Click "**New**" to generate a new region set. Input the set name "`setCulvert`".

Click “**Add Obj.**”, select three fan regions. Here the user can use default “Select All” filter  or the region filter . RIGHT-CLICK and select “**End Selection**” (Figure 2.4-12). The color of three fan regions changes to purple. Click “**Save & Exit**” to return the set management window.

Similarly, define another region set “`setSoil_all`” which includes the top and bottom soil layers.

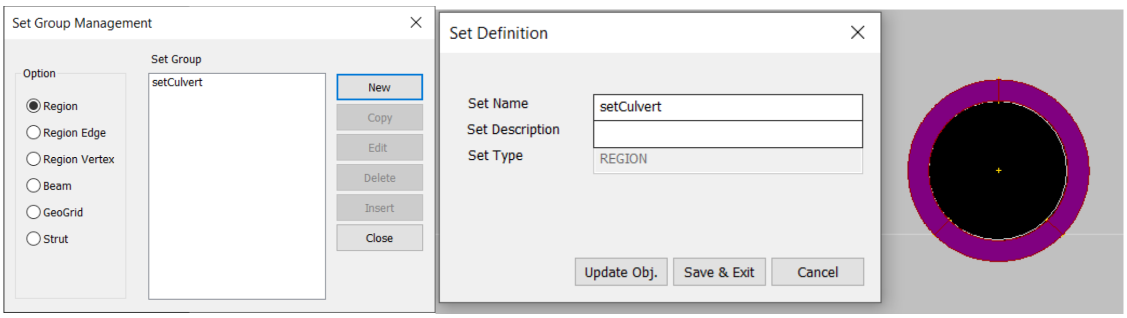


Figure 2.4-12: Define region-sets for Example 4

From the menu bar, click **"Tools->Display Tool"** to open the window for display control. Geometric entities can be shown or hidden conveniently by means of sets. Check the box before “setCulvert”, click “**Show Selected**” (Figure 2.4-13). See that all soil regions have been hidden and the display shows culvert regions only. Now, uncheck “setCulvert”, select "setSoil_all", and click **"Display"**. This time, all soil regions are displayed and culvert set is hidden. Leave the culvert regions hidden, and keep the display control dialog open.

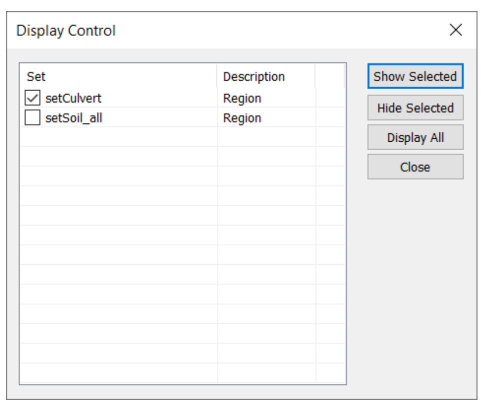


Figure 2.4-13: Use sets to control display/hide of regions

Back to the ‘Set Definition” dialog, change the ‘Set Option’ to **“Edge”**. Click **“New”** to create a new edge-set. Input the set name “`setSoil_Interface`” (seen in Figure 2.4-14). Clicking **“Add Obj.”** and then select the internal three arcs using a ‘contained’ selection window. RIGHT-CLICK and select **"End Selection"** to complete the new set definition. Click **“Save & Exit”** to exit the “Set Management” dialog. The new edge set will be used to output contact results in the post processor.

Back to the “Display Control” dialog, click **“Display All”**, and then **“Close”**. Users may also click  from the tool bar to uncover hidden geometry entities.

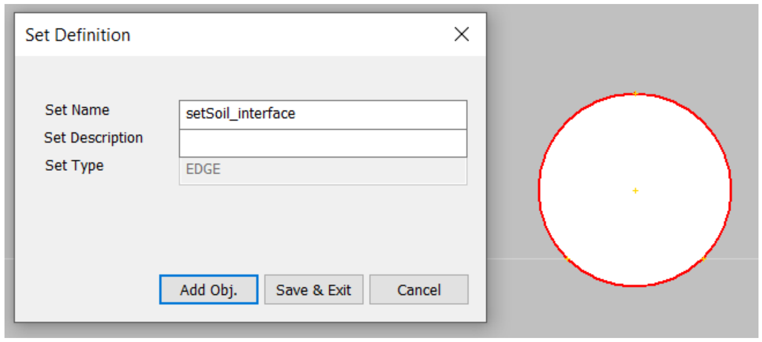


Figure 2.4-14: Define edge-set for Example 4

Step 3: Define stage data

From the menu bar, click **"Data->Stage Management"**, and create two new stages as shown in Figure 2.4-15. Input *"`ints`"* as the name of the first stage. Select "Initial Soil Stress" as the type. The self-weight of the soil and the culvert will be applied in this stage.

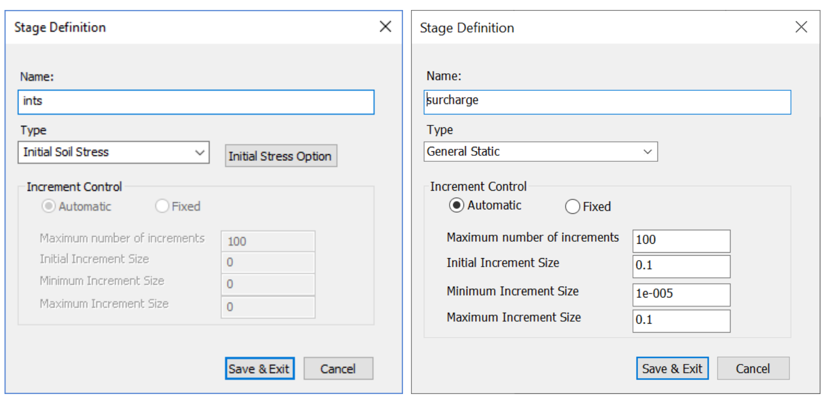


Figure 2.4-15: Define two stages in the example of soil-culvert interaction analysis

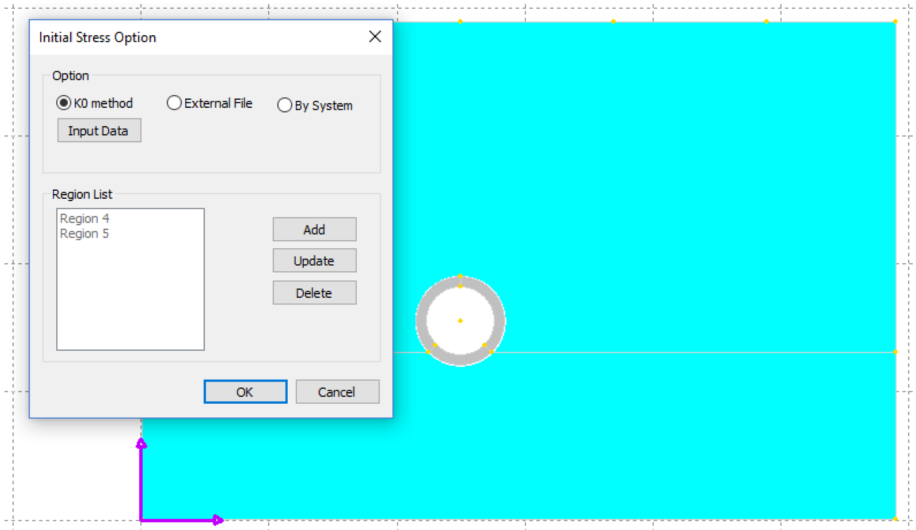


Figure 2.4-16: Set up options for initial soil stress in Example 4

Click **"Initial Stress Option"** to define initial soil stresses (Figure 2.4-16). Keep the **"K0 Method"** option. Click **"Add",** and then select both soil regions in the display window. Next, RIGHT-CLICK and select **"End Selection"**. Both soil layers are added to the "Region List" in this dialog. Next click **"Input Data"** to input vertical soil stresses at controlling vertices (Figure 2.4-17). The Poisson's ratio of the top soil is *0.45.* Lateral stress factor k_0 can be estimated:

$$k_0 = \frac{0.45}{1 - 0.45} = 0.818$$

Click **"Add Row"** and enter the z coordinate and vertical stress of each soil layer at top/bottom position as seen in Figure 2.4-17. Click **"Save & Exit"** from the input dialog (Figure 2.4-17) and then **"OK"** option dialog (Figure 2.4-16), and then **"Save & Exit"** again (Figure 2.4-15).

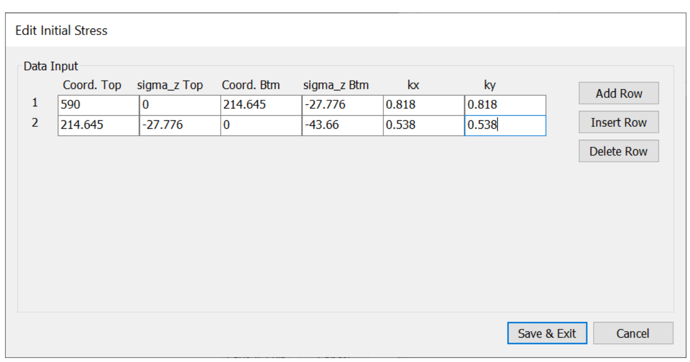


Figure 2.4-17: Input parameters for interpolation of initial soil stress

The second stage, named "`surcharge`", is a "General Static" type. A ground surcharge of 35-psi will be applied in this stage. Keep the default increment settings for the 'surcharge' stage, and click **"Save & Exit"** when complete (Figure 2.4-15).

Step 4: Define material properties

From the menu bar, click **“Data->Material Properties”** to open the window for material group management. Click **"New"** to create the first material named *"`concrete`"*, and enter in unit weight, the Young’s modulus and the Poisson ratio as seen in Figure 2.4-18. Click **"Save & Exit"** when complete.

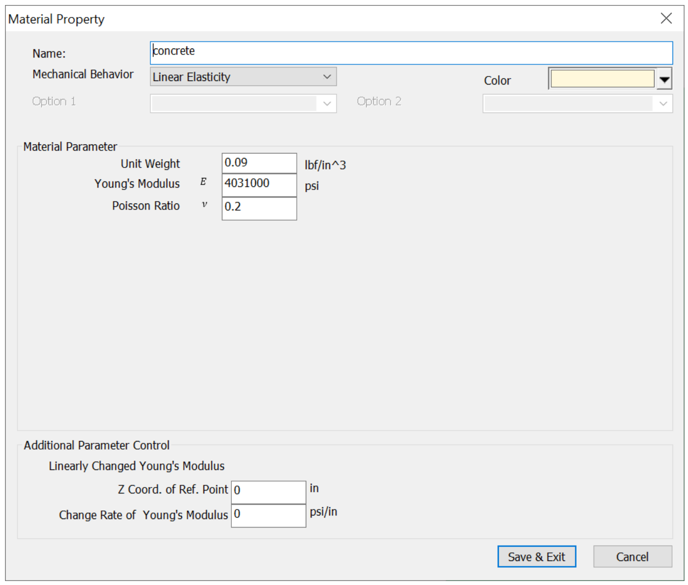


Figure 2.4-18: Define linear elastic type material for culvert

Repeat this process to create Mohr-Coulomb type materials for the top soil layer (first image of Figure 2.4-19), and the bottom soil layer (second image of Figure 2.4-19).

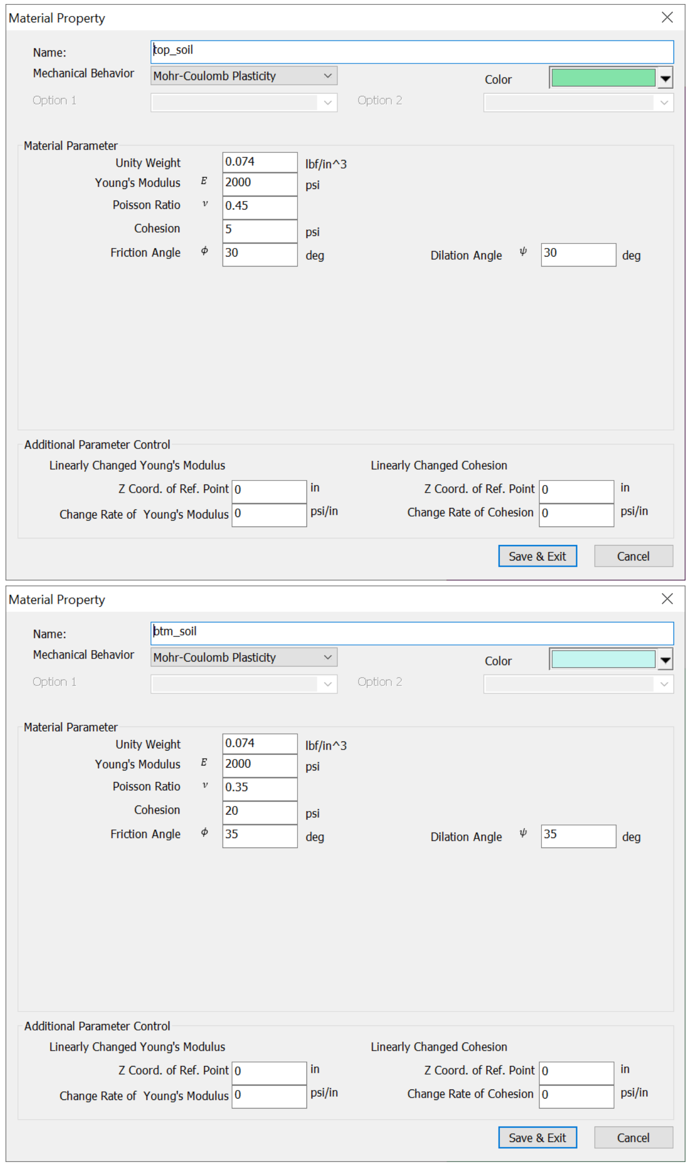


Figure 2.4-19: Define two Mohr-Coulomb type materials for soil layers

Next, click **"Data->Material Assignment"** from the menu bar to open the material assignment dialog. Select "By Set", and then select the region-set "setCulvert" from the set list. The culvert will be highlighted in cyan. Select "concrete" from the Material Group list and click **"Assign"**. The culvert will be highlighted in purple after a successful material assignment (Figure 2.4-20). Next, select "By Individual Region", select "Region 4" (bottom soil layer), and assign material "btm_soil" to the bottom soil layer. Similarly, assign material "top_soil" to the top layer. Click **"Close"** when complete.

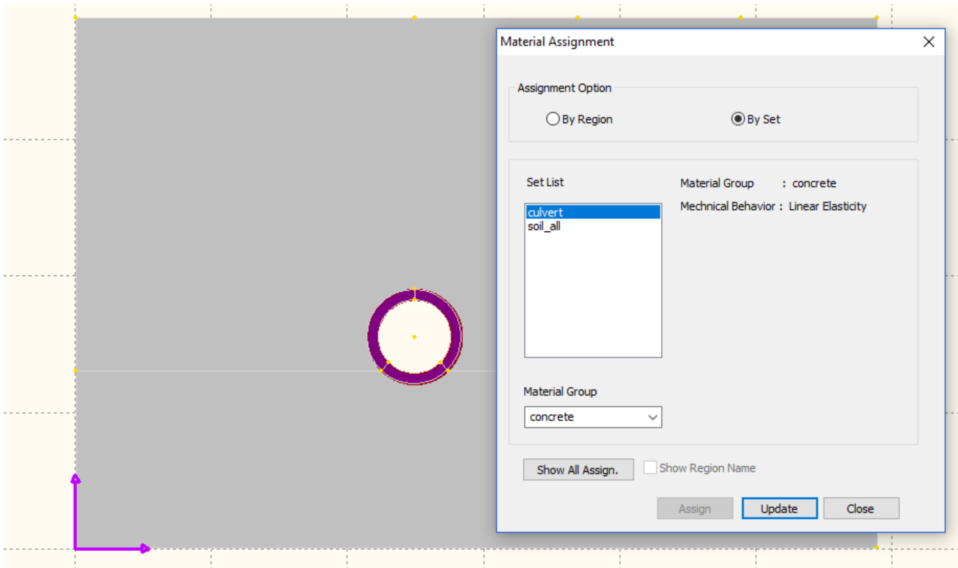
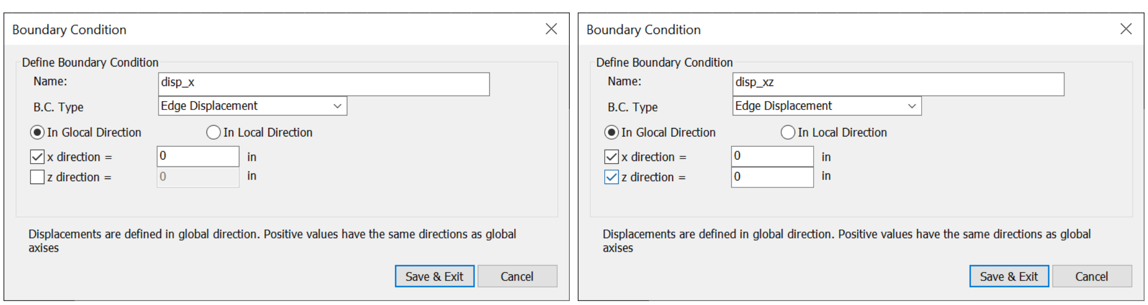


Figure 2.4-20: Assign material definitions using "By Set" option

Click **"Close"** to exit from this dialog.

Step 5: Define boundary conditions

From the menu bar, click **"Data->Boundary Conditions"** to open the boundary condition group management dialog. Define the following boundary conditions "`disp_x`", "`disp_xz`", "`gravity`" and "`surcharge`" as seen in Figure 2.4-21. Displacement boundaries are typically defined in global directions for horizontal and vertical edges. For inclined edges, defining them in local directions may be more practical. Surface loads should be defined in local directions to accommodate various edge inclinations.



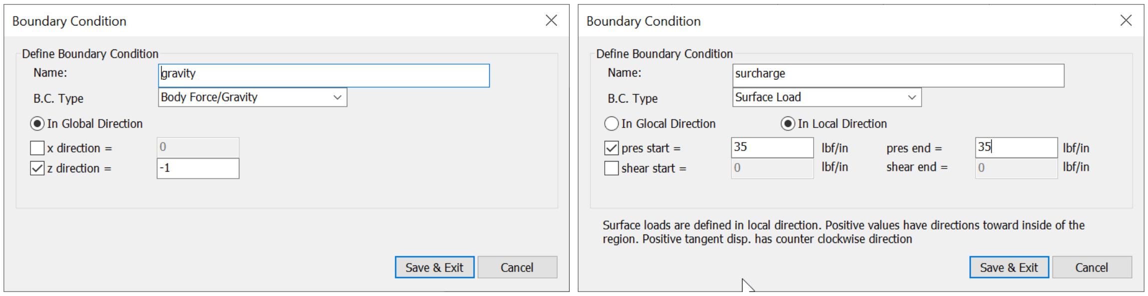


Figure 2.4-21: Define boundary conditions in culvert analysis

Next, from the menu bar, select "**Data → Boundary Assignment"** to assign boundary conditions to geometric entities:

- Assign "*disp_x"* to the left and right vertical soil edges.
- Assign "*disp_xz*" to the bottom soil edge.
- Assign "*gravity"* to the culvert and both soil regions.
- Assign "*surcharge"* to the ground edge (line C-D).

By default, all boundary conditions start from the first stage. To adjust the "surcharge*"* condition, select it, click "Created*"*, and use "**Move Right"** to set its start stage to the second stage. Figure 2.4-22 illustrates the stages and their related definitions.

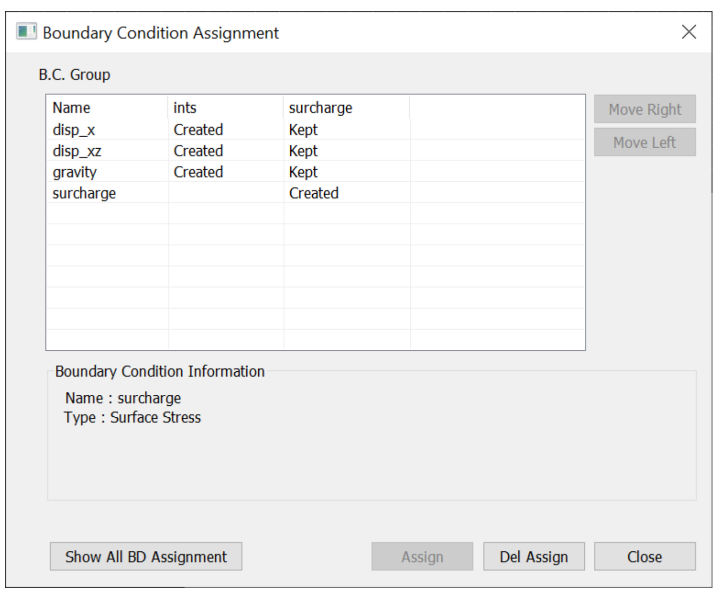


Figure 2.4-22: Assign boundary conditions in Example 4

When complete, click **"Show All BD Assignment"** to verify assignments (Figure 2.4-23).

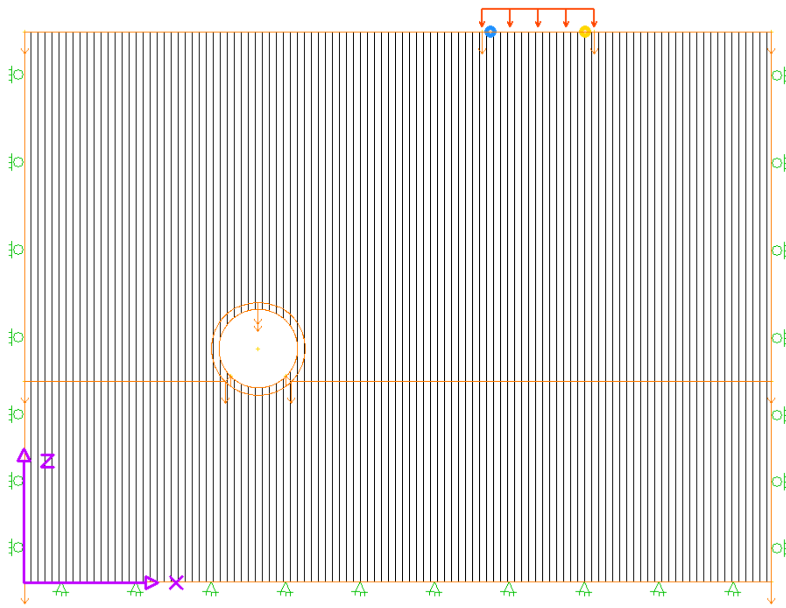


Figure 2.4-23: Show all assigned B.C. in Example 4

Step 6: Define contact property of soil-culvert interaction

When defining contact properties, exclusive display of either the soil layers or the culvert is necessary due to overlapping soil-culvert interaction surfaces. From the menu bar, select "**Tools → Display Tool"** to open the display control dialog. In the set list box, choose "setCulvert" and click "**Show Selected"** to display only the culvert and hide the soil layers (Figure 2.4-24).

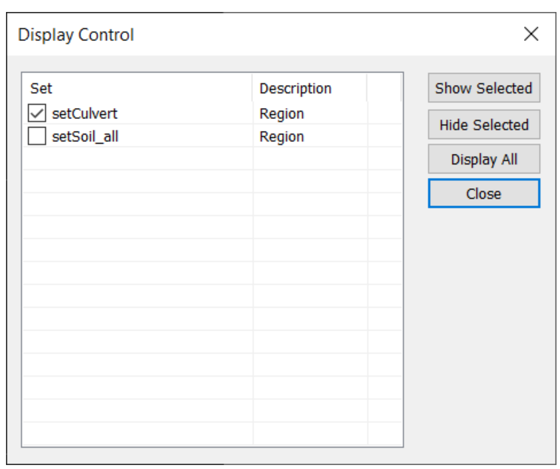


Figure 2.4-24: Use the display tool to hide soil layers

With the display control dialog open, navigate to menu "**Data-> Interaction…-> Contact Constrained Method"** to access the interaction group management window. Click "**New"** to create a contact group. Keep the default group name and interface option as "**contact**", then set the coefficient of friction to *0.36*.

Click "**Add Master Surf.**", hold SHIFT and LEFT-CLICK to select all three external arcs of the culvert, then RIGHT-CLICK and select "**End Selection"**. The selected arcs, now marked in blue, represent the master surfaces.

Switch back to the display control dialog and click "**Hide Selected"** to display only the soil regions. Return to the interaction definition window, click "**Add Slave Surf.**", and use a 'contained' selection window to highlight all three arcs of the soil regions. RIGHT-CLICK and select "**End Selection**" to complete the process. The selected arcs from the soil layers, marked in green, represent the slave surfaces (see Figure 2.4-25).

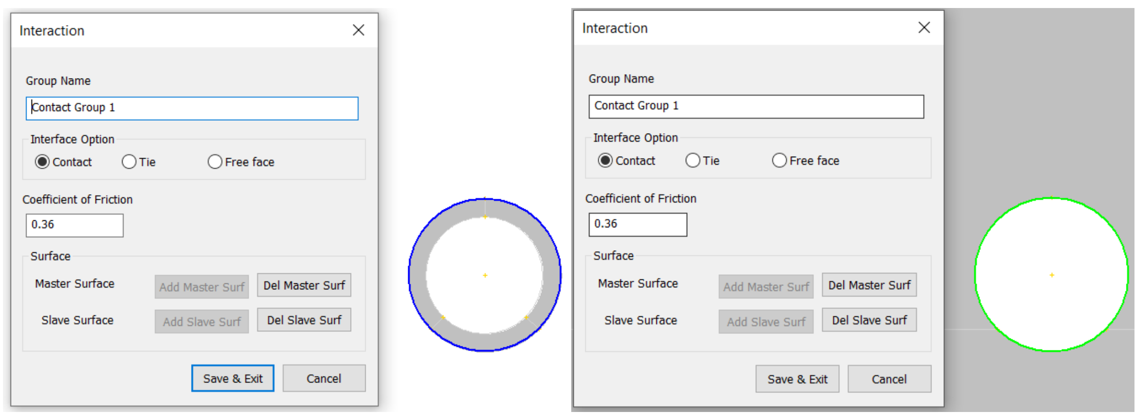


Figure 2.4-25: Define interaction property, and master/slave surfaces

Click **"Save & Exit"** to return to the interaction group management dialog. Click **"Close"** to exit.

Click to save the file.

Step 7: Mesh generation

In this example, the culvert is simulated using solid finite elements. Since EnFEM only supports only linear elements (Q4/T3) for contact analysis, careful mesh selection is crucial for accuracy. Click to display all soil layers and the culvert, then use the zoom window tool, , to focus on the culvert region. From the menu bar, click **"pre-Mesh->Advanced Mesh Control"**, and select the three 'fan' regions belonging to the culvert. RIGHT-CLICK and select "**End Selection**" to open the mesh control dialog, as shown in Figure 2.4-26.

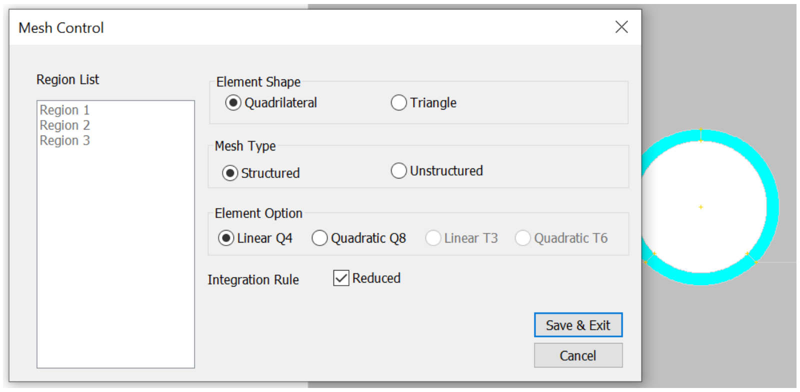


Figure 2.4-26: Advanced mesh control for culvert regions

Note: Retaining walls, piles, and culverts are common in geotechnical engineering, where soil-structure interactions (SSI) are a key focus when structures are present. Slim rectangles or rings, such as walls or culverts, behave more like beams due to dominant flexural deformation.

EnFEM offers two methods for addressing SSI problems:

1. **Contact Constraint Method** for interactions between solid finite elements.
2. **Zero-Thickness Interface Elements**, where structures are modeled as beam elements.

For models with contact interactions, using Structured Quadrilateral (Q4) elements is recommended to enhance accuracy. The integration rule—*full* or *reduced*—should be chosen based on whether the structure is confined by soil:

- Reduced integration minimizes shear locking in Q4 elements and is suitable for confined structures (Figure 2.4-26).
- If the structure is not fully confined, reduced integration is not recommended due to potential stability issues during computation.

Structured meshes also enable the post-processing of integrated forces, such as moments and shear forces, ensuring accurate analysis results.

Based on the above considerations, set the mesh control parameters as "Quadrilateral", "Structured", and a "Linear Q4" element with reduced integration rule, as seen in Figure 2.4-26. Click **"Save & Exit"** when complete.

Again, click **"pre-Mesh->Advanced Mesh Control"**, this time select the two soil regions, and then RIGHT-CLICK and select **"End Selection"**. In the mesh control dialog change the settings to "Quadrilateral", "Unstructured", and "Linear Q4" with full integration, as seen in Figure 2.4-27.

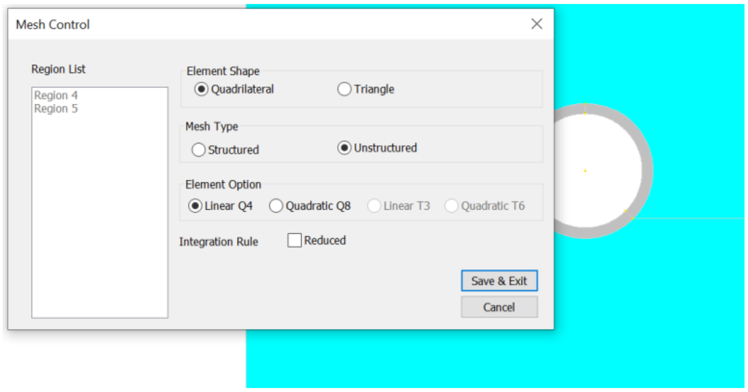


Figure 2.4-27: Advanced mesh control on soil layers

From the menu bar, click **"pre-Mesh->Show Mesh Assignment"** to display mesh properties on each region (Figure 2.4-28). Detailed information about referential colors for mesh assignments can be found in the user's manual.

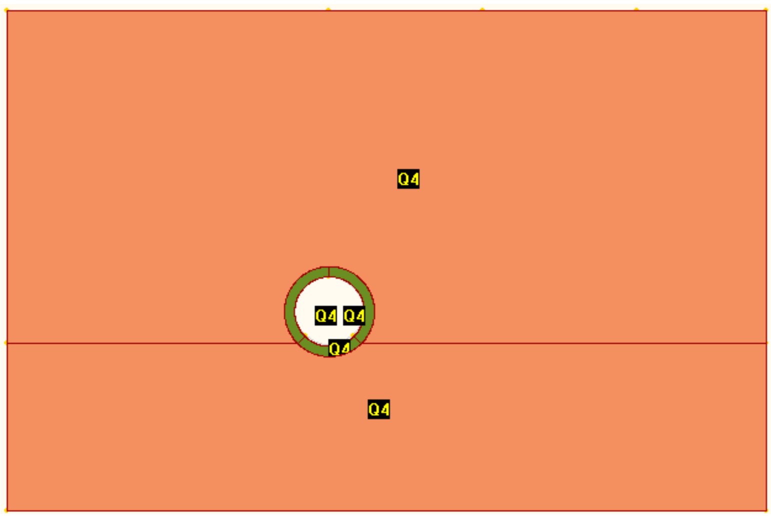


Figure 2.4-28: Show mesh assignment on regions

Click **"pre-Mesh->Show Mesh Assignment"** again to turn off the display of the mesh assignments.

The next step is to define the finite element size. A uniform mesh is unsuitable for this model because:

- **Coarse meshes** poorly discretize contact interfaces.
- **Fine meshes** significantly increase solving time.

A non-uniform mesh is recommended to balance accuracy and computational efficiency. Fine mesh elements should be applied at contact interfaces for smooth results, while larger elements can be used near external soil boundaries.

In this analysis, integrated forces (moment, shear, thrust) within the culvert are critical. To ensure accuracy, four element layers will be used across the culvert's cross-sectional thickness (radial direction). This multiple element layers are helpful to minimize shear-locking effects. With a culvert centerline perimeter of approximately 288 inches and a Q4 element height-to-width ratio of 2.5, around 58 divisions along the centerline will be applied.

From the menu bar, click **"pre-Mesh->Global Seed Control"** and enter *"30"* as the global seed size. Click **"Assign"**. Then assigned seeds will be displayed as in Figure 2.4-29. Click **"Close"** to exit from this dialog.

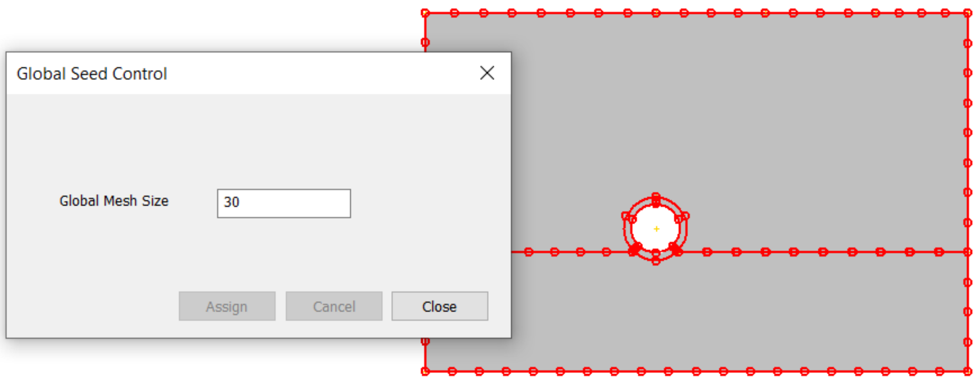


Figure 2.4-29: Assign global mesh size to control mesh on external boundaries of soil

Now to refine the mesh around the culvert. Click  and zoom into the culvert, with the zoom window.

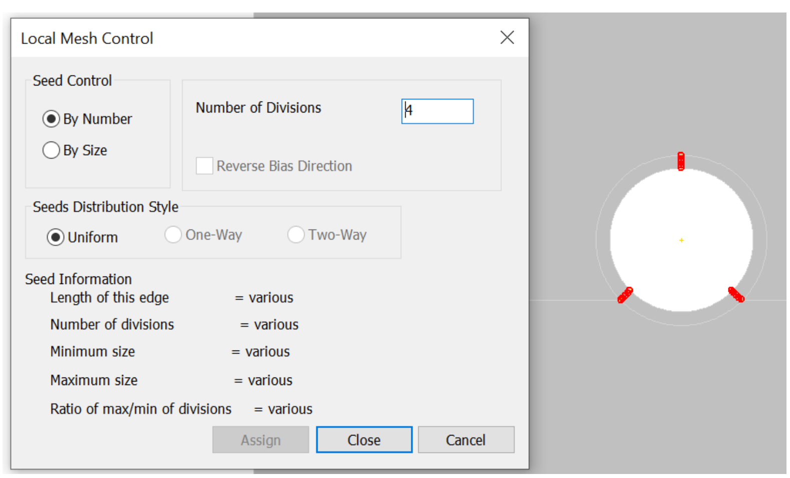
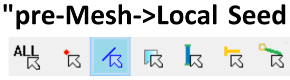


Figure 2.4-30: Define seed divisions on culvert cross-section

From the menu bar, click **"pre-Mesh->Local Seed Control"**, note that the line selection icon is automatically turned on, , is actived. Select the three, radial line-segments of the culvert, then RIGHT-CLICK and select **"End Selection"**. The local seed control dialog will open. Change the number of divisions to "4" and keep the other settings as default. Click **"Assign"**. The new distribution of seeds is shown in Figure 2.4-30. Click **"Close"** to exit.

Next, from the menu bar, click **"Mesh->Local Seed Control"** and select the four arcs of the two larger fans. RIGHT-CLICK and select **"End Selection"**. The local seed control dialog will open again.

NOTE: When culvert edges overlap with soil layer edges, both are selected simultaneously, but the overlapped edges are counted as one in seed control. This ensures a conforming mesh between the culvert and soil layer. If multiple non-overlapping edges are selected, only uniform seed distribution is possible.

In this example, seed control is based on the *number of divisions*. As the culvert uses a structured mesh, the same division number must be assigned to opposite sides. For more details about local seed control, refer to Chapter 11 of the user manual.

In the dialog of local seed control, keep the option of seed control as "By Number". Enter "22" as the "Number of Divisions" and click **"Assign"** to complete. Updated divisions of the selected arcs are shown in Figure 2.4-31.

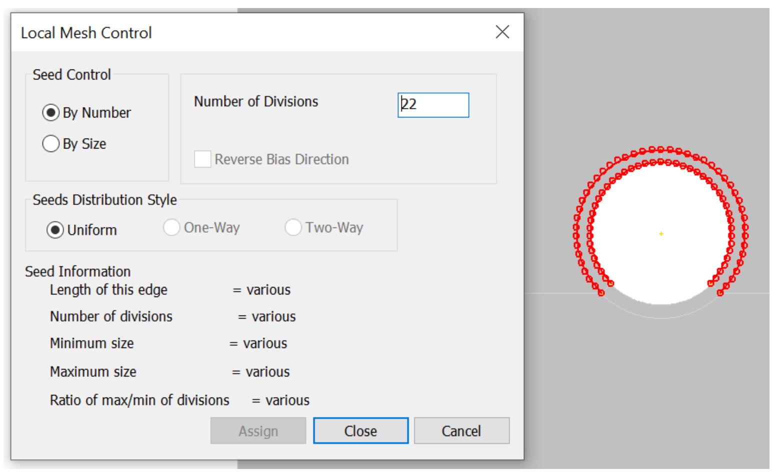


Figure 2.4-31: Local seed control on upper fan regions

From the menu bar, click **"pre-Mesh->Local Seed Control"** and select two arc edge of the lower fan region. RIGHT-CLICK and select **"End Selection"** to open the local seed control dialog as before (Figure 2.4-32). Keeping the "Seeds Control" option as "By Number", change the division number to "14" and click **"Assign"**. Click **"Close"** to exit.

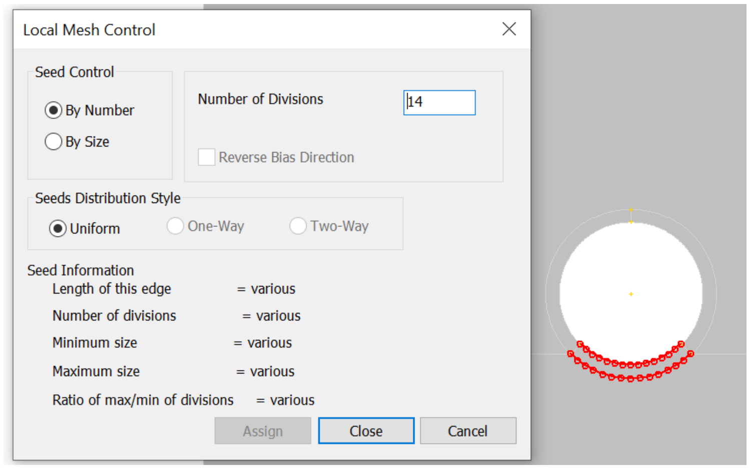


Figure 2.4-32: Local seed control on lower fan region

The mesh refinement on the culvert has been completed. Now adjust the seeds on the horizontal edges separating the upper and the lower soil layers to make the mesh change size gradually. This will create a smooth transition from the coarse mesh area to the fine mesh area (culvert arcs).

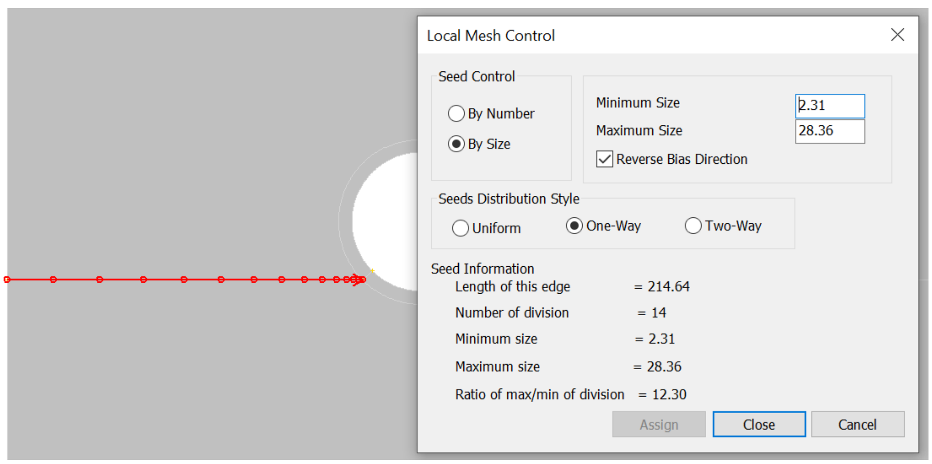


Figure 2.4-33: Local seed control on the left common edge between soil layers

From the menu bar, click **"pre-Mesh->Local Seed Control"** and select the left edge shared by the top and the bottom soil layers. RIGHT-CLICK and select **"End Selection"**. In the local seed control dialog, change the control option to "By Size". Next, select the seed style as "One Way" and check the option "Reverse Bias Direction", to have refinement go in the direction toward the culvert. Change the minimum size to "3" and maximum size to "30" (Figure 2.4-33). Click **"Assign"**. Similarly, change the local seed control on the right shared soil edge to "By Size", "One Way", minimum size= "3", and maximum size= "30". Leave the "Reverse Bias Direction" unchecked, to set the refining direction toward the culvert. The final seed distribution around the culvert is shown in Figure 2.4-34. It is normal that the final minimum and maximum spacing may be a little bit different from user's input.

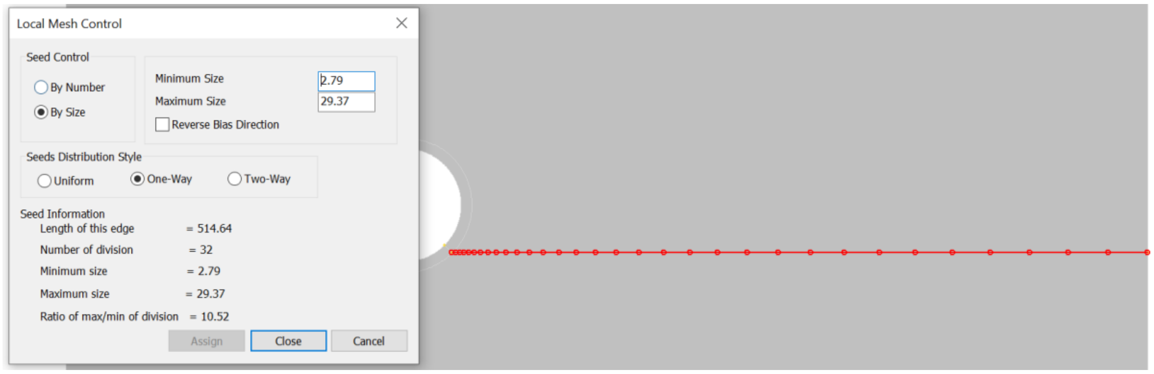


Figure 2.4-34: Final seed distribution around culvert

<u>NOTE:</u> When assigning 'bias' to a mesh refinement, a red arrow will appear on the selected line to give visual confirmation of the refinement direction.

From the menu bar, click **"Mesh->Generate Mesh"**. Since the reduced integration rule is used for Q4 elements, a warning message will pop up as shown in Figure 2.4-36. Click "**OK**" to continue. If the mesh is successfully generated, it will be loaded automatically (Figure 2.4-35).

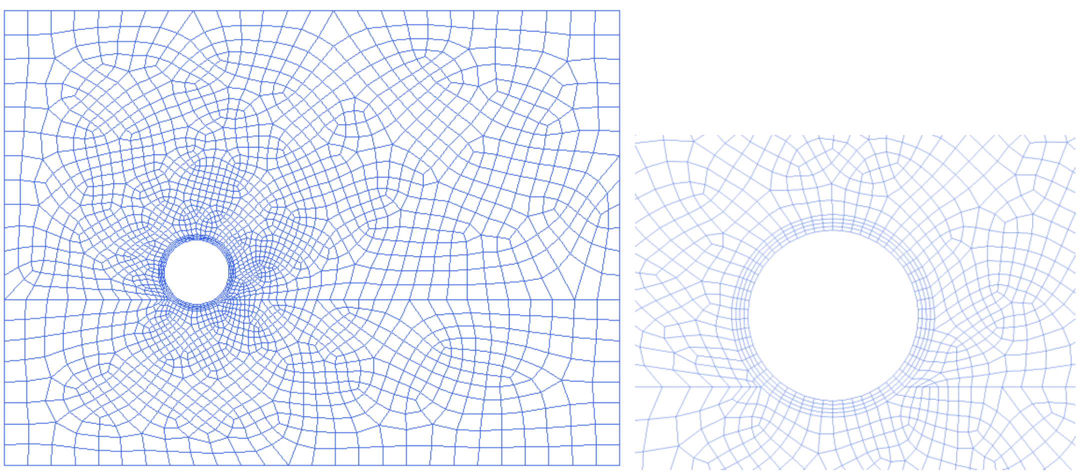


Figure 2.4-35: Mesh generated for buried culvert analysis

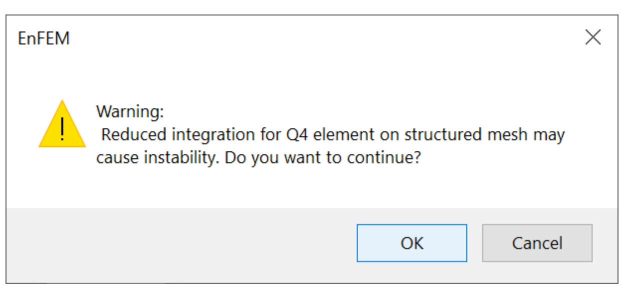


Figure 2.4-36: Warning message for reduced integration on Q4

Click to save the file.

NOTE: A high-quality mesh for a complex geometry may often take several trials/iterations to obtain. When attempting local mesh refinement, the most important rule is to keep the finite element size around one vertex as uniform as possible. More detail about this strategy and other useful information regarding mesh generation can be found in the user's manual.

Step 8: Analysis

From the menu bar, click **"Analysis->Analysis Setting"** to open the dialog for analysis options (Figure 2.4-37). Keep the default setting for convergence and interactions. Check the option for **"Contact"** in the output group, so that contact stresses will be output to the file *"analysis of buried culvert.out"*. Click **"OK"** to exit.

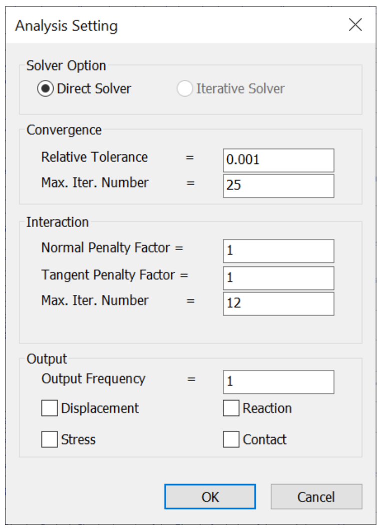


Figure 2.4-37: Analysis setting for buried culvert

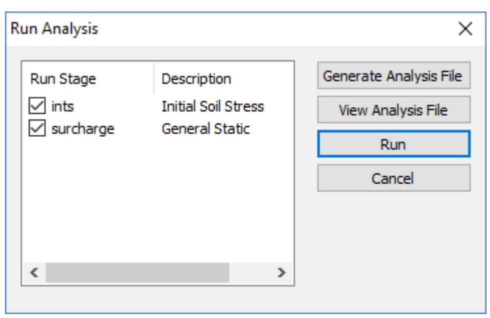


Figure 2.4-38: Run analysis in Example 4

From the menu bar, click **"Analysis->Run Analysis"** to open the window for analysis management. Set the check mark before each stage, click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. The file, *"analysis of buried culvert.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history.

Step 9: Post-processing of result

Click to switch to the "RESULT-MODE" and then click to open the Contour Plot dialog. The default contour is contour of the displacement in the X-direction (u1) at the last loading increment (Figure 2.4-39). Next, check **"**Show Contour on Deformed Body**"**, change the scale factor to "`10`", and click **"Apply"** (located below "contour limits"). According to this contour, the maximum lateral displacement for the soil is approximately -0.51-inches, and the maximum lateral displacement for the culvert is roughly -0.24-inches.

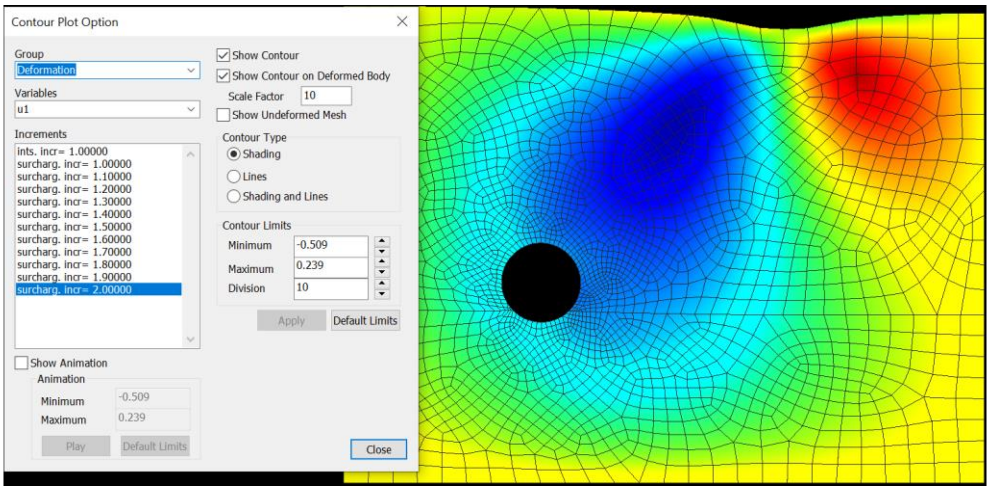


Figure 2.4-39: Displacement contour for soil-culvert interaction analysis

Leaving the Contour Plot dialog open, from the menu bar, click **"Tools->Display Tool"**. Select the "*setCulvert*" from set list and click **"Display"** to display only the culvert. Zoom-in as needed. Select **"**Stress**"** from the Group drop-down list and select **"**Equivalent Shear q**"** from the Variables drop-down list. This distribution of the Mises-stress on the culvert is plotted in Figure 2.4-40.

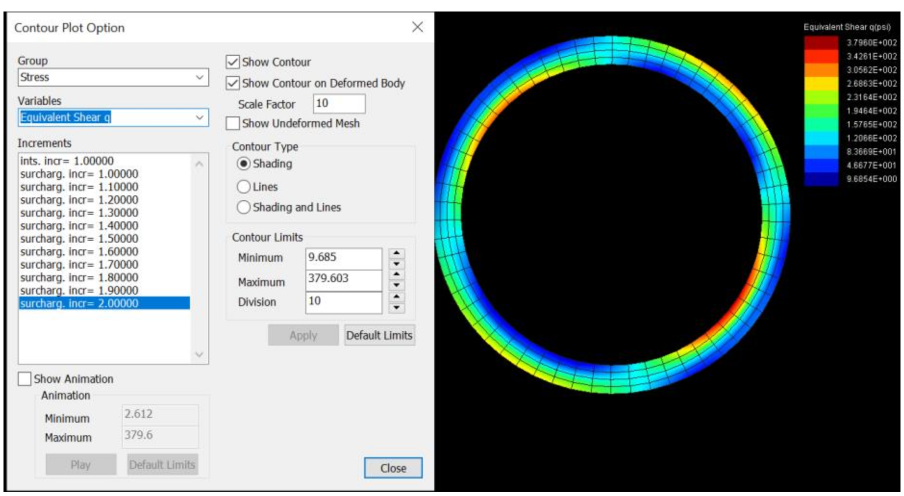


Figure 2.4-40: Mises stress inside culvert

Click **"Close"** to exit from the contour dialog.

With the soil-set still hidden, click **"Mesh->Show Node Numbers"** from the menu bar. Next click  to open the X-Y plot dialog. Select "Field Result" as the source data type. Input the following node list (corresponding to the node on the 'centerline' of the culvert thickness):

`47:69:1,144:157:1,248:228:-1`

Select "Integrated Force" as the Category and "Moment" as the Variable. Next, select the last increment, "Surcharge Incr=2.0000", and click **"Plot"** to plot the moment along the defined path (see Figure 2.4-41). The first node is located at the top of the culvert.

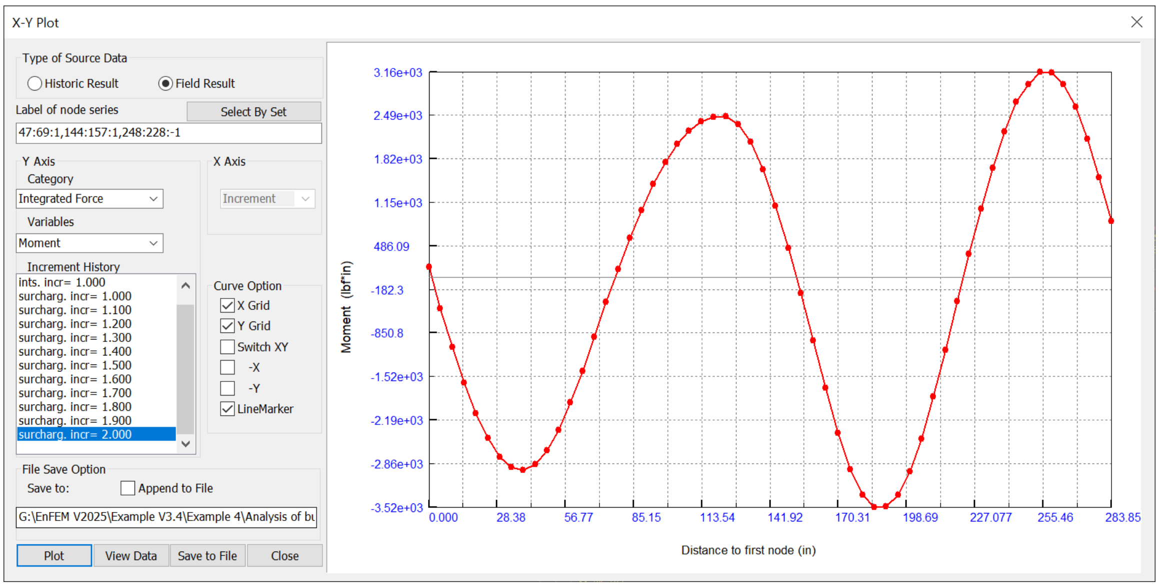


Figure 2.4-41: Moment of culvert subject to self-weight and surcharge

Keeping "Field Result" as the source data type, select "By Set". From the "Set Selection" dialog, select the previously defined edge-set, "setSoil_interface", and click **"OK"**. Select "Interaction" as the Category and "Normal Pressure" as the Variable. Select the last increment, "Surcharge Incr=2.0000", and click **"Plot"** to display the normal pressure along the defined node-path (see Figure 2.4-42).

<u>NOTE:</u> When plotting results of an "Integrated Force", only the region in which the integration shall occur should be displayed – in this case only the culvert is displayed.

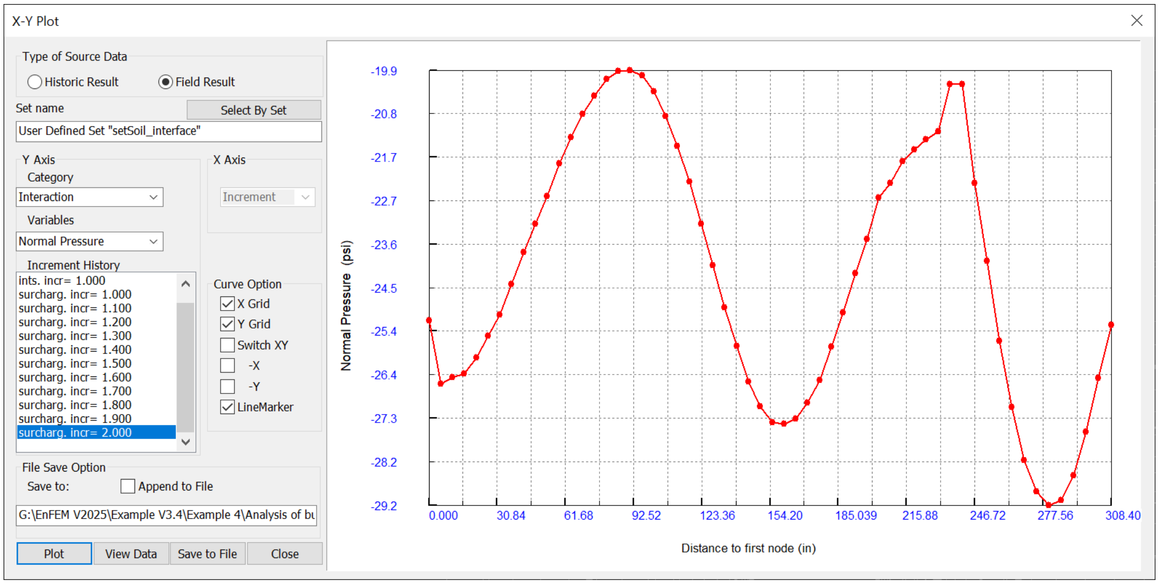


Figure 2.4-42: Normal contact pressure on soil side after surcharge

<u>NOTE:</u> Only nodes on the *'slave interaction-surface'* are available to plot contact stresses in EnFEM. Additionally, the start node in Figure 2.4-42 is determined by EnFEM automatically. If users want to control the sequence of nodes, they may input the node list - as similarly done for generating the plot in Figure 2.4-41.

Example 17 in this manual simulates the same buried culvert by means of beam elements and zero-thickness interface. Some results from both models are plotted in Figure 2.4-43~Figure 2.4-45. It's clear that results from both models agree well.

Figure 2.4-43: Comparison of moment of culvert between example 4 and 17

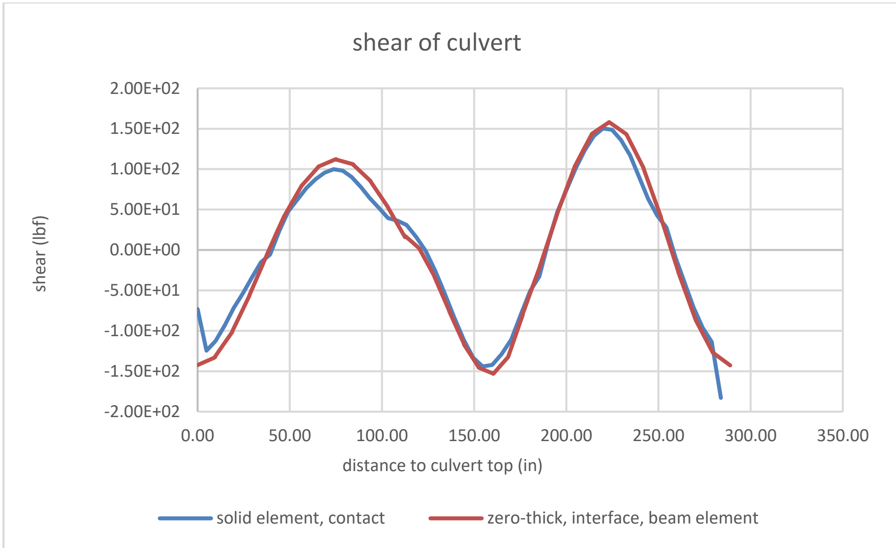


Figure 2.4-44: Comparison of shear of culvert between example 4 and 17

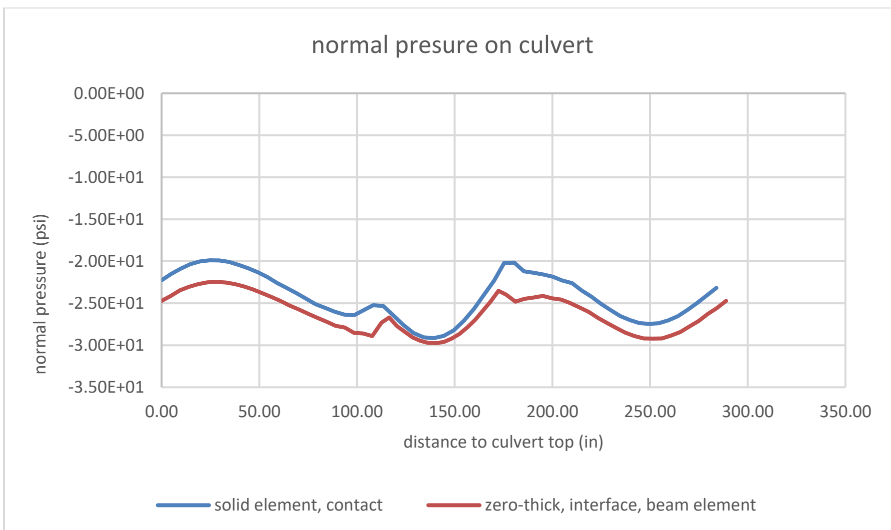


Figure 2.4-45: Comparison of normal pressure on culvert between example 4 and 17

Since the perimeters of the culvert are different in both models, the normal pressure from example 4 is normalized in Figure 2.4-45.

2.5 Example 5: Stability analysis of slope, strengthened with secant-pile wall

In this example, stability of pile-wall strengthened slope is analyzed. The physical geometry is shown in Figure 2.5-1. Sub-regions of the pile-wall are shown in the right image of Figure 2.5-1. The size of the pile-wall is 0.2m x 15.6m. Units in examples are: length (m), stress (kPa) and force (kN).

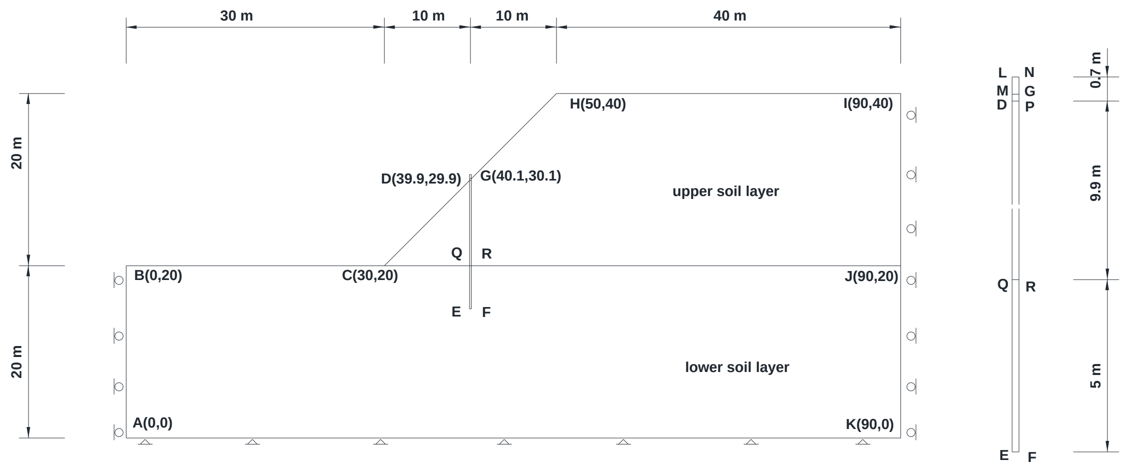


Figure 2.5-1: Configuration of stability analysis of slope, strengthened with pile

Controlling vertices for this example are listed in Table 2.5-1. The pile-wall configuration consists of four rectangle regions.

	A	B	C	D	E	F	G	H	I
X	0	0	30	39.90	39.90	40.10	40.10	50.0	90.0
Z	0	20	20	29.90	15.0	15.0	30.10	40.0	40.0
	J	**K**	**L**	**M**	**N**	**P**	**Q**	**R**	
X	90.0	90.0	39.90	39.90	40.10	40.10	39.90	40.10	
Z	20.0	0.0	30.60	30.10	30.60	29.90	20.0	20.0	

Table 2.5-1: Coordinates of controlling vertices of Example 5

Material Properties:

Top soil layer:

- Unit weight:.................................$\gamma = 20\ kN/m^3$
- Young's Modulus:.........................$E = 8.0 \cdot 10^4\ kPa$
- Poisson's ratio:............................$\nu = 0.4$
- Cohesive strength:........................$c = 35\ kPa$
- Friction angle:..............................$\phi = 25°$
- Dilation angle:..............................$\psi = 25\ °$

Bottom soil layer:

- Unit weight:.............................. $\gamma = 20\ kN/m^3$
- Young's Modulus:...................... $E = 1.0 \cdot 10^5\ kPa$
- Poisson's ratio:........................ $\nu = 0.3$
- Cohesive strength:..................... $c = 40\ kPa$
- Friction angle:........................... $\phi = 35°$
- Dilation angle:........................... $\psi = 35°$

Concrete (Pile-Wall):

- Unit weight:............................ $\gamma = 25\ kN/m^3$
- Young's Modulus:..................... $E = 2.8 \cdot 10^7\ kPa$
- Poisson's ratio:....................... $\nu = 0.3$

Both soil layers are simulated using the Mohr-Coulomb plasticity model. The pile-wall is made from concrete and will be simulated by the linear-elastic constative model.

Boundary Conditions:

- Displacement constraints on vertical external soil edges
- Displacement constraints on bottom edge of lower soil layer
- Gravity load
- Contact interface between pile-wall and soil.
    - The pile-wall is the master body and the soil is the slave body.
    - Coefficient of friction for the contact interface is 0.3.

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM to create a new project. Change the title to `"Stability analysis of slope strengthened by anti-sliding pile-wall"`. Keep the problem type as "Plane Strain" (Figure 2.5-2**Error! Reference source not found.**). Select preselected unit option "kN, m, second". Click **"OK"** to exit.

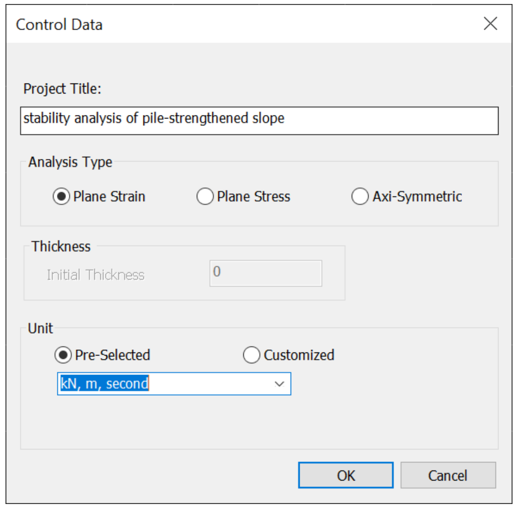


Figure 2.5-2: Control data for Example5

Following Figure 2.5-3, set up the drawing space and grid spacing.

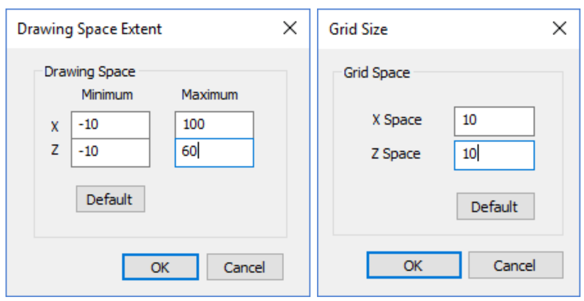


Figure 2.5-3: Setup drawing space and grid size for Example 5

Click  to activate the rectangle creation tool. In the coordinate input box (located beneath the working zone), key in the coordinate of the left lower corner of rectangle E-F-R-Q (pile-wall), *"39.9,15"*. Press RETURN to create the first Vertex "*E*". Next, key in the relative coordinate of vertex R, "*[0.2,5]*" (Figure 2.5-4), and press RETURN. The first rectangle E-F-R-Q (pile-wall) is now created.

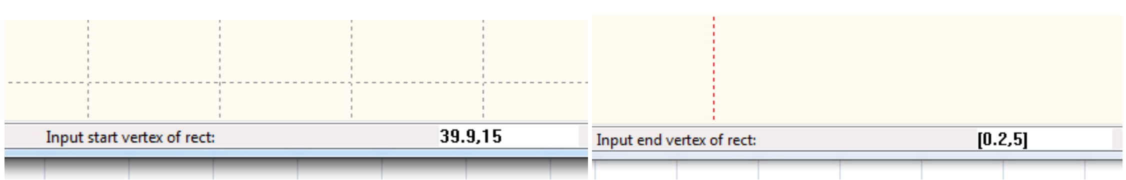


Figure 2.5-4: Create rectangle by keying in coordinates

Hold CTRL and then LEFT-CLICK when 'snapped' to Vertex "*Q*" to set the first vertex of the new rectangle. Key in the relative coordinate "*[0.2,9.9]*", and press RETURN to complete rectangle Q-R-P-D. Repeat this 'snap-method' at Vertex "*D*", for the first vertex of the third rectangle. Now enter in the relative coordinate `[0.2,0.2]` and press RETURN to complete the third rectangle (D-P-G-M). Follow similar steps to create the fourth rectangle of the pile-wall, M-G-N-L.

All regions of the pile-wall have been created.

Activate the polygon creation tool, . 'Snap' to Vertex "*R*" and then LEFT-CLICK to create the first vertex of the new polygon. Continuously create two more polygon vertices using the 'snapping method' at Vertex "*P*" and "*G*". Next, key-in the absolute coordinate of Vertex "*H*", `"50,40"`, and press RETURN. Now key-in the relative coordinate for vertex I, `"[40,0]"`, and press RETURN. Key-in the relative coordinate of Vertex "*J*", `"[0,-20]"`, and press RETURN. Finally, RIGHT-CLICK and select **"Close Polygon"** to complete the first polygon for the top soil layer (Figure 2.5-5).

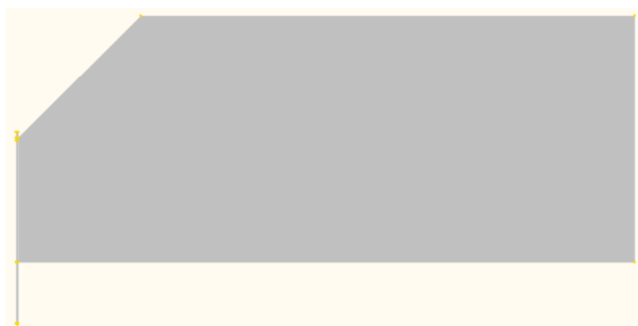


Figure 2.5-5: Create polygon of the top soil layer

The left, triangular region of the top soil layer has three vertices, "*C*", "*Q*", and "*D*". Still using the polygon creation tool, key in the absolute coordinate of vertex "*C*", `"30,20"`, and press RETURN. Create Vertex "*Q*" and "*D*" by 'snapping' to existing vertices of the pile-wall (Figure 2.5-6). RIGHT-CLICK and select **"Close Polygon"** when complete.

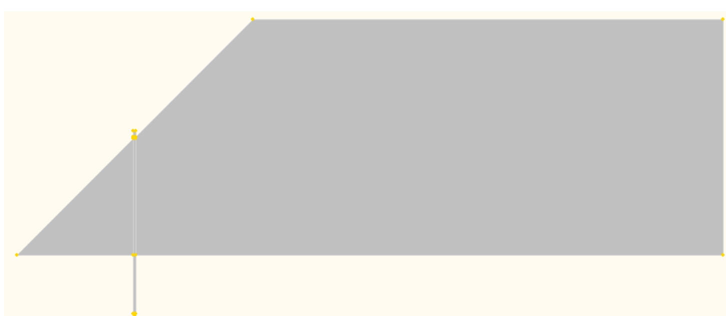


Figure 2.5-6: Create triangular region for upper soil layer

Click to activate the polygon-creation tool, . 'Snap' and then LEFT-CLICK on Vertex "*C*" of triangle C-Q-D as the start of the polygon. Key-in the relative coordinate `"[-30,0]"` of Vertex "*B*" and press RETURN. Key in the absolute coordinate of vertex "*A*", `"0,0"` and press RETURN. Key in the relative coordinate of "*K*", `"[90,0]"`, and press RETURN. Key in the relative coordinate of "*J*", *"[0,20]".* Create following vertices R,F,E,Q. RIGHT-CLICK and select **"Close Polygon"** when complete (Figure 2.5-7).

NOTE: For the vertices near the pile-wall, it may be beneficial to use the "zoom window", , operation. Notice that this zoom-function doesn't interrupt the creation of a continuous line. Users may zoom into the lower part of the pile-wall to make snapping operations easier.

Click to save the model file as "`Stability of strengthened slope.sep`".

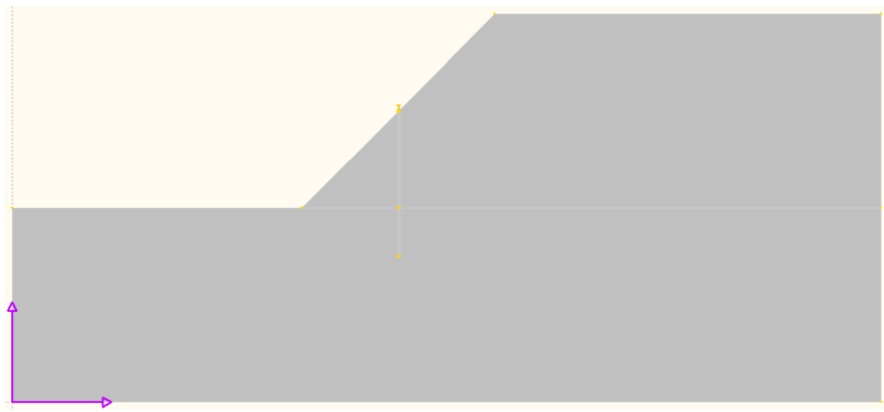


Figure 2.5-7: Create bottom soil layer as a general polygon

Step 2: Define region sets

From the menu bar, click **"Tools->Set Tool"** and create four region-sets as seen in Figure 2.5-8. Click "**New**" to create four region sets. The set "`setWall`" includes all pile-wall regions, the set "`setSoil_all`" includes all soil regions, and the set "`setSoil_top`" includes the two soil regions (triangular and trapezoidal) above coordinate Z=20-m. In the set generating dialog, enter in the set-name, click "**Add Obj**." to select appropriate geometric entities. RIGHT-CLICK and select **"End Selection".** Click **"Save & Exit"** to add new region-sets into model database.

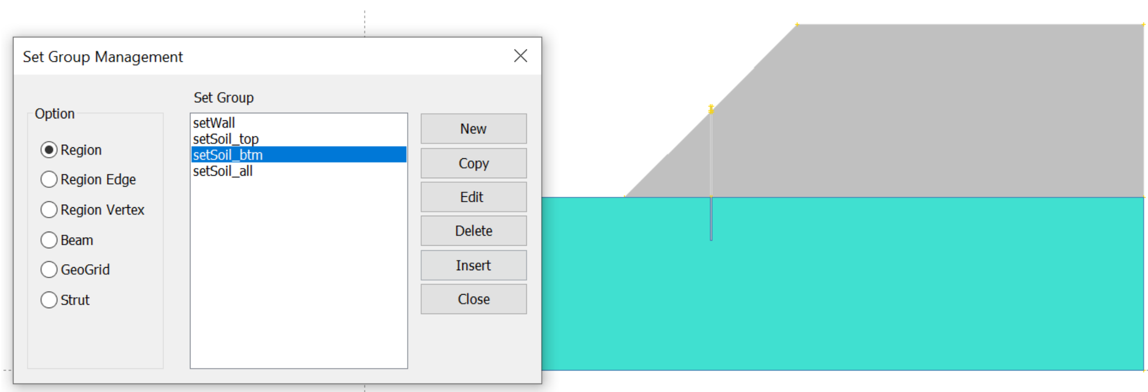


Figure 2.5-8: Define region sets in Example 5

Still within the *'Set Definition'* dialog, change the set option to **"Region Edge"** and create four edge-sets as seen in Figure 2.5-9. *'setWall_right'* has all vertical edges on the right side of the pile-wall, while *'setWall_left'* has similar for the left side of the pile-wall. *'setSoil_right'* and *'setSoil_left'* have the vertical edges that correspond with those from each of the respective sides of the pile-wall. These edges will be used later for plotting results on these faces.

Note: The user will need to use the **"Display Tool"** ("**Tools->Display Tool"**) to hide/display certain regions while making these edge-set definitions.

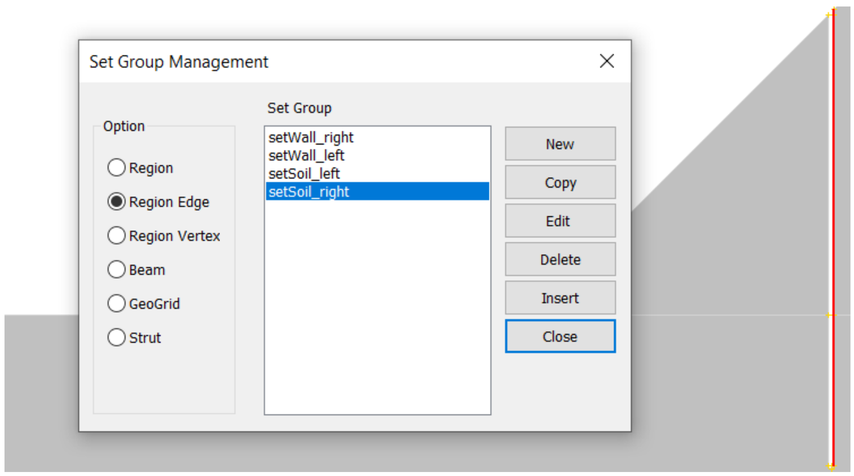


Figure 2.5-9: Create four edge sets on contact interfaces

Click **"Close"** to exit from the *'Set Definition'* dialog when complete.

Step 3: Define stage data

From the menu bar, click **"Data->Stage Management"**, to create two new stages. The first stage, named *"load"*, is an **"Initial Soil Stress"** type. See Figure 2.5-10 for the stage and initial stress option definitions. Click **"Add"** and select all of the soil regions, and then RIGHT-CLICK and select **"End Selection"** to add the soil regions to the initial stress definition.

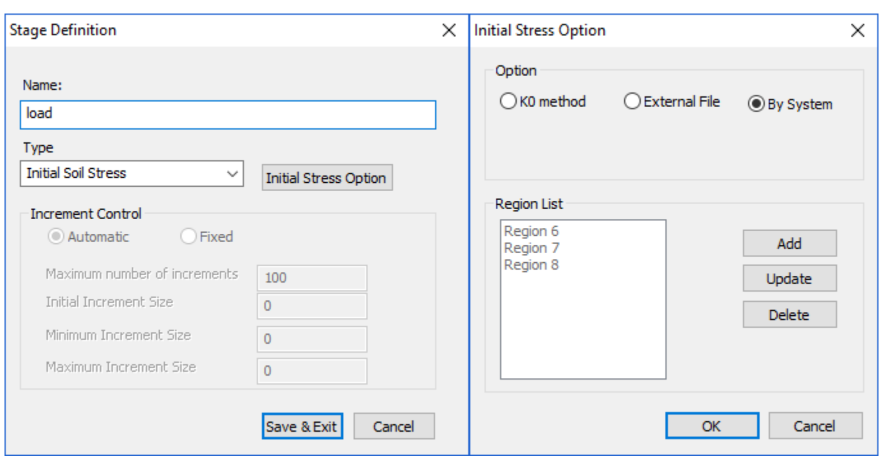


Figure 2.5-10: Define initial stress stage and options

The self-weight/gravity load is applied to the soil in the first stage, followed by a stability analysis stage.

Create the second stage, "`Stability Analysis`" as seen in Figure 2.5-11. The stage type is set to "**Stability (Strength Reduction Method)**". Click the button "**Define SRM**" to define start and end safety factor as `1.1` and `1.6` respectively.

Since a large amount of soil will become yielding, a small increment size is recommended to control development of yielding zone.

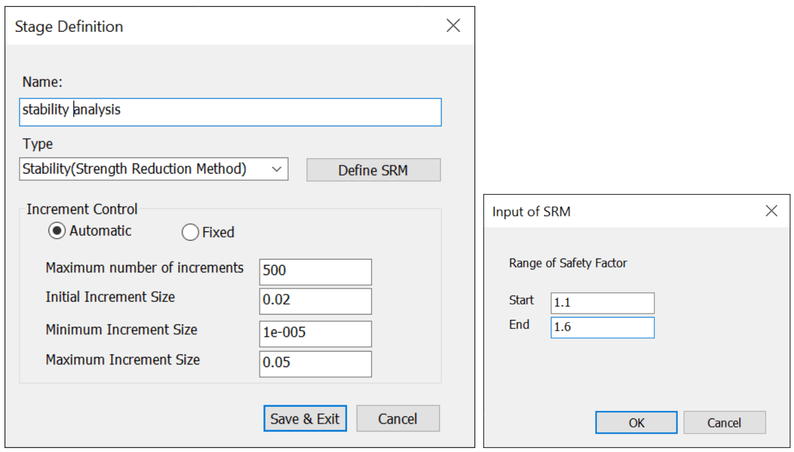


Figure 2.5-11: Create the second stage for strength reduction calculation

Step 4: Define material properties

From the menu bar, click **"Data->Material Properties"** and define three materials as shown in Figure 2.5-12 through Figure 2.5-14.

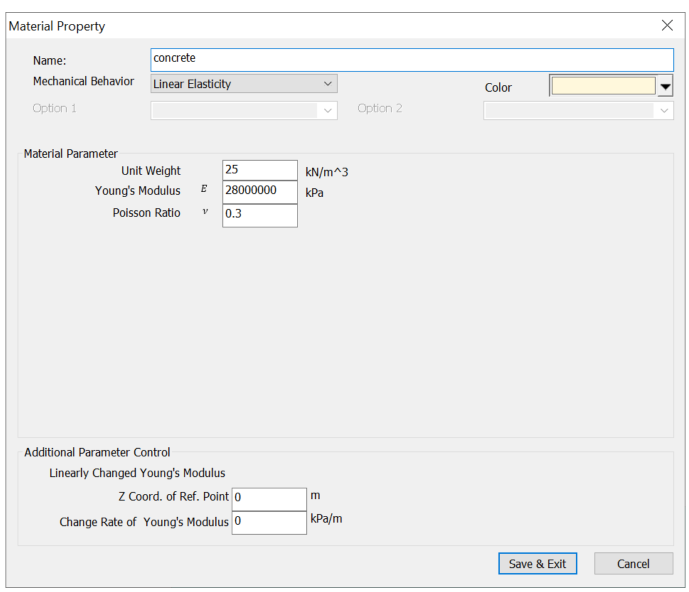


Figure 2.5-12: Linear elastic material for concrete of pile-wall

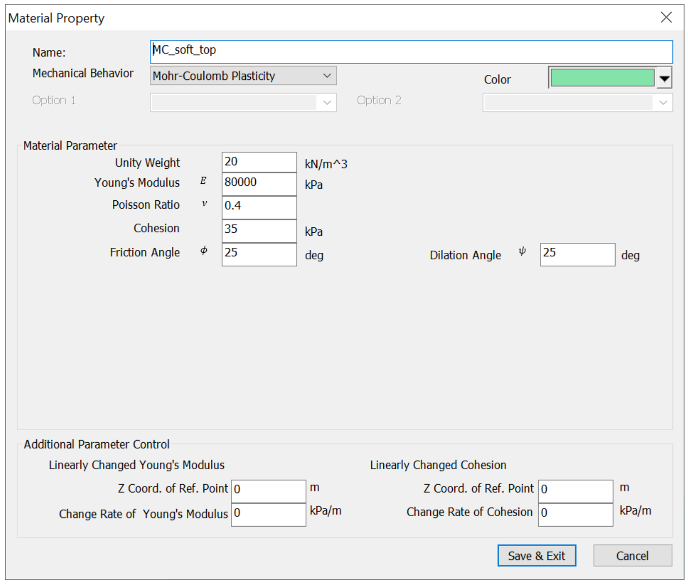


Figure 2.5-13: Mohr-Coulomb model and SRM parameters for top soil layer

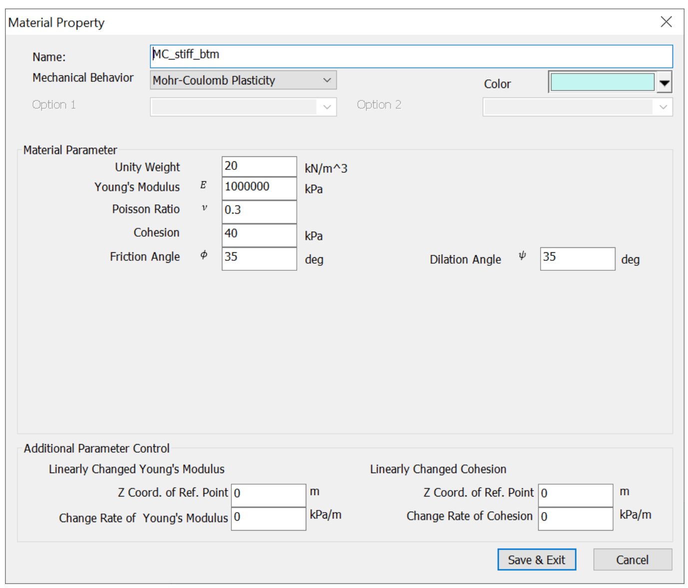


Figure 2.5-14: Mohr-Coulomb plasticity for bottom soil layer

Next, click **"Data->Material Assignment"** from the menu bar and assign the "`concrete`" material to the pile-wall regions, the "`MC_soft_top`" material to the top soil layer (2-regions), and the "`MC_stiff_btm`" material to the bottom soil layer (1-region).

Step 5: Define boundary conditions

From the menu bar, click **"Data->Boundary Conditions"** and define three new B.C.'s as shown in Figure 2.5-15.

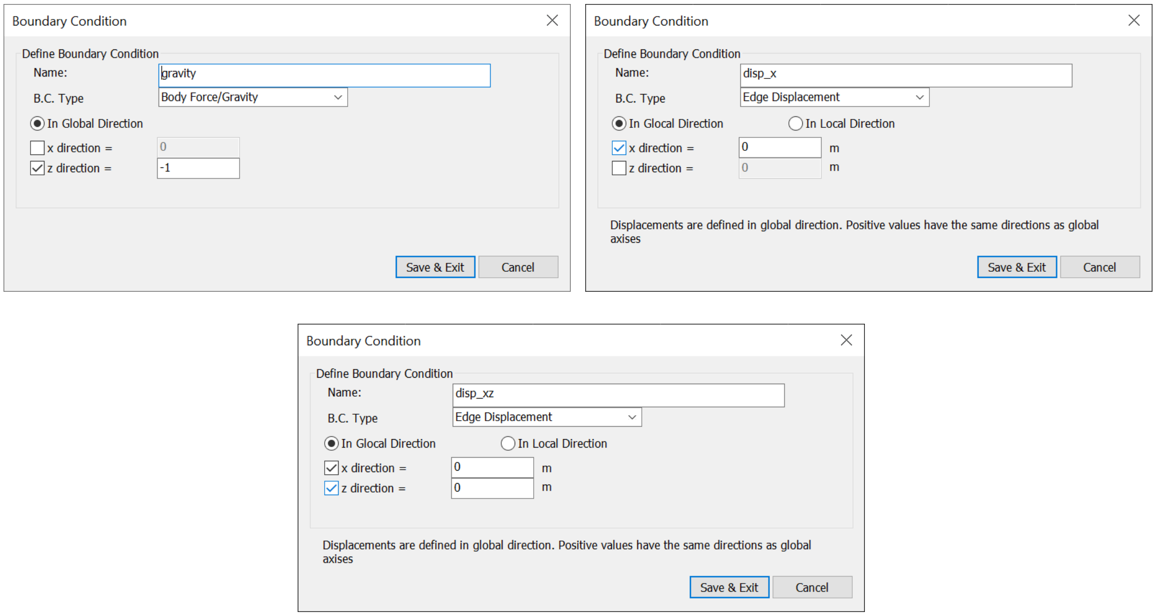


Figure 2.5-15: Define boundary condition of gravity load

From the menu bar, click **"Data->Boundary Assignment"** to assign BC's to geometric entities. All boundaries are shown as in Figure 2.5-16

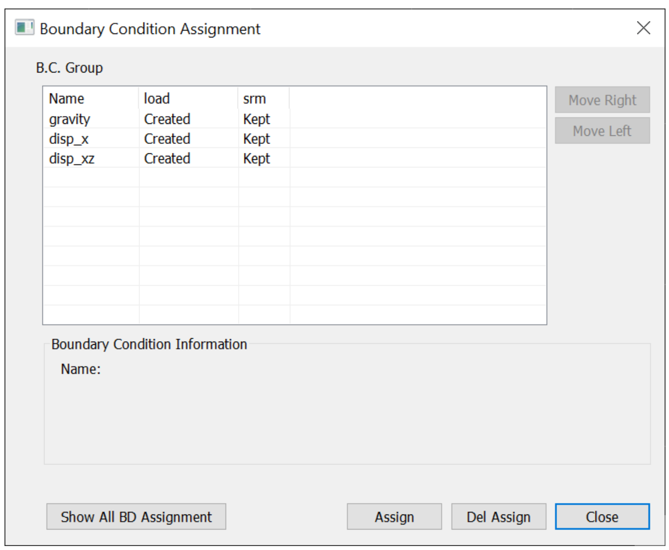


Figure 2.5-16: Assign boundary conditions in example 5

Assign "gravity" to all pile-wall and soil regions. Assign "disp_x" to the vertical soil edges. Assign "disp_xz" to soil bottom edge. Click "**Show All BD Assignment**". The assigned BC should look like those in Figure 2.5-17.

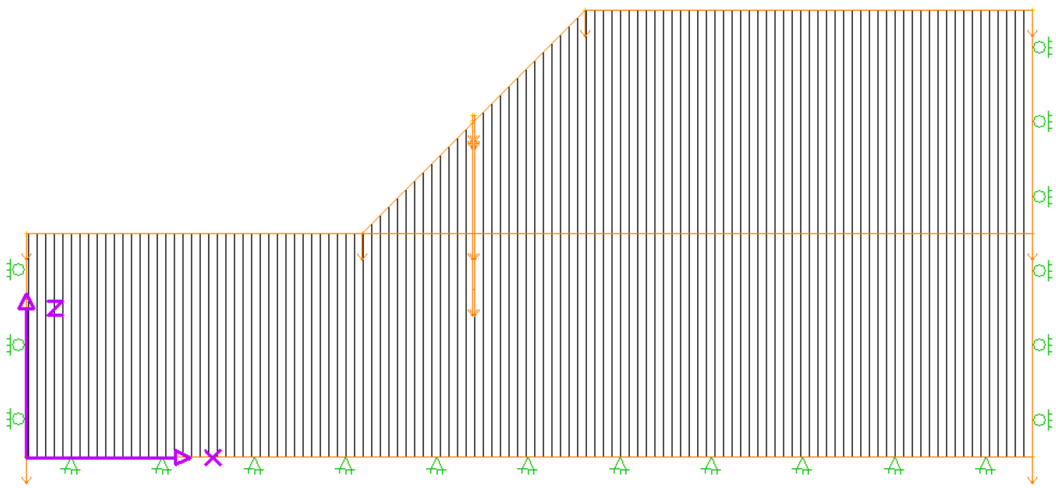
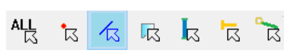
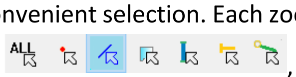


Figure 2.5-17: Assigned boundary conditions

Click  to save the file.

Step 6: Define soil-wall interaction property

From the menu bar, click **"Tools->Display Tool"**. Select "setWall" from set list and click **"Display"** to hide all regions except for the pile-wall. Next, click **"Data->Interaction"**, from the menu bar, to open the dialog to define an interaction surface. Click **"New"** and keep the default name `"Contact Group 1"`. Change the coefficient of friction to `"0.3"`. Click **"Add Master Surf."** and make sure the edge-selection icon, , is active. Next, select the six vertical edges, and the two bottom arcs, of the pile-wall. RIGHT-CLICK and select **"End Selection"**. Do not select the top horizontal edge of the pile-wall. The selected pile-wall edges will change from red, to blue in color to signify the master surface assignment (Figure 2.5-18).

<u>NOTE:</u> During this operation, users may need to zoom-in/out frequently to enlarge detail of the pile-wall for convenient selection. Each zoom-in/out command will turn off selection icons. Thus, the edge-selection icon, , may need to be re-activated after zoom-in/out operation.

Next, go back to the *'display tool'* dialog (**"Tools->Display Tool"**), and display the soil only. Click **"Add Slave Surf."**, and select the four vertical soil edges, and the two arcs around the pile-wall tip (activate the edge-selection icon, , if necessary). RIGHT-CLICK and select **"End Selection"**. After being assigned, slave surfaces will be marked in green color, as seen in Figure 2.5-19.

Click **"Save & Exit"** and then **"Close"**, when complete.

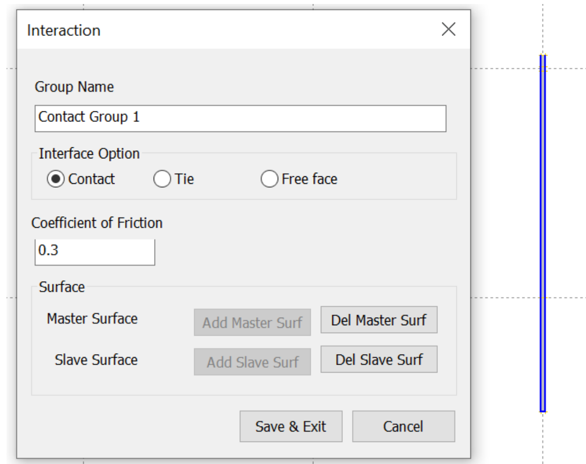


Figure 2.5-18: Define interaction property and master surfaces

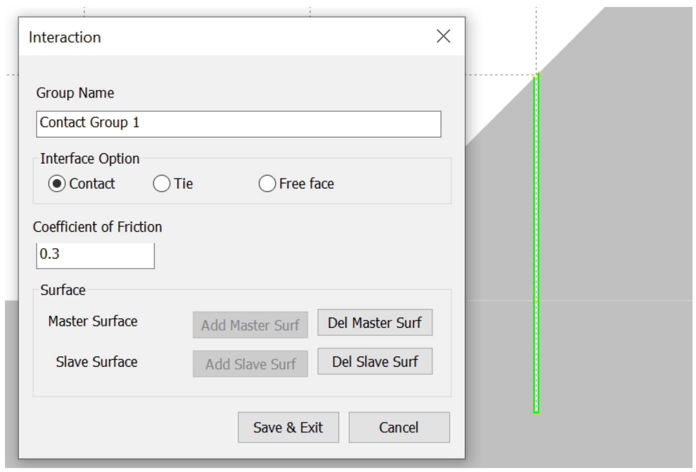


Figure 2.5-19: Define slave surfaces on soil layers

Click to save the file.

Step 7: Mesh generation

Since the pile-wall shape is slim and flexural deformation is dominant, following mesh properties are recommended:

- To reduce the influence of any shear-locking with flexural deformation, define four element layers for the cross-section of the pile-wall or more (local seed edge-division of 4, or greater) in the vertical direction. Use full integration rule because reduced integration rule does not work well in this example.
- Define a quadrilateral element with a height-width ratio of 5, or less.
    - This results in a larger number of finite elements on the pile-wall to maintain a more acceptable element aspect-ratio, without drastically increasing computation time.

- In general, computation time increases more, with an increasing number of elements.

- Additionally, soil around the pile-wall tip may suffer from local stress concentrations. Distorted finite elements (from insufficient refinement) can easily cause local material failure before overall sliding of the slope is fully developed. Therefore, it is difficult to balance the number of elements and accuracy of analysis in this example.

NOTE: It is good practice to perform some comparative studies to determine the effects of the mesh and determine optimum refinement.

Based on the previous references:

- The number of divisions for the horizontal edges of the pile-wall (cross-section) is set to 4.
- If the height-width ratio is set between 3-5, the height of the finite element for the pile-wall will be approximately 0.25-meters.
- External edges of soil do not need as much refinement as the mesh surrounding the pile-wall, so defining an element size=3m on the external soil boundaries will be appropriate.

EnFEM provides "Linear Q4" (quadrilateral) and "Linear T3" (triangular) elements for contact/interaction simulation. In this case, the "Q4" element is the recommended choice because it provides better accuracy than the "T3" elements using lesser number of finite elements. The pile-wall regions are rectangular and assigned structured meshes. Unstructured meshes will be applied to all soil regions.

Click  to display all regions of soil layers and the pile-wall. From the menu bar, click **"pre-Mesh->Advanced Mesh Control"**, and select all pile-wall regions except for the half-circle of the pile-wall tip. RIGHT-CLICK and select **"End Selection"** from the popup menu to open the global mesh control dialog. Keep the default **"**Quadrilateral**"** for the element shape, change the mesh type to **"**Structured**"**, and select **"**Linear Q4**"** with "full integration" as the element option (Figure 2.5-20). Click **"Save & Exit"**.

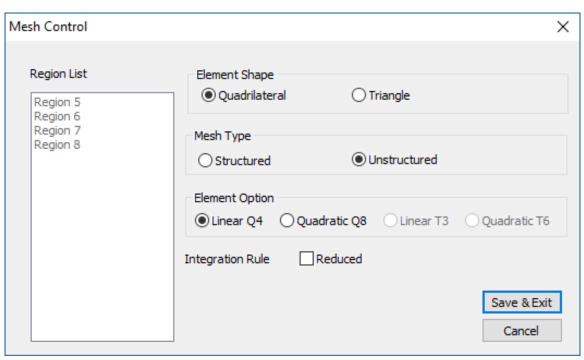


Figure 2.5-20: Set up global mesh control for rectangle regions of wall

Again, click **"pre-Mesh->Advanced Mesh Control"** and select all three soil regions and the half-circle of the pile-wall tip. RIGHT-CLICK and select **"End Selection"**. In the mesh control dialog, keep the default "Quadrilateral" and "Unstructured", and change the element option to "Linear Q4" with "full integration" (Figure 2.5-21). Click **"Save & Exit"** to close the dialog.

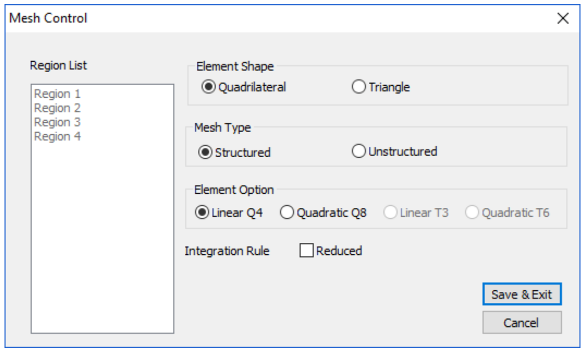


Figure 2.5-21: Set up global mesh control for soil layers and the pile-wall tip

The global mesh size should be defined as the size needed for the majority of the edges in this model. In this case, the size along pile-wall-soil interaction is selected as the control parameter for the global mesh size. From the menu bar, click **"pre-Mesh->Global Seed Control"** to open the dialog for global mesh size control. Change the default mesh size to "*0.2*", and click **"Assign"**. Click **"Close"** to exit from the dialog.

From the menu bar, click **"pre-Mesh->Local Seed Control"** and select all five, horizontal edges of the pile-wall regions. RIGHT-CLICK and select **"End Selection"** to open the control dialog. Keep the control option as "By Number" and change the "Number of Divisions" to "*4*" (Figure 2.5-22). Click **"Assign"**, and then **"Close"** to exit.

NOTE:

- In EnFEM, biased seed distribution is available only if the selection set includes only a single line or arc. If the selection contains multiple lines/arcs, only a uniform distribution of seeds is available. For this example, a uniform and matched mesh is highly recommended on the contact interface.
- After click "**Close**" to exit from the local seed control dialog, EnFEM is still in 'local seed control' mode. Users can select more edges, RIGHT-CLICK and select "**End Selection**". The local seed control dialog will pop up the again. Otherwise, RIGHT-CLICK and select "**Cancel Command**" will exit from local seed control mode actually.
- 'Seed Information' is not displayed if multiple edges are selected (in this case both arcs from the soil and pile-wall).

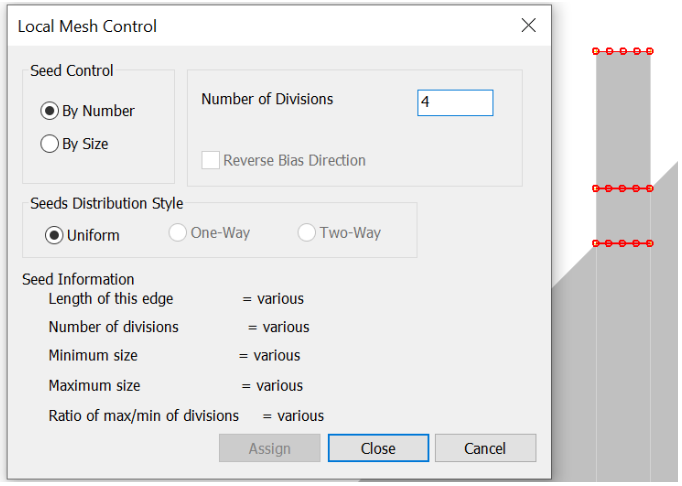


Figure 2.5-22: Assign seed divisions across pile-wall section

<u>NOTE:</u> Since the global mesh size was defined by the size for the vertical edges of the pile-wall, 0.2-m, there is no need to redefine seeds on these vertical interaction edges.

Next click **"pre-Mesh->Local Seed Control"** from the menu bar and select line segment 'C-Q'. RIGHT-CLICK and select **"End Selection"** to open the control dialog. Change the seed control to **"By Size"** and Select **"One Way"** as the distribution style. If the red arrow doesn't point in the direction of the pile-wall, check the option for "Reverse Bias Direction". Input minimize size as "`0.2`" and maximum size as "`1`", and click **"Assign"** then **"Close"** to exit. Since the ratio of the maximum/minimum is approximately, final minimum and maximum sizes may differ from the user's exact input (Figure 2.5-23).

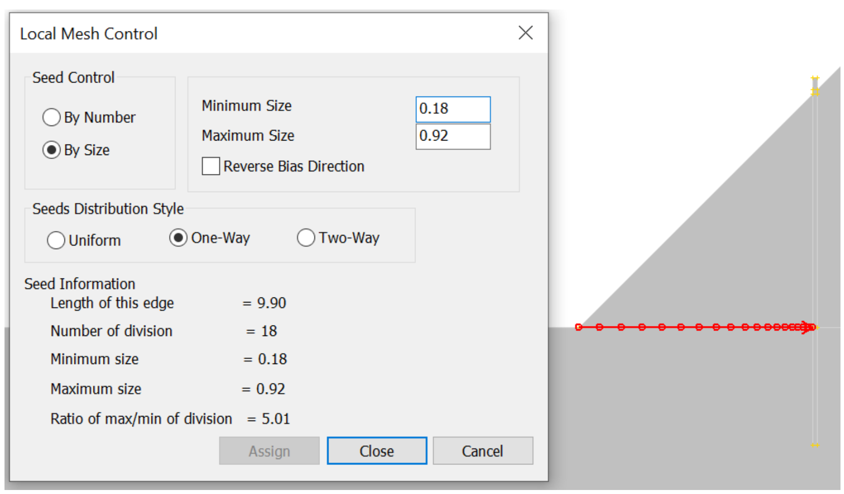


Figure 2.5-23: Apply biased seed distribution to obtain smooth transition

Similarly, select line 'R-J' and open the local seed control dialog. Change the seed control to "By Size" and select option "One Way". Input the minimum size as "`0.2`" and the maximum size as "`3`" (Figure 2.5-24). Click button **"Assign"** than click **"Close"** to exit.

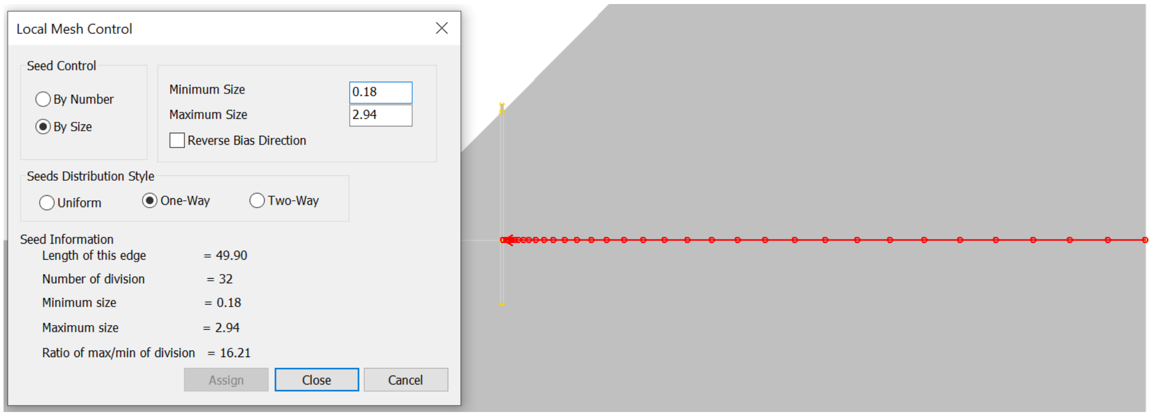


Figure 2.5-24: Local mesh control on common edge between soil layers

<u>NOTE:</u> Users may want to use the keyboard shortcut **"CTRL+L"**, to call the local seed control command for repeated definitions.

Use the following local seed controls for the specified lines:

1. Line C-D: "By Size", " One Way ". Min. size= "*0.2*", and max. size=1.0. Refine towards the wall top.
2. Line G-H: "By Size", "One Way" towards Vertex "*G*". Min. size= "*0.2*". Max. size= "*1*".
3. Line C-B: "By Size", "One Way" towards Vertex "*C*". Min. size= "*1*". Max. size= "*3*".
4. Line H-I: "By Size", "One Way" towards Vertex "*H*". Min. size= "*1*". Max.= "*3*".
5. Lines B-A, A-K, K-J, and I-J (Hold SHIFT key and select to define all at once): "By Size", "Uniform". Uniform size= "*3*".

All lines have now been assigned to appropriate local seed definitions. From the menu bar, click **"Mesh->Show Seeds"** to plot all seed definitions and check if any changes are necessary.

Click  to save the file.

Finally, click **"Mesh->Run Mesh Generation"** from the menu bar. The final mesh looks like that shown in Figure 2.5-25.

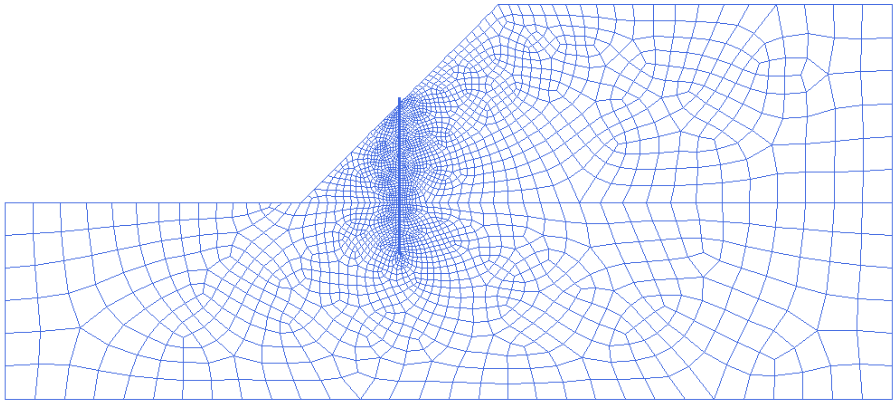


Figure 2.5-25: Quadrilateral mesh for Example 5

Click  to save the file.

Step 8: Analysis

From the menu bar, click **"Analysis->Analysis Setting"**. Change the default factors for normal and tangent penalty to *"2"*. Click **"OK"** to exit.

From the menu bar, click **"Analysis->Run Analysis"** to open the "Run Analysis" window. By default, each stage is selected to run. Click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. Note, that due to the significant mesh refinements, and the end goal of finding a slope failure, the analysis may take some more time than previous examples. The file, *"stability analysis of strengthened slope.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history and any error information, if it exists.

Step 9: Post-processing of result

Click , to switch to the result-mode and then click  to open the contour plot dialog. The default contour initially displayed is the displacement in the x-direction (u1) at the last loading increment (Figure 2.5-26). Since the analysis stopped at increment=1.9385, the safety factor can be estimated by linear interpolation following equation (2.3-1), and seen below:

$$\boldsymbol{SF} := \mathbf{1.1} + \frac{\mathbf{1.6} - \mathbf{1.1}}{\mathbf{2} - \mathbf{1}} * (\mathbf{1.73352} - \mathbf{1}) = \mathbf{1.467}$$

(2.5-1)

From the contour plot, notice that the dark blue zone indicates significant horizontal movement to the left side, a sign of slope failure. Select "Strain" from the Group drop-down, and select **"PEEQ"** from the Variables drop-down. The contour of the equivalent plastic strain is plotted, like that seen in Figure 2.5-27. The zone of plastic strain starts from slope-toe and goes through the pile-wall towards the top of the slope.

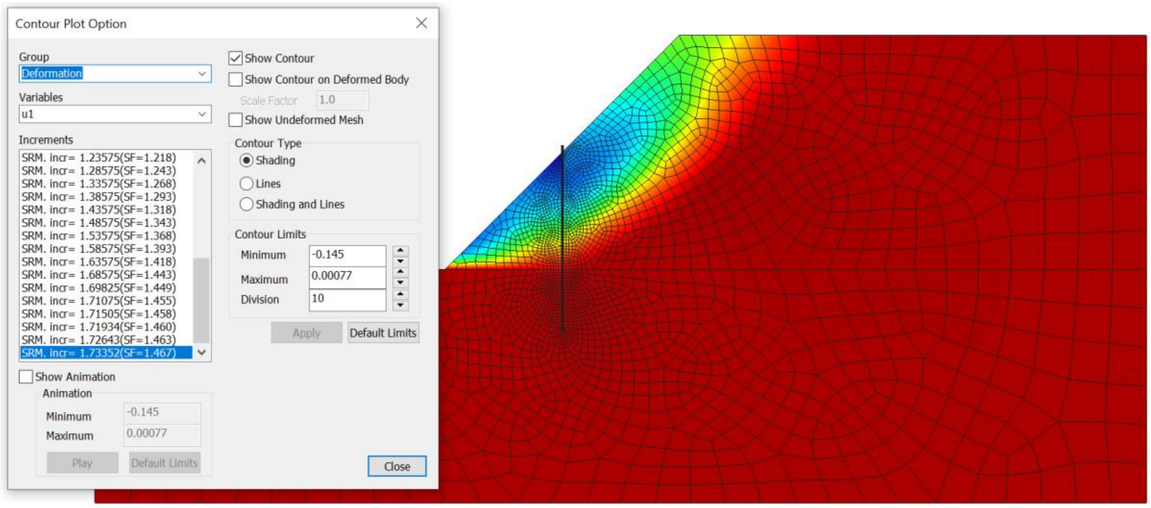


Figure 2.5-26: Contour of horizontal displacement at last increment

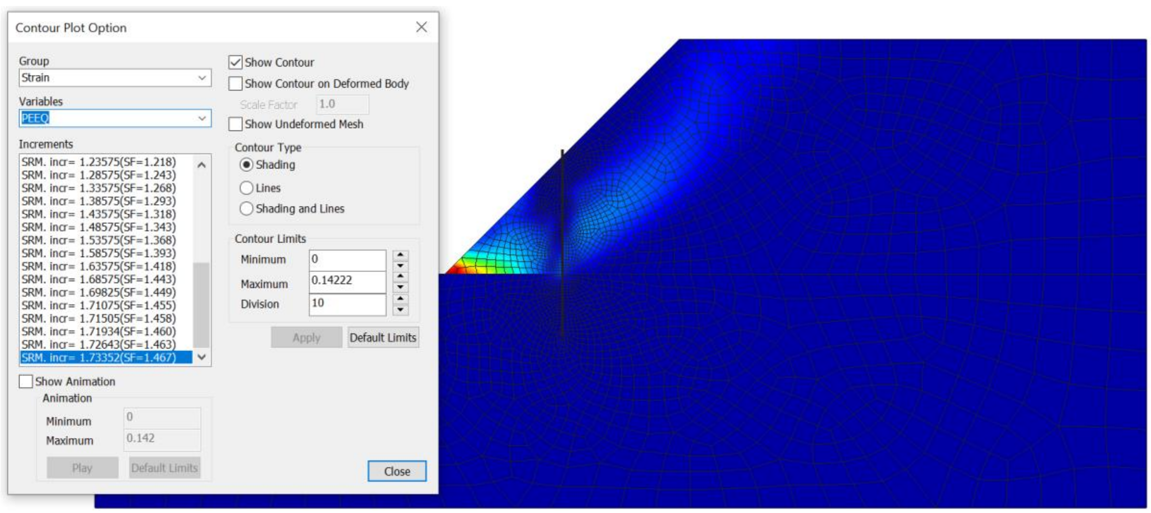
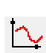


Figure 2.5-27: Contour of equivalent plastic strain at last increment

From the menu bar, click **"Mesh->Show Node Numbers"** to find the node number with the maximum lateral displacement (hide regions as necessary). This node is number *'60* (to the left of the pile-wall, where the slope intersects with the pile-wall).

Click the X-Y plot icon, . Keep source data as "Historic Result", input "60" as the node set. Select "Primary Variable" from the Category list, and select "u1" from the Variables list. Click **"Plot"** to plot the Displacement in the x-direction VS Load Increment – seen in Figure 2.5-28.

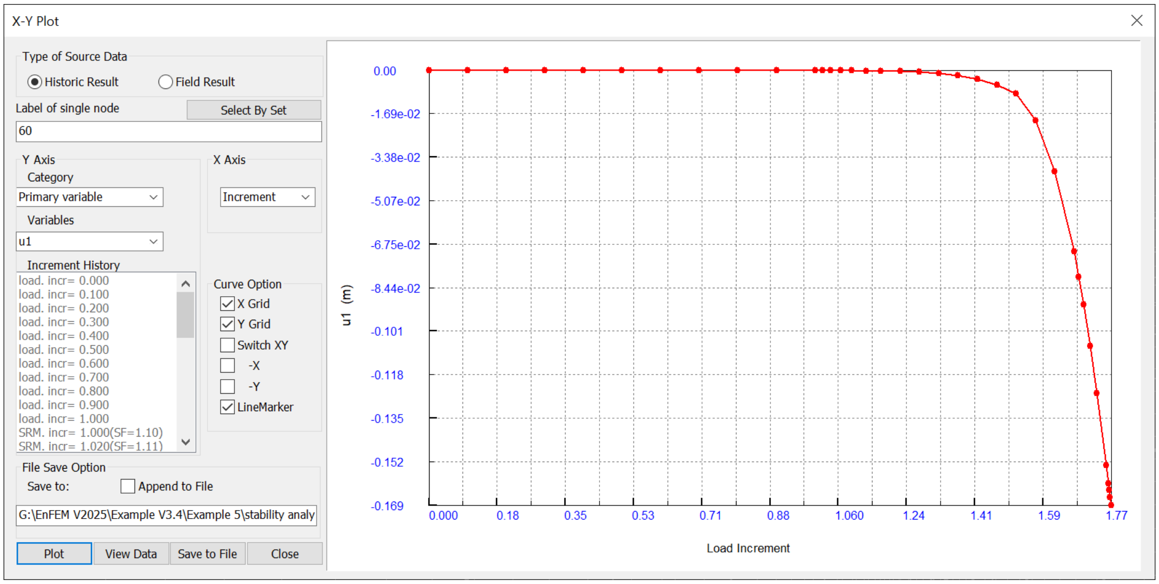


Figure 2.5-28: Horizontal displacement VS Load increment at node 2566

Turn on the nodal numbers, by clicking **"Mesh->Show Node Numbers"**. Next, click **"Tools->Display Tool"** to show the pile-wall only. Zoom into the pile-wall to find the node path numbers along the center axis of the pile-wall, from top to tip. This node path is as follows:

`2927,2928,2917,2918,2793:2832:1,2650:2670:1`

While keeping the soil regions hidden, click  to open X-Y plot dialog again. Select "Field Result" as the data source. Input the node set previously listed. Select "Integrated Force" as the category and "Moment" as the Variable. Select the last increment. Check options for **"Switch XY"**, **"-X"** and **"-Y"**. Click **"Plot"** to plot moments along pile-wall, from top to tip, as seen in Figure 2.5-29. Next select "Shear" as the Variable and click **"Plot"** to plot shear from pile-wall top to-pile-wall tip, as seen in Figure 2.5-30.

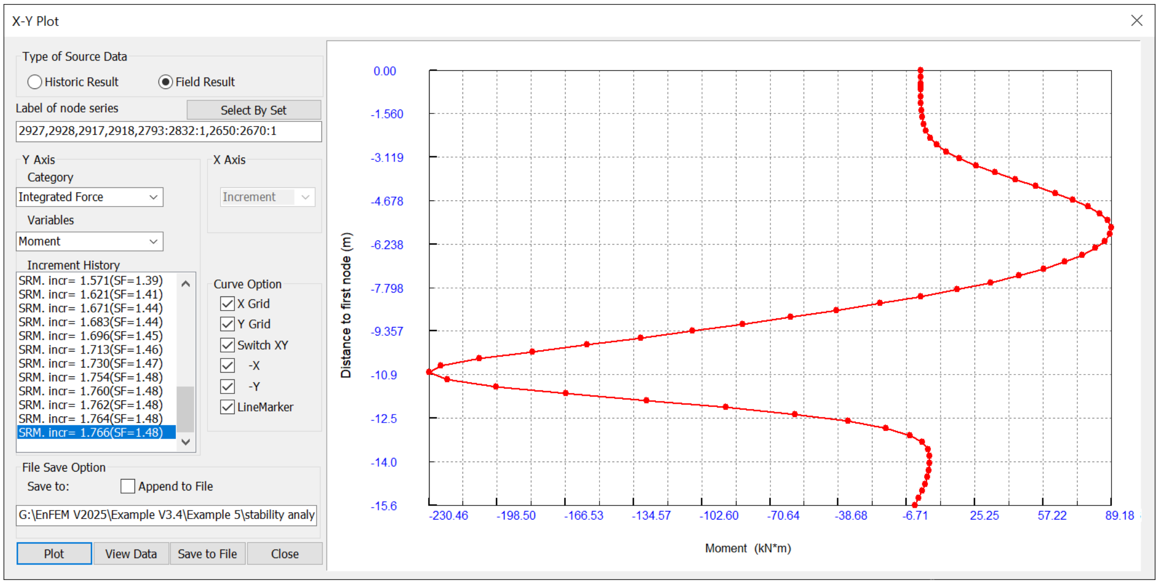


Figure 2.5-29: Moment of pile VS depth

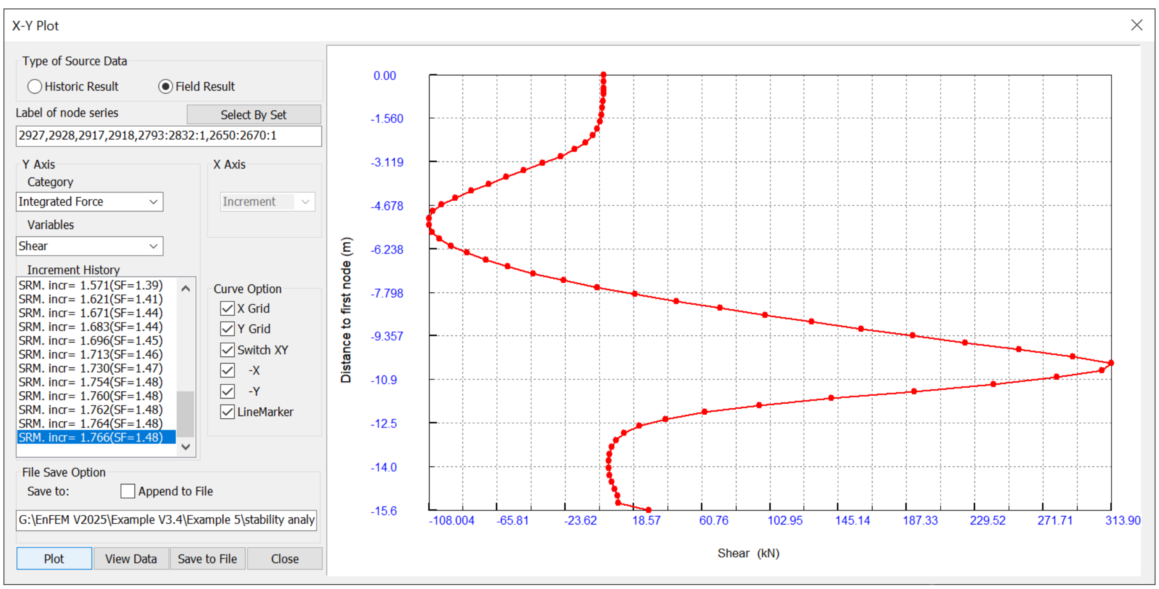


Figure 2.5-30: Shear distribution VS depth

Keeping "Field Result" as the source data type, select “By Set”. From the “Set Selection” dialog, select the previously defined edge-set, "soil_right", and click **“OK”.** Select "Interaction" from Category list. Select "Normal Pressure" from Variable list. Click the last increment and keep checked the options for "Switch XY", **"-x"** and **"-y"**. Click **"Plot"** to display the distribution of soil pressure on the right side of the soil-pile-wall interface, as seen in Figure 2.5-31.

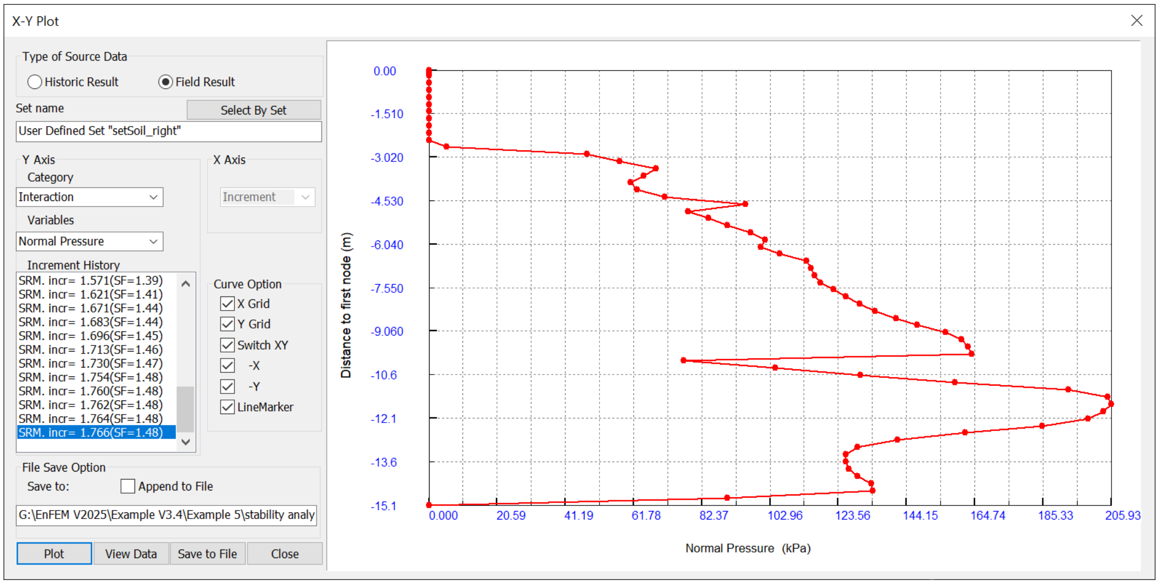


Figure 2.5-31: Distribution of contact pressure on right interface

Repeat the previous procedures, but use the "`setSoil_left`" edge-set, to plot the soil pressure on pile-wall on the left interface (Figure 2.5-32):

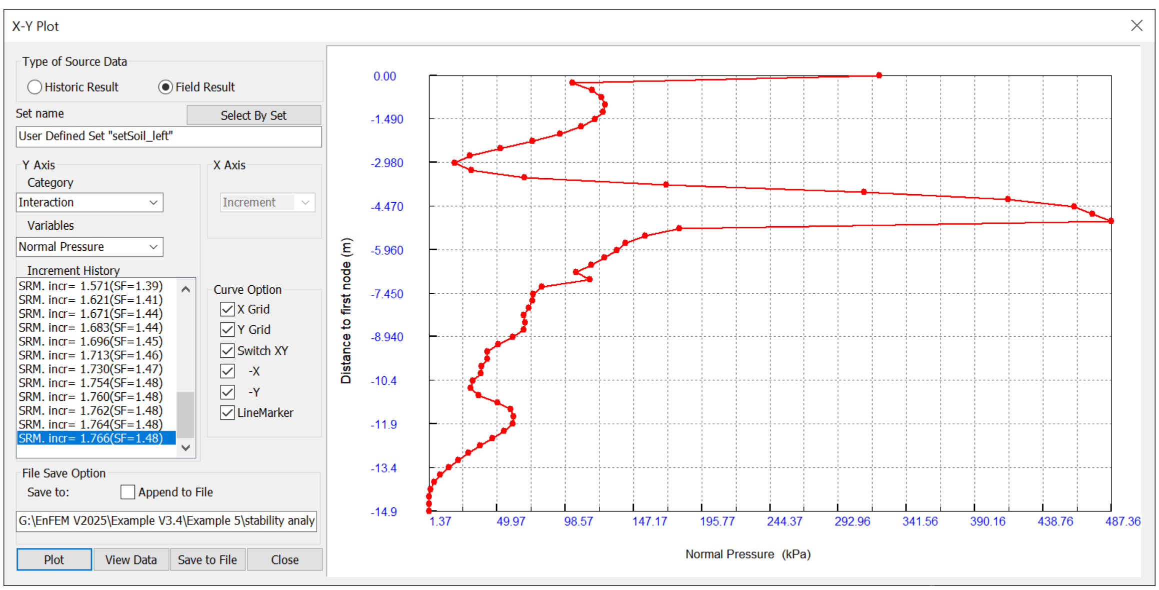


Figure 2.5-32: Distribution of normal contact pressure along the left interface

2.6 Example 6: Simulation of CD tri-axial test using Duncan-Chang hyperbolic elastic model for Loose Sand

Conventional-drained (CD), tri-axial tests of loose and dense sand are simulated in example 6 and example 7. Duncan-Chang hyperbolic elastic model is applied to soil. The physical configuration of the sand sample is shown in the left image of Figure 2.6-1. The sand sample is a 40mm x 60mm cylinder subjected to uniform pressure σ_1 and σ_3. Due to symmetry, a simplified axi-symmetric geometry can be used as shown the right image of Figure 2.6-1. Units used in this example are "N, mm, second".

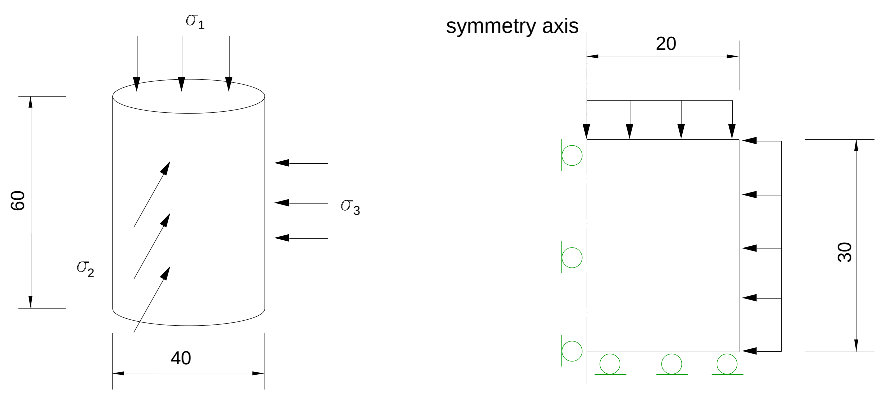


Figure 2.6-1: Simulation of CD tri-axial test using Duncan-Chang model

Material Properties:

Loose sand ($E - \upsilon$ model):

- Modulus number:.. $K = 295$
- Modulus exponent:.. $n = 0.65$
- Failure ratio:.. $R_f = 0.9$
- Cohesion:.. $c = 0\ MPa$
- Friction angle:.. $\phi = 30.4°$, $d\phi = 0°$
- Unloading modulus number:.. $K_{ur} = 1090$
- Parameters: .... $G = 0.32$, $F = 0$, $D = 0$
- Atmosphere pressure:.. $p = 0.1\ MPa$

Boundary Conditions:

- Displacement constraint on the vertical symmetric axis
- Displacement constraint on horizontal, bottom edge of the sample
- External pressure boundary conditions, as listed in Table 2.6-1.

	Stage 2	Stage 3	Stage 4	Stage 5	Stage 6	Stage 7	Stage 8
σ_1 (MPa)	0.20	0.271	-0.422	0.515	-0.074	-0.412	0.059
σ_3 (MPa)	0	0	0	-0.074	0.196	0	-0.196

Notes: (1) $\sigma_2 = \sigma_3$;

(2) initial stresses are $\sigma_1 = \sigma_2 = \sigma_3 = 0.294\ MPa$

Table 2.6-1: Stress B.C. for conventional drained tri-axial test

Model development steps:

Step 1: Create geometric entity

Start EnFEM, or Click **"File->New"** to start a new project. Change the title to *"CD tri-axial test for loose sand"*. Change the analysis type to "Axi-symmetric" (Figure 2.6-2**Error! Reference source not found.**). Select preselected unit "N, mm, second". Click **"OK"** to exit.

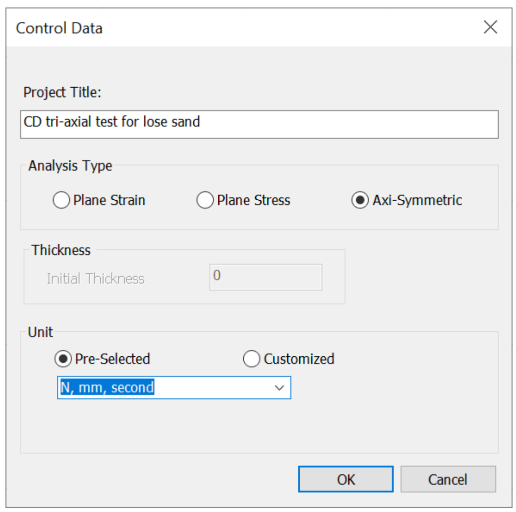
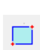


Figure 2.6-2: Define control data for Example 6

Click **"Options->Drawing space"** and setup the drawing space like that shown on the left image of Figure 2.6-3. Click **"OK"** to exit.

Next, click **"Options->Grid Spacing"** and change the default grid spacing to *"10"* in both the X and Z-directions (right image of Figure 2.6-3).

Click  to activate the rectangle creation tool. 'Snap' to the origin of the working zone (coord. x=0, z=0), and LEFT-CLICK to create the first vertex of the rectangle. Next, 'snap' to grid intersection (x=20, z=30) and LEFT-CLICK to create the second vertex for the rectangle. The created rectangle is shown in Figure 2.6-4.

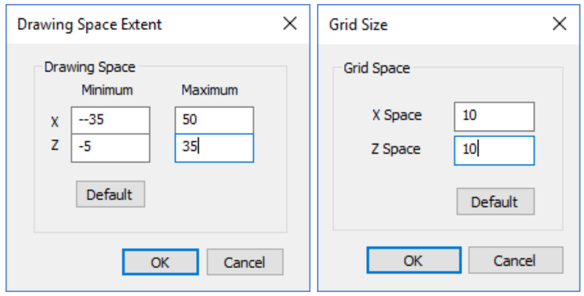


Figure 2.6-3: Modify the drawing space and grid spacing in Example 6

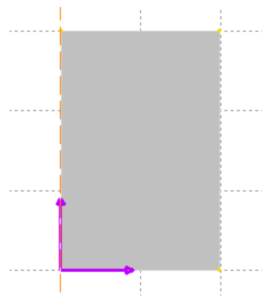


Figure 2.6-4: Created rectangular region of sand sample in Example 6

Click  to save the model file as "`Tri-axial test loose sand Duncan-Chang.sep`".

Step 2: Define stage data

From the menu bar, click **"Data->Stage Management"**, to open the stage group management dialog. Define eight stages, as seen in Figure 2.6-5.

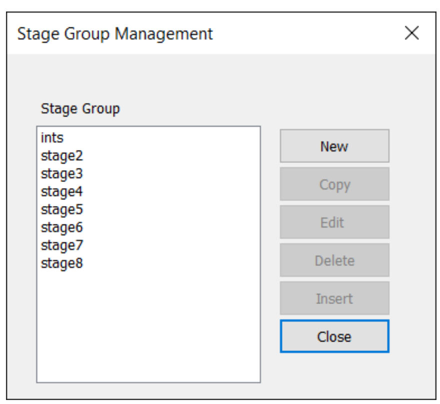


Figure 2.6-5: Eight stages defined in Example 6 to simulate CD test

The first stage, "ints" should be defined as an "Initial Soil Stress" type. The settings for the initial stress options are shown in Figure 2.6-6. The other seven stages should be defined as "General Static" type. Keep the "Automatic" option for increment control, and change the initial and maximum increment size to "`0.02`" (Figure 2.6-7) for 'stage2' through 'stage8' - <u>except for</u>

'stage6'. '*Stage6*' shall have the initial and the maximum increment size set to "`0.005`". Confining pressure will be changed in this stage and a smaller increment size will be helpful (Figure 2.6-7) for the solver at this stage.

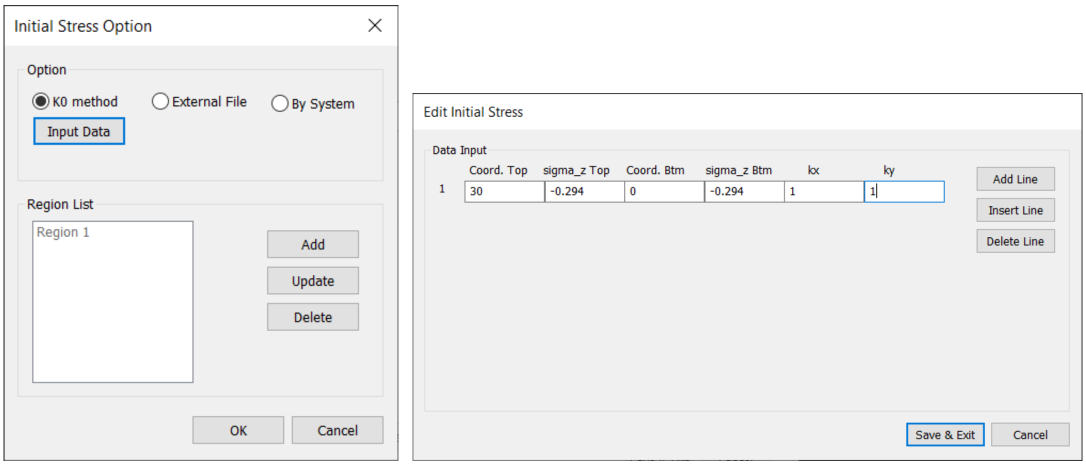


Figure 2.6-6: Setting of initial soil stress for Example 6

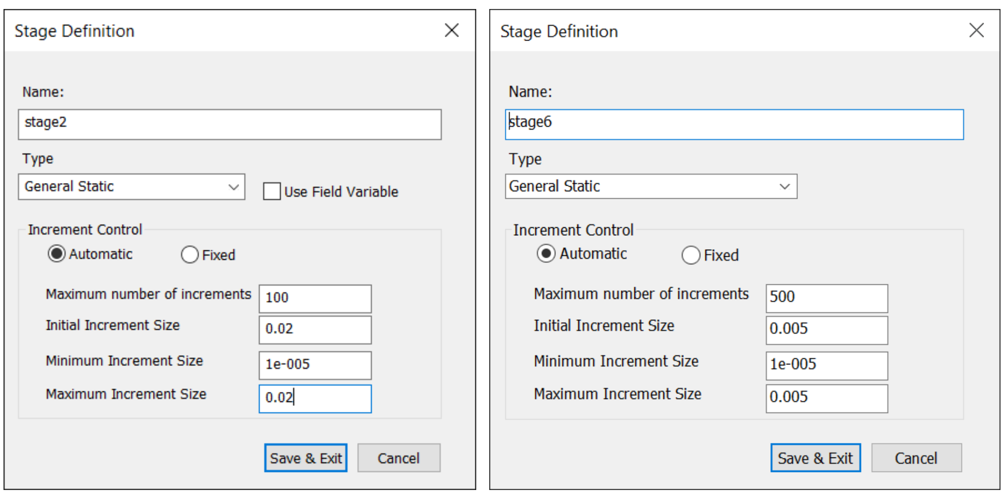


Figure 2.6-7: Define Stage2 through Stage8 in Example 6

Stage 3~8 can be generated by "**Copy**" method. Select "`stage2`", click "**Copy**". New stage has the name "*stage2_copy1*". Select the new stage and click "**Edit**" to change the stage name to "`stage3`".

Step 3: Define material properties

From the menu bar, click **"Data->Material Properties"** to open the material group management dialog. Click **"New"** to define the material seen in Figure 2.6-8. Select the "$E-\nu$ Model" for "Option 1". Click **"Save & Exit"** to close this dialog. when complete, click **"Close"** to exit.

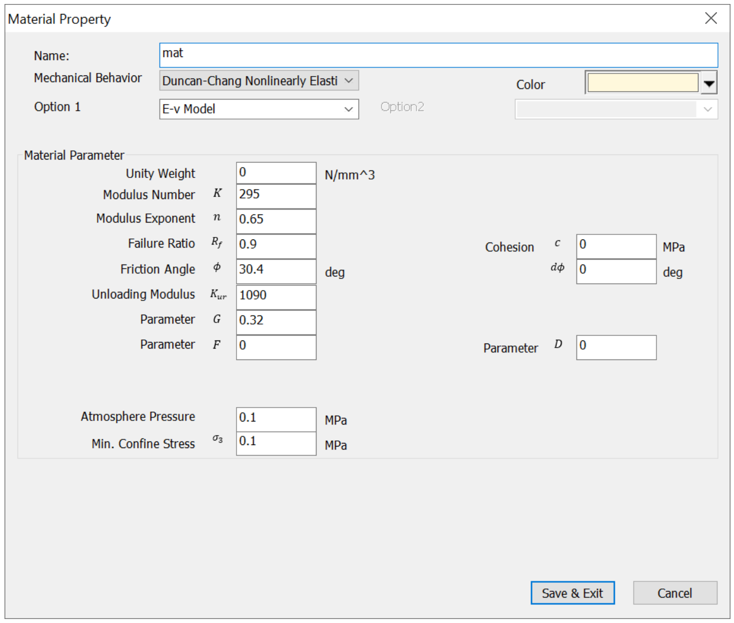


Figure 2.6-8: Define material properties for Duncan-Chang hyperbolic elasticity

Next, click **"Data->Material Assignment"** from the menu bar to open the material assignment dialog. Assign the previously defined material "`mat`" to the rectangle region in this model. Click **"Close"** to exit.

Step 4: Define boundary conditions

Click **"Data->Boundary Conditions"** from the menu bar to open the B.C. group management dialog. Define twelve boundary conditions as seen in Figure 2.6-9, and the images of Figure 2.6-10.

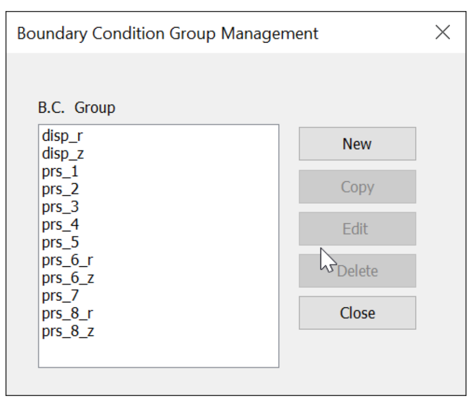
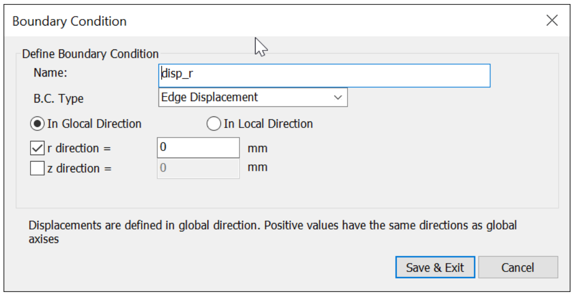
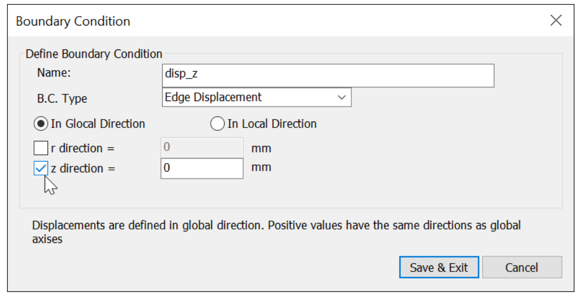
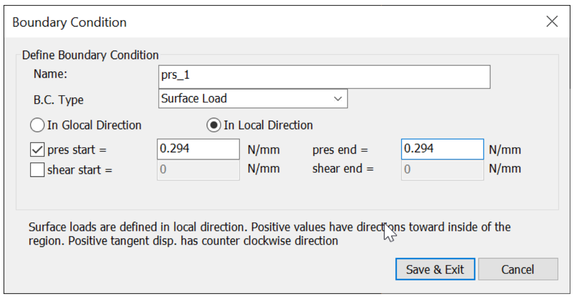
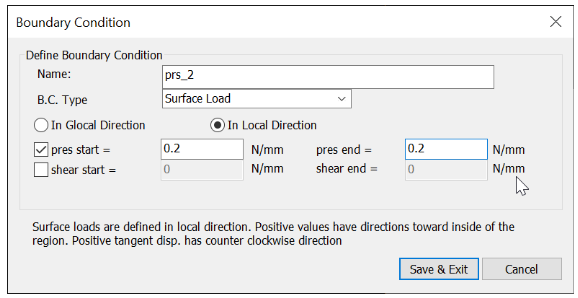


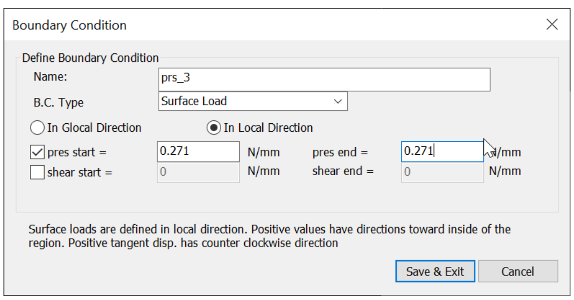
Figure 2.6-9: Define twelve B.C. groups in Example 6

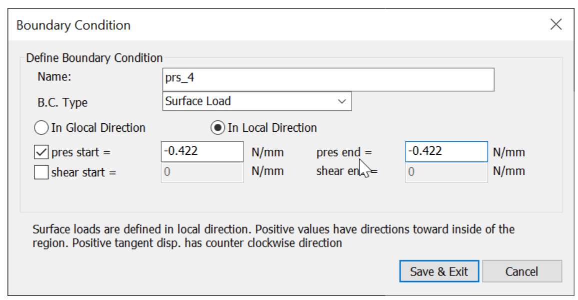

Boundary Condition
Define Boundary Condition
Name: disp_r
B.C. Type Edge Displacement
In Glocal Direction
In Local Direction
r direction = 0 mm
z direction = 0 mm
Displacements are defined in global direction. Positive values have the same directions as global axises
Save & Exit
Cancel

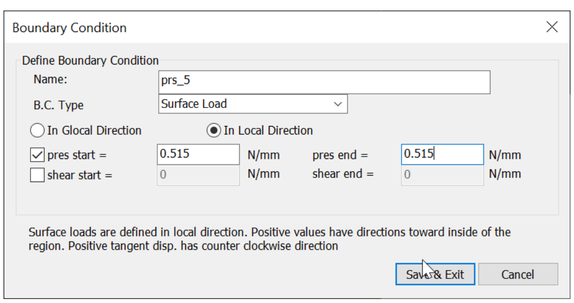

Boundary Condition
Define Boundary Condition
Name: disp_z
B.C. Type Edge Displacement
In Glocal Direction
In Local Direction
r direction = 0 mm
z direction = 0 mm
Displacements are defined in global direction. Positive values have the same directions as global axises
Save & Exit
Cancel

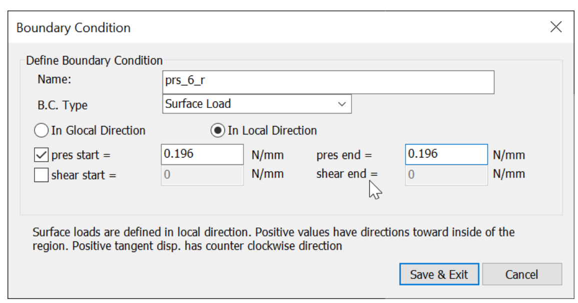

Boundary Condition
Define Boundary Condition
Name: prs_1
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = 0.294 N/mm
pres end = 0.294 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel

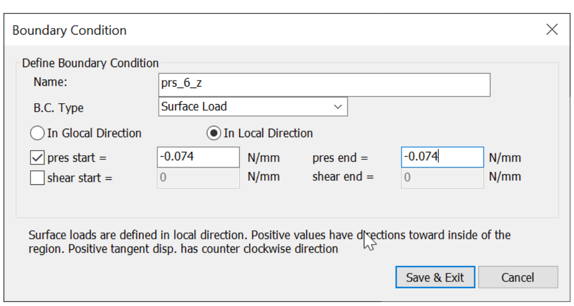

Boundary Condition
Define Boundary Condition
Name: prs_2
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = 0.2 N/mm
pres end = 0.2 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel

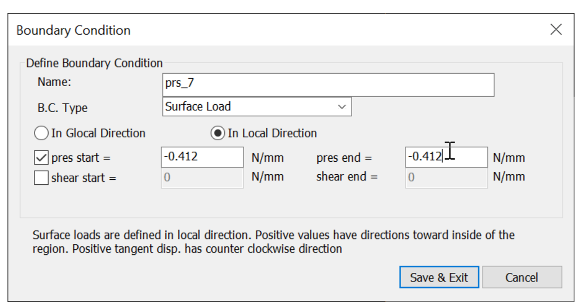

Boundary Condition
Define Boundary Condition
Name: prs_3
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = 0.271 N/mm
pres end = 0.271 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel


Boundary Condition
Define Boundary Condition
Name: prs_4
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = -0.422 N/mm
pres end = -0.422 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel


Boundary Condition
Define Boundary Condition
Name: prs_5
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = 0.515 N/mm
pres end = 0.515 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel


Boundary Condition
Define Boundary Condition
Name: prs_6_r
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = 0.196 N/mm
pres end = 0.196 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel


Boundary Condition
Define Boundary Condition
Name: prs_6_z
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = -0.074 N/mm
pres end = -0.074 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel


Boundary Condition
Define Boundary Condition
Name: prs_7
B.C. Type Surface Load
In Glocal Direction
In Local Direction
pres start = -0.412 N/mm
pres end = -0.412 N/mm
shear start = 0 N/mm
shear end = 0 N/mm
Surface loads are defined in local direction. Positive values have directions toward inside of the region. Positive tangent disp. has counter clockwise direction
Save & Exit
Cancel

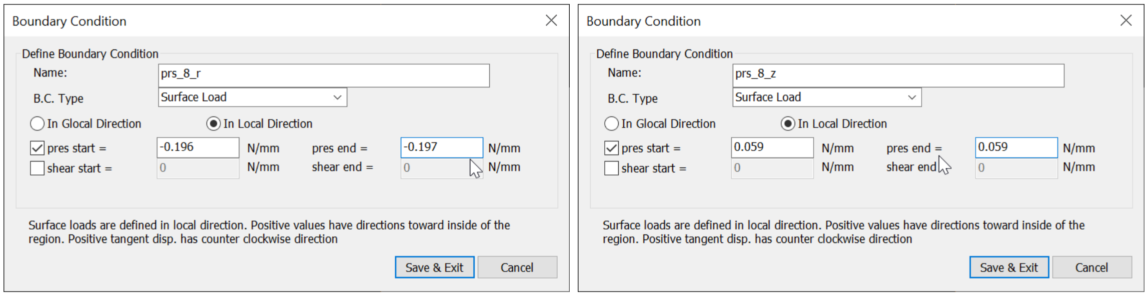


Figure 2.6-10: Boundary conditions in Example 6

These stages and boundary conditions follow the loading-unloading procedures as referenced in the example description and loading applications of Table 2.6-1.

<u>NOTE:</u> There is no gravity BC defined in this example. The initial stresses are $\sigma_1 = \sigma_2 = \sigma_3 = 0.294\ MPa$, which are obtained by applying uniform pressure on all external surfaces of the sample.

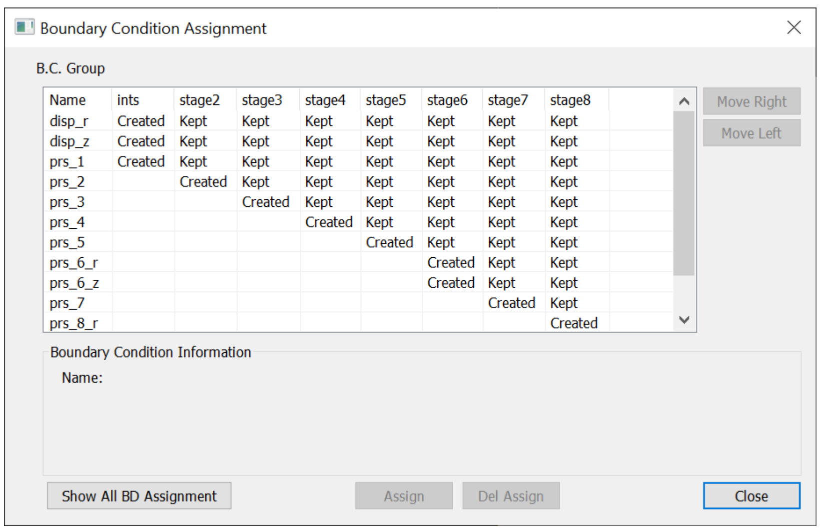
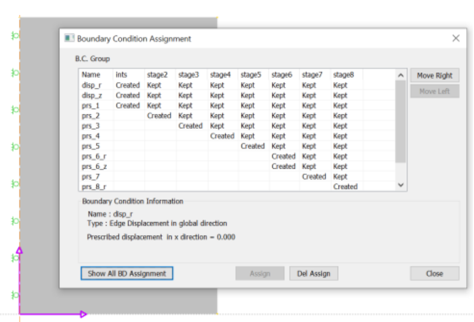
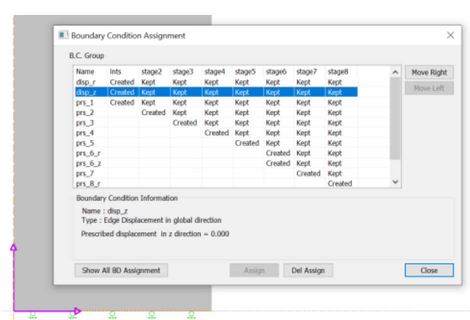


Figure 2.6-11: Assignment of boundary conditions in Example 6

From the menu bar, click **"Data->Boundary Assignment"** to assign the BC's to their corresponding geometric entities, and stage numbers (i.e. *'prs2'* in *'stage2'*. Assignments are shown in each image of Figure 2.6-11 to Figure 2.6-12:





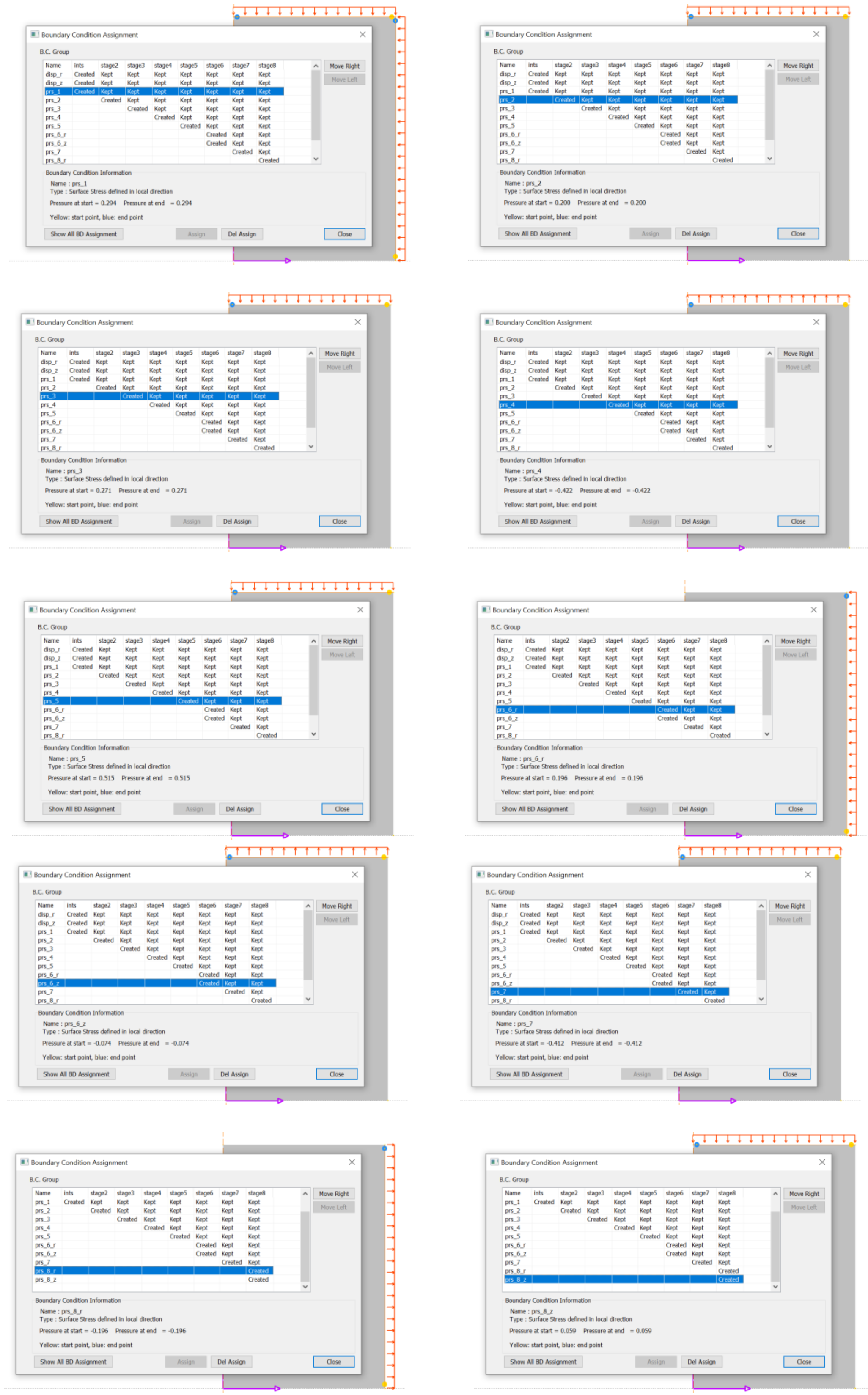


Figure 2.6-12: Assignment of boundary conditions

Click  to save the file.

Step 5: Mesh generation

Click **"pre-Mesh->Advanced Mesh Control"** from the menu bar. If the region selection icon is not activated, , click to activate it. Select the only region in this model and then RIGHT-CLICK and select **"End Selection"**. The global mesh control dialog will open up as shown in Figure 2.6-13.

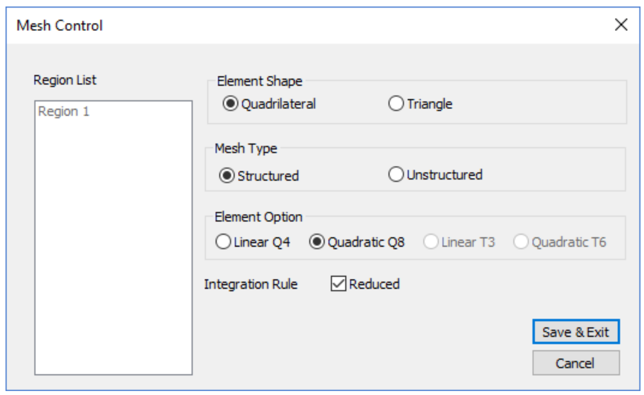


Figure 2.6-13: Global mesh control for Example 6

Select "Quadrilateral", "Structured" and "Quadratic Q8" with the “Reduced integration rule” checked. Click **"Save & Exit"** to close.

From the menu bar, click **"pre-Mesh->Global Seed Control"**. Change the default size to "5" as shown in Figure 2.6-14. Click **"Assign"** and then **“Close”** to exit.

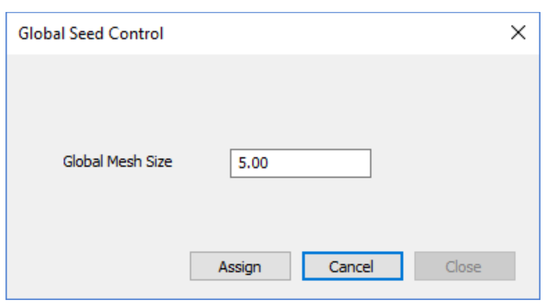


Figure 2.6-14: Global mesh size control in Example 6

For this model there is no need to apply any local seed control.

Click **"pre-Mesh->Run Mesh Generation"** from the menu bar. If the mesh is successfully generated, it will be loaded automatically.

Step 6: Analysis

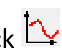
From the menu bar, click **"Analysis->Analysis Setting"** to open the dialog for analysis options. Keep the default settings for this example and click **"OK"** to exit.

From the menu bar, click **“Analysis->Run Analysis”** to open the “Run Analysis” window. By default, each stage is selected to run. Click **“Run”** to submit the analysis job to the solver.

NOTE: A warning message may appear, since an initial stress stage is defined in which no gravity /body force is defined. In this example it is ok to click **“OK”** to continue.

Next, a command window will open to show information related with each analysis increment. The file, *"DC_Loose_Sand.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history and any error information, if it exists.

Step 7: Post-processing of result

Click  to switch to the result mode. From the menu bar, click **"Mesh->Show Node Numbers"**. There are seven ‘corner-nodes’ on the symmetric axis numbered *‘1‘*, through *‘7.’* Click  to open the X-Y plot dialog. Keep the type of source data as "Historic Result", and input one of the ‘corner-node’ numbers on the symmetric axis as the node set – in this case the top node, node “7″. Select "Primary variable" from the Category list and select "E22" from the Variables drop-down list. Click **"Plot"** to plot the vertical Elastic Strain VS Load Increments (Figure 2.6-15). Click **"Save to File"** to export the X-Y data for this curve to a file with user-specified path on the hard drive.

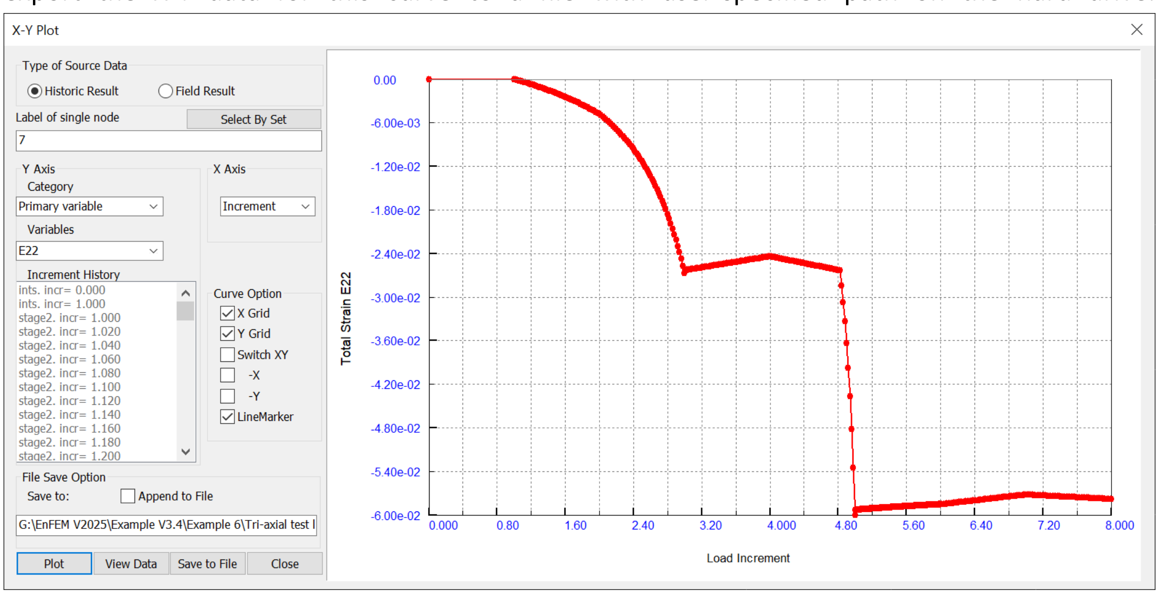


Figure 2.6-15: X-Y plot of E22 VS Load Increments in Example 6

Change variable to "Equivalent Shear q" and click **"Plot"** again, as shown in Figure 2.6-16. Now plot the curve E22 vs q. Keep the variable for Y axis to be equivalent shear q and select “E22” from the X axis. Click “Plot” to show the relationship between vertical strain E22 and shear force q as shown in **Error! Reference source not found.**Figure 2.6-17.

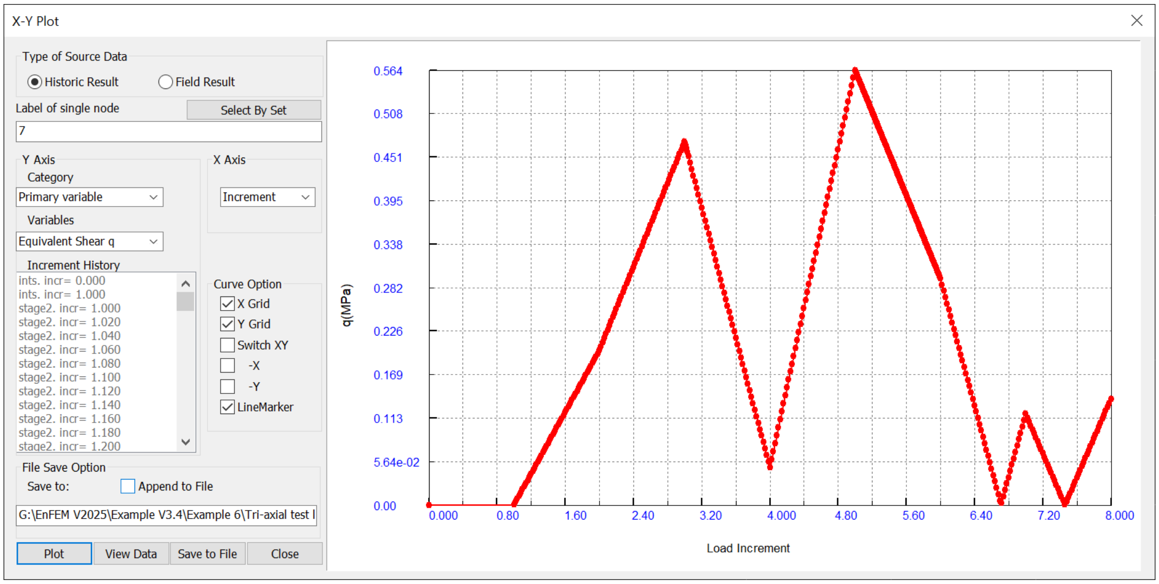


Figure 2.6-16: X-Y curve plot of equivalent shear q VS increments in Example 6

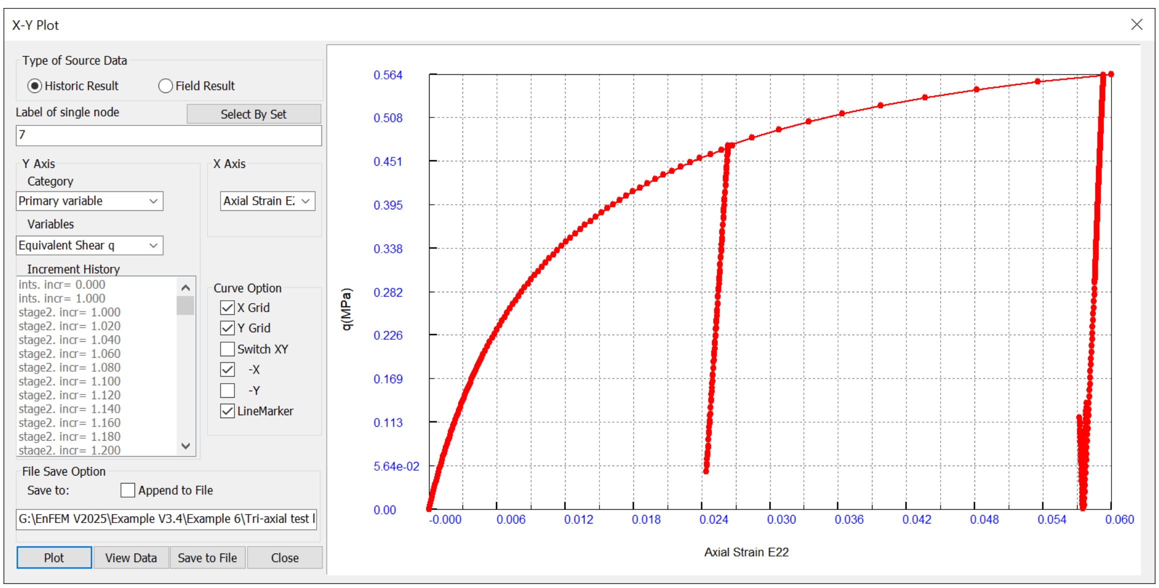


Figure 2.6-17: Combined plot of Shear Stress, q VS E22 for loose sand in Example 6

2.7 Example 7: Simulation of CD tri-axial test using Duncan-Chang hyperbolic elastic model for dense sand

Example 7 uses the same geometry configuration and meshing controls as Example 6 described above. Detailed below are the changes necessary for *'Tri-axial test Dense Sand Duncan-Chang'*. With the loose-sand model opened, click **"File->Save As"**, and name the new file "`Triaxial test dense sand Duncan-Chang.sep`".

Material Properties

Dense sand ($E - \upsilon$ model):

- Modulus number: .... $K = 2000$
- Modulus exponent: .... $n = 0.54$
- Failure ratio: .... $R_f = 0.91$
- Cohesion: .... $c = 0\ MPa$
- Friction angle: .... $\phi = 36.5°$, $d\phi = 0°$
- Unloading modulus number: .... $K_{ur} = 2120$
- Parameter: .... $G = 0.32$, $F = 0$, $D = 0$
- Atmosphere pressure: .... $p = 0.1\ MPa$

Boundary Conditions:

- Displacement constraint on the vertical symmetric axis (same as loose sand)
- Displacement constraint on horizontal, bottom edge of the sample (same as loose sand)
- External pressure boundary conditions, as listed in Table 2.7-1

prs_1	prs_2	prs_3	prs_4	prs_5	prs_6_z	prs_7	prs_8_x	prs_8_z	prs_9
0.294	0.392	-0.343	0.603	-0.603	0.628	-0.294	-0.098	0.029	-0.363

Table 2.7-1: Pressure boundary conditions for dense sand test

Further detail of boundary conditions and assignments can be referenced in the model example file, under the directory:

\ ENFEM2024-Examples\ Example 7\ Tri-axial test dense sand Duncan-Chang.sep

Once analyzed, follow a similar procedure from Example 6 for exporting the appropriate results from the XY-Plot to create the curve of Shear Stress, q VS Axial Strain, E22 for the dense sand case as shown in Figure 2.7-1.

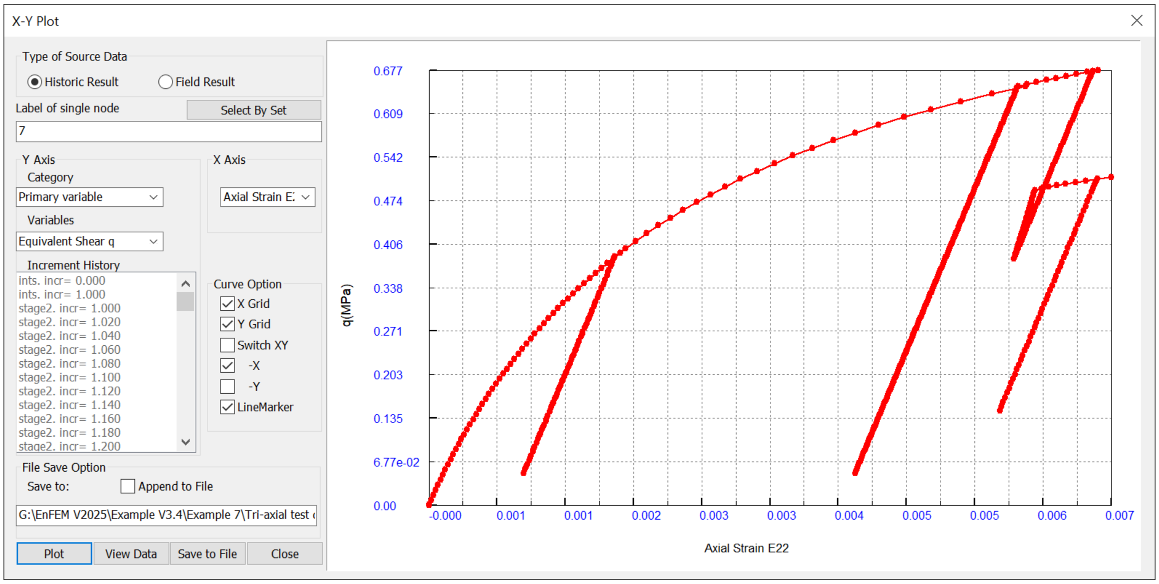


Figure 2.7-1: Curve of Equivalent Shear Stress, q VS E22

2.8 Example 8: Simulation of CD test using modified cam-clay model (OCR=1.2)

This example presents procedures to use the modified cam-clay model in simulating a conventional drained test for normally consolidated soil. The physical configuration is shown in Figure 2.8-1. Due to symmetry, a simplified axi-symmetric geometry can be used in this example (right image of Figure 2.8-1). Units used in this example are “kN, m, second”.

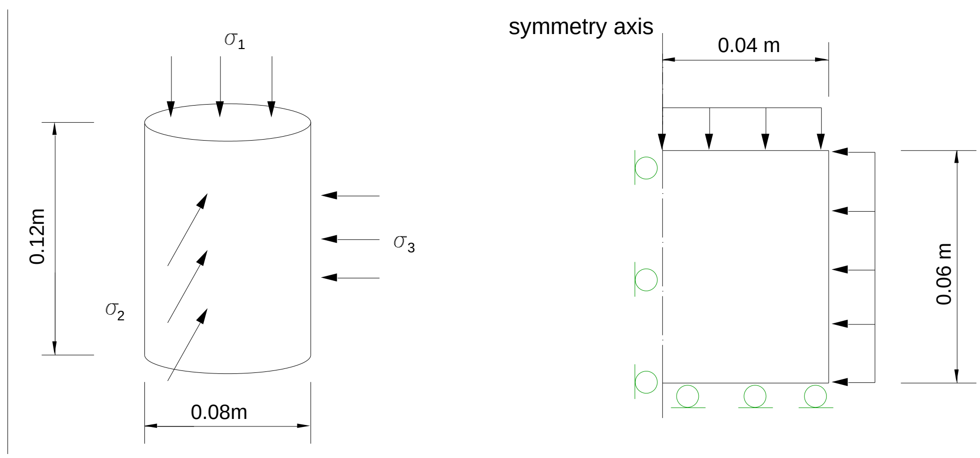


Figure 2.8-1: Configuration of conventional tri-axial test for modified cam-clay model

Material Properties:

- Slope of the normal compression line:....................... lambda $\lambda = 0.077$
- Slope of the swelling line: .... kappa $\kappa = 0.0054$
- Slope of critical line: .... $M = 1.2$
- Specific volume number:.. $N = 1.788$
- Poisson's ratio:... $\upsilon = 0.3$
- Over-consolidation ratio: .... $OCR = 1.2$

Boundary Conditions:

- Displacement constraint on the vertical symmetric axis
- Displacement constraint on horizontal, bottom edge of the sample
- Pressure for initial soil stress stage, $\sigma_1 = \sigma_2 = \sigma_3 = 333.33kPa$
  - Multiplied by OCR=1.2 gives a pre-consolidation stress = $400kPa$
- Loading pressure on top: $\sigma = 800kPa$

Model development steps:

Step 1: Create geometric entity

Start EnFEM, or click **"File->New"** to start a new project. Change the title to *"CD tri-axial test using modified Cam-Clay model"*. Change the analysis type to “Axi-symmetric" (Figure

2.8-2**Error! Reference source not found.**). Select pre-selected unit “kN, m, second”. Click **"OK"** to exit.

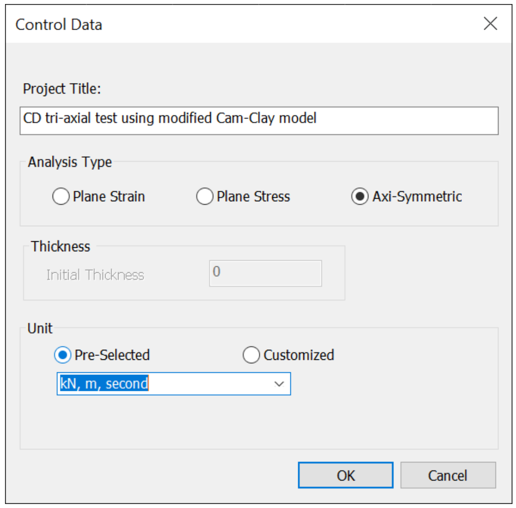


Figure 2.8-2: Control data for example 8

Click **"Options->Drawing space"** and setup the drawing space like that shown in Figure 2.8-3.

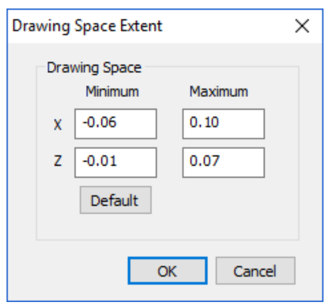


Figure 2.8-3: Drawing space for Example 8

Click  to activate the rectangle creation tool. Key in the coordinates of the first vertex of the rectangle, *"0,0"*, into the coordinate input box. Press RETURN to create the first vertex. Next, key in the relative coordinate of the second rectangular vertex, *"[0.04,0.06]"*, and press RETURN to complete the rectangle.

Click  to save the model file as "*Tri-axial test Modified Cam Clay.sep*".

Step 2: Define stage data, material properties, and boundary conditions

From the menu bar, click **"Data->Stage Management"** and create two stages. The first, named *"ints"*, should be defined as an "Initial Soil Stress" type, and the second, named *"prs2"* should be set to a "General Static" type. Set the “Initial Stress Option” to that seen in Figure 2.8-4. Keep the "Automatic Control" option for the ‘prs2’ stage definition and change the initial increment size, as well as the maximum increment size, to “`0.05`”. Click **"Save & Exit"** to close.

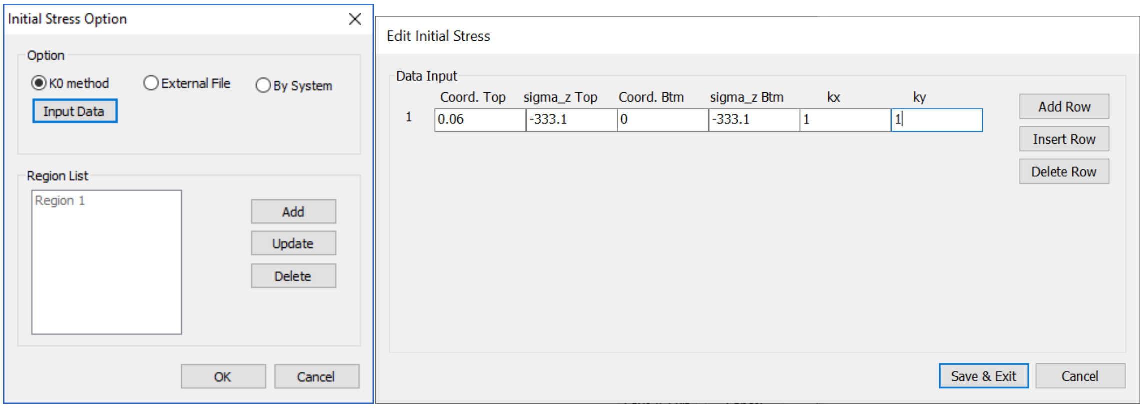


Figure 2.8-4: Define initial stress for modified cam-clay material in Example 8

Click **"Data->Material Properties"** from the menu bar and define one material, named “`mat1`”, as a "Modified Cam-Clay" type. Set the minimum preconsolidation stress to “*0*” as shown in Figure 2.8-5.

<u>NOTE:</u> The setting for initial the Young's Modulus and the initial Poisson's Ratio are shown in Figure 2.8-5, and are computed by the program (‘**By Default**’) for this case. These two parameters are used to determine the initial soil stresses and they are not related with behavior after the initial soil stress stage. If these parameters are known, the user may enter by selecting the ‘**User Defined**’ option. Otherwise, it is recommended to let the program compute this value.

Click **"Save & Exit"** to close.

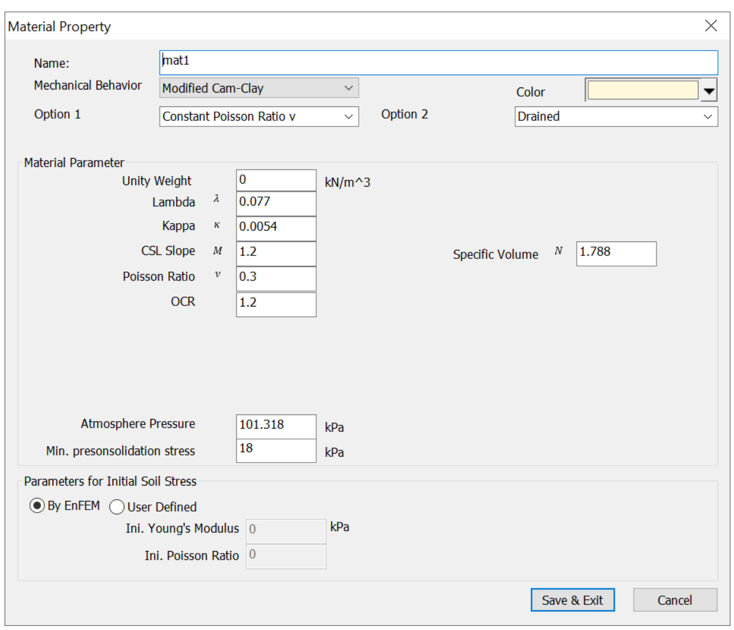


Figure 2.8-5: Define modified Cam-Clay material in Example 8

From the menu bar, click **"Data->Material Assignment"** and then assign the previously defined material to the only rectangular region in this model. Click **"Close"** to exit this dialog when complete.

Click **"Data->Boundary Conditions"** from the menu bar to open the BC group management dialog. Create four boundary conditions as seen in Figure 2.8-6.

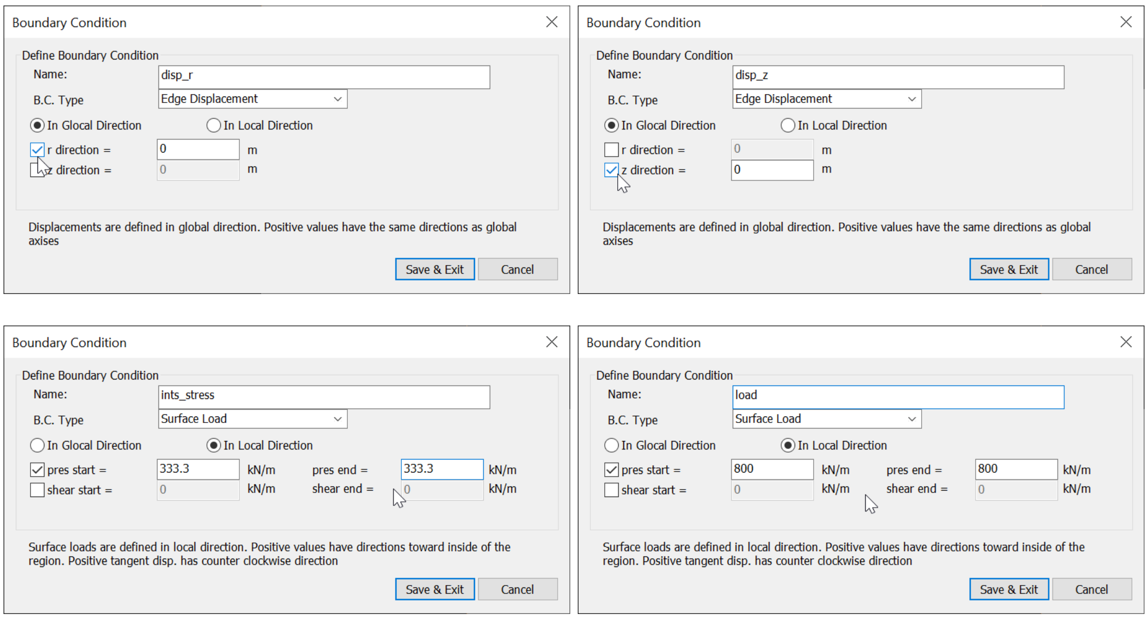


Figure 2.8-6: Define horizontal displacement constraint on symmetric axis

Next, click **"Data->Boundary Assignment"** from the menu bar, to assign these BC's to corresponding geometric entities as seen in Figure 2.8-7 through Figure 2.8-10. Note "prs2" is created at the second stage "load".

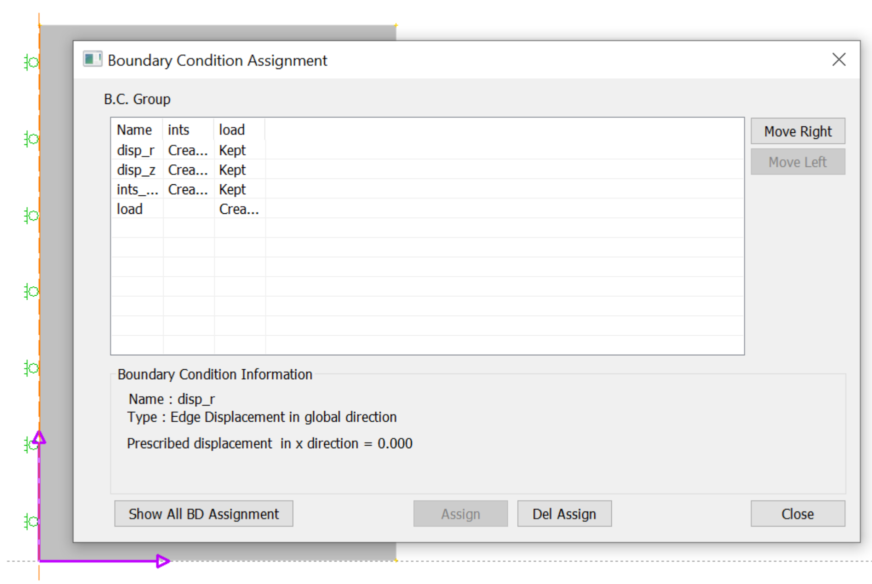


Figure 2.8-7: Apply displacement constraint to symmetric axis

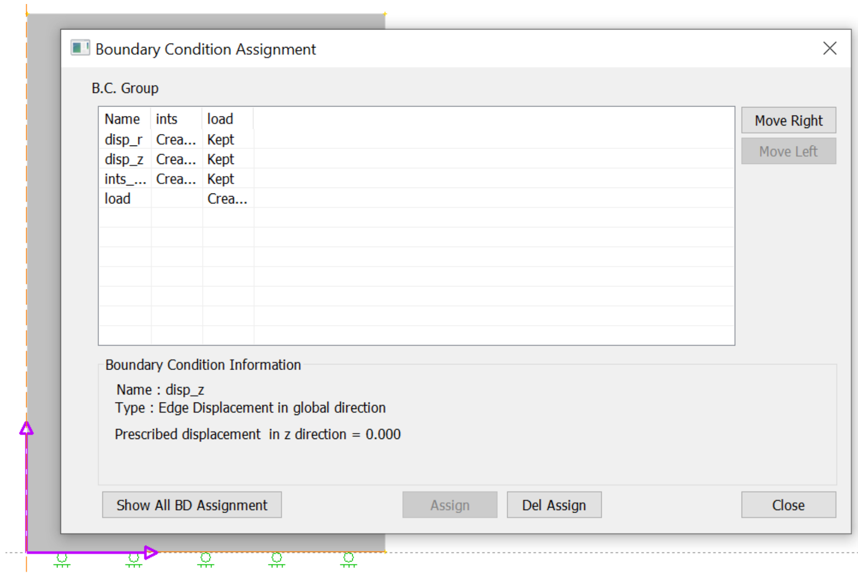


Figure 2.8-8: Apply vertical displacement constraint

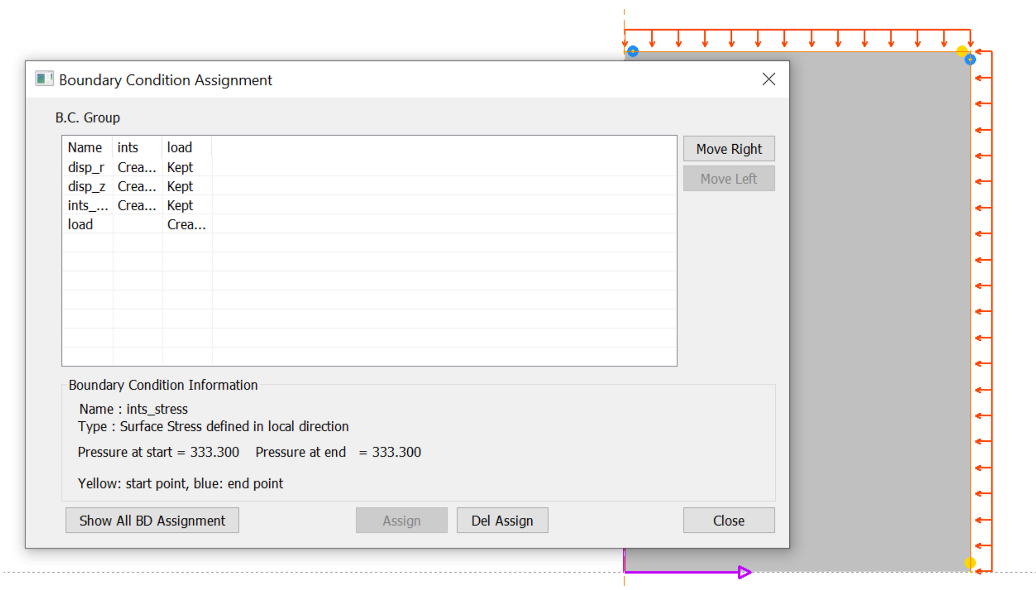


Figure 2.8-9: Apply B.C. to build initial soil stresses

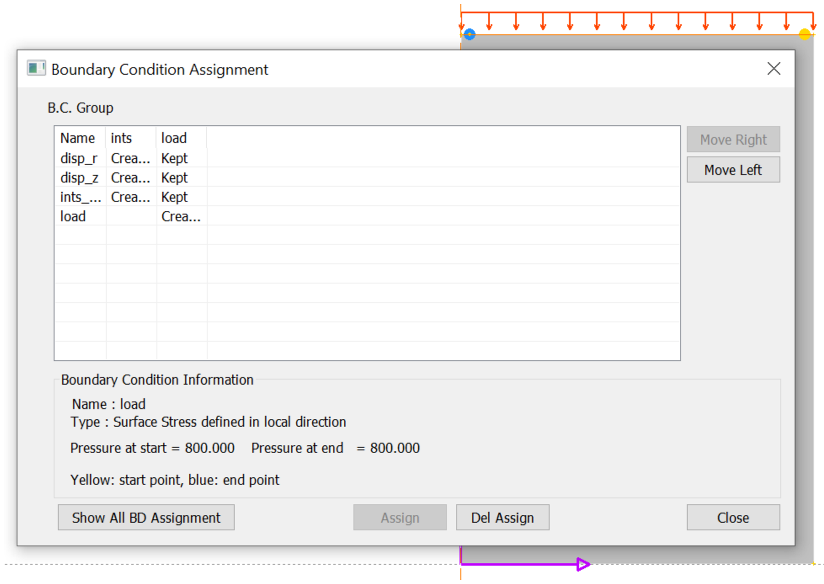


Figure 2.8-10: Apply loading pressure on top of sample cylinder

Click  to save the file.

Step 3: Mesh generation

Click **"pre-Mesh->Advanced Mesh Control"** from the menu bar. If the region selection icon is not activated, , click to activate it. Select the only region in this model and then RIGHT-CLICK and select **"End Selection"**. The global mesh control dialog will open up as shown in Figure 2.8-11. Select "Quadrilateral", "Structured", and **"Quadratic Q8"** with the "Reduced Integration Rule" checked. Click **"Save & Exit"** to close, when completed.

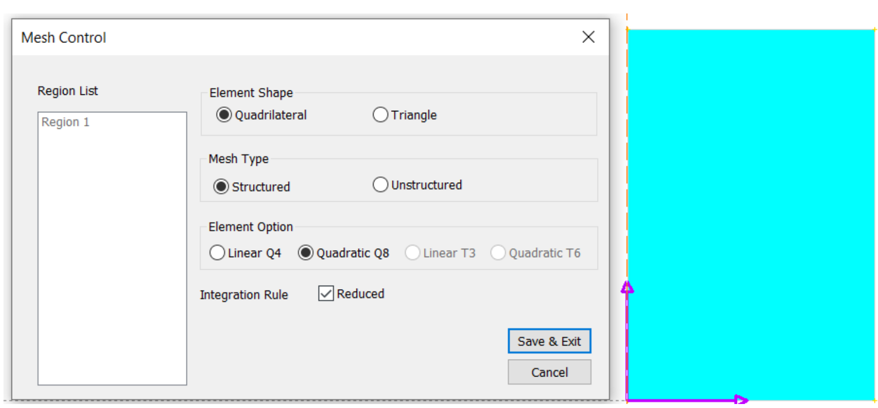


Figure 2.8-11: Global mesh control in Example 8

Click **"Mesh->Global Seed Control"** from the menu bar, to open the dialog to assign the global element size. Change the default size to "0.02" and click **"Assign"**. Click **"Close"** to exit.

For this model there is no need to apply any local seed control.

Click **"Mesh->Run Mesh Generation"** from the menu bar. If the mesh is successfully generated, it will be loaded automatically.

Step 4: Analysis

From the menu bar, click **"Analysis->Analysis Setting"** to open the dialog for analysis options. Input *"25"* for the *'Max. Iter. Number'* for convergence, and keep the other settings as default. Click **"OK"** to exit.

From the menu bar, click **"Analysis->Run Analysis"** to open the *'Run Analysis Dialog'*. By default, each stage is selected to run. Click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. The file, *"Modified_Cam_Clay.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history and any error information, if it exists.

Step 5: Post-processing of result

Click  to switch to the result mode. From the menu bar, click **"Mesh->Show Node Numbers"**. There are five 'corner-nodes' on the symmetric axis numbered *'1'* through *'5.'* This information will be used later in this section for X-Y Plots.

Click **"Result->Stage Selection"** from the menu bar to open the dialog for stage output controls, as seen in Figure 2.8-12. Uncheck the first stage "*ints*" to ignore results of this stage, during the following post-processing exercises.

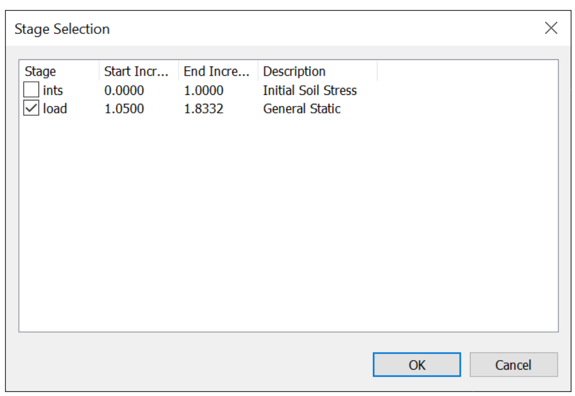


Figure 2.8-12: Select stage for result output in Example 8

Click  to open the X-Y plot dialog. Keep the type of source data as "Historic Result", and input one of the 'corner-node' numbers on the symmetric axis as the node set - in this case *"2"*. Select "Primary variable" from the Category list and select "E22" from the Variables list. Click **"Plot"** to display the Vertical Elastic Strain VS Load Increments (Figure 2.8-13).

NOTE: Notice that the *"ints"* stages (previously set to be hidden) are not displayed in the increment history.

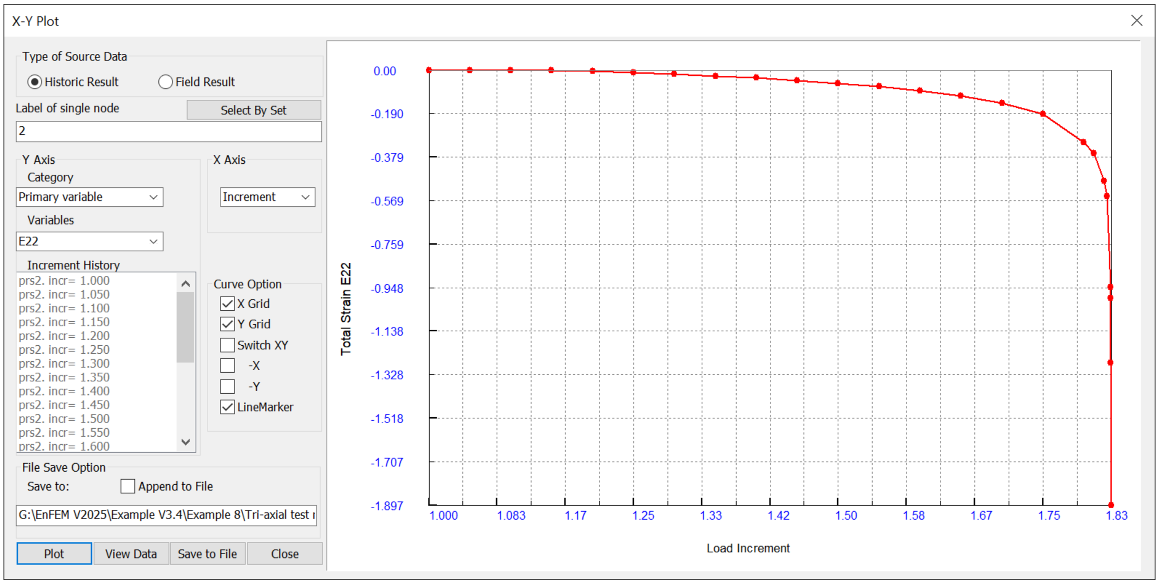


Figure 2.8-13: X-Y plot E22 VS Increment of Node 2

Click **"Save to File"** to export the X-Y data for this curve to file with user-specified path on the hard drive (*filename_xy.dat*). Next, click **"**Equivalent Shear q**"** from variable list and then **"Plot"** to display the curve of Equivalent Shear VS Increments, as seen in Figure 2.8-14. Check the option **"**Append to File**"** and click **"Save to File"** to save this new curve data to the existing X-Y data file for this example. Use a third-party plotting software to plot the curve Shear, **q** VS **E22**, as seen in Figure 2.8-15.

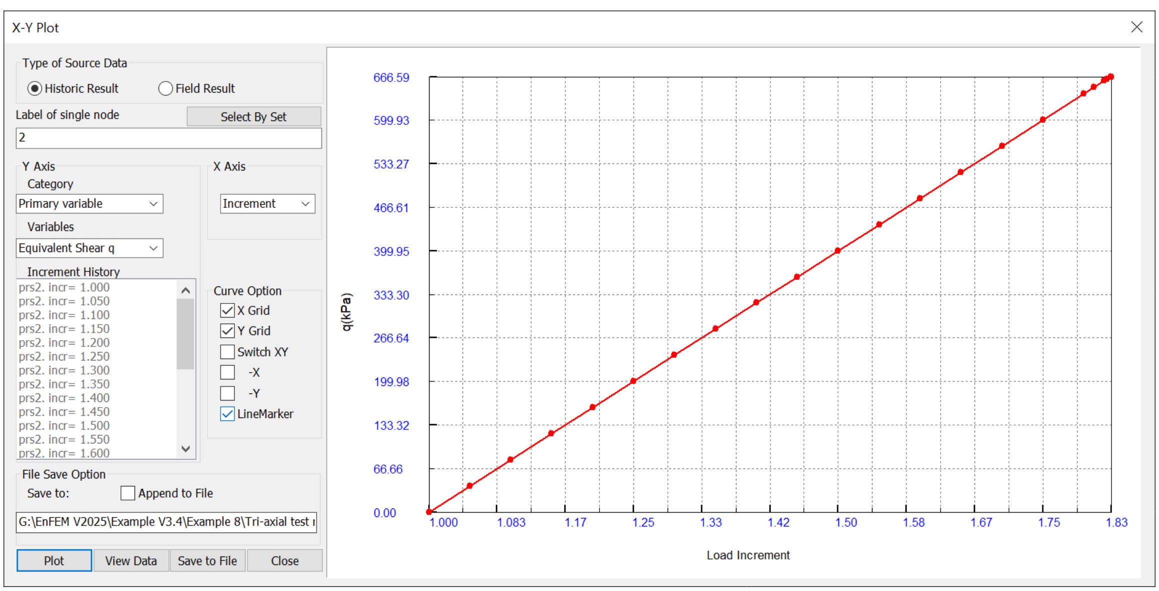


Figure 2.8-14: X-Y plot of Equivalent Shear, q VS Increments of Node 2

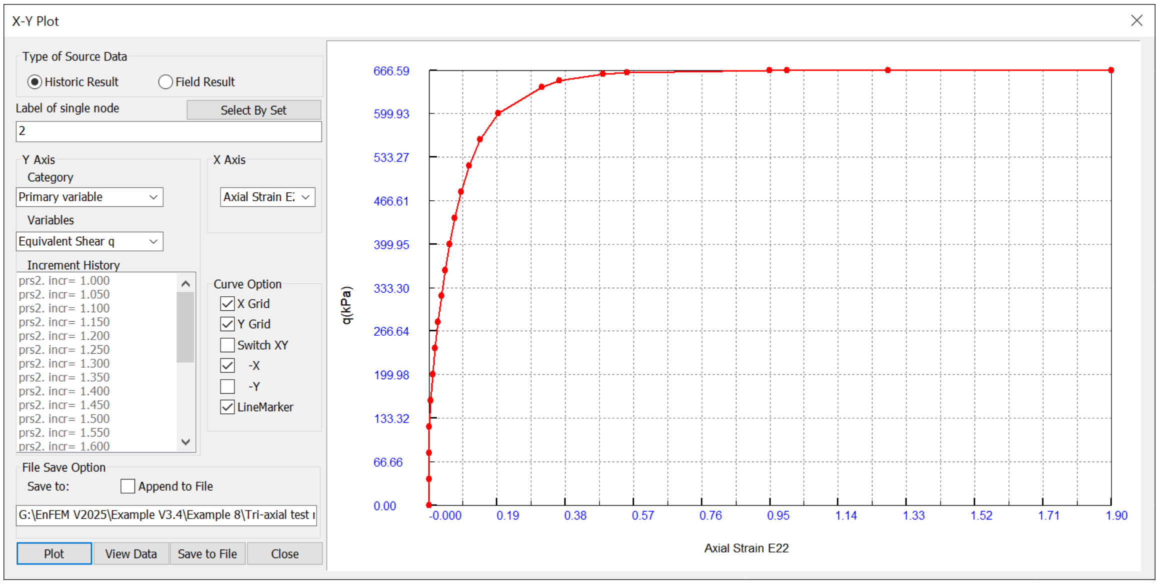


Figure 2.8-15: Combined plot of Shear, q VS E22 in Example 8

Follow similar procedures to plot the curves of q VS mean stress P' (Figure 2.8-16), and void ratio VS E22 (Figure 2.8-17).

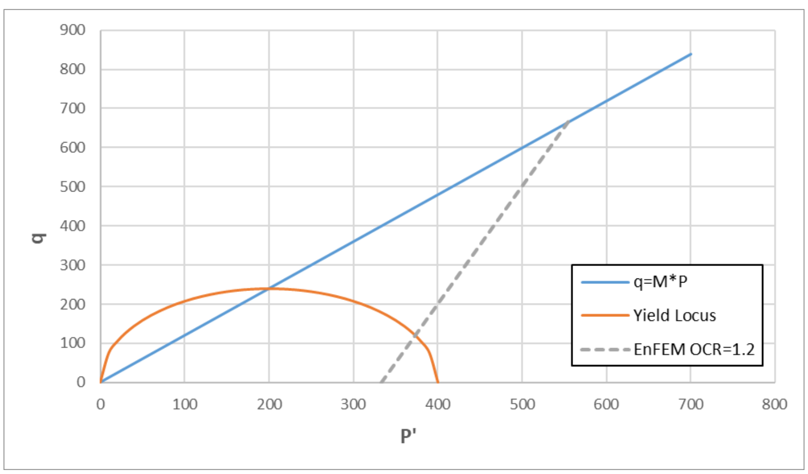


Figure 2.8-16: Combined plot of Shear Stress, q VS Mean Stress, P'

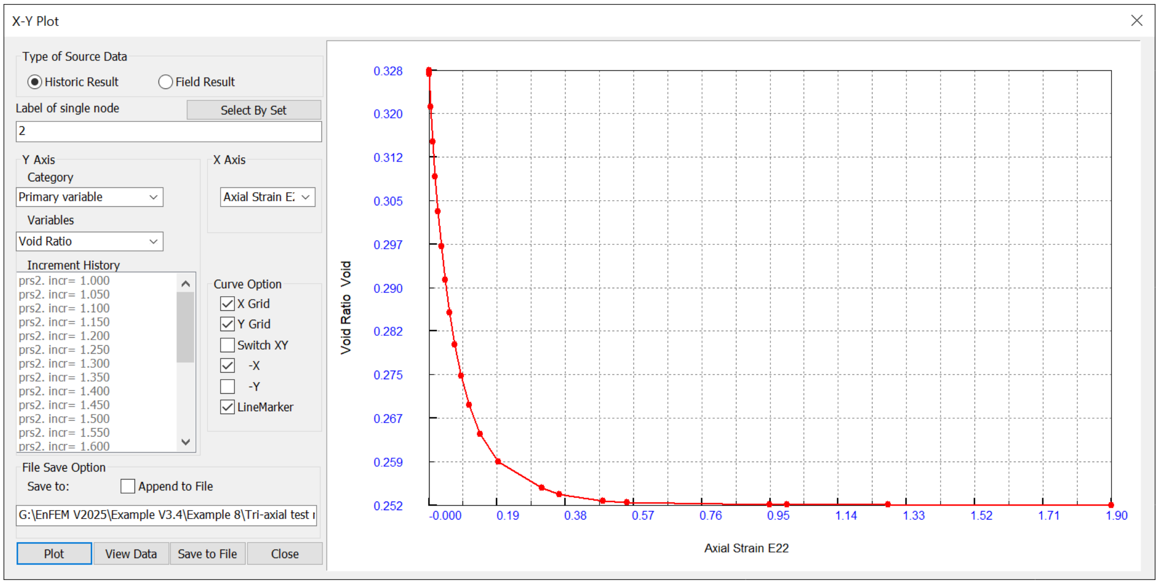


Figure 2.8-17: Combined plot of Void Ratio VS Axial Strain, E22 in Example 8

2.9 Example 9: Analysis of axi-symmetric pile subjected to point load

This example simulates the behavior of an axi-symmetric pile subjected to a point load. Soil-pile interaction is simulated using contact constrained method. The pile is made from concrete which is assumed to be elastic. Dense sand around the pile is simulated by the Mohr-Coulomb plasticity model. Geometric dimensions are displayed in Figure 2.9-1. The pile has 1.0-meter diameter and 9.5-meter length (including a rounded pile tip). The soil geometry is a 25-m by 25-m polygon. Units in this model are “N, m, second”.

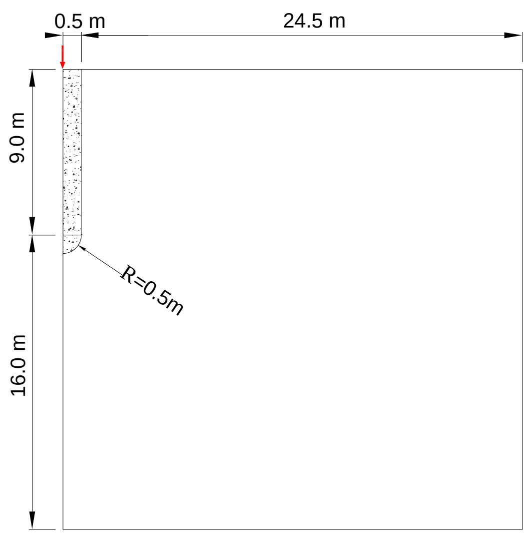


Figure 2.9-1: Configuration of analysis of axi-symmetric pile

Material Properties:

<u>Concrete (Pile):</u>

- Unit weight:.. $\gamma = 24000\ N/m^3$
- Young’s Modulus:...................................... $E = 2.5e^{10} Pa$
- Poisson’s ratio:.. $\nu = 0.2$

<u>Soil (Dense Sand):</u>

- Unit weight:.. $\gamma = 18000\ N/m^3$
- Young’s Modulus:...................................... $E = 5e^{7} Pa$
- Poisson’s ratio:.. $\nu = 0.3$
- Cohesive strength:.................................... $c = 5000\ Pa$
- Friction angle:.. ϕ=36 º
- Dilation angle:... ψ=25 º

Boundary Conditions:

- Displacement constraint on vertical symmetric axis.
- Displacement constraint on bottom edge of the soil region.
- Point load applied to pile on symmetric axis.
- Gravity load
- Contact interface between pile and soil.
    - The pile is the master body and soil is the slave body.
    - Coefficient of friction for the contact interface is 0.5.

Model development steps:

Step 1: Create geometric entities

Start EnFEM, or Click **"File->New"** to start a new project. Change the title to "`axial-symmetric pile subject to point load`". Select "Axial-Symmetric" as analysis type. Select "Customized" unit option. Then select unit "N" for force, "meter" for length and "second" for time.

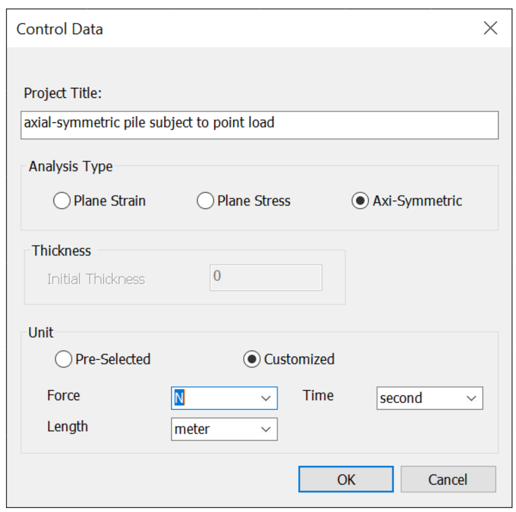


Figure 2.9-2: Control data for Example 9

From the menu bar, click **"Options->Drawing Space"** and change the default space to the values shown in Figure 2.9-3.

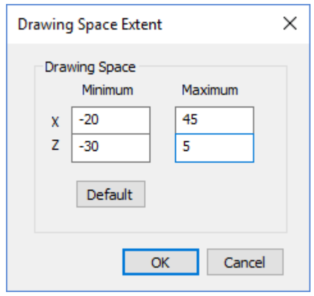


Figure 2.9-3: Setup drawing space for Example 9

Similarly, click **"Options->Grid Spacing"** and change the default grid spacing to "*1*" in both the X and Z-directions.

The pile has two regions: a rectangle, and a one-quarter circle. First, click to activate the rectangle creation tool. Using the 'snap method' create the first rectangular vertex at grid intersection (x=0, z=-9). Next, key in the relative coordinate "*[0.5,9]*" into the coordinate input box and press RETURN to complete the rectangle for the pile. Next, click to activate the arc-creation tool. Use the 'snap method' to select the bottom left corner of the pile-rectangle (coord. x=0, z=-9) as the center-vertex of the arc. Next, enter in the absolute coordinate "*0,-9.5*" as the start-vertex, and then 'snap' to the bottom right corner of the pile-rectangle (coord. x=0.5, z=-9) to create the end-vertex. Click to activate the line creation tool, , and use the 'snapping method' to create new horizontal and vertical lines from the arc endpoints, back to the arc center point (Figure 2.9-4). From the menu bar, click **"Create ->Polygon from Lines/Arcs"**, and select the arc and the new independent lines, then RIGHT-CLICK and select **"End Selection"** from the popup menu. A new quarter circle region, representing the rounded pile-tip, is created as seen in Figure 2.9-4.

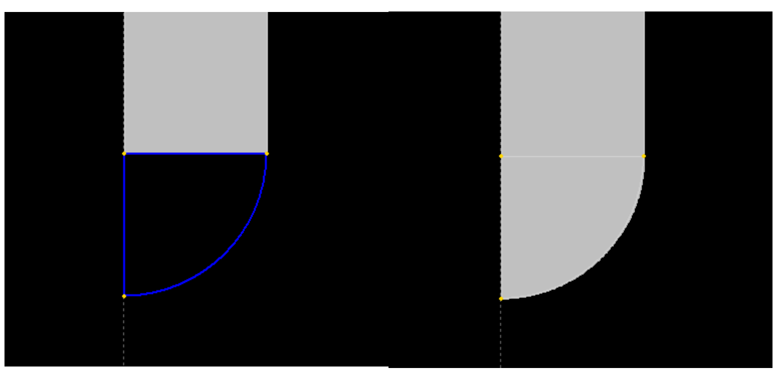


Figure 2.9-4: Pile-tip geometric entities in Example 9

The soil region will be created next. First, use the 'snap method' to recreate the arc of the pile-tip. This will be used in creation of the soil region. Click to activate the line creation tool, and 'snap' to the bottom vertex of the pile-arc as the start point of the first line. Enter in the following absolute coordinates to create new lines continuously for the soil region: "*0,-25*"; "*25,-25*";

`and "25,0"`. Next, use the 'snap method' and click on the top right corner of the pile-rectangle (coord. x=0.5, z=0), and then the bottom right corner of the pile-rectangle (coord. x=0.5, z=-9). RIGHT-CLICK and select **"End Line Creation"** from the popup menu to finish the continuous line-creation. The new independent lines of the soil region should look like those in Figure 2.9-5.

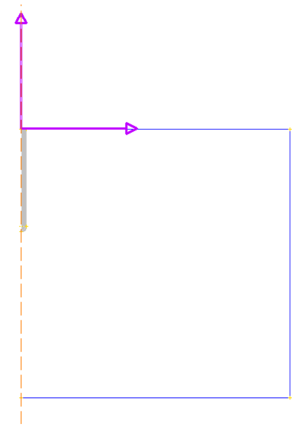


Figure 2.9-5: Created geometric entities of the soil region in Example 9

From the menu bar, click **"Create ->Polygon from Lines/Arcs"**, and select the arc and the new independent lines of the soil region, then RIGHT-CLICK and select **"End Selection"** from the popup menu. The final geometry configurations look like those in Figure 2.9-6.

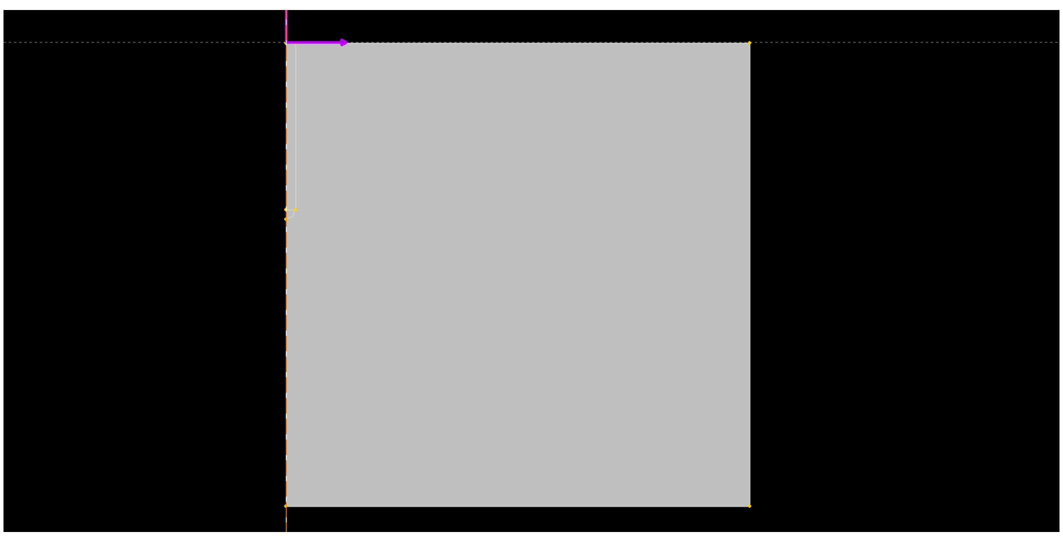


Figure 2.9-6: Geometric entities of the pile and soil in Example 9

Click the icon to save the model file as "`Axial symmetric pile.sep`".

Step 2: Define stage data, material properties, and boundary conditions

From the menu bar, click **"Data->Stage Management"**, and create two new stages – `'ints'` and `'load'`. The "Initial Soil Stress" type should be selected for `'ints'`, and the **"K0 Method"** selection should be made for the soil region, under "'Initial Stress Option" (Figure 2.9-7). Click **"Add"**, select the soil region, and then RIGHT-CLICK and select **"End Selection"** to add the region to the "Region List". Click **"Input Data"**, and then **"Add Row"** to enter in the data seen in (Figure 2.9-7)

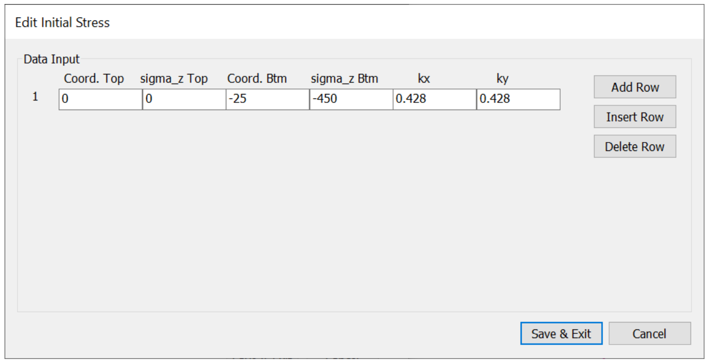


Figure 2.9-7: Initial Stress Stage definition in Example 9

For the second stage, *'load'*, the "General Static" type should be selected. Set the stage parameters to those seen in Figure 2.9-8. Click **"Save & Exit"** and then **"Close"** to exit.

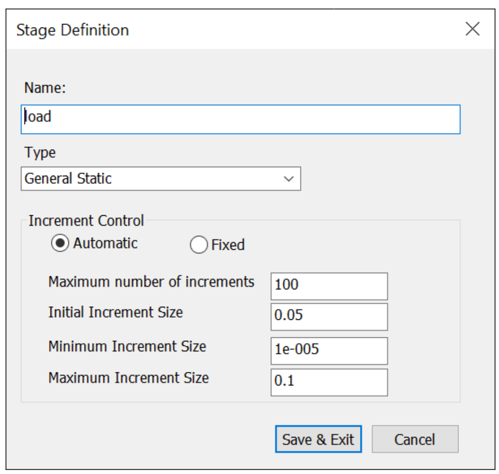


Figure 2.9-8: Loading Stage definition in Example 9

From the menu bar, click **"Data->Material Properties"** and define two new materials as seen in Figure 2.9-9.

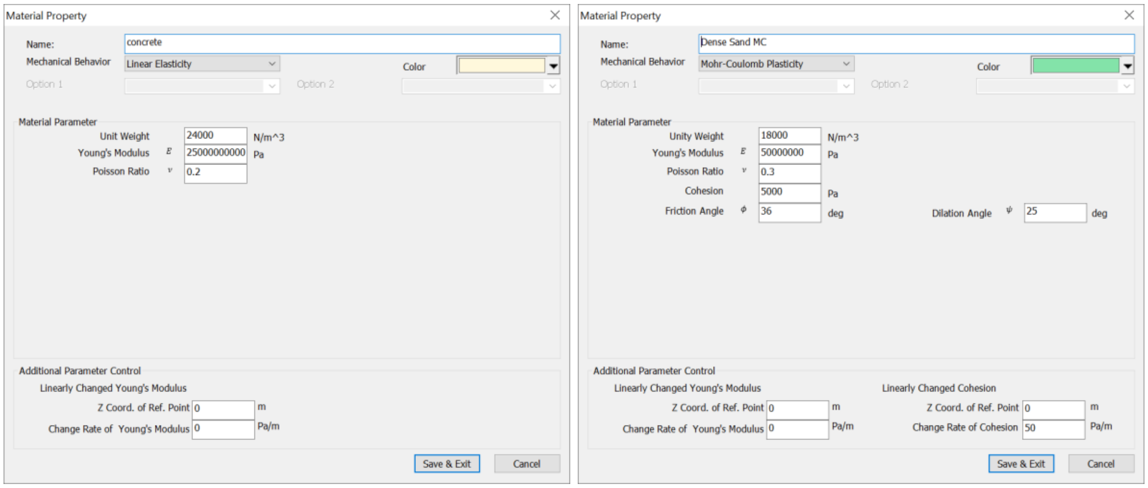


Figure 2.9-9: Define materials for pile and soil in Example 9

Next, click **"Data->Material Assignment"** from the menu bar. Assign the "Concrete" material to the pile regions and assign the "DenseSand MC" material to the soil polygon.

From the menu bar, click **"Data->Boundary Conditions"** and define four boundary conditions as shown in Figure 2.9-10 through Figure 2.9-13.

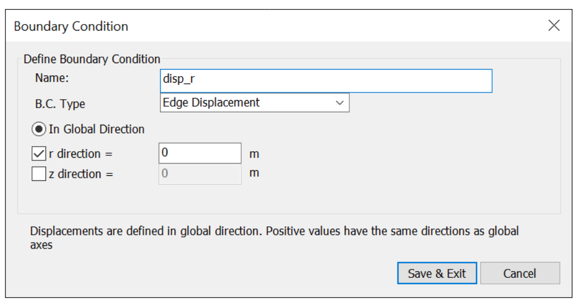


Figure 2.9-10: Define displacement B.C. for vertical soil/pile edges in Example 9

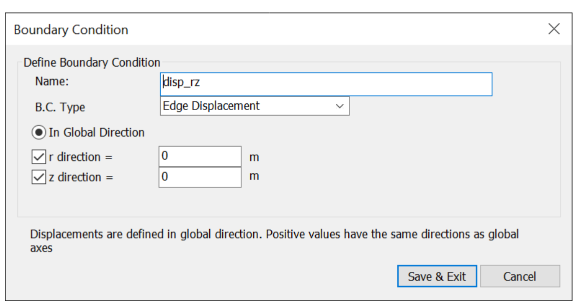


Figure 2.9-11: Define displacement B.C. for bottom soil edges in Example 9

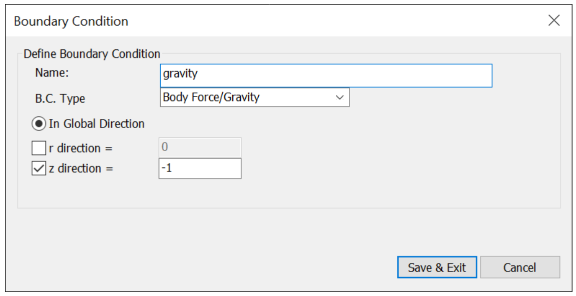


Figure 2.9-12: Define gravity load B.C. in Example 9

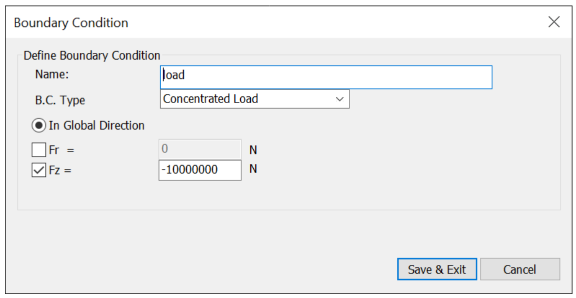


Figure 2.9-13: Define point load on top of pile in Example 9

From the menu bar, click **"Data->Boundary Assignment"** to assign boundary conditions to geometric entities and stages, both as seen in Figure 2.9-14 through Figure 2.9-17. Note that both displacement and gravity boundary conditions start from the 'ints' stage, while the point load starts from 'load'.

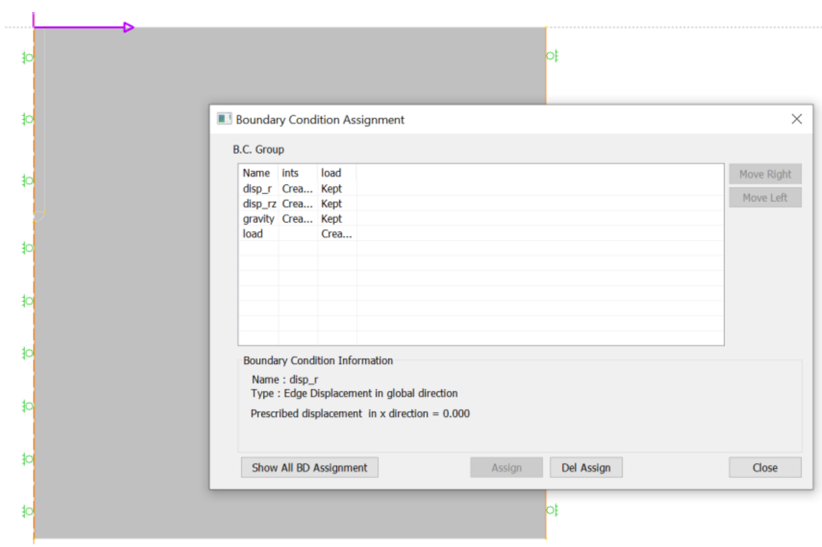


Figure 2.9-14: Assign displacement constraint to vertical soil/pile edges

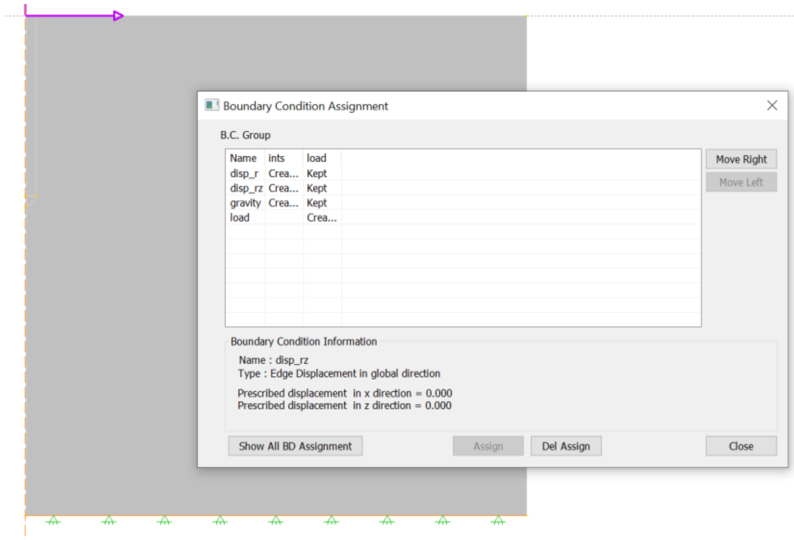


Figure 2.9-15: Assign displacement constraint to bottom edge of soil

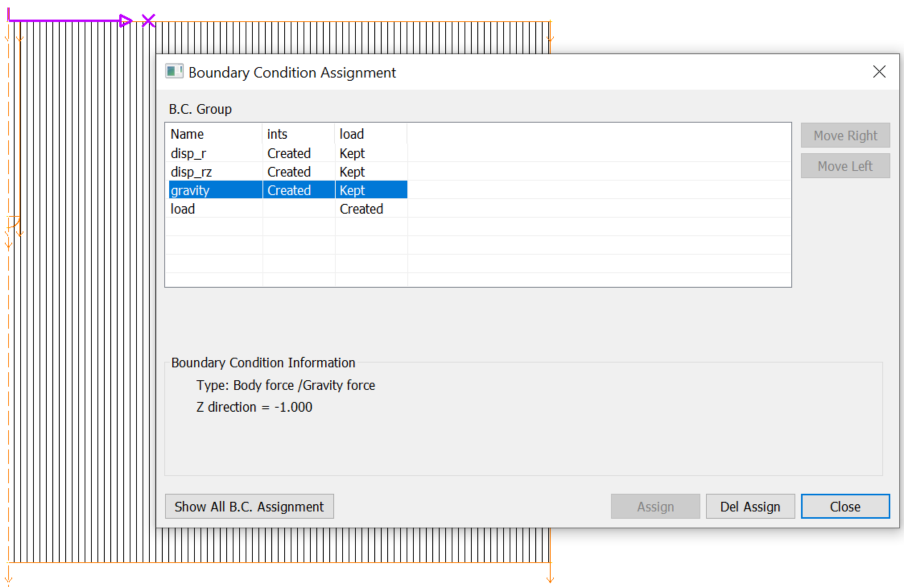


Figure 2.9-16: Assign gravity B.C. to pile and soil regions

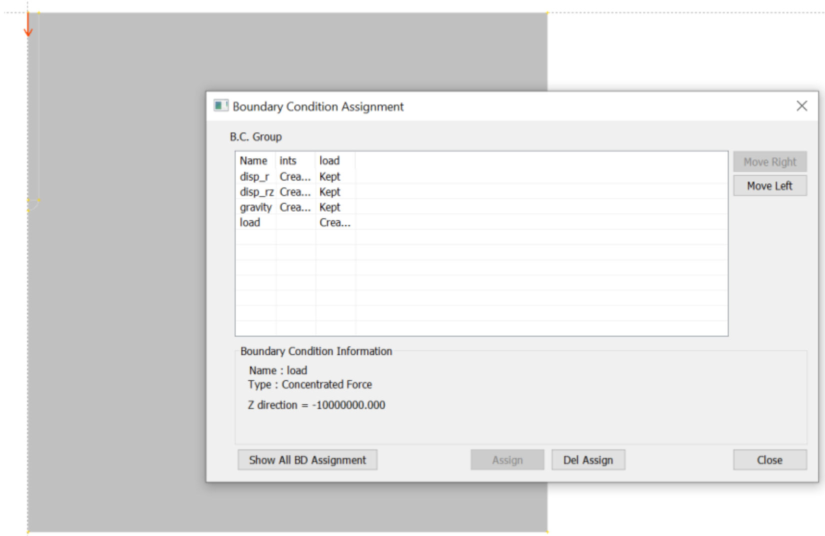


Figure 2.9-17: Assign point load to pile top along symmetric axis

Step 3: Define region sets

From the menu bar, click **"Tools->Set Tool"** and create two region-sets, as seen in Figure 2.9-18. The set *"Pile"* includes both pile regions, and the set *"Soil"* includes the only soil region. Click **"Add Set"**, enter in the set-name, make the appropriate selections, and RIGHT-CLICK and select **"End Selection"** to add new region-sets into model database. Click **"Close"** to exit when complete.

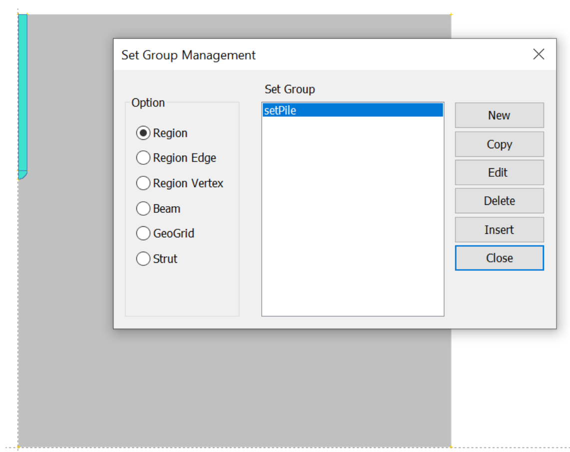


Figure 2.9-18: Define region-sets in Example 9

Step 4: Define contact property

From the menu bar, click **"Tools->Display Tool"**. Select **"pile"** from set list and click **"Display"** to display only the pile regions. Next, click **"Data->Interaction…->Contact Constrained Method"**, from the menu bar, to open the dialog to define an interaction surface. Click **"New"** and keep the default name *"Contact Group 1"*. Change the coefficient of friction to *"0.5"*. Click **"Add Master Surf."**. Next, select the vertical edge and bottom arc of the pile (either with a selection window or SHIFT+CLICK). RIGHT-CLICK and select **"End Selection"**. Do not select the top horizontal edge of the pile nor the symmetric edge. The selected pile edges will be highlighted to blue to signify the master surface assignment (Figure 2.9-19). Use zoom commands as needed to ease selections.

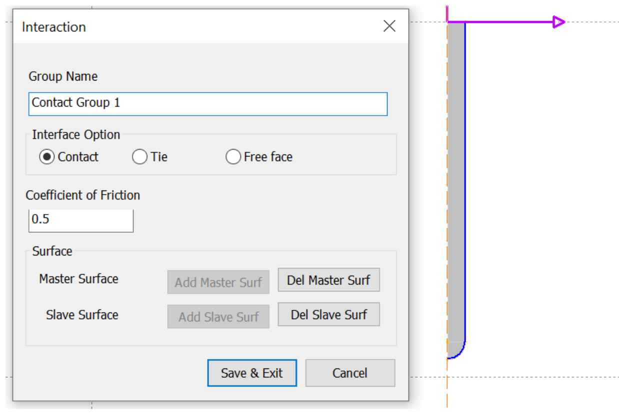


Figure 2.9-19: Define master surface on pile edges

Next, go back to the "display tool" dialog (**"Tools->Display Tool"**), and display the soil only. Click **"Add Slave Surf."**. RIGHT-CLICK and select **"End Selection"**. After being assigned, slave surfaces will be marked in green color, as seen in Figure 2.9-20. Click **"Save & Exit"** to exit from this dialog. Click **"Display All"** on the display control dialog (to show both the pile and soil), and then click **"Close"** to exit from the dialog.

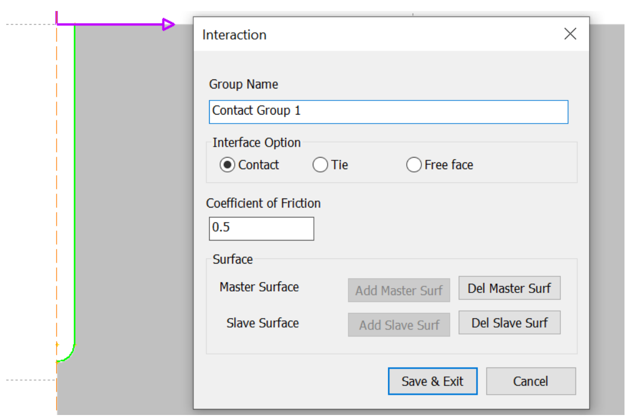


Figure 2.9-20: Define slave surface on soil edges

Click  to save the file.

Step 5: Mesh generation

From the menu bar, click **"pre-Mesh->Advanced Mesh Control"**, and select the rectangular region of the pile. RIGHT-CLICK and select **"End Selection"**. Setup the global mesh control on the pile region as shown in Figure 2.9-21. Click **"Save & Exit"** to close.

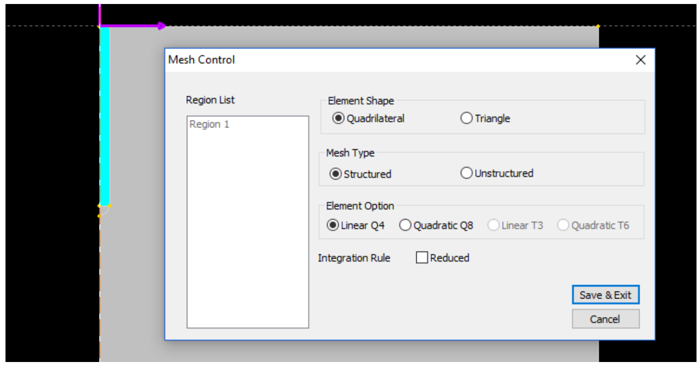


Figure 2.9-21: Setup global mesh control on rectangular pile region

Click **"pre-Mesh->Advanced Mesh Control"** again, and select the one-quarter circle region of the pile. RIGHT-CLICK and select **"End Selection"**. Setup the global mesh control on the pile region as shown in Figure 2.9-22. Click **"Save & Exit"** to close.

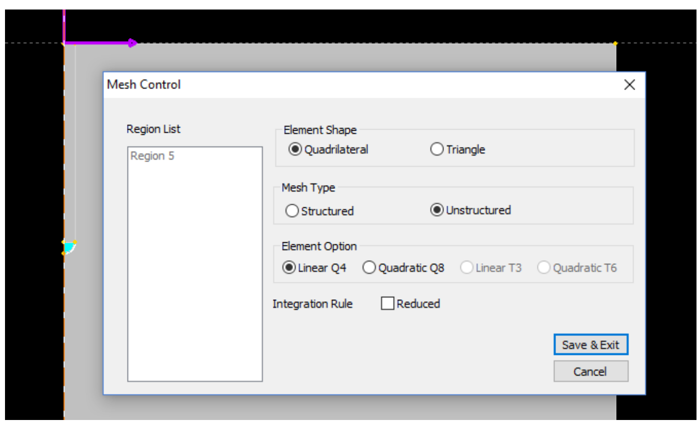


Figure 2.9-22: Setup global mesh control on quarter-circle pile region

Again, click **"pre-Advanced Mesh->Mesh Control"**, select the soil region, and then RIGHT-CLICK and select **"End Selection"**. Setup the mesh control on the soil region as shown in Figure 2.9-23. Click **"Save & Exit"** to close.

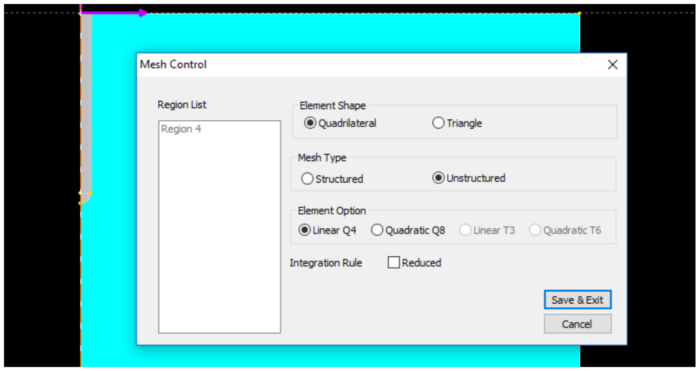


Figure 2.9-23: Setup mesh control on soil region

From the menu bar, click **"pre-Mesh->Global Seed Control"** to open the dialog for global mesh size control. Change the default size to *"0.18"*. Click **"Assign"**, and then **"Close"** to exit.

Click to recover display of all regions if necessary. From the menu bar, click **"pre-Mesh->Show Edge Division"** and then **"Show Seeds"** to turn on display of the edge division and seeds, marked by a red circle.

NOTE: Since the pile is subjected to a vertical point load and the problem is an axi-symmetric model, flexural effects within the pile are not significant, therefore four layers of elements across the pile cross-section will be adequate in this case.

Click **"pre-Mesh->Local Seed Control"**, from the menu bar. Select the top/bottom edges of the pile, then RIGHT-CLICK and select **"End Selection"**. In the popup dialog change the division number on these edges to *"4"* (Figure 2.9-24). Similarly set the division on the arc to be *"4"*.

NOTE: In order to ease selection at the top of the pile (under coordinate axis icon), the user may want to *'uncheck'* the Global Axes display, from **"View->Global Axes"**.

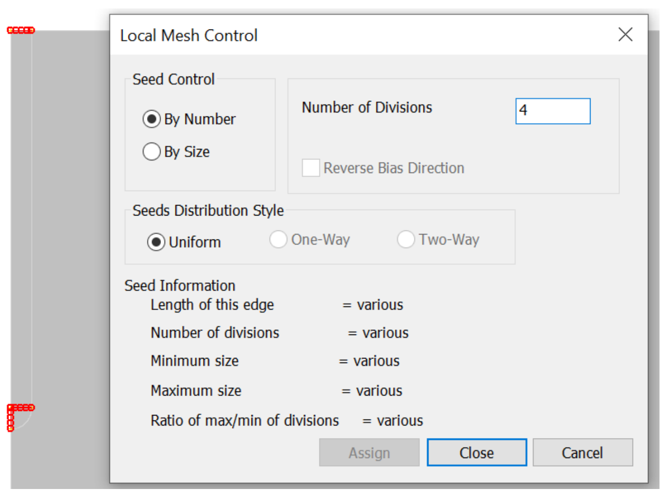


Figure 2.9-24: Define division number for the pile cross-section

Again, click **"pre-Mesh->Local Seed Control"** and select the top edge of the soil region. RIGHT-CLICK and select **"End Selection"**. In the local seed control dialog, adjust the setting according to Figure 2.9-25. Click **"Assign"** and then **"Close"** to exit.

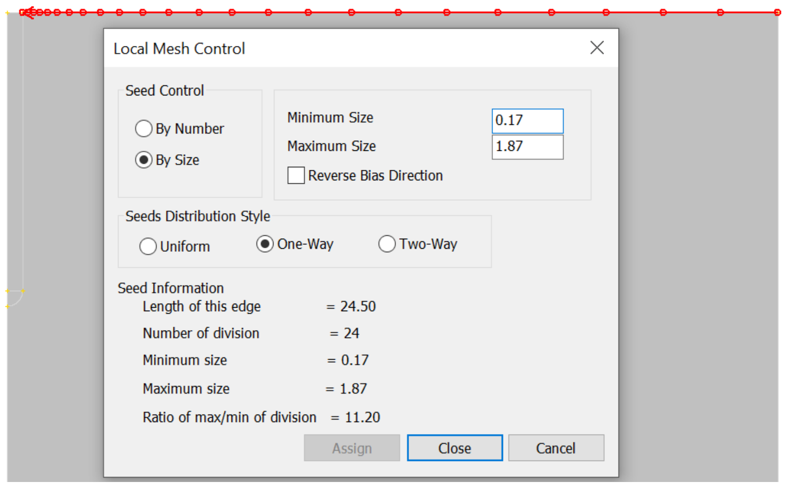


Figure 2.9-25: Set up local seed control to top edge of soil

Click **"pre-Mesh->Local Seed Control"** again, and select the lower left edge of the soil polygon. RIGHT-CLICK and select **"End Selection"**. In the local seed control dialog, adjust the setting according to Figure 2.9-26. Click **"Assign"** and then **"Close"** to exit.

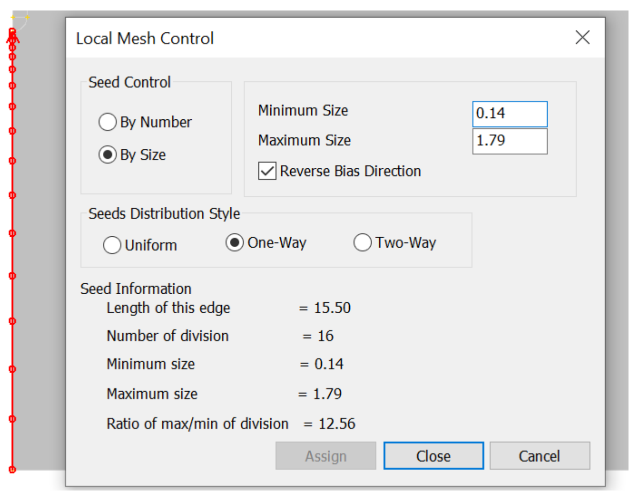


Figure 2.9-26: Setup local seed control on lower left edge of soil

Similarly, set uniformly distributed seeds with size = "2" for the right and bottom edges of the soil region.

Finally, click **"pre-Mesh->Run Mesh Generation"** from the menu bar. A successfully generated mesh will be loaded automatically, as shown in Figure 2.9-27.

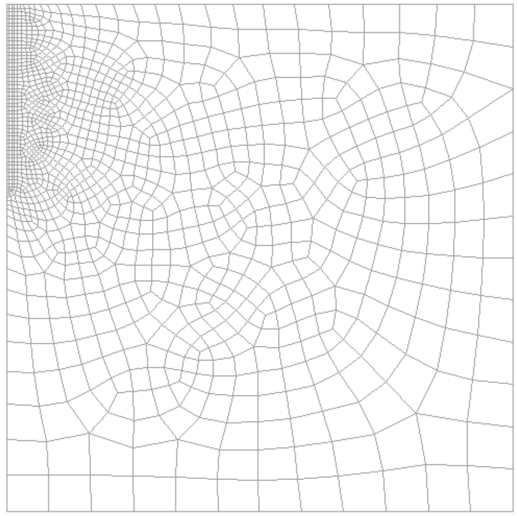


Figure 2.9-27: Quadrilateral mesh in Example 9

Click  to save the file.

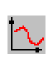
Step 6: Analysis

From the menu bar, click **"Analysis->Analysis Setting"**. For this example, the default setting is appropriate for the analysis. Click **"OK"** to exit.

From the menu bar, click **"Analysis->Run Analysis"** to open the "Run Analysis" dialog. By default, each stage is selected to run. Click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. The file, *"axial_pile_ints_K0.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history and any error information, if it exists.

Step 7: Post-processing of result

Click , to switch to the result-mode and then from the menu bar, click **"Mesh->Show Node Numbers"**. Zoom-in to the pile top to check node numbers. In this example, node-1 is located on the symmetric axis at the point of load. Next, click **"Results->Stage Selection"** from the menu bar, and uncheck **"**ints**"** (initial soil stresses). Click **"OK"** to close the stage selection dialog.

Next, click the X-Y curve plot icon, . Note that the stage "ints" is now removed from the increment history list. Keep the source data type as **"**Historic Result**"** and input *"1"* to the node list. Select **"**Primary variable**"** from the Category drop down and select **"**u2**"** from Variable drop list. Click button **"Plot"** to plot the Displacement in the Z-Direction VS Increments of the 'Load' stage (Figure 2.9-28).

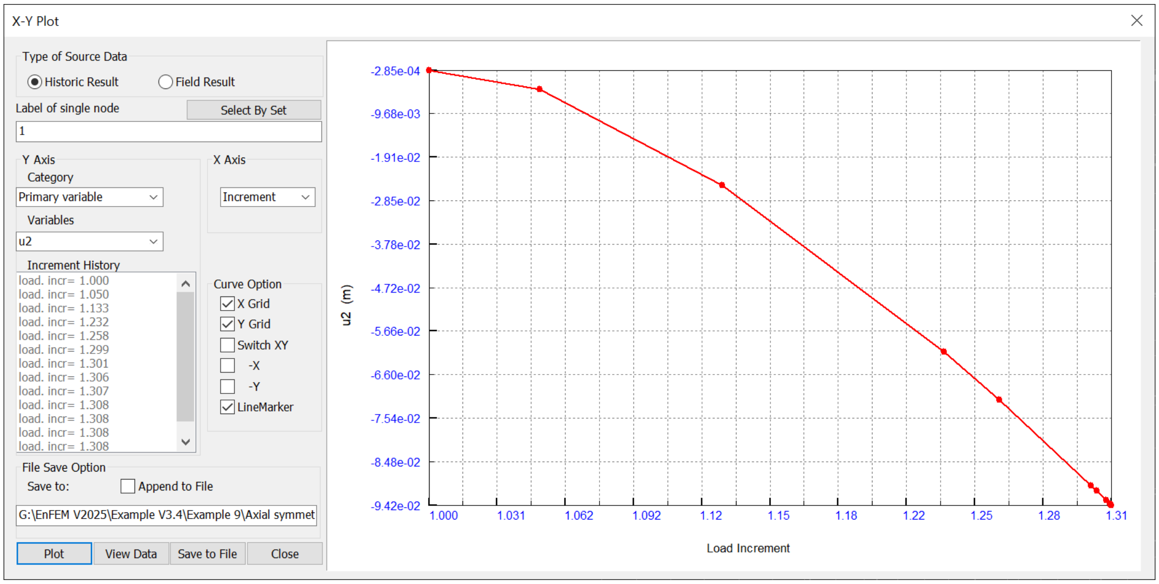


Figure 2.9-28: Vertical Displacement VS Increments of Stage 2

Next, enter in the node set "`319:367:1`" (nodes along the slave/soil interaction surface, not including the bottom-arc). Select "Interaction" from the Category list, and "Tangent Shear" from Variable drop down list. The source data switches to "Field Result" automatically and the increment window is activated. Click **"Plot"** to plot the tangent shear stress on the Vertical Interaction Interface (Figure 2.9-29). End nodes are dropped in this plot.

<u>NOTE:</u> More information can be found on the interaction status as detailed in the User Manual

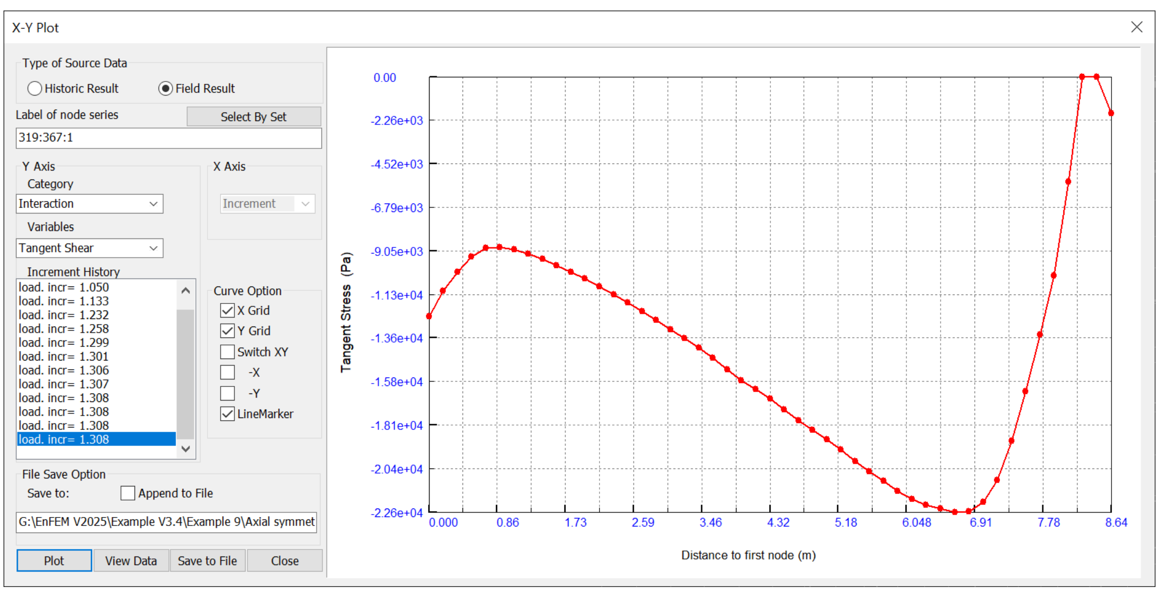


Figure 2.9-29: Tangent status on the vertical interface

2.10 Example 10: Analysis of excavation with cantilever retaining wall

This example presents how to simulate procedures of soil excavation. The retaining wall is discretized by solid elements. Soil-structure-interaction is simulated by contact constrained method. The retaining wall is assumed to be elastic. Modified Cam-Clay model is applied to the soil layer.

The geometric configuration is shown in Figure 2.10-1. The model consists of an uniform soil region, and a 0.6m concrete diaphragm wall. The excavated portion has 10-m in depth, which will be divided into four layers. A reinforcement concrete slab (thickness=0.15 m) will be constructed on the excavated base. Units used in this example are “KN, m, second”.

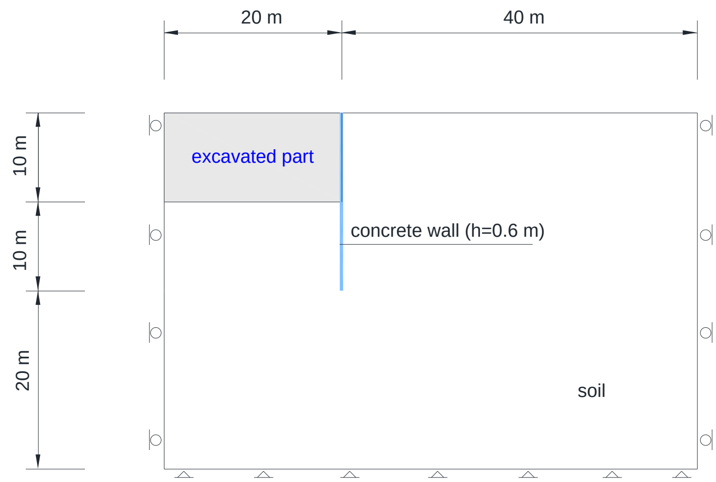


Figure 2.10-1: Configuration of excavation analysis example

Material property:

<u>Concrete (Diaphragm wall - linear elastic):</u>

- Unit weight:.. γ=$22kN/m^3$
- Young's Modulus:.................................. $E = 2.8e^7 kPa$
- Poisson's ratio:..................................... $\upsilon = 0.2$

<u>Soil (modified Cam-clay):</u>

- Unit weight:.. $\gamma = 18kN/m^3$
- Slope of the normal compression line:.......... lambda λ =0.1
- Slope of the swelling line: .... kappa κ =0.025
- Slope of critical line: .... M =1.2
- Specific volume number: .... N =3
- Poisson's ratio: .... υ =0.35
- Over-consolidation ratio:.......................... *OCR=1*

Boundary conditions:

- Displacement constraint on vertical soil boundaries.
- Displacement constraint on bottom edge of the soil region.
- Gravity load
- Contact interfaces between wall and soil configurations.
    - The wall is the master body and the soil is the slave body.
    - Coefficient of friction is 0.6.

Model development and analysis steps:

Step 1: Create geometric entities

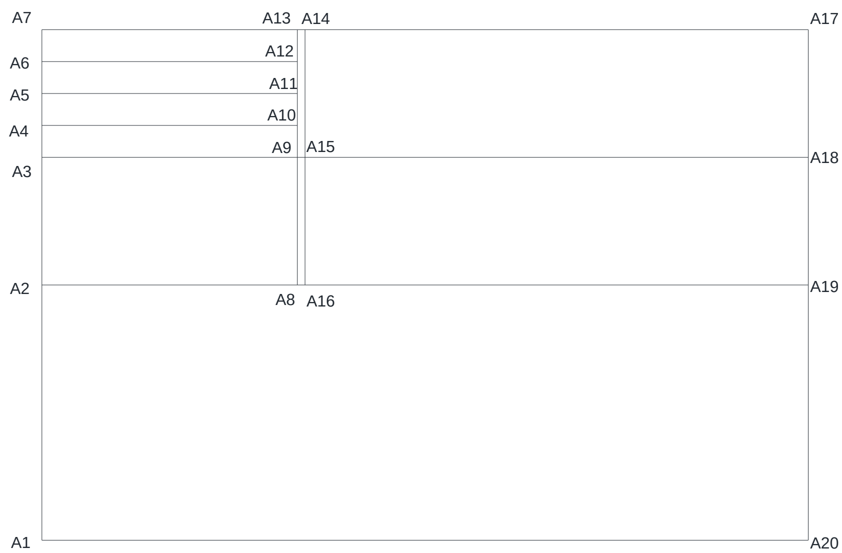


Figure 2.10-2: Region division and vertex label in Example 10

The soil and wall regions are divided into several sub-rectangles, as shown in Figure 2.10-2. A structured mesh can be assigned to a rectangle region easily to generate high quality finite elements. Controlling vertexes are labeled in Figure 2.10-2 and coordinates are shown in Table 2.10-1.

Start EnFEM, or Click **"File->New"** to start a new project. Change the title to "`excavation with cantilever retaining wall`". Select preselected unit "kN, meter, second" (Figure 2.10-3).

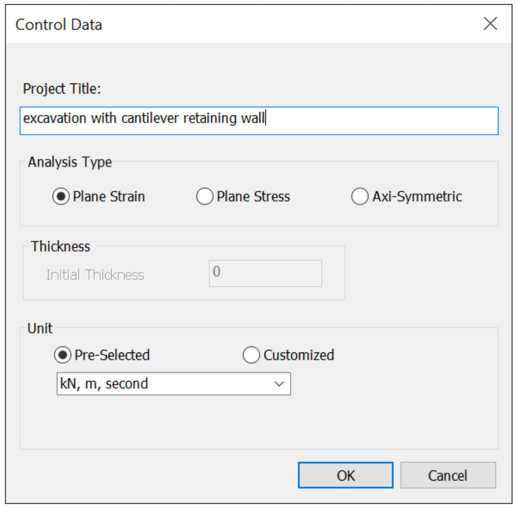


Figure 2.10-3: Control data in example 10

	A1	A2	A3	A4	A5	A6	A7	A8	A9	A10
x	0	0	0	0	0	0	0	20	20	20
z	0	20	30	32.5	35	37.5	40	20	30	32.5
	A11	A12	A13	A14	A15	A16	A17	A18	A19	A20
x	20	20	20	20.6	20.6	20.6	60	60	60	60
z	35	37.5	40	40	30	20	40	30	20	0

Table 2.10-1: Coordinates of controlling vertexes

Click **"Options->Drawing space"** and setup the drawing space like that shown in Figure 2.10-4. Click **"OK"** to exit.

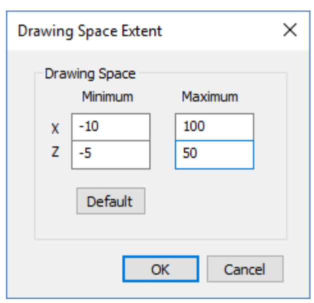


Figure 2.10-4: Setup drawing space for Example 10

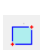
Click  to activate the rectangle creation tool and key in the coordinate of vertex "*A7*", *"0,40"*, into the coordinate input box. Press RETURN to create the first vertex of this rectangle. Next, key in the relative coordinate of the opposite corner, vertex "A12", *"[20,-2.5]"* and press RETURN to complete the rectangle. 'Snap' to vertex "*A6*", and LEFT-CLICK to create first vertex of the next rectangle. Then, key in the relative coordinate of the opposite corner, vertex "A11", *"[20,-2.5]"* and press RETURN to complete the second rectangle. Similarly, create the remaining rectangles for the excavated portion. Click to create three polygons for soil and two rectangles for the wall as shown in Figure 2.10-5.

Figure 2.10-5: Region creation in Example 10

Click to save the model file as "`Excavation and cantilevered retaining wall.sep`".

Step 2: Define region sets

From the menu bar, click **"Tools->Set Tool"** and create region-sets as shown in Figure 2.10-6. Each excavated soil layer is defined as one region set ("`setExcava_1`" to "`setExcava_4`"). These excavation sets are necessary to define excavation as 'deactivated'. The set "`setWall`" includes all regions of the retaining wall. The set "`setSoil_all`" includes all soil sub-regions. The later two region sets will be used to display wall or soil individually to define contact interaction conveniently.

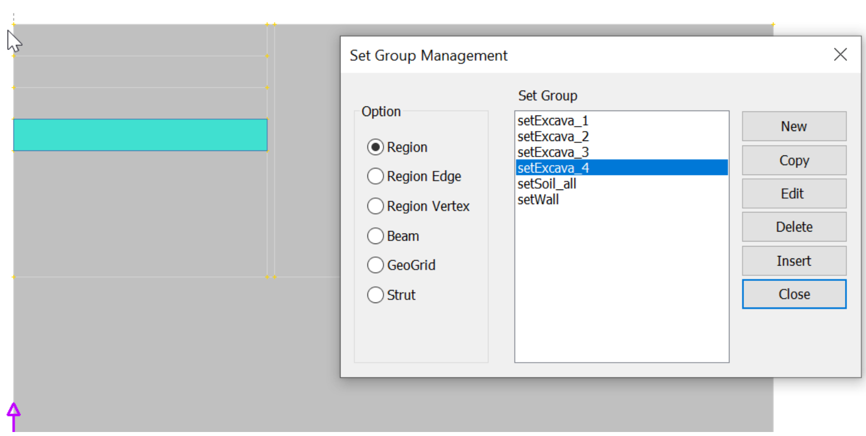


Figure 2.10-6: Define region sets in Example 10

Click **"Close"** to exit.

Step 3: Define stage data, material properties, and boundary conditions

From the menu bar, click **"Data->Stage Management"**, to open the dialog of stage group management. Create the first stage, *"ints"*, as shown in Figure 2.10-7, and select **"**Initial Soil Stress**"** as the type. Next click **"Initial Stress Option"** to open the Initial Stress dialog as seen in Figure 2.10-8. Select the **"**K_0 method**"**.

NOTE: Flat ground lets the K_0 method to be applicable. The vertical stress at any point will be interpolated linearly from stress values at top and bottom positions of the soil layer.

Click **"Add"**, select all of the soil regions, and then RIGHT-CLICK and select **"End Selection"** to add the soil regions to the 'Region List'. A 'crossing' selection window may be useful because it helps to select multiple regions simultaneously. Press and hold the SHIFT If all soil regions cannot be selected by one selection window. If soil regions are not correctly added, users may click "**Clear**" to refresh the region list, or click "Update" and reselect objects again. Click **"Input Data"** to open the initial stress window. Click **"Add Row"** to enter in the data seen in Figure 2.10-9. The k_x and k_y are coefficients of lateral stress which can be estimated by:

$$k_x = k_y = \frac{\nu}{1-\nu}$$

where ν is the Poisson's Ratio. Click **"Save & Exit"**.

Next, users will define four construction stages, named *"Excava_1"* through *"Excava_4"* as an **"Excavation"** type, with parameters as seen in Figure 2.10-10.

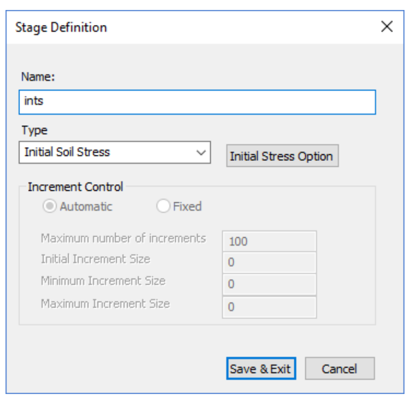


Figure 2.10-7: Define stage type as "Initial Soil Stress

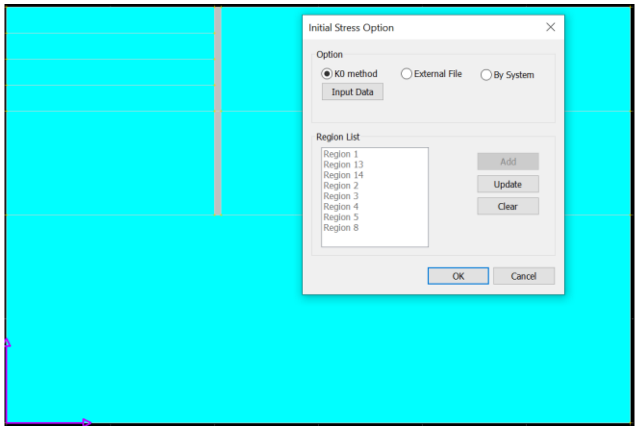


Figure 2.10-8: Options of initial soil stress

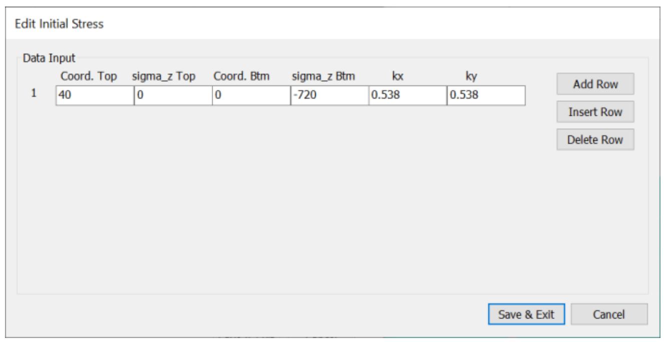


Figure 2.10-9: Input vertical soil stresses at controlling points and coefficient of lateral stress

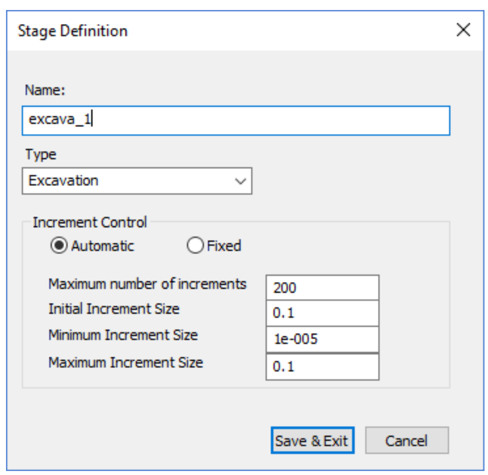


Figure 2.10-10: Typical 'Excavation' stage

The last stage "`surcharge`" is defined as "General Static" in which weight of concrete slab will be applied to the excavated ground (Figure 2.10-11).

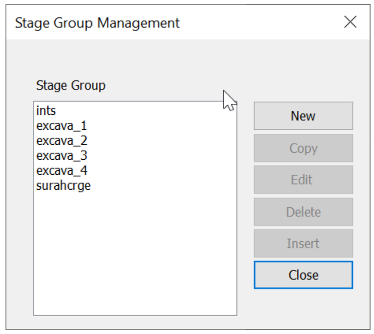


Figure 2.10-11: Stage groups in Example 10

Step 4: Define material properties

From the menu bar, click **"Data->Material Properties"** and define one material, named "`soil`", as a "Modified Cam-Clay" type, like that seen in Figure 2.10-12. Select "Constant Poisson's ratio **υ**" for "Option 1" and "Drained" for "Option 2". Keep the option **"Use Default"** for the "Parameters for Initial Soil Stress" selection. Click **"Save & Exit"** to exit.

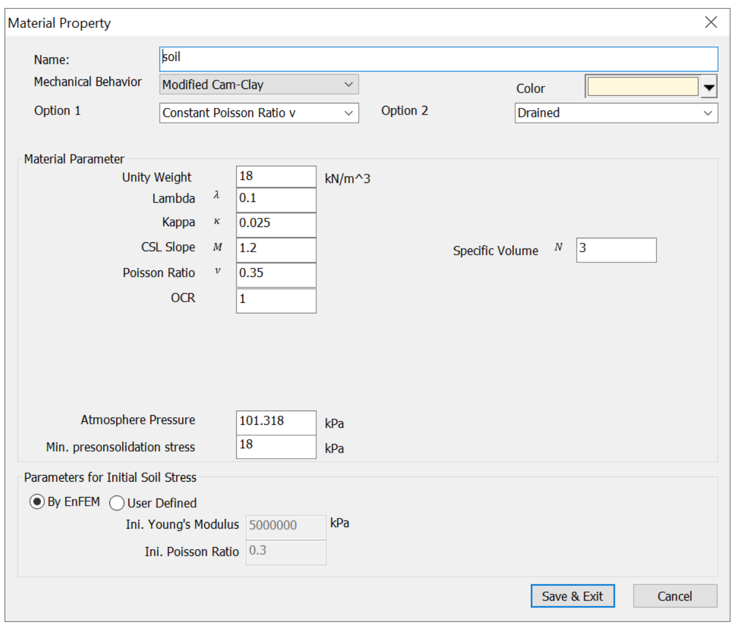


Figure 2.10-12: Define modified cam-clay material

Create another elastic material, named "`concrete`" for the retaining wall, with parameters as seen in Figure 2.10-13.

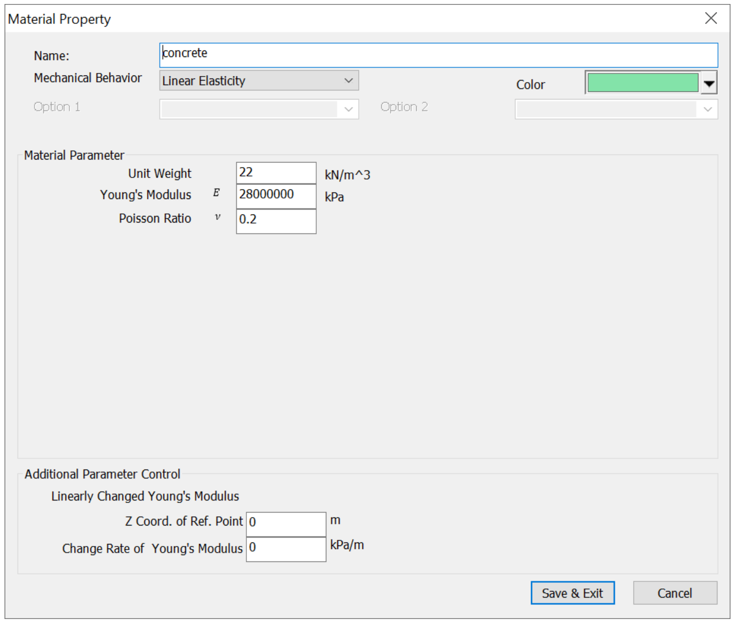


Figure 2.10-13: Define linear elastic concrete material

Next, click **"Data->Material Assignment"** from the menu bar to open the material assignment dialog. Select the option "By Set" to assign previously defined materials to existing set definitions. Click the set "setWall" and select "concrete" from the "Material Group" drop-down list. Click **"Assign"** to finish assigning the "concrete" material to the "setWall" region-set, as shown in Figure 2.10-14. Similarly assign the material "soil" to the "setSoil_all" region-set. Click **"Close"** to exit the dialog window when complete.

<u>NOTE:</u> If a set is not assigned any material previously, the button **"Assign"** will be activated. Otherwise, if an assignment already exists, the button **"Update"** will be activated instead.

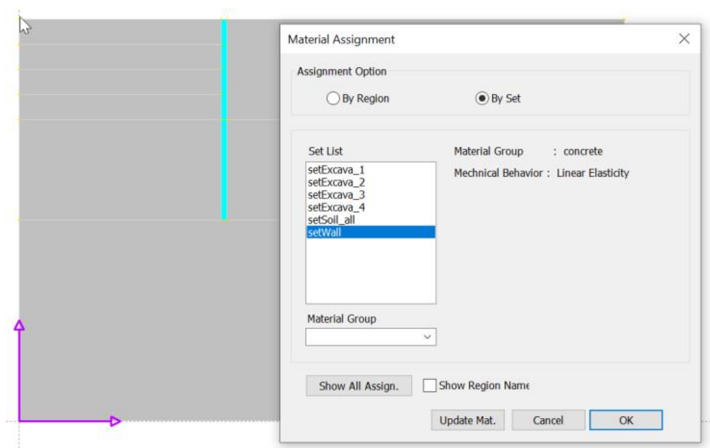


Figure 2.10-14: Assign material to previously defined set

Step 5: Define boundary conditions

From the menu bar, click **"Data->Boundary Conditions"** to open the dialog for the boundary condition group management. Create three boundary conditions as seen in Figure 2.10-15 through Figure 2.10-18.

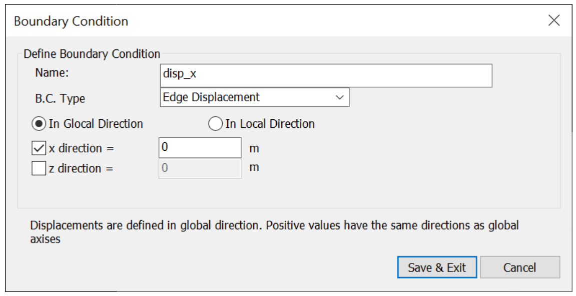


Figure 2.10-15: Define displacement constraint on vertical soil boundaries

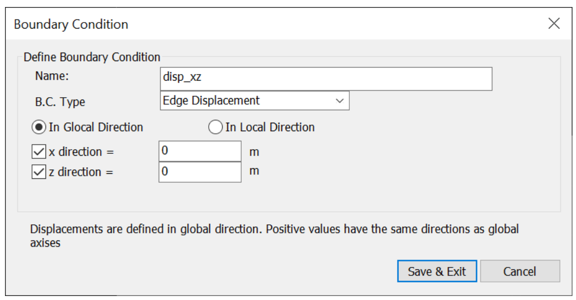


Figure 2.10-16: Define displacement constraint on bottom soil boundary

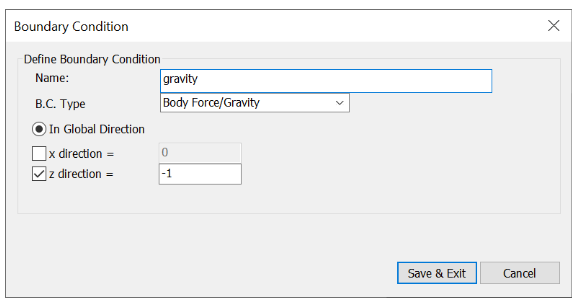


Figure 2.10-17: Define gravity load

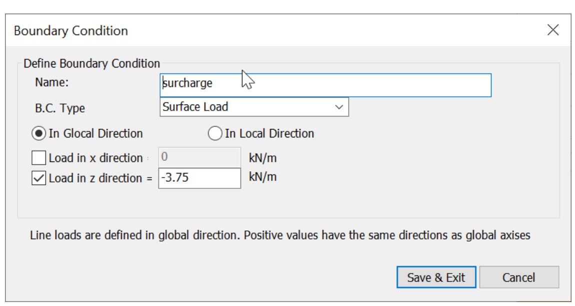


Figure 2.10-18: Define surcharge on excavation base

Note the surcharge `-3.75` is defined in the global direction. This surcharge is applied to the exposed ground after excavation, which is an internal edge at beginning. If this load is defined in the local direction, EnFEM will report an error message due to difficulty to find the region object to apply this surface load.

From the menu bar, click **"Data->Boundary Assignment"** to assign BC's to geometric entities appropriately. Assign **"**disp_x**"** to the vertical boundaries of the soil layer, assign **"**disp_xz**"** to the bottom edge of the soil layer. **"**gravity**"** is assigned to both soil and wall regions. Set all boundary conditions to start from stage "ints", except the last 'surcharge'. When assignments are completed, click "**View->Show Boundary Assign.**" To turn on display of all assigned boundary conditions as shown in Figure 2.10-19. Verify each assignment, then click "**Close**" to exit.

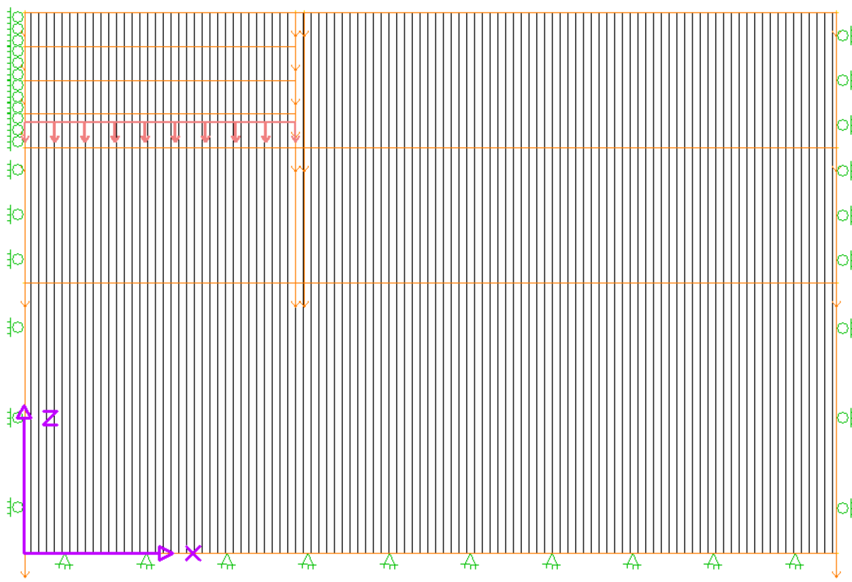


Figure 2.10-19: Boundary condition assignments

Step 6: Define excavation

From the menu bar, click **"Data->Deactivate"** to open the window to define excavation constructions, as shown in Figure 2.10-20. All defined stages will be displayed under column "Stage". Under the column "Set Deactivate", click drop list on the right side and then select the appropriate set name that represents the region to be excavated in current stage. Define these deactivated sets as seen in Figure 2.10-21. After one set is selected as "deactivated", it is not necessary to reselect it in later stages.

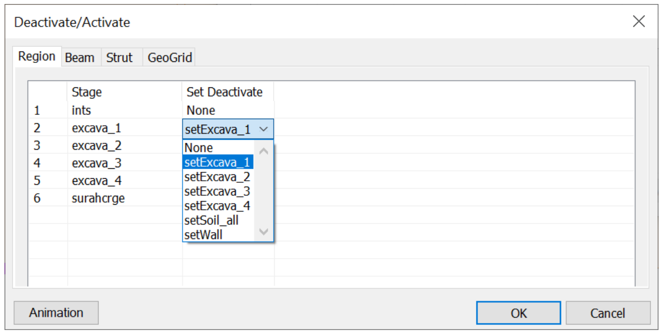


Figure 2.10-20: Define deactivated regions to simulate excavation

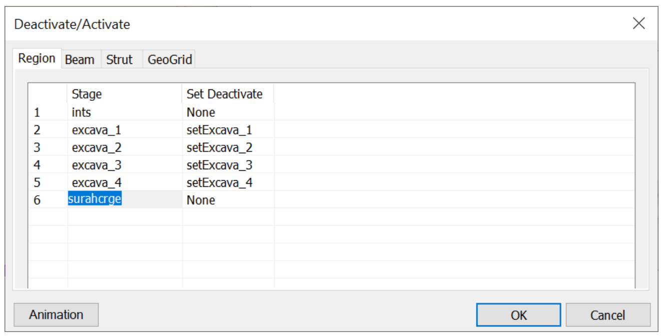


Figure 2.10-21: Final deactivated regions definitions

NOTE: Click **"Animation"** to let EnFEM play animation of the excavation procedures. The animation will run once after users click '**Animation**" button.

Click **"OK"** to exit from this dialog and save the model.

Step 7: Define soil-wall interaction property

Since the soil-wall interaction surfaces are overlapping, to define this interaction, it is necessary to display soil layers or the wall exclusively. From the menu bar, click **"Tools->"Display Tool"** to open the *'Display Control'* dialog. Hide all soil regions. Keep the *'Display Control'* dialog open. From the menu bar, click **"Data->Interaction"** to open the interaction group management dialog. Define the property as seen in Figure 2.10-22. Select **"Contact"** as the interaction option and input "`0.6`" as the coefficient of friction. With only the wall displayed, click **"Add Master Surf."** (left image of Figure 2.10-22) and select the vertical edges and the bottom horizontal edge of the wall configuration by a 'contained' selection window. RIGHT-CLICK and select **"End Selection"** to complete the definition of the master surface. Go back to the *'Display Control'* dialog, hide the wall and show all soil regions. Click **"Add Slave Surf."** in the interaction dialog (right image of Figure 2.10-22) and select the corresponding soil edges, RIGHT-CLICK and select **"End Selection"**. This will complete the definition of the slave surface and thus the interaction property. Click **"Save & Exit"** to close the interaction dialog.

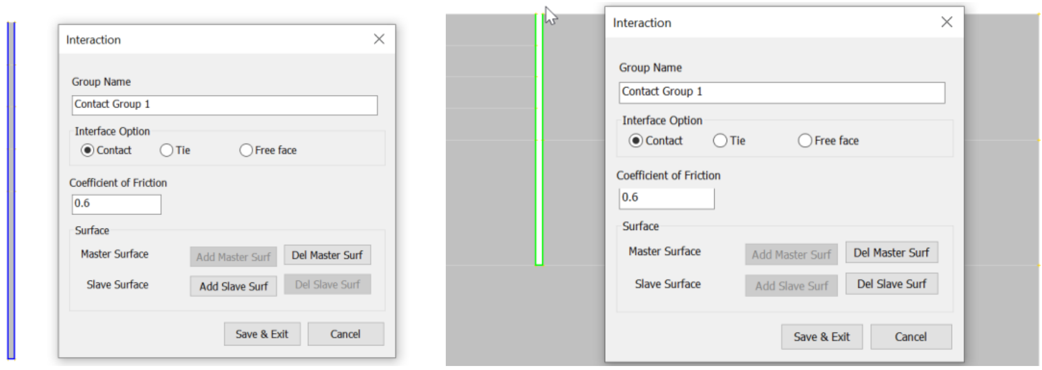


Figure 2.10-22: Define master and slave contact surfaces

Step 8: Mesh generation

Click  to display all regions. From the menu bar, click **"Mesh->Advanced Mesh Control"**, and select all of the rectangular regions, except for the bottom soil region below the wall. RIGHT-CLICK and select **"End Selection"** to open the “Mesh Control” dialog. Define “Quadrilateral” as element shape, “Structured” as mesh type, and “Linear Q4” with full the mesh like that shown in Figure 2.10-23. Click **"Save & Exit"** to exit from this dialog.

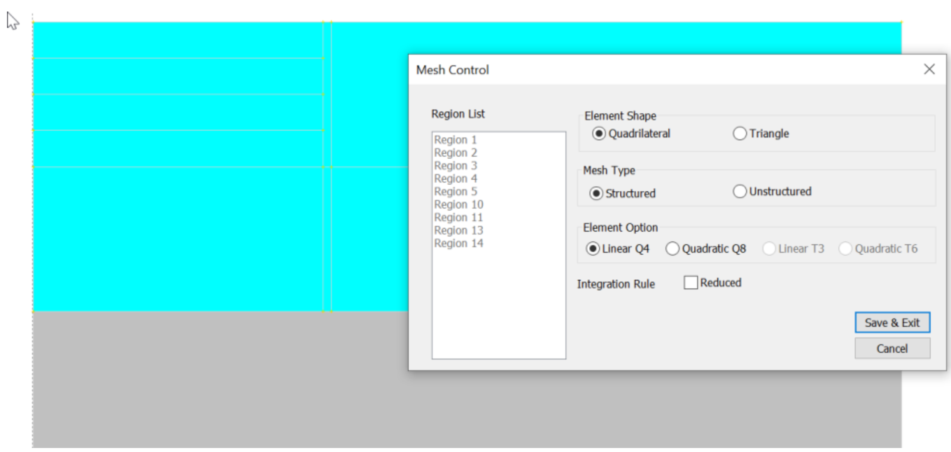


Figure 2.10-23: Mesh control for regions next to wall

Repeat this procedure to define mesh controls for the lower soil region only. Since the lower soil region will not meet the requirements for a structured mesh, an unstructured mesh will be defined for this layer – see Figure 2.10-24. Click **"Save & Exit"** when complete.

From the menu bar, click "**Mesh->Global Seed Control**" to open the dialog to assign the global element size. Change the default size to `“1.5”` and click **"Assign"**. Click **"Close"** to exit.

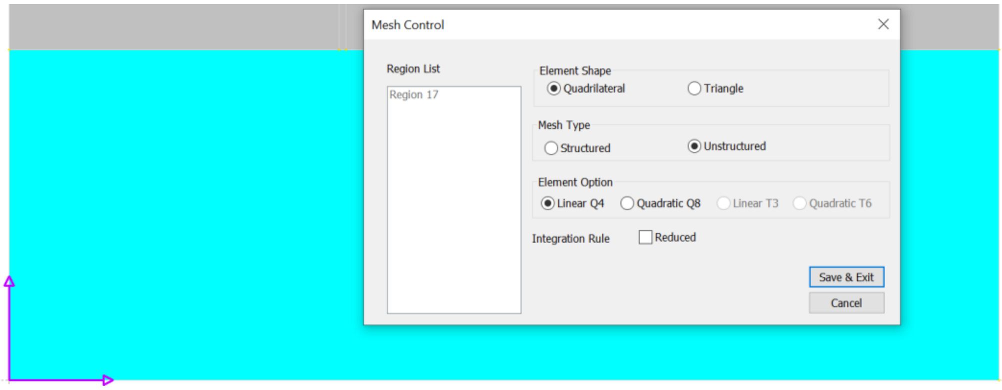


Figure 2.10-24: Mesh control for region below wall

Next, click "**Mesh->Run Mesh Generation**" to generate the finite element mesh, as seen in Figure 2.10-25. Click  to save the file.

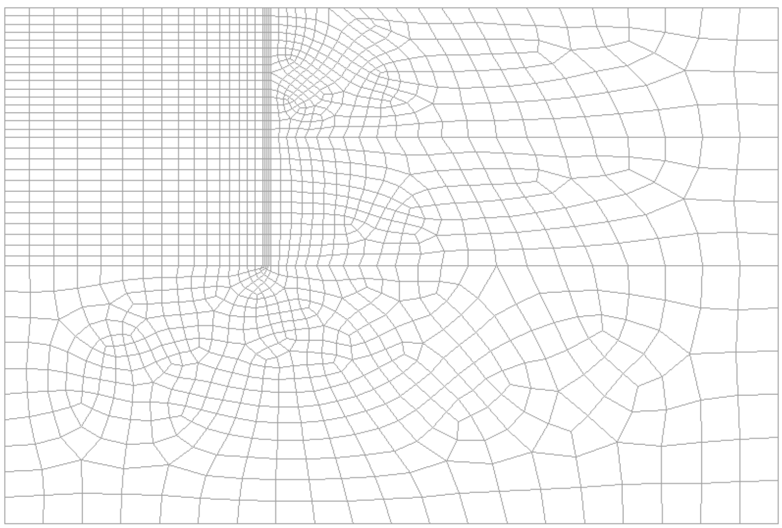


Figure 2.10-25: Fully generated mesh

Step 9: Analysis

From the menu bar, click **"Analysis->Analysis Setting"** to open the dialog for analysis options. Use the settings shown in Figure 2.10-26 for this analysis. Click **"OK"** to exit.

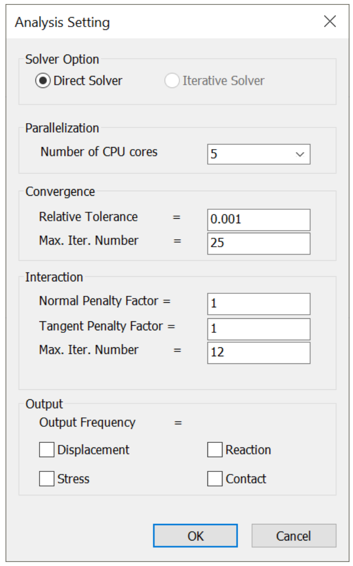


Figure 2.10-26: Analysis settings

From the menu bar, click **"Analysis->Run Analysis"** to open the "Run Analysis" window. Check the square box before each stage (Figure 2.10-27). Unchecked stage will not be submitted to the solver. Click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. The file, *"Excavation and cantilevered retaining wall.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history or error information, if it exists.

NOTE: The button **"Generate Analysis File"** can be clicked to generate the analysis input file without starting an analysis. This may be necessary for troubleshooting and checking the input data file, but should only be done as directed by the EnFEM support team.

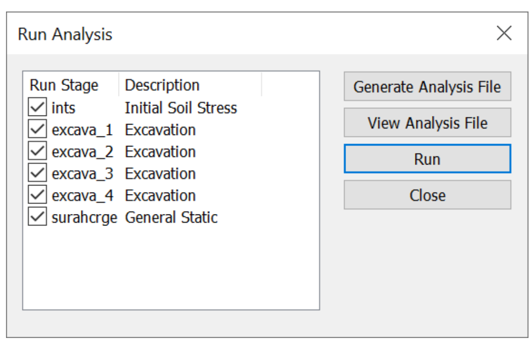


Figure 2.10-27: Run analysis dialog

Step 10: Post-processing of result

Click 

to switch to the result-mode and use the *'Display Control'* dialog (**"Tools->Display Tool"**) to hide the wall regions. Next, click  to open the contour plot dialog. Select "Stress" from the

Group drop-down list and "S22" from Variable drop-down list. Select the first increment, "Ints. incr=1.0", from the Increments list. The contour of the initial vertical stresses should look like those seen in Figure 2.10-28. The vertical stress at the bottom of the soil should agree with to the theoretical value of -720. Notice that in this example, the distribution of initial soil stresses should be uniform in the horizontal direction as expected with a flat soil geometry.

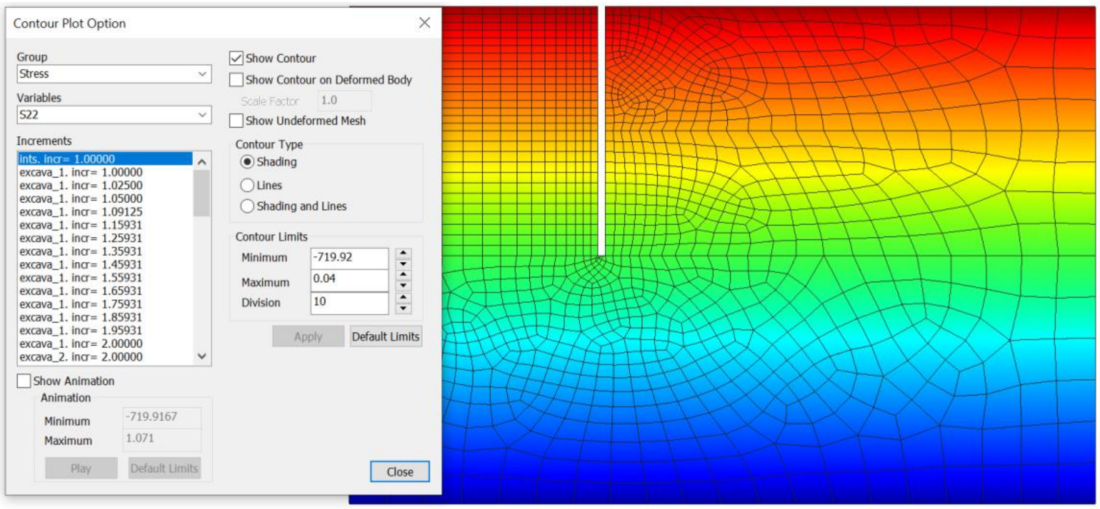


Figure 2.10-28: Contour of vertical initial soil stresses

Close the contour plot and click the cutline plot icon . Input the coordinates of a vertical cutline, like those seen in Figure 2.10-29. Click **"Add"** to add this cutline to the list. Select "Void Ratio" from Field Variable drop-down list and select the first increment "Ints. incr=1.00". Keep all other settings unchanged. Click **"Plot"** to display the distribution of the initial void ratio with depth, as seen in Figure 2.10-29. This agrees with exponential function of the initial void ratio, in theoretical terms.

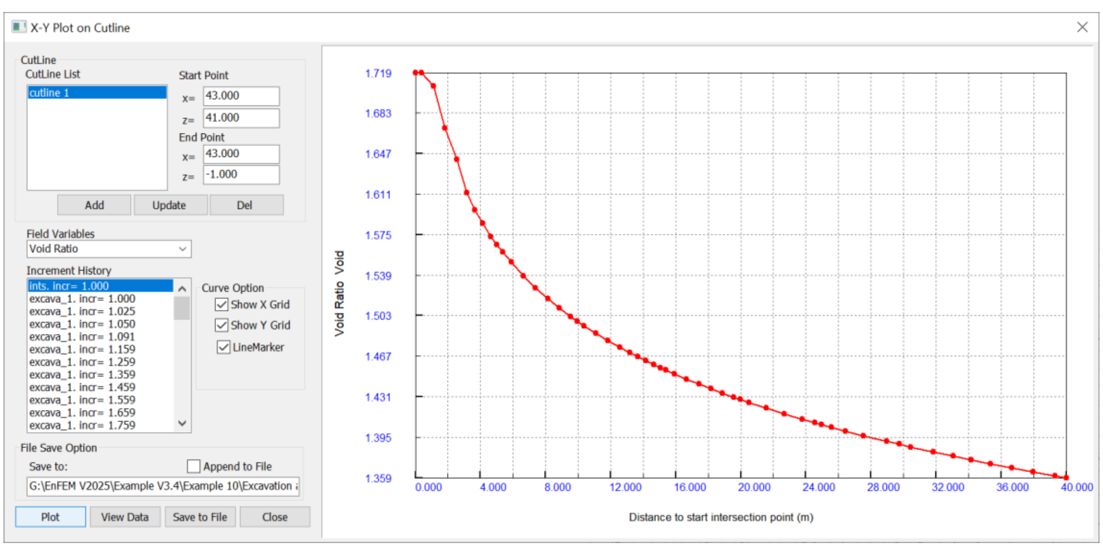


Figure 2.10-29: Plot distribution of initial void ratio with depth

Click to switch back into the 'CAD-MODE'. Hide the display of the retaining wall and create a new edge-set (**"Tools->Set Tool"**) called *"setSoil_heave"*. Select the final edge of excavation surface, as seen in Figure 2.10-30. Define another edge set named *"setCT1"*, which includes right-hand side soil edges of the contact interface (Figure 2.10-31).

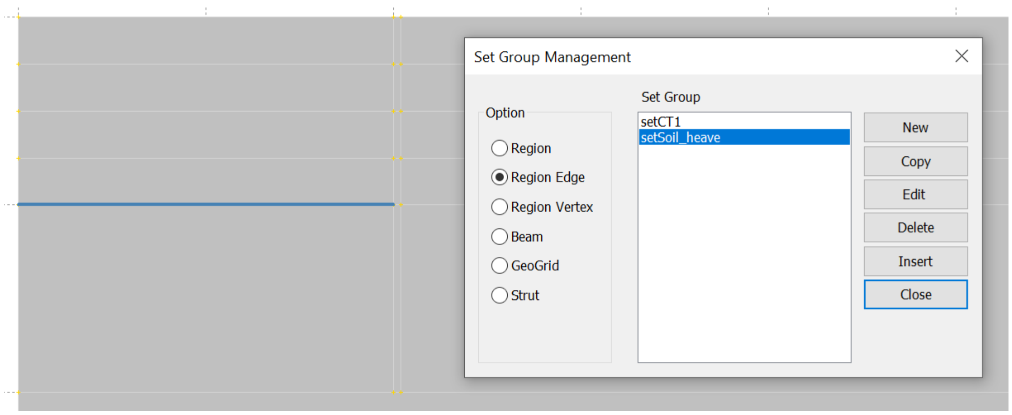


Figure 2.10-30: Define set of final excavation surface

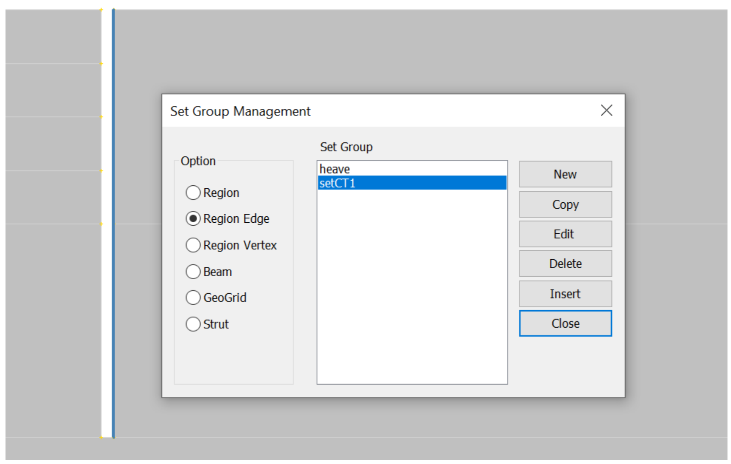


Figure 2.10-31: Define set of contact interface

Click to go back into the result window. Next click to open the X-Y plot dialog. Select the option "Field Result", select "By Set", and select the previously defined edge-set "*setSoil_heave*" as the source and click **"OK"**. Select "Primary variable" as the Category and "u2" as the Variable. Finally, select the last increment and click **"Plot"**. The soil heave on the excavation surface after all excavation are completed, is shown in Figure 2.10-32. Note that nodal distance (X-axis), as determined by the software, starts from soil-wall contact interface.

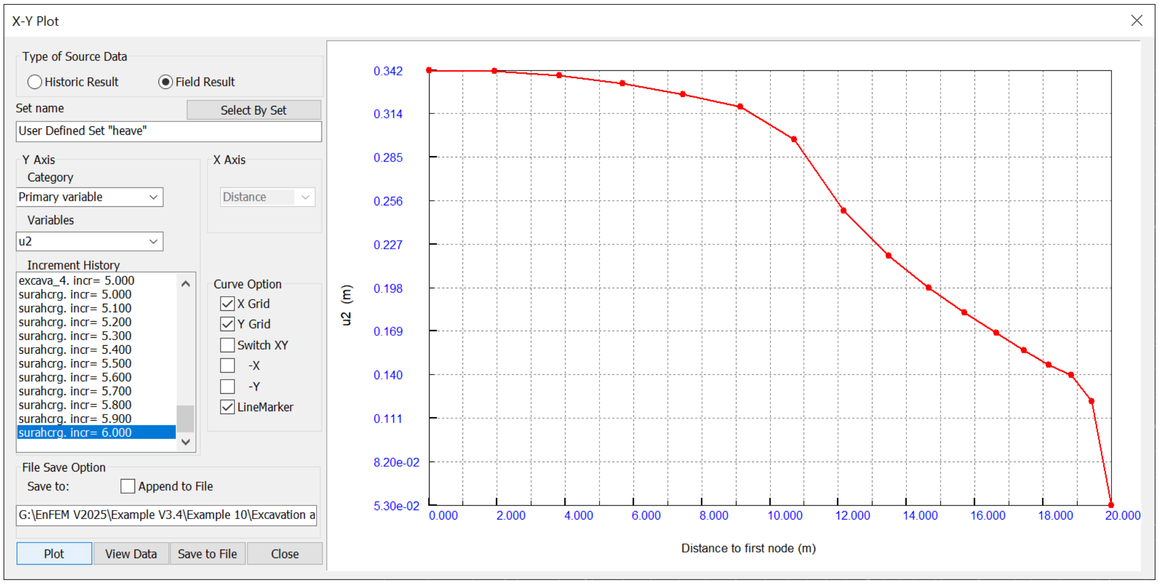


Figure 2.10-32: Soil heave on excavation surface

Next, again click "By Set" and this time select edge-set "`setCT1`". Select "Interaction" as the Category and "Normal Pressure" as the Variable. Select the last increment and click "**Plot**". The active soil pressure supported by the retaining wall is seen in Figure 2.10-33.

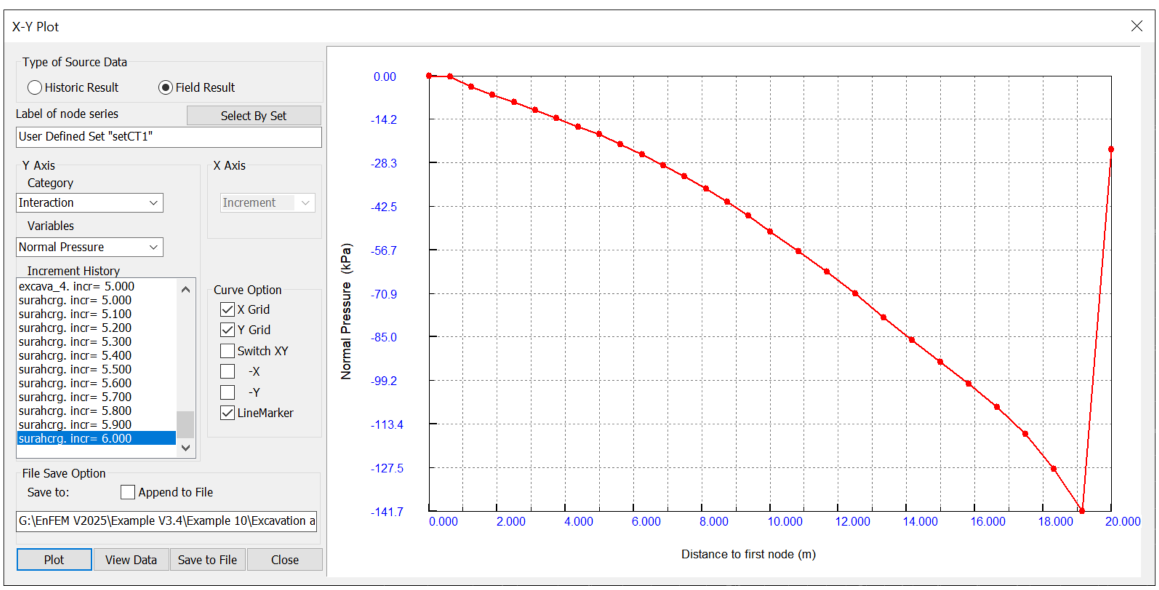


Figure 2.10-33: Active soil pressure on retaining wall VS depth

NOTE: The jump seen in the normal pressure at the last nodal point is caused by the sharp corner of the retaining wall. This jump can be reduced by using 'rounded' tip (half-circle region) at the bottom of the wall, as seen in other examples, to smooth the contact interface.

Next, use the display tool to display only the retaining wall. Turn on the display of the node numbers. Keep the "Field Result" option and find the node list on the middle of the wall:

1568:1488:-5,1628:1573:-5

Select "Integrated Force" as the Category and "Moment" as the Variable. Select the last increment from the increment list and click **"Plot"**. The plot of the Moment vs Depth of the retaining wall is seen in Figure 2.10-34. The maximum moment appears at approximately 1-m below the excavation surface.

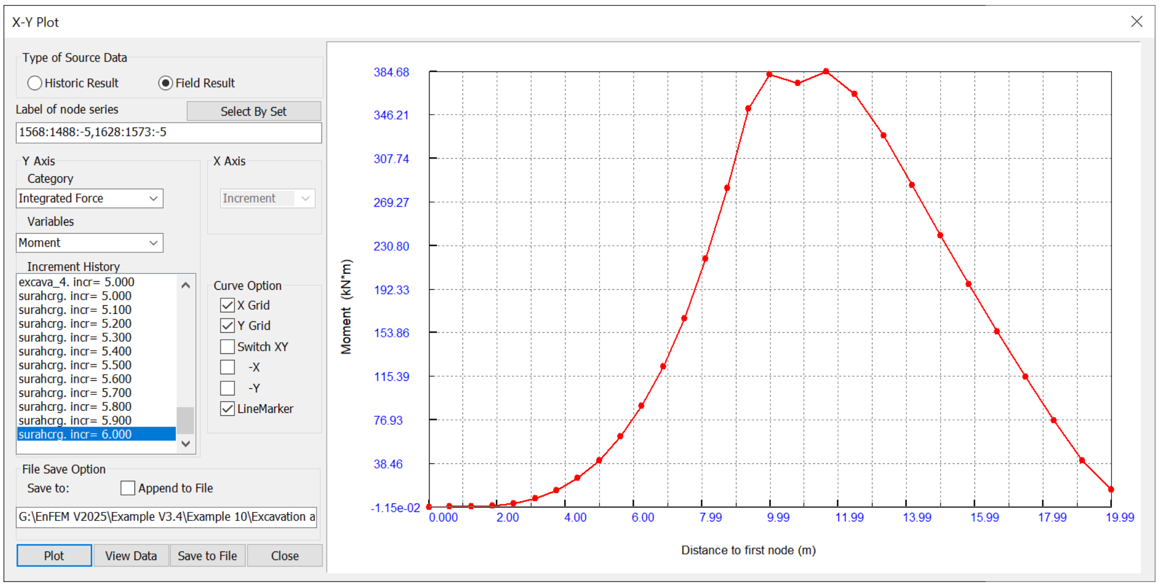


Figure 2.10-34: Moment along retaining wall after excavation

<u>NOTE:</u> If the user wishes to plot nodes along the centerline, an alternative method, and good practice, is to create the geometry such that there is a 'physical' line at this location. For this example, that would require splitting the retaining wall regions into two rectangular regions, rather than one – as shown above. That way an edge-set can be defined easily and then selected for plotting. This method would be especially helpful is a very fine mesh is used.

2.11 Example 11: Excavation on inclined soil

In this example, an excavation on uneven ground is simulated. Flat ground makes it possible to interpolate initial soil stress with the k_0 method as demonstrated in the previous example. However, if the ground is not flat, or the soil layers are not flat, interpolation method is not applicable because it cannot generate initial soil stresses correctly. This example presents a second approach to generate initial soil stresses from an external linear elastic analysis, and then using this external file as the source of initial soil stresses.

The geometry is shown as in Figure 2.11-1. The right side has a surface elevation that is 2.5-m higher than the left side. The new model can be obtained by modifying the model in previous example. Open the previous example (*Excavation and cantilevered retaining wall.sep)* and select **"File->Save As"**. Save the new file using the filename *"Excavation on inclined soil.sep"*.

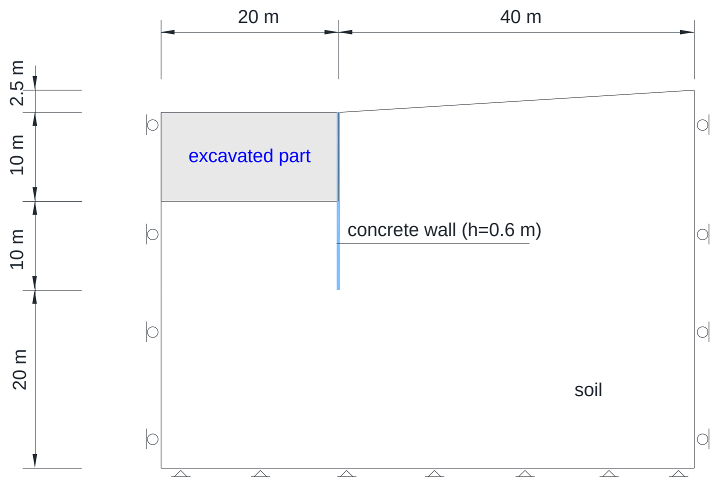


Figure 2.11-1: Excavation on inclined soil ground

Step 1: Modify geometry, stage data, and regenerate mesh

Switch to the "CAD-MODE" () if necessary. From the menu bar, click **"Edit->Edit Vertex Coordinate"**. Use the 'snap method' to select vertex "*A17*" (see Figure 2.10-2), and then right click and select **"End Selection"**. In the newly opened dialog, check the option **"**Apply to Z**"** and change the Z-coordinate to *"42.5"*, as shown in Figure 2.11-2.

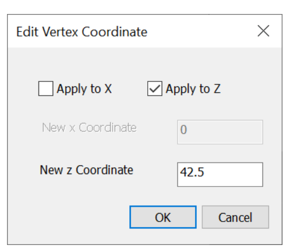


Figure 2.11-2: Edit vertex coordinate to raise right-hand soil vertex elevation

After modification, the new model should look like those seen in Figure 2.11-3.

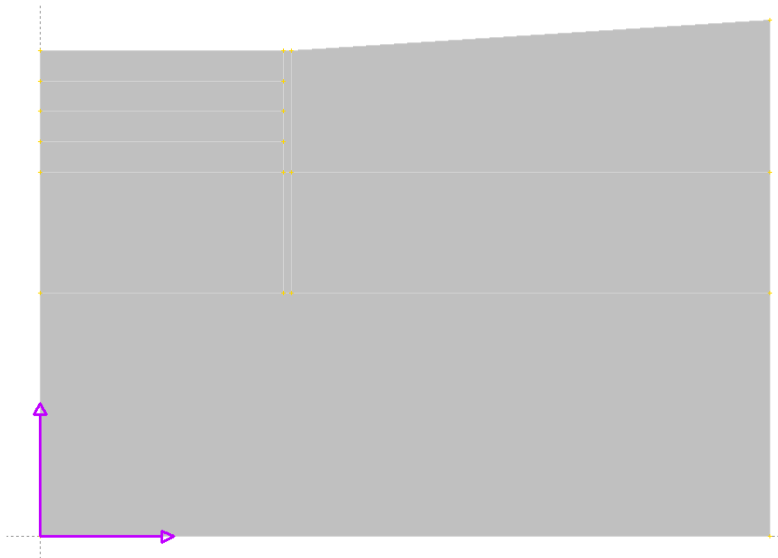


Figure 2.11-3: Modified soil layers

Click to save the file.

Step 2: Modify material properties for generating initial stress, and regenerate mesh

First, select **“File->Save As”** again and save the new file under the filename *"Excavation on inclined soil ints.sep"*. This file will be used to generate initial soil stresses.

From the menu bar, click **"Data->Stage Management"**, to open the dialog for stage group management. Select the first stage "*ints*" and click **"Edit"**. Change the stage type from “Initial Soil Stress” to "General Static" (Figure 2.11-4).

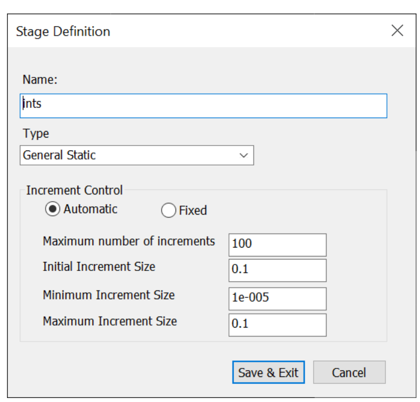


Figure 2.11-4: Modify type of first stage in example 11 to generate initial soil stress

Materials, boundary conditions and contact interfaces are kept as before without change. Click **"pre-Mesh->Run Mesh Generation"** to regenerate the mesh for the new geometry (Figure 2.11-5).

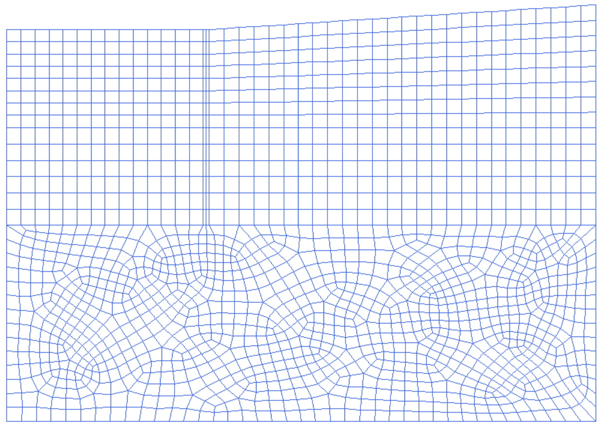


Figure 2.11-5: Regenerated mesh for the inclined soil layers

Click  to save the file.

Step 3: Analysis of initial stress file

From the menu bar, click **"Analysis->Analysis** Setting", click **"Analysis->Run Analysis"** to open the "Run Analysis" window. Uncheck all stages except the first one **"**ints**"**, as seen in Figure 2.11-6. Click **"Run"** to submit the analysis job to the solver and perform the calculation for the first stage only.

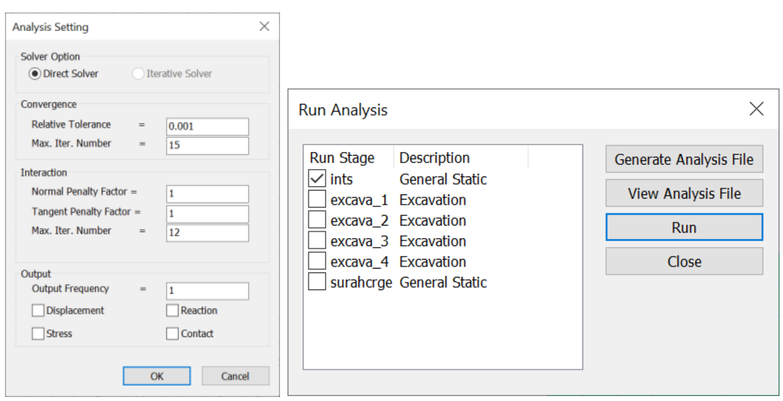


Figure 2.11-6: Select/Analyze the first stage only

Step 4: Post-processing of initial stress file

Click  to switch to the "RESULT-MODE". From the menu bar, click "**Results->Initial Soil Stress Output**" to open the dialog as seen in Figure 2.11-7. Click "**Select Element Set**" and select the set “`setSoil_all`”. Click “OK” to exit from set selection dialog. In the initial stress output window, click symbol **“+”** to expand the increment selection, and select the last increment, **“**1.0**”**, of stage *"ints"*. The default output file name is *‘***.ints’* where *** is the same as the project name. If necessary, change the output file name and path, which is located at the bottom of the dialog. Click **"Output"** to write the results to the specified file. A dialog message will appear to indicate the data file was saved. Click **"Close"** to exit.

The stress file locates under the same directory as the model file. Use Windows Explorer to search for this file (in this case *“Excavation on inclined soil ints.ints”)*. Then open it using any text editor as shown in Figure 2.11-8. Stress components S11,S22,S33 and S12 are output at each integration point.

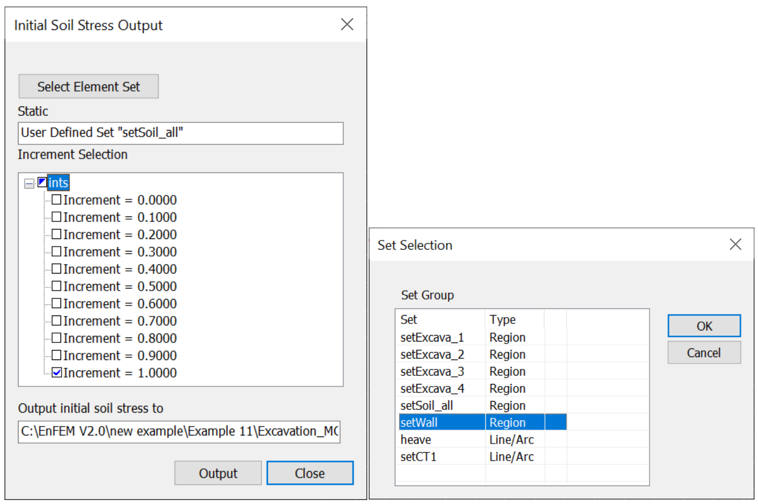


Figure 2.11-7: Use report output to write stresses to an external file

4

1	1	-4.66522338	-11.19268378	-0.05482173941	-5.550267506
1	2	-4.655575694	-11.17476665	0.02042888691	-5.54061982
1	3	-4.950548957	-11.34632063	-0.05843962534	-5.703904355
1	4	-4.94090127	-11.3284035	0.01681100097	-5.694256669
2	1	-17.15094643	-33.78477486	-0.07203424827	-17.82750245
2	2	-17.14316251	-33.770319	-0.02738796357	-17.81971853
2	3	-17.32023043	-33.87592778	-0.07495322128	-17.91865537
2	4	-17.31244651	-33.86147193	-0.03030693657	-17.91087145
3	1	-4.763344119	-11.2141765	-0.08735943451	-5.592132217
3	2	-4.734416245	-11.16045331	-0.01567854191	-5.563204343
3	3	-5.035134442	-11.36052514	-0.09820739811	-5.738480853
3	4	-5.006206568	-11.30680194	-0.02652650551	-5.709552979
4	1	-17.23054308	-33.80226912	-0.1695119102	-17.86148427
4	2	-17.20704076	-33.75862196	-0.1281559262	-17.83798195
4	3	-17.38735134	-33.88670434	-0.1783252886	-17.94591949
4	4	-17.36384902	-33.84305718	-0.1369693045	-17.92241717
5	1	-4.949290675	-11.2389933	-0.1177078331	-5.66589939
5	2	-4.9012269	-11.149732	-0.05316067688	-5.617835615
5	3	-5.194032221	-11.37077721	-0.1357317669	-5.7976833
5	4	-5.145968445	-11.28151591	-0.07118461058	-5.749619524

Figure 2.11-8: File of Initial soil stress

Back in EnFEM, click to save the file.

Step 5: Use external file to generate initial stress

After the initial soil stress file has been generated successfully, go back to open the original analysis model. Click **“File->Open”** and select the original analysis file, *" Excavation on inclined soil.sep".*

In “CAD-MODE”, click **"Data->Stage Management"**, select the first stage **“**ints**”**, and click **“Edit”**. Click **"Initial Stress Option"** to open the dialog for initial soil stress options. Select the second option **"External File"** and click **“Browse”** to select previously generated stress file ***.ints (Figure 2.11-9). Click **"OK"**, then **“Save & Exit”**, and then **“Close”** to complete the stage edit.

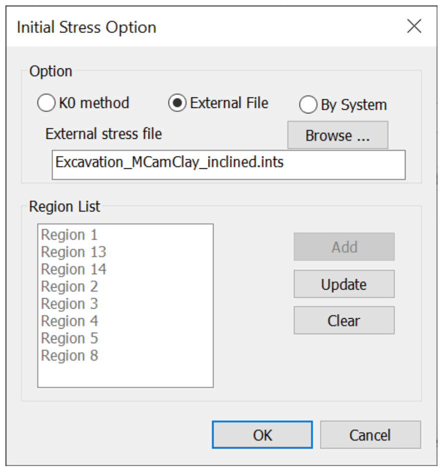


Figure 2.11-9: Define an external file as the source of initial soil stress

Click to save the file.

Step 6: Mesh generation and analysis

There is no need to change any material properties or assignments, boundary conditions or assignments, or the interaction definitions. However, since the geometry was changed and a mesh was never regenerated, a new mesh must be created before this analysis is run.

Click **"pre-Mesh->Run Mesh Generation"** to regenerate the mesh for the new geometry. Since geometric entities and mesh assignments are not changed. The newly generated mesh should be the same as that one in Figure 2.11-5. Click to save the file.

<u>NOTE</u>: Since initial soil stresses are input from an external file, meshes in both models should be identical. If the original model “*Excavation on inclined soil.sep*” has any change in geometry or mesh assignment (mesh type, element shape, seed), the initial soil stress file should be regenerated as described in steps 2~step6.

From the menu bar, click **"Analysis->Run Analysis"** to open the "Run Analysis" window. Make sure each stage is selected to run (Figure 2.11-10) and click **"Run"** to submit the analysis job to the solver.

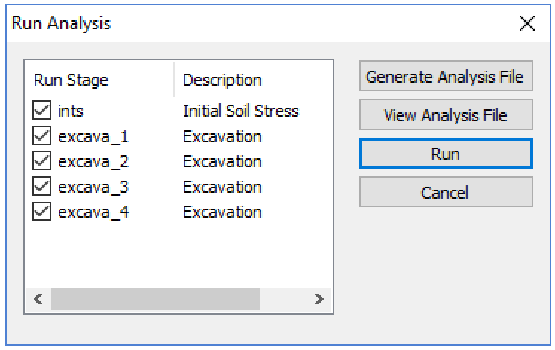


Figure 2.11-10: Check all stages and run analysis

Step 7: Post-processing of result

Click to switch to the "RESULT-MODE" and then click to open the contour plot dialog. Select **"**Stress**"** from the Group drop-down list and select **"**S22**"** from the Variables drop-down list. Select the interval **"**ints. Incr=1.00**"**. Notice that the initial stresses follow the profile of the incline, as opposed to the uniformity of the flat geometry, until well below the bottom of the retaining wall.

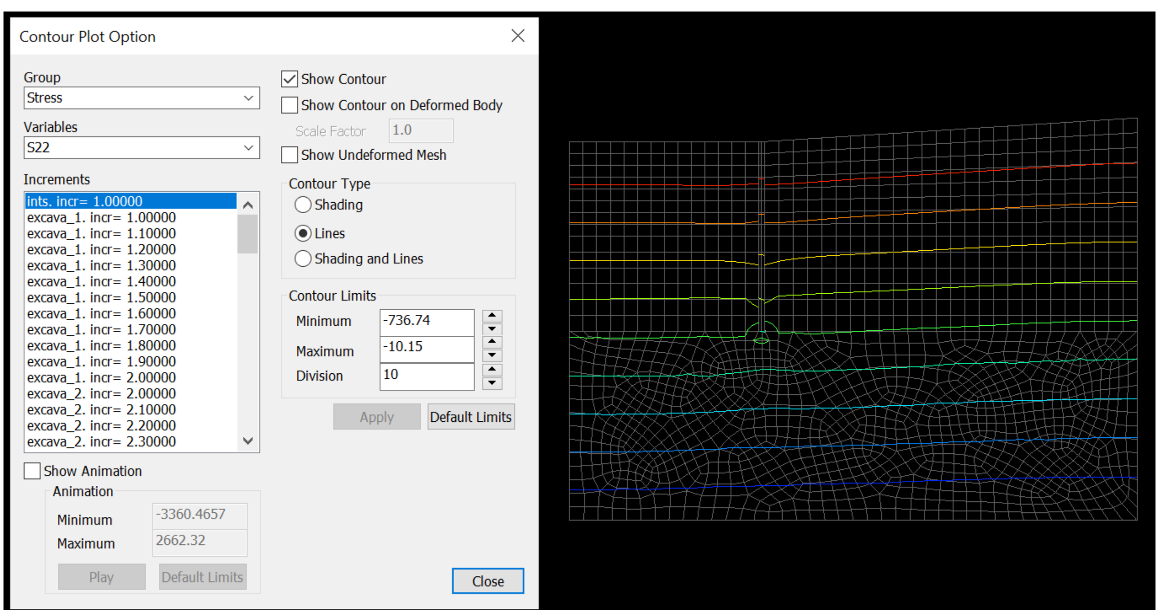


Figure 2.11-11: Initial stress of inclined soil model

2.12 Example 12: Investigation of design loads on a box culvert

In this example, design loads on a buried box culvert are investigated. Some useful skills to simplify model development are introduced. English units are used in this example – length in units of inches, and pressure in units of "psi".

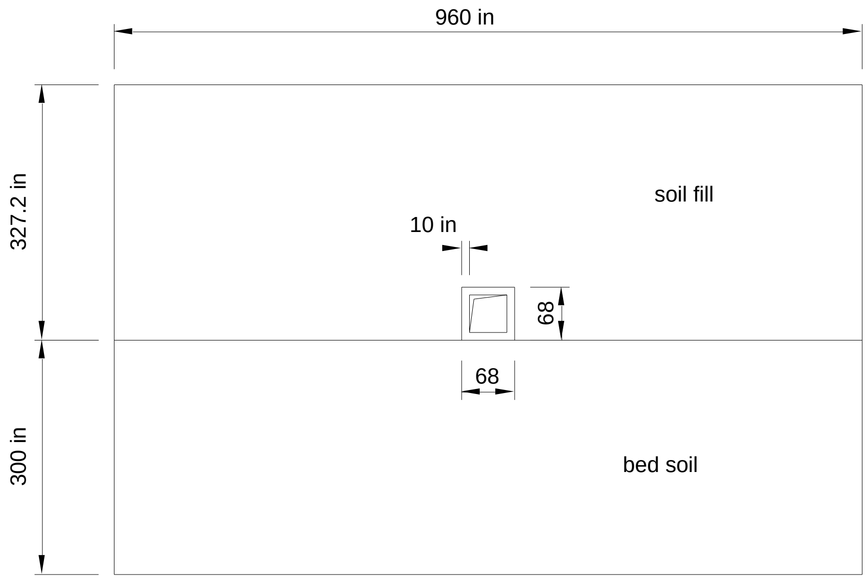


Figure 2.12-1: Geometry of the box culvert

The geometry of the box-culvert is shown in Figure 2.12-1. The geometry is symmetric about the Z-axis (vertical) of the culvert. However, the full geometric model is used to investigate the influence of an unsymmetrical mesh. The culvert is made of concrete which has 4-ft by 4-ft internal opening and a 10-in wall thickness. Soil will be simulated by Mohr-Coulomb model. Soil-structure-interaction is simulated by means of contact constrained method.

Material property:

Concrete (linear elastic):

- Unit weight:... $\gamma = 0.087\ lb/in^3$
- Young's Modulus:.. $E = 3600000\ lb/in^2$
- Poisson's ratio:.. $\nu = 0.2$

Soil fill (Mohr-Coulomb):

- Unit weight:... $\gamma = 0.069\ lb/in^3$
- Young's Modulus:.. $E = 2000\ lb/in^2$
- Poisson's ratio:.. $\nu = 0.4$
- Cohesive strength:..................................... $c = 5.5 lb/in^2$
- Friction angle:... ϕ=14°
- Dilation angle:... ψ=1°

Bed soil (Mohr-Coulomb):

- Unit weight: $\gamma = 0.072\ lb/in^3$
- Young's Modulus: $E = 15000\ lb/in^2$
- Poisson's ratio: $\nu = 0.3$
- Cohesive strength: $c = 0.1\ lb/in^2$
- Friction angle: ϕ=35°
- Dilation angle: ψ=1°

Boundary Conditions:

- Displacement constraints on vertical external soil edges
- Displacement constraints on bottom edge of bottom soil layer
- Gravity load
- Contact interface between soil and culvert.
    - The culvert is the master body and the soil is the slave body.
    - Coefficient of friction on the contact interface is 0.4

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM, or Click **"File->New"** to start a new project. Change the title to "`Design loads on buried square culvert`". Select preselected unit "lbf, in, second" (Figure 2.12-2).

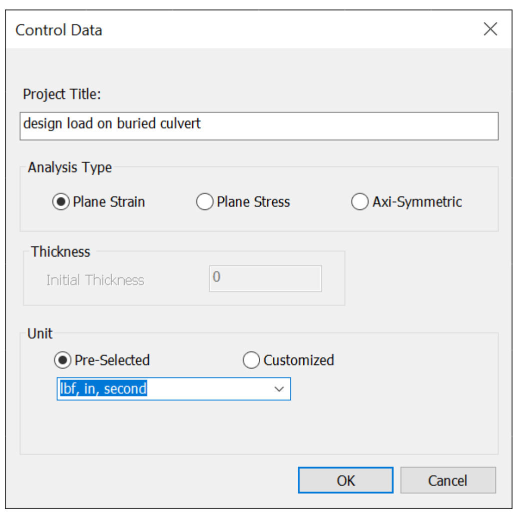


Figure 2.12-2: Control data in example 12

The cross section of the culvert can be divided into four rectangles. On a rectangle region, structured mesh is always preferred. Structured mesh can maintain very good element shape and total element number can be minimized without sacrifice of accuracy. Start EnFEM, or click **"File->New"** to start a new project. Set the grid spacing (**"Options->Grid Spacing"**) to be "*20*" in both X and Z-directions. Set the working zone ranges (**"Options->Drawing Space"**) from "*-600*" to "*600*" in the X-direction and from "*-400*" to "*400*" in the Z-direction.

Click  to create the bottom rectangle of the culvert. Key in *"-34,0"* as the left-lower corner of the rectangle, then press RETURN to create the vertex. Key in coordinate of the diagonal vertex as *"[68,10]"*. Press RETURN to complete the first rectangle (Figure 2.12-3). Here relative coordinates are used by means of square bracket [].

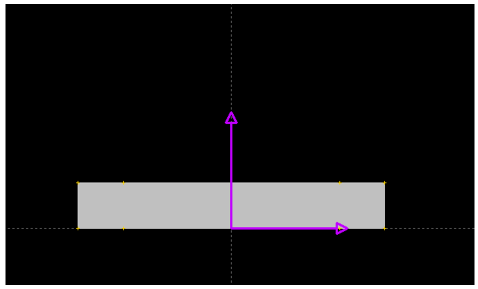


Figure 2.12-3: Create bottom section of box culvert

From the menu bar, click **"Edit->Copy to"** and then select the bottom rectangle, RIGHT-CLICK and select **"End Selection"**. A new dialog window will be opened up to define the copy-vector as in Figure 2.12-4. Define this vector as seen in the figure, and click **"OK"** to finish the operation. The finished 'copy' should look like that seen in Figure 2.12-4.

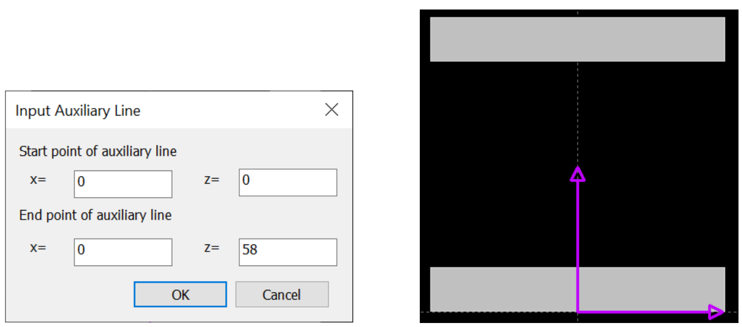


Figure 2.12-4: Use "Copy to" to create top section of the box-culvert

Click  to start a new rectangle. Use the 'snapping method' to create the first vertex at "A"(left image in Figure 2.12-5). Key in the relative coordinate *"[10,-48]"* to complete the third rectangle. After one new region is created, ENFEM will check geometry compatibility among regions. Here extra vertices are added on the existing rectangles automatically to maintain geometry compatibility (right image in Figure 2.12-5). Similarly, create the fourth rectangle on the right of the culvert.

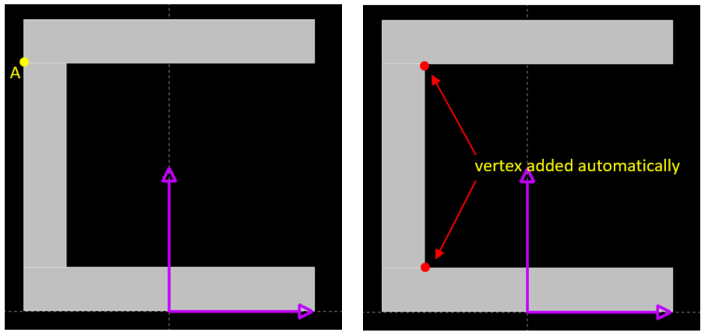


Figure 2.12-5: Completed section of the box-culvert

From the toolbar, click to activate the polygon creation tool to create the top soil layer. Combine the 'snapping method' and key in coordinates, this soil region can be built quickly(Figure 2.12-6). Only corner vertices of the culvert are necessary to pick up by 'snap' method. Intermediate vertices, like "A" in Figure 2.12-5, will be inserted automatically by EnFEM.

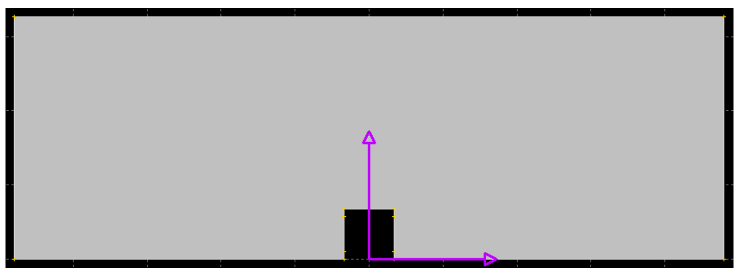


Figure 2.12-6: Create the polygon region for soil fill

Click to create the bottom soil layer by means of two diagonal vertices (x=-480, z=0) and (x=480, z=-300). The top edge of the new region has been inserted two extra vertices to keep geometry compatibility with the culvert (Figure 2.12-7).

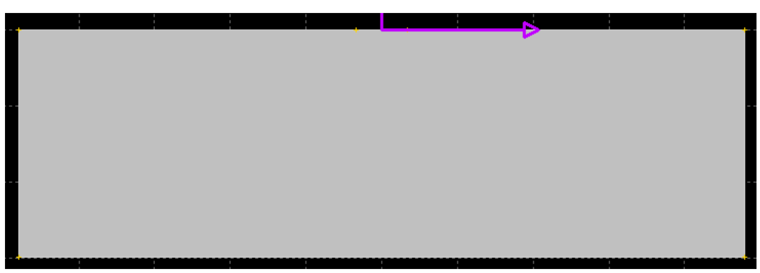


Figure 2.12-7: Create the polygon region for bed soil layer

Click to save the model file as "`Design load on buried box culvert.sep`".

Step 2: Define region sets

Click **"Tools->Set Tool"** to create the region and edge-sets as seen in Figure 2.12-8. Create the following three region-sets: "`setCulvert`", which includes all square and rectangular regions of the culvert; "`setFill`", for the top soil layer region; and "`setBed`", for the bottom soil layer region. Use the *'Display Tool'* (**"Tools->Display Tool")** to hide the culvert. Back in the *'Set Definition Dialog'* select the **"Edge"** option and create four edge-sets as seen in Figure 2.12-9. Note that these edge-sets should be defined on the soil bodies, as they are to be used for outputting results of the contact interaction, and interaction-specific results are only available on the *'slave'* bodies.

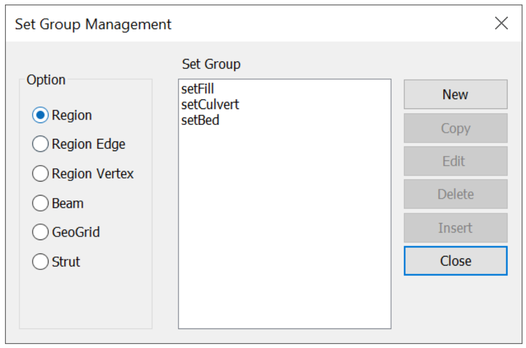


Figure 2.12-8: Create region and edge-sets

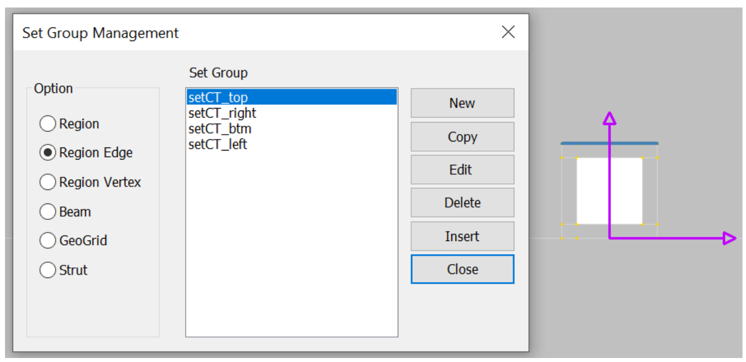
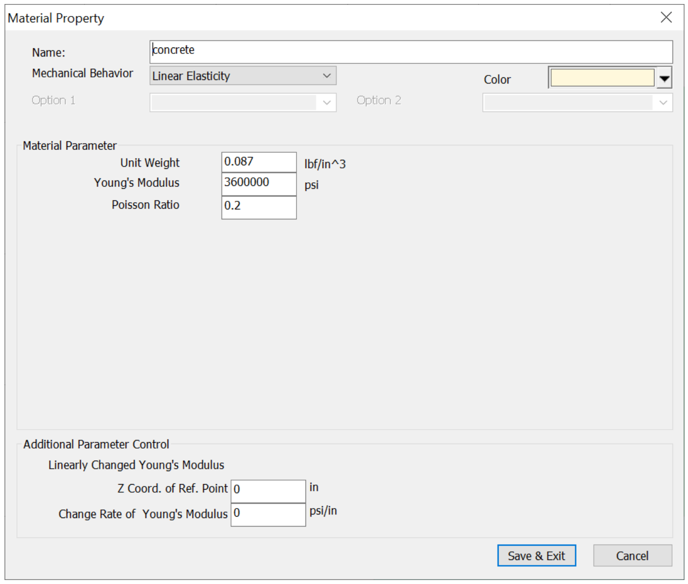


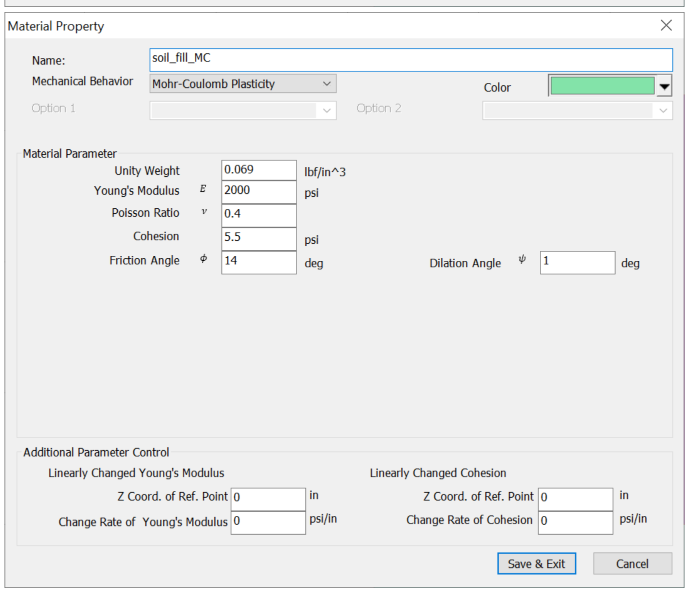
Figure 2.12-9: Create four edge sets on soil regions for contact output

Step 3: Define control data, stage data, material properties, and boundary conditions

From the menu bar, click **"Data->Stage Management"**, to define a stage, named "`stage1`", and of the type "General Static". Keep the default increment setting and click **"Save & Exit"** when complete.

From the menu bar, click **"Data->Material Properties"** to open the material group management dialog. Define three new materials named as those seen in **Error! Reference source not found.**. Enter in the details of each material as seen in Figure 2.12-10.


Material Property
Name:
concrete
Mechanical Behavior
Linear Elasticity
Color
Option 1
Option 2
Material Parameter
Unit Weight
0.087
lbf/in^3
Young's Modulus
3600000
psi
Poisson Ratio
0.2
Additional Parameter Control
Linearly Changed Young's Modulus
Z Coord. of Ref. Point
0
in
Change Rate of Young's Modulus
0
psi/in
Save & Exit
Cancel


Material Property
Name:
soil_fill_MC
Mechanical Behavior
Mohr-Coulomb Plasticity
Color
Option 1
Option 2
Material Parameter
Unity Weight
0.069
lbf/in^3
Young's Modulus
E
2000
psi
Poisson Ratio
ν
0.4
Cohesion
5.5
psi
Friction Angle
φ
14
deg
Dilation Angle
ψ
1
deg
Additional Parameter Control
Linearly Changed Young's Modulus
Z Coord. of Ref. Point
0
in
Change Rate of Young's Modulus
0
psi/in
Linearly Changed Cohesion
Z Coord. of Ref. Point
0
in
Change Rate of Cohesion
0
psi/in
Save & Exit
Cancel

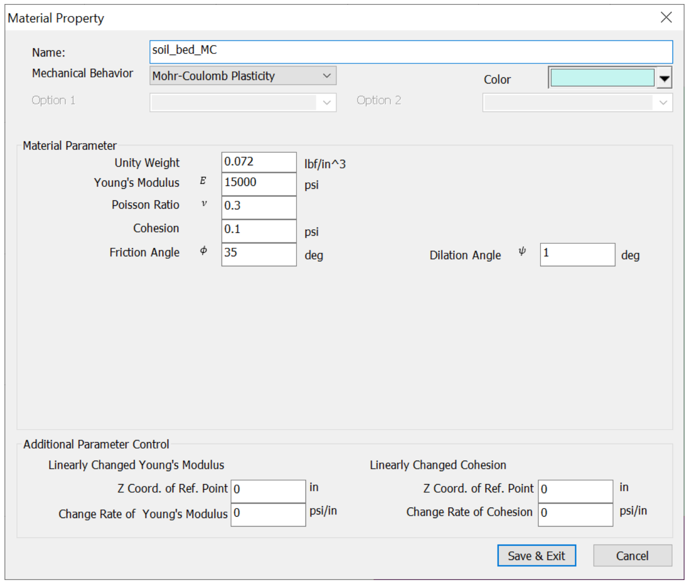


Figure 2.12-10: Material properties for the box-culvert analysis

Next, click **"Data->Material Assignment"** from the menu bar to open the material assignment dialog. Assign **"**concrete**"** to the culvert regions, **"**soil_fill_MC**"** to the upper soil layer, and **"**soil_bed_MC**"** to the lower soil layer. Note that using previously defined region-sets, may aide with this assignment.

From the menu bar, click **"Data->Boundary Conditions"** to open the dialog for the boundary condition group management. Create three new boundary conditions, as seen in the images of Figure 2.12-11.

Click **"Data->Boundary Assignment"** to assign B.C.'s to their corresponding geometric entities. Assign **"***gravity***"** to all of the geometric regions, assign **"***disp_x***"** to the external vertical edges of the soil regions, and assign **"***disp_xz***"** to the bottom edge of the lower soil layer (Figure 2.12-12).

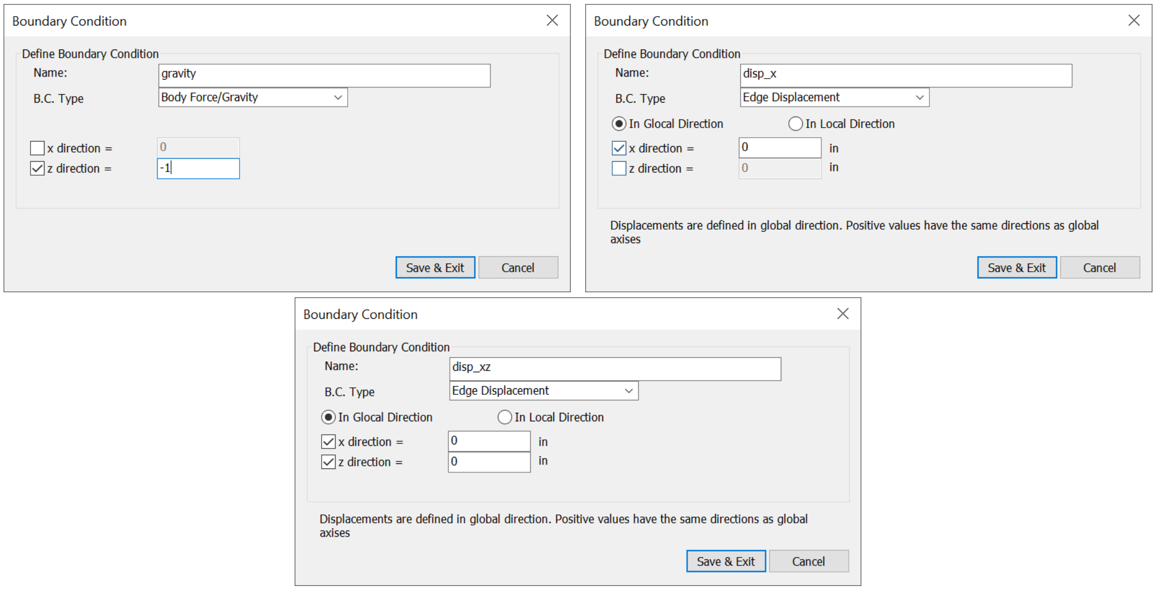


Figure 2.12-11: Create displacement and gravity load boundary conditions

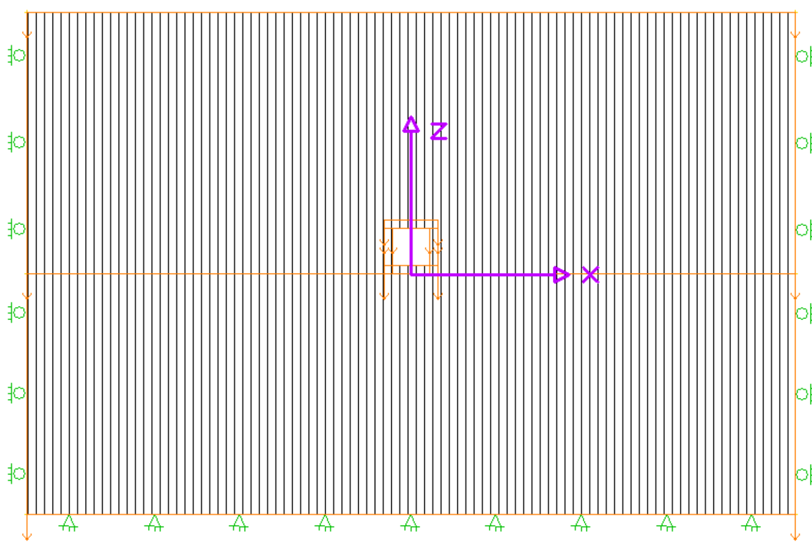


Figure 2.12-12: Assign displacement and gravity load boundary conditions

Step 4: Define soil-culvert interaction property

From the menu bar, click **"Tools->"Display Tool"** to open the “Display Control” dialog, and hide the soil layers. Keep the display control dialog open, and from the menu bar, click **"Data->Interaction"** to open the interaction group management dialog. Click **"New"** to create a new interaction definition. Keep the default group name, the default **"contact"** as the interface option, and change the coefficient of friction to “0.4”. Click **"Add Master Surf."**, LEFT-CLICK to select all of the external edges of the culvert, and then RIGHT-CLICK and select **“End Selection”**. The master surfaces will be highlighted in blue. Now display only the soil layers, and similarly define the corresponding slave-edges on the soil (Figure 2.12-13). Click **"Save & Exit"** to return to the interaction group management dialog. Click **"Close"** to exit, and click to save the file.

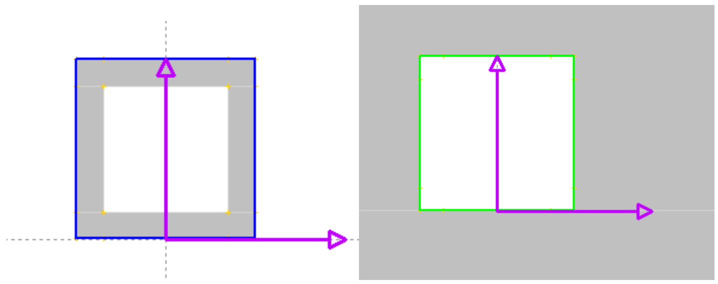


Figure 2.12-13: Define master surfaces on culvert(blue) and slave surfaces on soil (green)

Step 5: Mesh generation

First, click the icon and ensure all regions of soil layers and the culvert are displayed. Then from the menu bar, click **"pre-Mesh->Advanced Mesh Control"**, and define unstructured quadrilateral meshes on two soil regions, as seen in Figure 2.12-14. Click **"pre-Mesh->Advanced Mesh Control"**, and define four structured quadrilateral meshes for all culvert regions. Since a structured mesh has strict requirement on the shape, the user may need to define the structured mesh by individual selection, as seen in Figure 2.12-15. Linear quadrilateral elements Q4 with reduced integration are selected for the culvert regions.

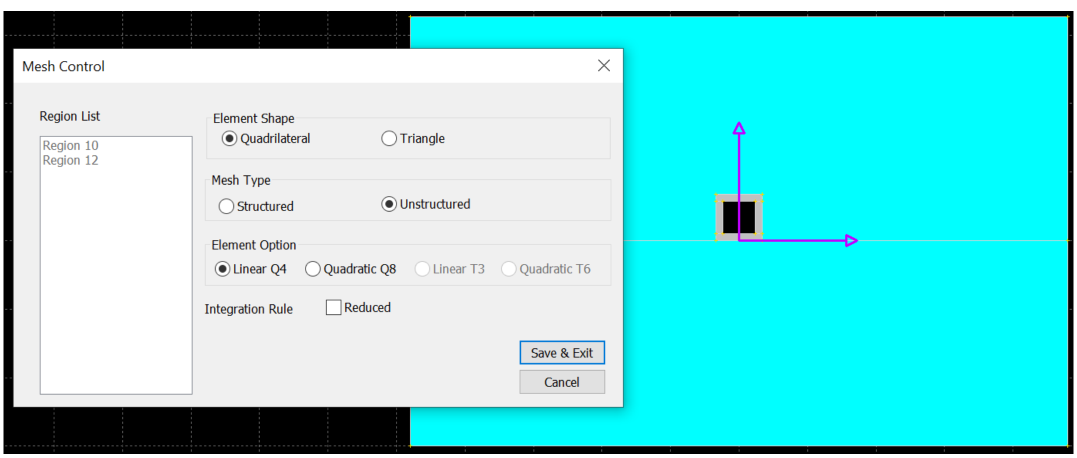


Figure 2.12-14: Mesh control for soil layers in Example 12

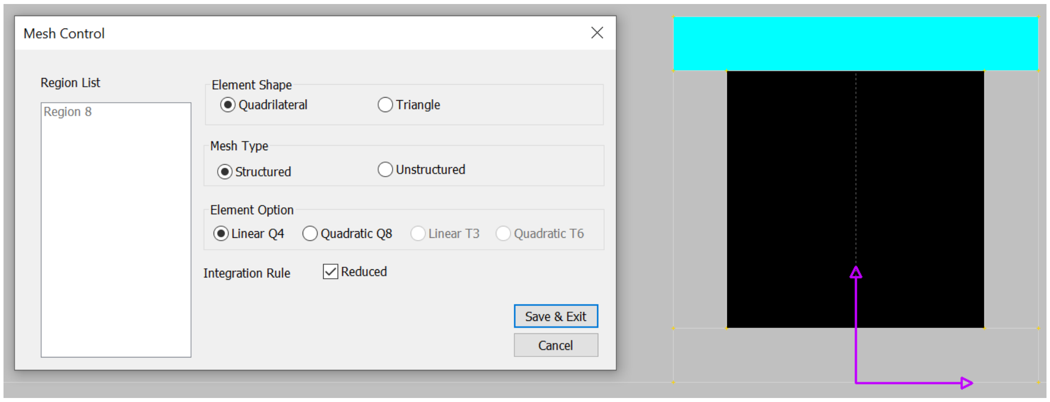


Figure 2.12-15: Mesh control for culvert regions in Example 12

In this example, the algorithm of mesh size control is to define fine mesh around the culvert, while the mesh can be coarse at location far from the culvert. From the menu bar, click **"pre-Mesh->Global Seed Control"** and enter "*32*" as the global seed size. This size will be used to define element size on the external boundaries of soil layers (Figure 2.12-16).

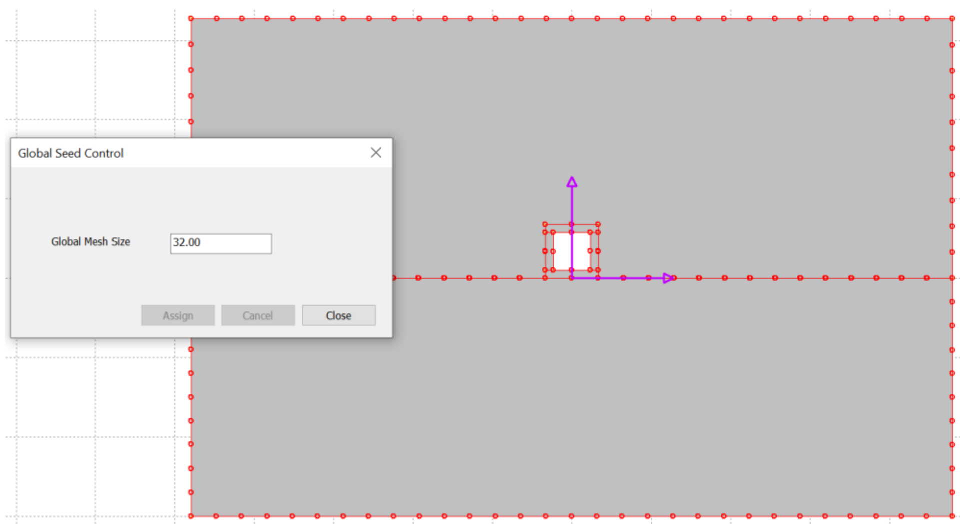


Figure 2.12-16: Use coarse mesh for external edges

Next, elements around the culvert will be refined and smooth transition from fine mesh to coarse mesh is expected. Click "**pre-Mesh->Local Seed Control**", select the top and bottom edges of the culvert. RIGHT-CLICK and select "**End Selection**". Keep the default "**By Number**". Input the division 20 as shown in Figure 2.12-17. Similarly, define element layers on the four culvert regions as "*4*" by selecting controlling edges as shown in Figure 2.12-18.

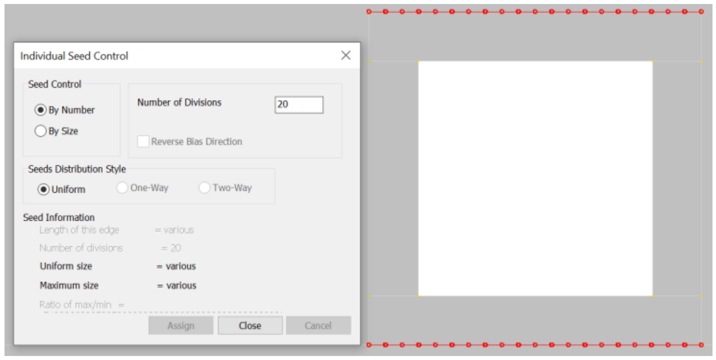


Figure 2.12-17: Local seed control on top/bottom of the culvert

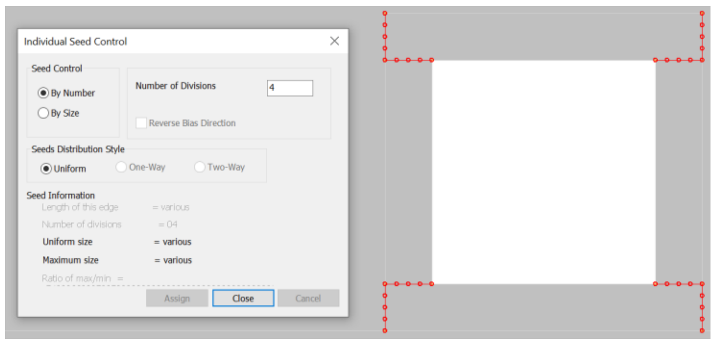


Figure 2.12-18: Define four element layers on regions of culvert

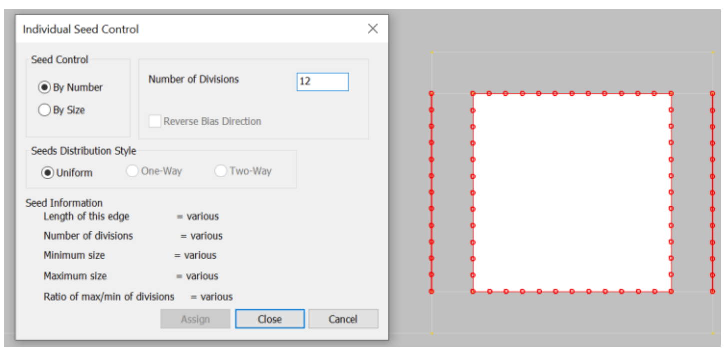


Figure 2.12-19: Define division on remaining edges of culvert

The last step is to define division number on remaining edges of the culvert to satisfy equal divisions on opposite edges of a structured mesh (Figure 2.12-19).

In order to create a high-quality mesh, the element size should transition smoothly from coarse to fine areas. Click **"pre-Mesh-> Local Seed Control "**, and select the left, horizontal region-edge between the two soil layers. Define the local seed control as shown in Figure 2.12-20. Note that the red arrow in Figure 2.12-20 represents refining direction. Similarly, change the local seed control for the right, horizontal internal soil edge, and set the refining direction back in towards the culvert.

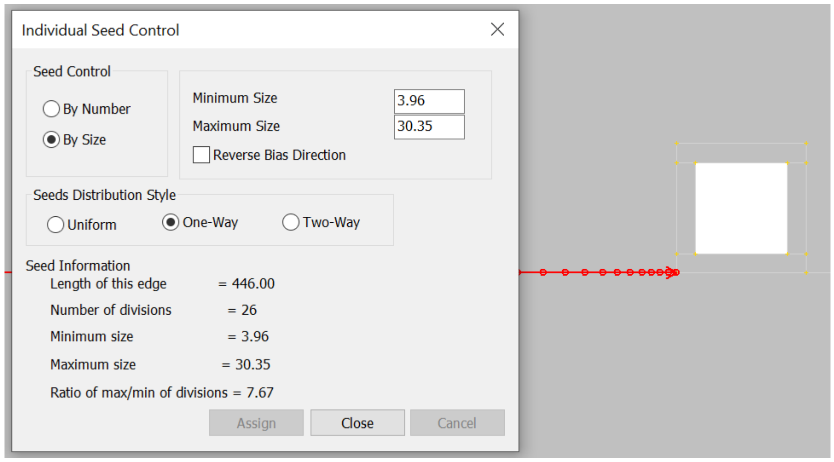


Figure 2.12-20: Control mesh size to transition smoothly from coarse to fine areas

It is always recommended to check seed distributions by turning on seeds and divisions on each edge. From the menu bar, click **"pre-Mesh->Show Edge Division"** and **"pre-Mesh->Show Seeds"** to verify these mesh assignments. Click each command again to turn off. With all mesh definitions complete, click **"pre-Mesh->Run Mesh Generation"** to generate the mesh, as shown in Figure 2.12-21. Click to save the file.

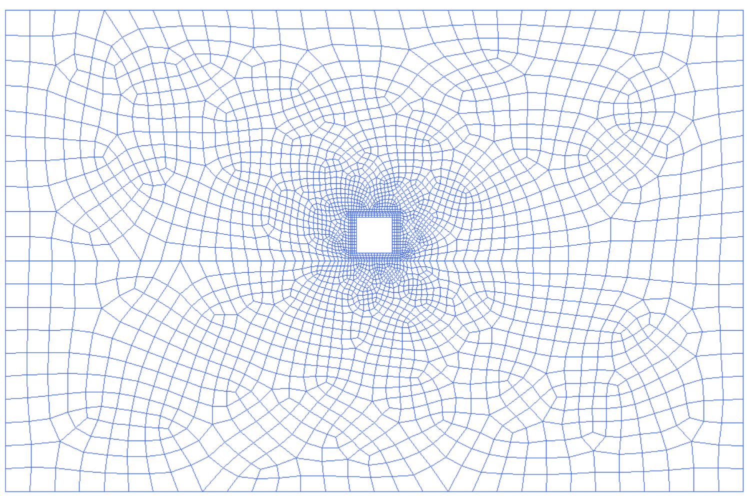


Figure 2.12-21: Q4 mesh for box culvert analysis

Step 6: Analysis

From the menu bar, click **"Analysis->Analysis Setting"** to open the dialog for analysis options. Keep the default setting for convergence and interactions, and click **"OK"** to exit.

From the menu bar, click **"Analysis->Run Analysis"** to open the "Run Analysis" window. Click **"Run"** to submit the analysis job to the solver. A command window will open to show information related with each analysis increment. The file, *"BoxCulvert_MC.log"*, will be opened automatically when the analysis job is done. Review this file to check convergence history and any error information, if it exists.

Step 7: Post-processing of result

Click  to switch to the result-mode. In this example, the scope of the problem is to investigate contact stresses for a comparison of using an unsymmetrical mesh for a symmetric problem. Due to the sharp corners, the box-culvert has discontinuities on the contact interfaces.

Click  to open the X-Y plot dialog. Select "Field Result" as the source data type. Click "By Set", select the previously defined edge-set "setCT_right", and click **"OK"**. Select "Interaction" from the Category list and "Normal Pressure" from Variables list. Select the last increment and click **"Plot"** to plot the Normal Contact Pressure along the top edge of the culvert (Figure 2.12-22). Differences in the contact pressure are again noticed for the nodes located at the sharp corners, and may not be as reliable due to this geometric discontinuity. Click **"Save to File"** and plot this curve using a third-party software if desired.

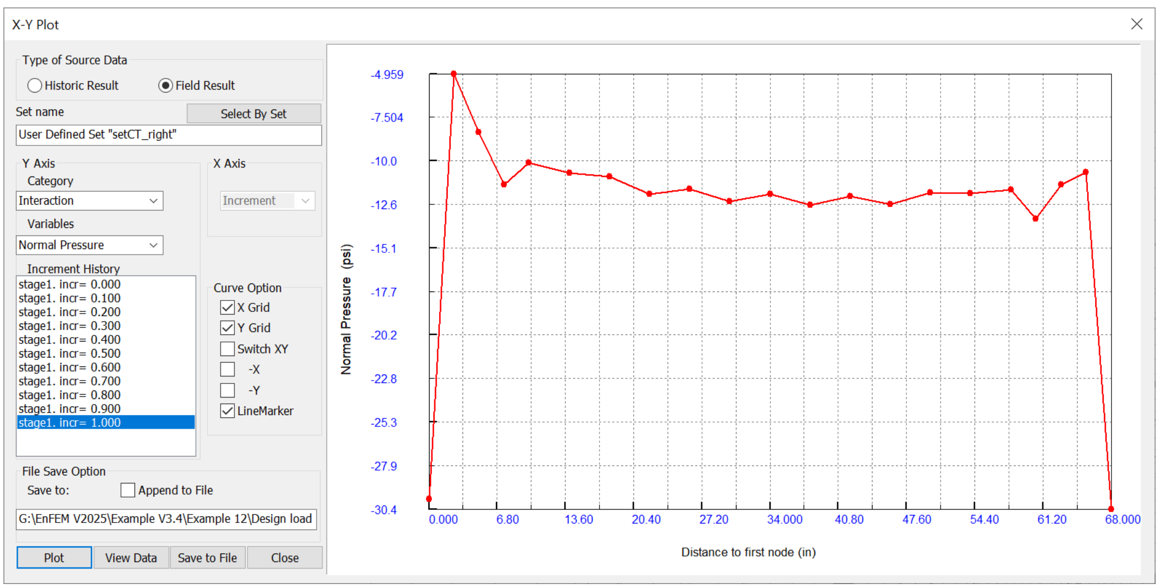


Figure 2.12-22: Plot contact pressure on top of the culvert by XY plot tool

The comparison of contact pressures on the left and right sides is shown in Figure 2.12-23. Although the mesh is not symmetric, the contact pressures on both vertical sides agree well with one another.

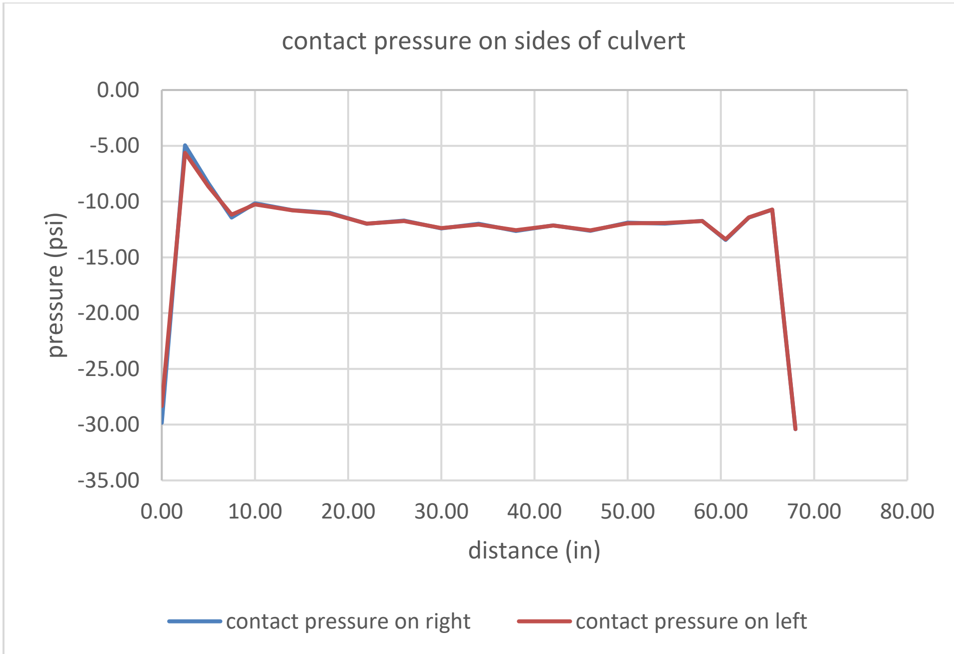


Figure 2.12-23: Comparison of contact pressure on the right and left vertical sides

2.13 Example 13: Structural Analysis of T shape Frame

This example presents how to use EnFEM to do structural analysis for plane frames subjected to various beam loads. The dimension of the T frame is shown in Figure 2.13-1. Units used in this example are “lbf, in, second”.

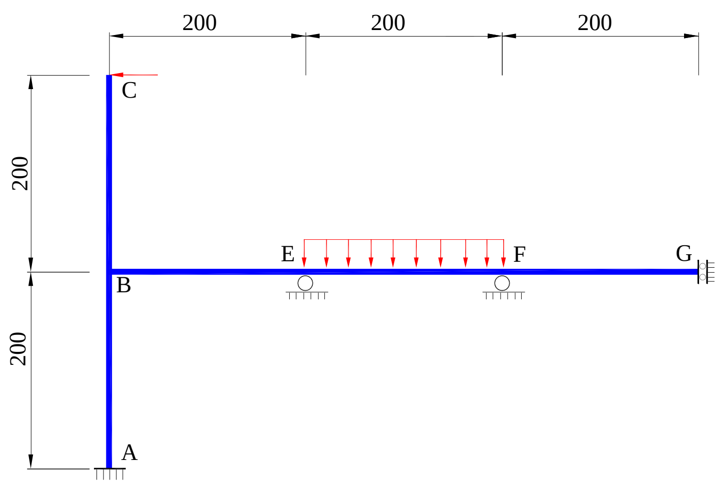


Figure 2.13-1: Analysis of a frame with straight segments

Section and material:

- Beam height:... $h = 18\ in$
- Beam material: concrete:.............................. $E = 4.060 \cdot 10^6\ psi$
- Beam area:... $A = 18\ in^2$
- Beam spacing:... $1\ in$

Loads and supports:

- A: embedded
- C: concentrated force $F = 1000\ \ lbf$
- E and F: roller
- G: rigid roller (no rotation and translation in x direction)
- EF: distributed load $p = 10\ \frac{lbf}{in}$
- gravity

Entity creation, control data, stage data and set definition

Start EnFEM, or slick menu ‘**File->New**”. Change title to “`Analysis of T-shape straight frame`”. Select unit “lbf, in, second” as shown in Figure 2.13-2.

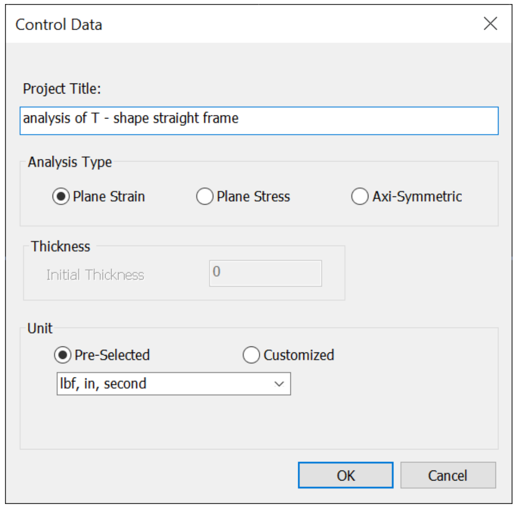


Figure 2.13-2: Control data in example 13

From the menu bar, click **"Create->Straight Beam"** or click  on the toolbar to activate beam creation. In the input box under the working zone, key in the first coordinate of point A as "`0,0`" and press RETURN. Key in the relative coordinate of point B as "`[0,200]`" and press RETURN. Continuously input the coordinate of the third point C as "`[0,200]`". RIGHT-CLICK and select the option "**End Creation**" to complete creation of the vertical frame. Hold CTRL and snap to grid intersection point B(x=0,z=200). Continuously create the three beam segments at coordinates "`[200,0]`", "`[200,0]`", "`[200,0]`". RIGHT-CLICK and select the option "**End Creation**" to terminate beam entity creation.

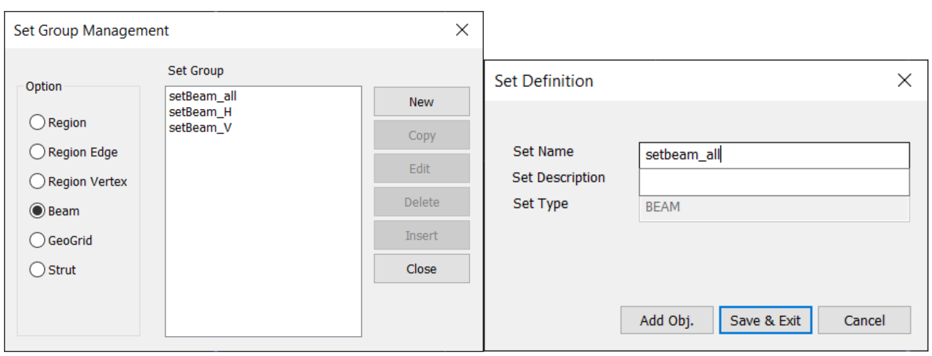
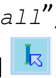


Figure 2.13-3: Define beam sets

Click the menu "**Tool->Set Management**" to define several beam sets (Figure 2.13-3). Select option "Beam", click the button "**New**" to open set definition dialog as the right of Figure 2.13-3. Input the set name "*`setBeam_all`*", click the button "**Add Obj.**". Notice that the beam selection icon is automatically activated . Using a crossing window to select all five beam segments. RIGHT-CLICK and select "**End Selection**". All selected beams will change to deep purple color. Click the button "**Save & Exit**" to save this set definition to the model database. Similarly define one new set for three horizontal beam segments, and another new set for two vertical beam segments.

Click the menu "**Data->Stage Management**". In the popup dialog, click the button "**New**" to generate a new stage in type of "General static" (Figure 2.13-4). Since this is a linear elastic analysis, the initial increment and maximum increment size can be set to "*1*".

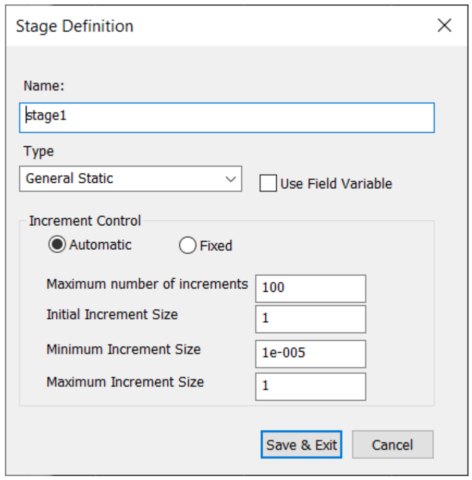


Figure 2.13-4: Stage definition for plane frame analysis

Definition and assignment of structural section

Click the menu "**Data->Structural Section**" to start the section group management (Figure 2.13-5). Click the button "**New**" to generate a new structural section "*bm1*" and input section parameters as in Figure 2.13-5.

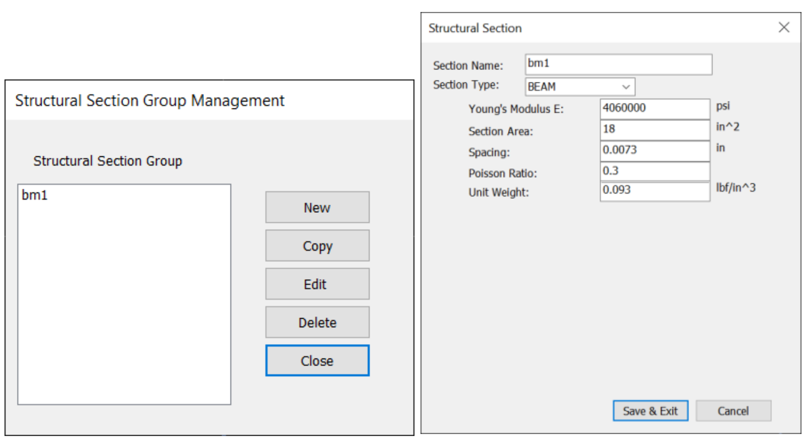


Figure 2.13-5: Section group management and section definition for plane frame

Assume this plane frame is made of reinforced concrete. The density of reinforced concrete is about 160 *pcf*. The Young's modulus is estimated as $E = 4.060 \cdot 10^6 \cdot psi$. Section parameters *EA, EI* and unit weight are calculated as:

$$EA = 4.061 \cdot 10^6 \cdot psi \cdot 18 \cdot in = 7.31 \cdot 10^7 psi \cdot in$$

$$EI = 4.061 \cdot 10^6 \cdot psi \cdot \frac{1}{12} \cdot (18 \cdot in)^3 = 1.974 \cdot 10^9 \cdot psi \cdot in^3$$

$$unit\ weight = 160 \cdot \frac{lbf}{ft^3} = 0.093\ pci$$

The shear section factor for a rectangle section is 0.8333 by default.

Click "**Save & Exit**" to save section parameters to model database and exit to section group management dialog. Click "**Close**" to exit.

Click the menu "**Data->Structural Section Assignment**" to pop up the section assignment dialog as shown in Figure 2.13-6. Select appropriate section type (default is "Beam") and selection option (default by "Set"). From "Set list" window select the set "**SetBeam_all**", select section definition "**bm1**" from the drop list of section type, click the button "**Assign**" to finish this section assignment operation.

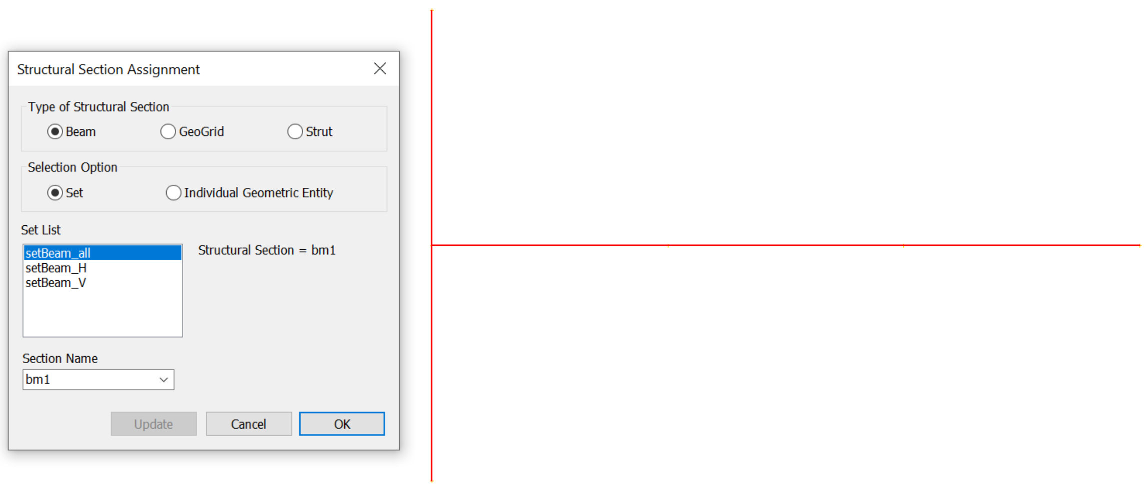


Figure 2.13-6: Assign structural section to beam set

Definition and assignment of boundary conditions

Click the menu "**Data->Boundary Conditions**" to start boundary condition definition. Define three displacement boundary conditions and three loads as shown in Figure 2.13-7.

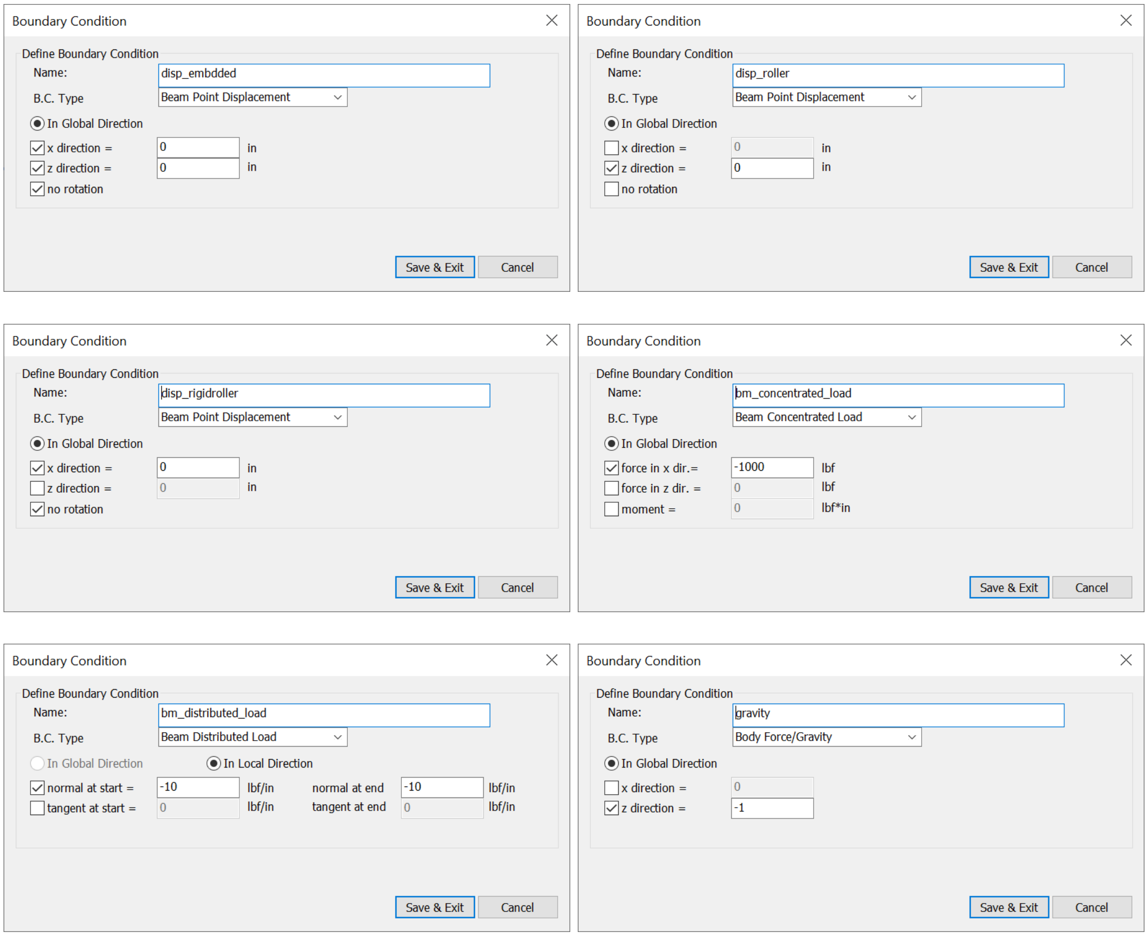


Figure 2.13-7: Displacement and load boundary conditions for plane frame

Click the menu “**Data->Boundary Assignment**” to pop up the BC assignment dialog (Figure 2.13-8). Followings are the list of boundary conditions in this example:

- Point A: ″**disp_embedded**″
- Point C: “*bm***concentrated_load**″
- Point E and F: “**disp_rolller**″
- Segment EF: “**distributed_load**″
- Point G: “**disp_rigidroller**″
- All beam segments: “**gravity**″

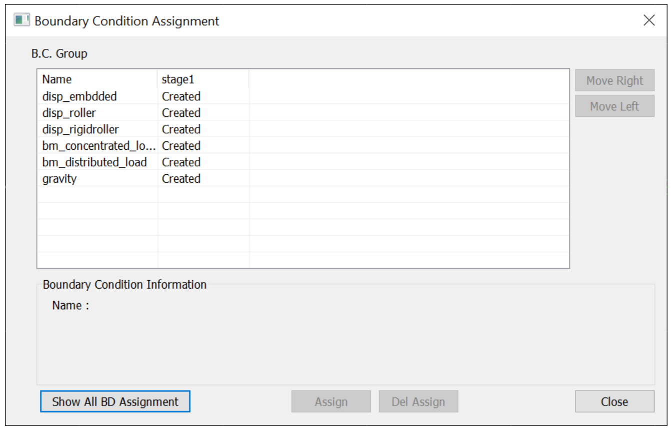


Figure 2.13-8: Assign boundary conditions to plane frame

Select one BC from the group window, select start stage, click “**Assign**”, select appropriate geometric entities. RIGHT-CLICK and select “**End Selection**” to complete current assignment operation.

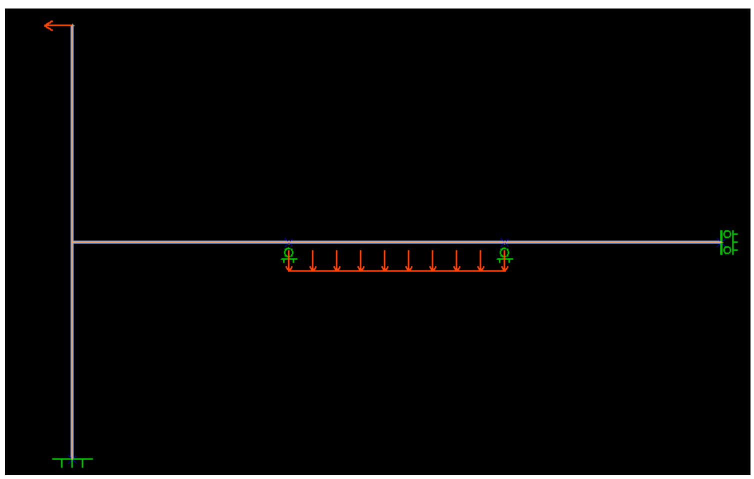


Figure 2.13-9: Show all boundary conditions to plane frame

Click “**Show All**” in Figure 2.13-8 to verify each boundary condition to be assigned correctly (Figure 2.13-9).

Mesh generation

In EnFEM, beam elements are three-node quadratic element. There is no necessary to do mesh control operation. Click the menu “**pre-Mesh->Global Mesh Control**”. In the pop-up dialog, input “*10*” as the global element size. Each beam segment will be discretized to 10 finite beam elements.

Analysis and result output

The default analysis setting is appropriate. There is no necessary to change the default setting. Click the menu “**Analysis->Run Analysis**”. In the popup dialog, check the only stage, click “**Run**” (Figure 2.13-10). When analysis is completed, a message file will be pop up automatically. Review this message file for any warning/error messages.

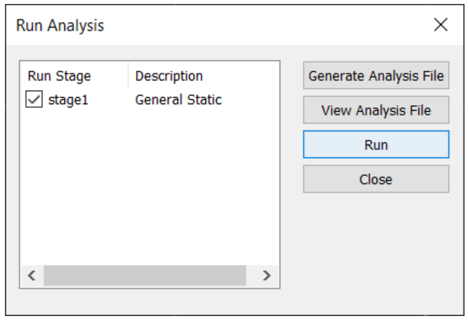


Figure 2.13-10: Run analysis for plane frame analysis

Click to switch to result mode. For a frame analysis, primary output is displacements and internal forces of each beam segments. Click the menu “**Results->Stage Selection**” to activate the only stage for output. Click the menu “**Results-> Curve Plot for Structural Elements**” or click on the toolbar. The curve plot dialog looks like that in Figure 2.13-11. Select “BEAM” as the type of structural element. Click “Select By Set”. In the popping up dialog, select the set “setBeam_H” which are three horizontal beams. Select “Internal Forces” as the category. Select “Moment” as the variables. Select the increment “stage1 incr=1.000”. Click “Plot” to plot the moment curve. Note that moments are always plotted on the tension side. Click “**Close**” to exit.

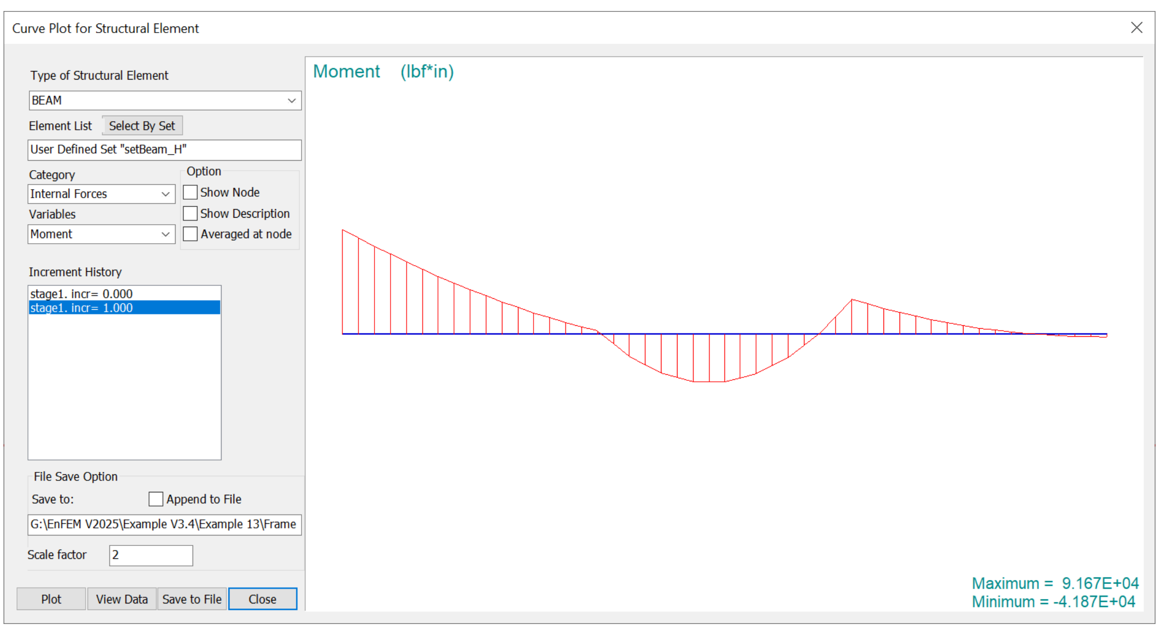


Figure 2.13-11: Plot result curves for plane frame

2.14 Example 14: Structural analysis of a quarter-ring beam subjected to concentrated load

In this example, a cantilevered quarter-ring beam is simulated. The geometry and boundary conditions are displayed in Figure 2.14-1. Radius of the ring is R=10ft, and concentrated load is P=1 kips which is applied to the free end of the beam. Units in this example are “kips, foot, second”.

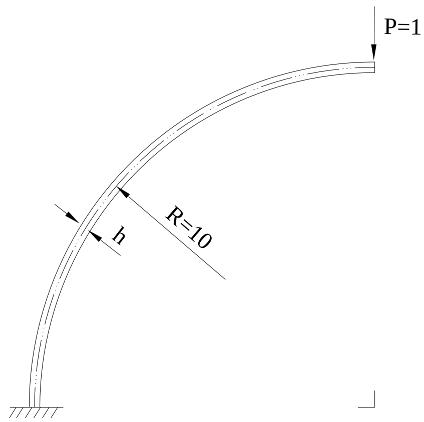


Figure 2.14-1: Cantilevered quarter-ring beam subjected to point load

Parameters for the beam section:

- Young’s modulus:................ $E = 5.8 \cdot 10^5\ ksf$
- Poisson’s ratio:.................. $\nu = 0.3$
- Beam height:...................... $h = 2\ ft$
- Shear factor:....................... $k = 0.833$

For plane strain analysis, this beam has unit thickness in the direction perpendicular to the screen. Section parameters in EnFEM are calculated as

$$EA = 5.8 \cdot 10^5 ksf \cdot 2ft = 1.16 \cdot 10^6\ kip/ft$$

$$EI = 5.8 \cdot 10^5 ksf \cdot \frac{1}{12} \cdot (2ft)^3 = 3.867 \cdot 10^5\ kip * ft$$

Entity creation, control data, stage data and set definition

Start EnFEM, or click “**File->New**” to open a new project. Click the menu Click the menu “**Options->Drawing Space**” and set the drawing limit as “*-20*” to “*20*” in both horizontal and vertical directions. Click the menu “**Options->Grid Space**” and set the grid space to 10 in both directions. Click the arc beam creation icon . Hold CTRL key and snap the grid intersection (x=10, z=0). LEFT-CLICK to create the arc center vertex. Similarly snap to the grid intersections (x=10, z=10) and (x=0, z=0) to create the arc start and arc end vertices (Figure 2.14-2).

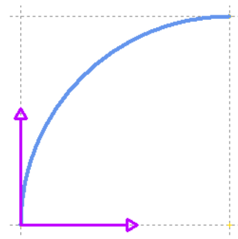


Figure 2.14-2: Create arc beam entity

Click the menu "**Tools->Set Management**" to generate a new beam set "*setBeam*". Add the beam arc segment to this set.

Click the menu "**Data->Control Data**" to generate global control data for this model. Input project title as "*Quarter ring beam analysis*". Keep the default "Plane Strain" option. Click "**OK**" to exit.

Click the menu "**Data->Stage Management**" to define a new stage with the type of "General Static". Keep the default "Automatic" option for increment control. Change the increment size and maximum increment size to "*1*" since this is a linearly elastic analysis.

Click the menu "**Data->Structural Section**". Generate a new structural section"*bm1*" with parameters shown in Figure 2.14-3.

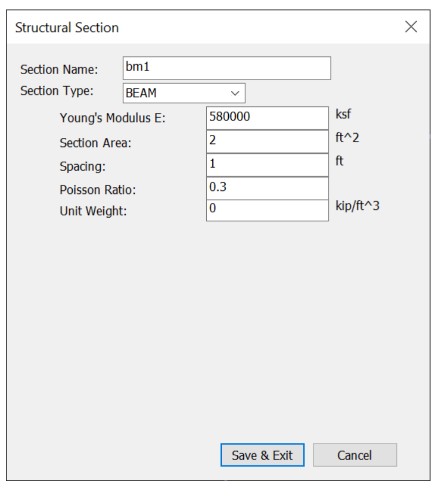


Figure 2.14-3: Define section for quarter-ring beam

Click the menu "**Data->Structural Section Assignment**". In the popping up dialog, select "Geometric Entity" as the selection option. Select the beam name "*beam 0-1*" from the object list window, select the section type "*bm1*", click "**Assign**", then click "**Close**" to exit (Figure 2.14-4).

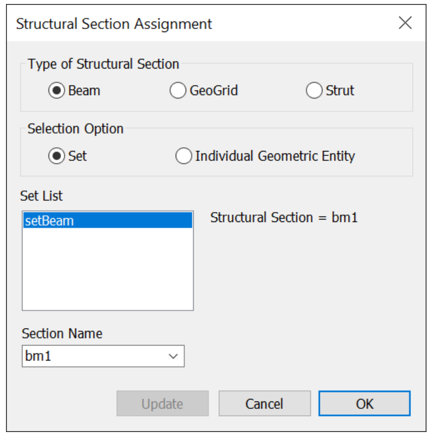


Figure 2.14-4: Assign section to quarter-ring beam

Generation and assignment of boundary conditions

Click the menu “**Data->Boundary Conditions**”. Generate two new boundary conditions “`embedded`” and “`pt_load`” as shown in Figure 2.14-5.

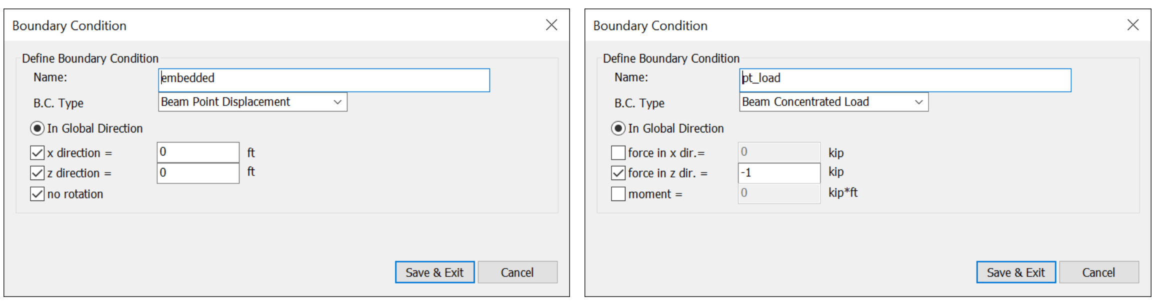


Figure 2.14-5: Define embedded BC and load BC for quarter-ring beam analysis

Click the menu “**Data->Boundary Assignment**”. In the popping up dialog, select BC “`embedded`”, then click “**Assign**”, select the lower vertex of the beam. The vertex selection icon  should be activated automatically. If it is not activated, activate it manually. Select the arc beam segment, RIGHT-CLICK and select “**End Selection**” from pop-up menu. Similarly assign the BC “`pt_load`” to the high vertex of the beam. Click “**Show all**” and verify correctness of boundary assignment (Figure 2.14-6).

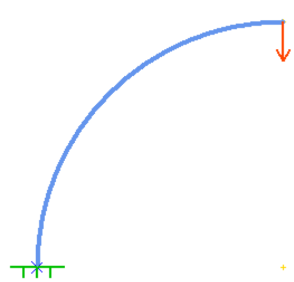


Figure 2.14-6: Assigned boundary conditions for quarter-ring beam analysis

Mesh generation

Click the menu “**pre-Mesh->Global Seed Control**”. Input value “*4*” as the global element size. Click the menu “**pre-Mesh->Run Mesh Generation**” to generate the beam mesh. For curved beam elements, middle nodes always locate on the real curve even they look to be not in the mesh mode.

Analysis and result output

The default analysis setting is appropriate for this model. Click the menu “**Analysis->Run Analysis**”, check the stage “stage1”, then click “**Run**”. After analysis, review the message file if error occurs.

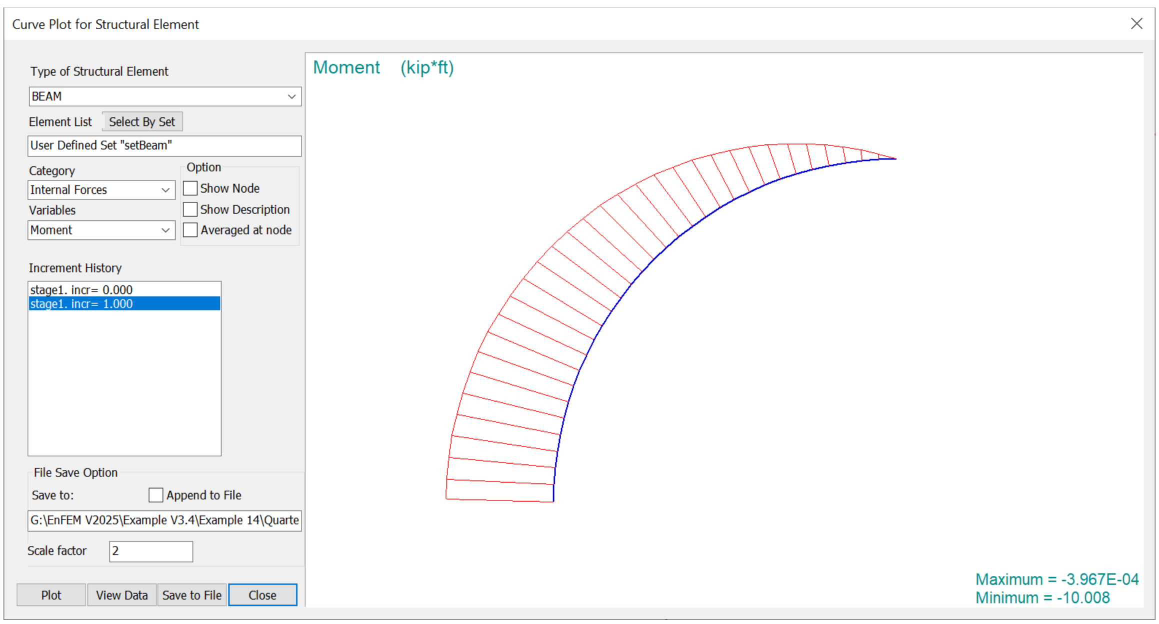
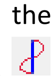


Figure 2.14-7: Moment plot for quarter-ring beam analysis

Click  to switch to result mode. Click the menu “**Results->Stage Selection**” to activate the “stage1” for output. Click the menu “**Results-> Curve Plot for Structural Elements**” or click  on the toolbar. Select “BEAM” as the type of structural element. Click “**Select By Set**”. In the popping up dialog, select the set “setBeam”. Select “Internal Forces” as the category. Select “Moment” as the variables. Select the increment “stage1 incr=1.000”. Click “**Plot**” to plot the moment curve. Note that moments are always plotted on the tension side. Click “**Close**” to exit.

Click “View *Data*” to review moment values at each beam nodes (Figure 2.14-8).

	Ele...	Node	Distance	Values
1	1	1	0.00	-3.97E-04
2	1	18	0.49	-0.49
3	1	2	0.98	-0.98
4	2	2	0.98	-0.98
5	2	19	1.47	-1.47
6	2	3	1.96	-1.95
7	3	3	1.96	-1.95
8	3	20	2.45	-2.43
9	3	4	2.93	-2.90
10	4	4	2.93	-2.91
11	4	21	3.42	-3.37
12	4	5	3.90	-3.83
13	5	5	3.90	-3.83
14	5	22	4.38	-4.27
15	5	6	4.86	-4.72
16	6	6	4.86	-4.72
17	6	23	5.33	-5.14
18	6	7	5.81	-5.56
19	7	7	5.81	-5.56
20	7	24	6.27	-5.95
21	7	8	6.74	-6.35
22	8	8	6.74	-6.35
23	8	25	7.20	-6.71
24	8	9	7.65	-7.08
25	9	9	7.65	-7.08
26	9	26	8.10	-7.41
27	9	10	8.55	-7.74
28	10	10	8.55	-7.74
29	10	27	8.99	-8.03
30	10	11	9.43	-8.32

Figure 2.14-8: Moment data table

2.15 Example 15: Excavation and analysis of cantilevered retaining wall using beam elements

This example simulates the same construction stages as in example 10. The concrete retaining wall is simulated by beam elements. The soil-structure-interaction is simulated using zero-thickness interface elements. Geometry of this example is displayed in Figure 2.15-1. Excavated soil is divided into two layers. The retaining wall is 20 meters in height and 0.7m in thickness. Units used in this example are “kN, m, second”.

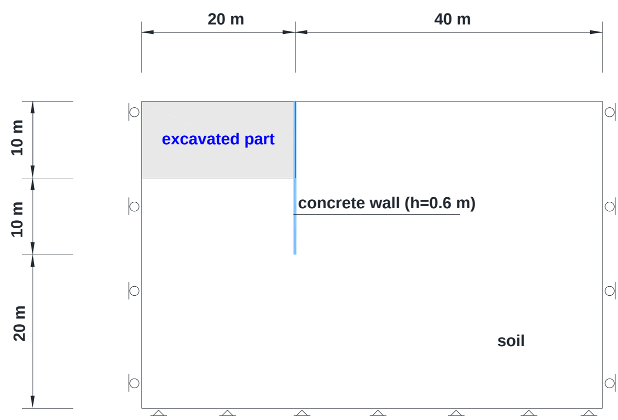


Figure 2.15-1: Excavation simulation and analysis of cantilevered retaining wall

Material property:

Diaphragm wall:

- Unit weight:................................ $\gamma = 0\ kN/m^3$
- Young's Modules:................................ $E = 2.8 \cdot 10^7\ kPa$
- Section area: .... $A = 0.6\ m^2$
- Poisson's ratio:................................ $\nu = 0.2$
- Section width:................................ $h = 0.6\ m$
- Spacing:................................ $1\ m$

Soil:

- Unit weight:................................ $\gamma = 18\ kN/m^3$
- Slope of the normal compression line:................ lambda λ =0.1
- Slope of the swelling line:................................ kappa κ =0.025
- Slope of critical line:................................ M =1.2
- Specific volume number:................................ N =3
- Poisson's ratio:................................ υ =0.35
- Over-consolidation ratio:................................ *OCR=1.1*
- Material model: .... Modified Cam-Clay

Boundary conditions:

- displacement constraint on vertical soil boundaries.
- displacement constraint on bottom edge of the soil region.
- gravity load
- zero-thickness interface between wall and soil body.

Zero-thickness interface for soil-structure-interaction:

- Normal stiffness:.. use default
- Tangent stiffness:... use default
- Friction angle:.. $\theta = 15\,°$
- Dilation angle: .... $\varphi = 0\,°$
- Cohesion:.. $c = 5\ kPa$

Model development and analysis steps:

Step 1: Create geometric entities

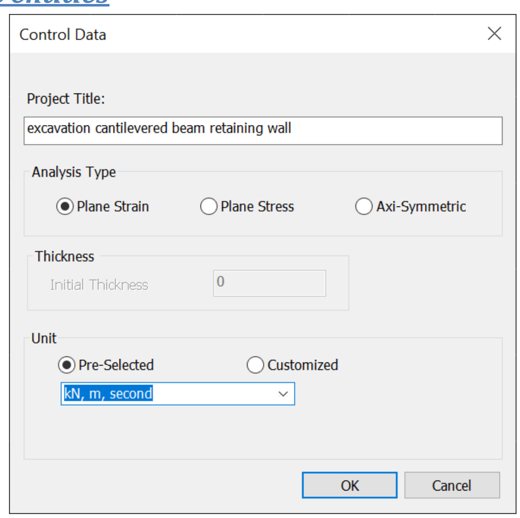


Figure 2.15-2: Control data in example 15

Start EnFEM, or click **"File->New"** to start a new project. Change title to "`excavation cantilevered beam retaining wall`". Select unit "kN, m, second" (Figure 2.15-2).

Click **"Options->Drawing space"** and setup the drawing space like that shown in Figure 2.15-3. Click **"OK"** to exit.

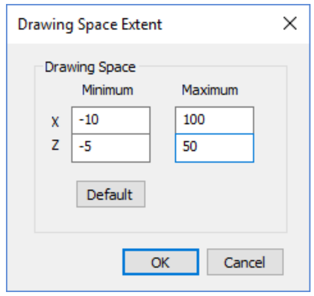


Figure 2.15-3: Setup drawing space for Example 15

Click  to activate the rectangle creation tool. In this model, soil parts are simulated by 5 rectangles in which high quality structural meshes are easy to generated (Figure 2.15-4). Key in the coordinate "`0,0`" in the coordinate input box, press RETURN to create the left-bottom rectangle vertex. Then key in the relative coordinate of the diagonal vertex "`[20,30]`". Press RETURN again to finish the first rectangle. Key in the left-bottom coordinate of the second rectangle as "`20,0`", or "snap" to the left-top vertex of the first rectangle. Key in the diagonal vertex in relative coordinate as "`[40,30]`" to create a width=40, height=30 rectangle. Similarly create the remaining rectangles using coordinates shown in Figure 2.15-4. EnFEM keeps edge compatibility automatically.

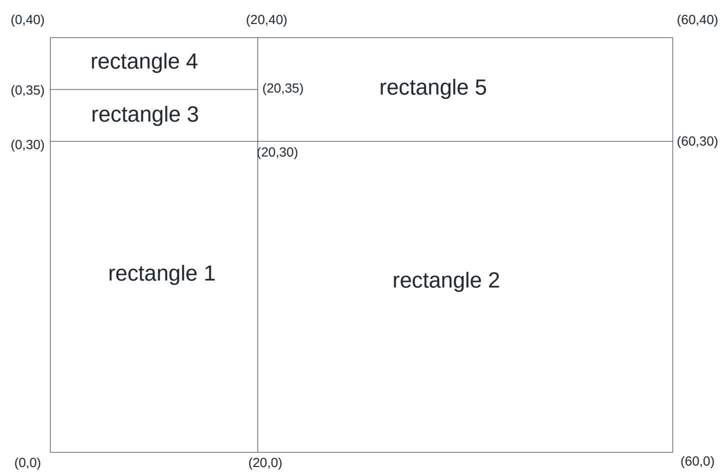


Figure 2.15-4: Create eight rectangles for soil part

Click to activate straight bean creation. Hold CTRL to snap to vertex (20,40) and LEFT-CLICK to create the start beam vertex. Using coordinates (20,20) to create the end beam vertex. RIGHT-CLICK and select "**End Creation**" to end beam creation.

After the beam is created, EnFEM will check geometry compatibility automatically. New vertices will be added on the region edge at beam vertices. This algorithm significantly makes the model development convenient and fast.

Step 2: define region set and beam set

Click the menu "**Tools->Set Management**" to open the set management dialog. Select the option "**Region**" and click "**New**" to create a new region set. In the popping up dialog input the set name "`exca_1`". Click "**Add Obj.**" and select the left-top region, RIGHT-CLICK and select "**End Selection**". Click "**Save & Exit**" to return to the set management dialog. Similarly create a new set "`exca_2`" for the second excavation part and a set "`soil_all`" for all eight regions (Figure 2.15-5).

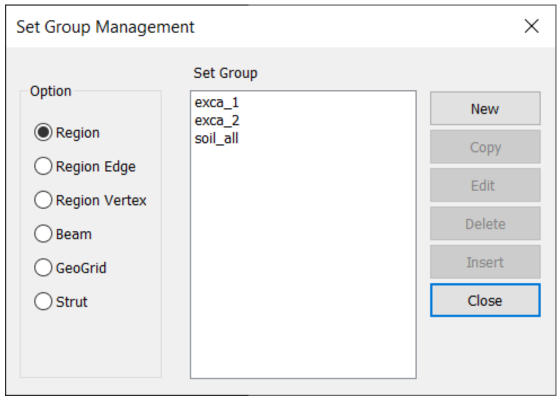


Figure 2.15-5: Create region sets

Select the option "**Beam**" in Figure 2.15-5 to create a new beam set "`setBeam`". This beam set should include all three beam segments. Beam segments in the same set can be head-toe connected or not. However, only the set with head-toe connected beams can be used for result out. Additionally, these head-toe connected beam segments should keep the same direction. To view beam directions, click the menu "**View->Show Direction**". The tangential direction (in red color) is defined from the beam start vertex to the end vertex. The normal direction (in green color) is counter clockwise 90° from the tangential direction (Figure 2.15-6). If some beams have inverted directions, users may click the menu "**Modify->Reverse Beam Direction**" and select these beams to reverse them.

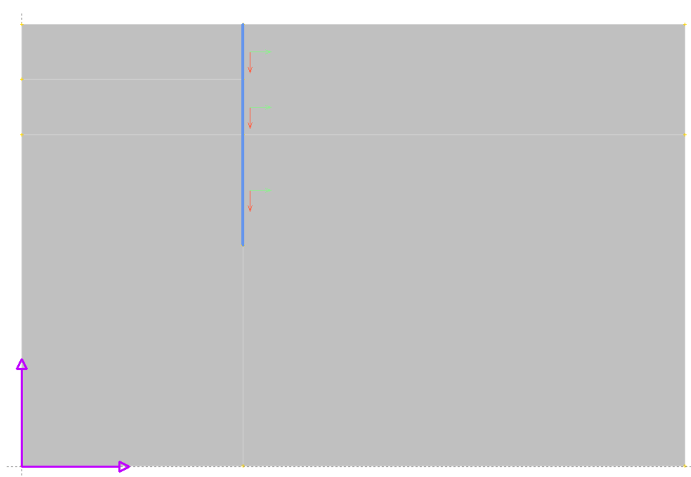


Figure 2.15-6: Definition of positive beam direction

Step 3: Define stage data

Click the menu "**Data->Stage Management**". Click "**New**" to create a new stage "`ints`" in type of "**Initial Soil Stress**". Click "**Initial Soil Stress Option**", keep the default option as "**K0 method**", click "**Add**" to add all eight soil regions into the region list. Click "**Input Data**" to open a dialog for initial soil stress interpolation. Click "**Add Row**" to add a new data row. Input the vertical coordinates of top and bottom soil body, input the component of vertical stress and lateral stress factor as in Figure 2.15-7. Click "**Save & Exit**" to exit to initial stress option dialog. Then click "**OK**" to exit to stage management dialog.

Create two new stages "`Excava_1`" and "`Excava_2`" in type of "Excavation". Keep the default settings for increment option and increment size.

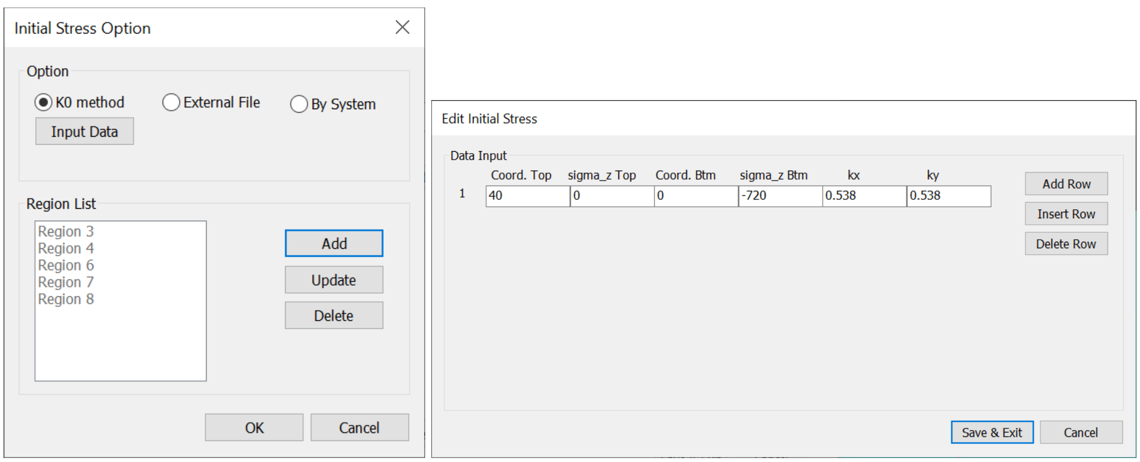


Figure 2.15-7: Define initial soil stress for example 15

Step 4: define soil material and beam section

Click the menu "**Data->Material Properties**" to open the material management window. Click "**New**" to define a new material as shown in the left image of Figure 2.15-8. Click the menu "**Data->Material Assignment**". Check the option "By Set", select the set "setSoil_all" from the set list, select "soil" from the material group list, click "**Assign**" if regions have no material assignment, or click "**Update**" if regions do have previously assigned materials. If regions in the same set have different material assignments, listed material group on the right of the set list will be displayed as "mixed".

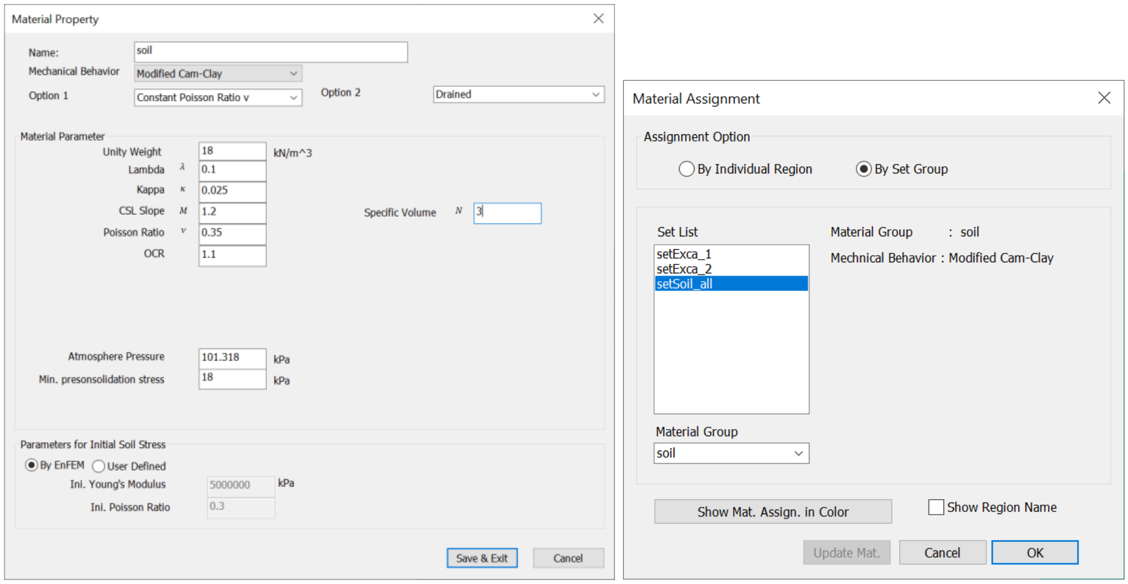


Figure 2.15-8: Define and assign modified CAM-Clay to soil

Click the menu "**Data->Structural Section**" to open the structural section management window. Click "**New**" to define a new beam section as shown in Figure 2.15-9. Click "**Save & Exit**" to save the new beam section into the model database. Click the menu "**Data->Structural Section Assignment**". Keep the default "BEAM" option for type of structural section. Users may select assign by "Set" or by "Geometric Entity". Click the name of the set or name of the geometric entity, note corresponding entities will be highlighted. Select "`bm1`" from the section type drop list, click "**Assign**" if no section has been assigned before, or click "**Update**" to override previously assigned section.

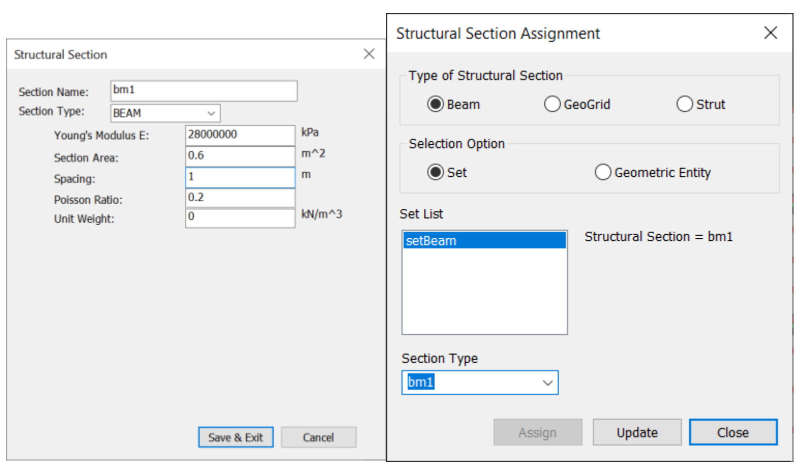


Figure 2.15-9: Define and assign beam section

Step 5: define and assign boundary conditions

Click the menu "**Data->Boundary Conditions**" to open the boundary condition management window. Click "**New**" to create two boundary conditions and one gravity load boundary as shown in Figure 2.15-10.

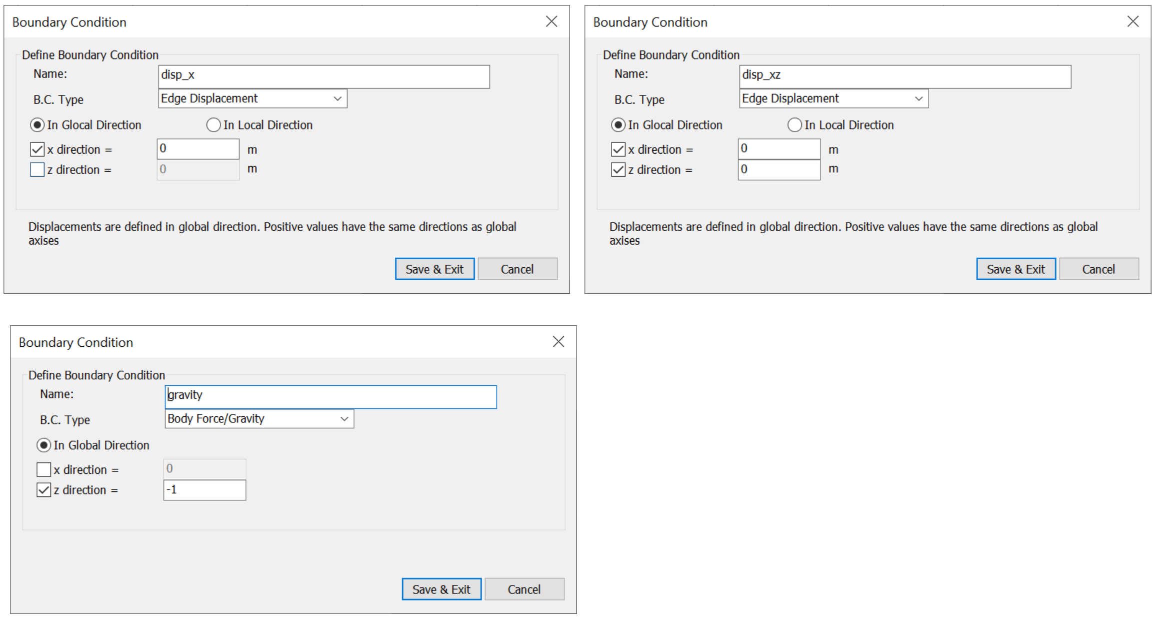


Figure 2.15-10: Define displacement and load boundary conditions

Click the menu "**Data->Boundary Assignment**". In the popping up dialog, click the BC "disp_x". Keep the default start stage setup. Click "**Assign**", select the left and right edges of the whole soil part, RIGHT-CLICK and select "**End Selection**" to finish this BC assignment. Similarly, assign BC "disp_xz" to the bottom edges of the whole soil body, and assign "gravity" to all eight soil regions.

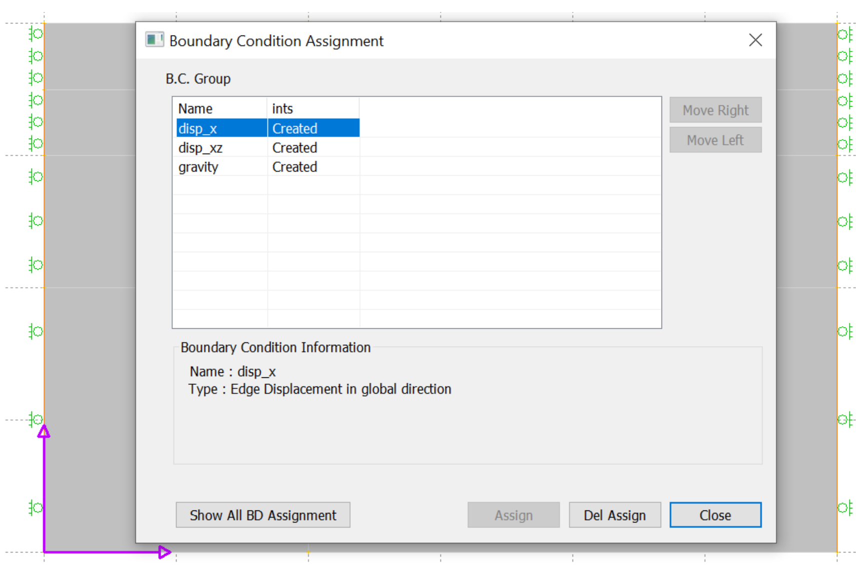


Figure 2.15-11: Assign lateral displacement boundary

After assignment, click each B.C. group, then corresponding boundary condition symbols will be displayed adjacent to selected geometric entities. Users may also click "**Show All**" to review all B.C. assignment simultaneously.

Step 6: Define soil-structure-interaction interface

Click the menu "**Data->Interaction->Zero Thickness Interface**" to open interface management window. Click "**New**" to create a new interface group. Keep the default group name, then input interface parameters as in Figure 2.15-12. The dilation angle is recommended to use the default value or keep to be "0". Click "**Add Beams**". Click or using a selection window to select all three beam segments. RIGHT-CLICK and select "**End Selection**" from the popup menu to finish adding beams to the interface. Click "**Add Regions**". Click or use a crossing window to select all eight soil regions. RIGHT-CLICK and select "**End Selection**" from the popup menu. Click "**Create Interface**". If interface is successfully identified, two deep green dash lines will be added adjacent to the beam segments (Figure 2.15-12**Error! Reference source not found.**). Click "**Save & Exit**" to return to interface group management dialog. Click "**Close**" to exit.

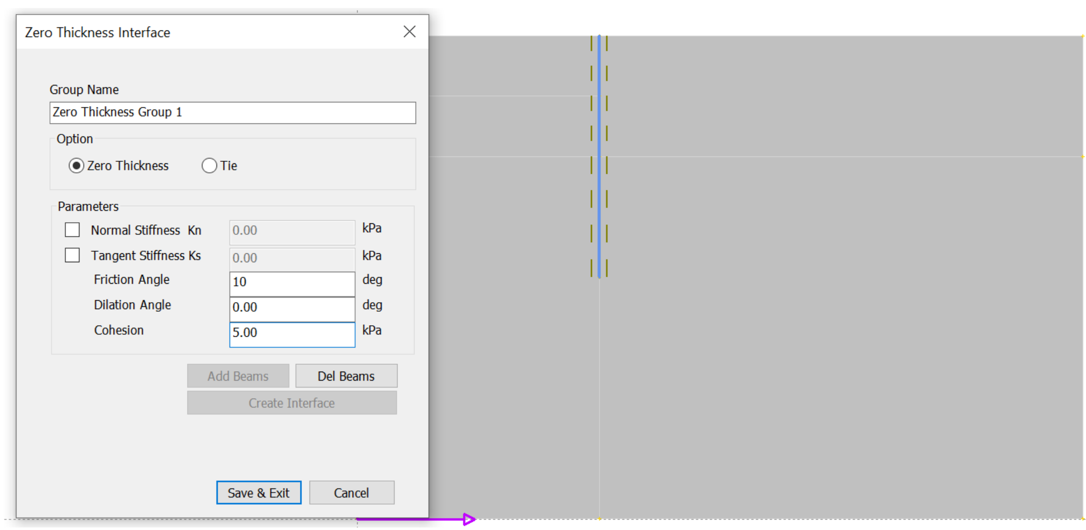


Figure 2.15-12: Define zero-thickness interface in example 15

Step 7: Define deactivation/activation for excavation construction

Click the menu "**Data->Deactivation/Activation**". Click the tab "Region". In the stage column, all three defined stages should be listed. Double-click the cell locating on the "Set Deactivated" column and stage "Excava_1" row, select the set name "setExca_1" (left image of Figure 2.15-13). Similarly set the region set "setExca_2" to be deactivated in the stage "Excava_2". Click the tab "Beam". Set the beam set "setBeam" to be activated in the initial soil stress stage "ints" (right image of Figure 2.15-13).

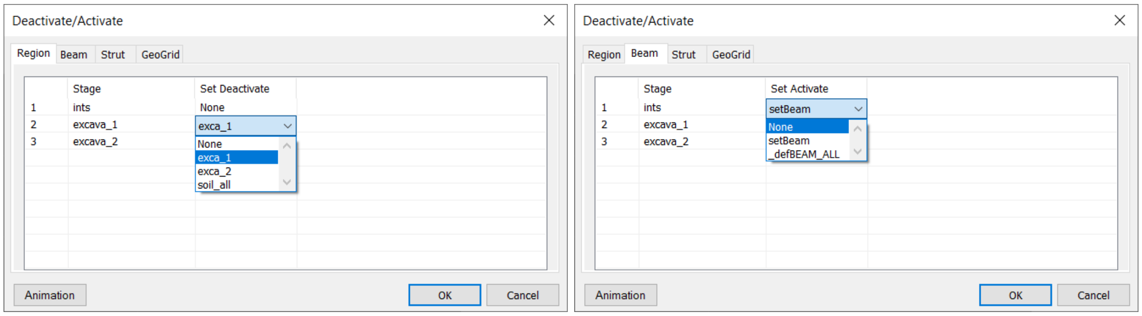


Figure 2.15-13: Set deactivated region and activated beams

Click "**Animation**" to verify excavated parts and construction procedures to be correctly defined.

Step 8: Mesh generation

Click the menu "**pre-Mesh->Advanced Mesh Control**". Select all five soil regions. RIGHT-CLICK and select "**End Selection**" from the popup menu. The mesh control dialog will be opened. Since all five regions are rectangles, the default setting will work well (Figure 2.15-14). Beam elements are always three-node with quadratic interpolation functions in EnFEM, paired solid elements should be quadratic too (Q8 or T6). Click "**Save & Exit**" to save mesh data to the model database.

Click the menu "**pre-Mesh->Global Seed Control**". Input number "*2*" as the global element size. This size is applicable to both 2D regions and 1D beam entities. Click "**Assign**" then click "**Close**" to exit. Click the menu "**pre-Mesh->Show Edge Division**" and "**pre-Mesh->Show Seeds**" to turn on display of edge discretization and locations of seeds. Verify seed space and concentration to be appropriate. In this model, the top two beam segments have only two divisions. It's better to refine these beam segments. Click the menu "**pre-Mesh->Local Seed Control**". Use a crossing window to select three vertical edges of top two regions and the overlapping beam segment. Hold SHIFT and select three vertical edges and the second beam segment from the lower two regions. RIGHT-CLICK and select "**End Selection**". Local seed control dialog will be opened automatically (left image of Figure 2.15-15). Change the division number from "*2*" to "*4*". Click "**Assign**", then click "**Close**" to exit from this dialog. New division and seed distribution will look like Figure 2.15-15the right image of Figure 2.15-15.

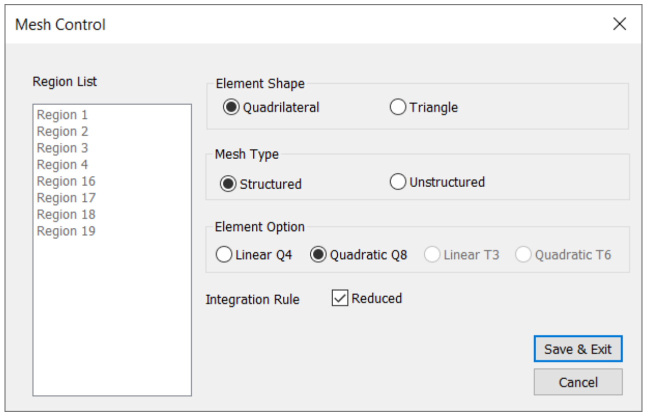


Figure 2.15-14: Mesh control for regions in example 15

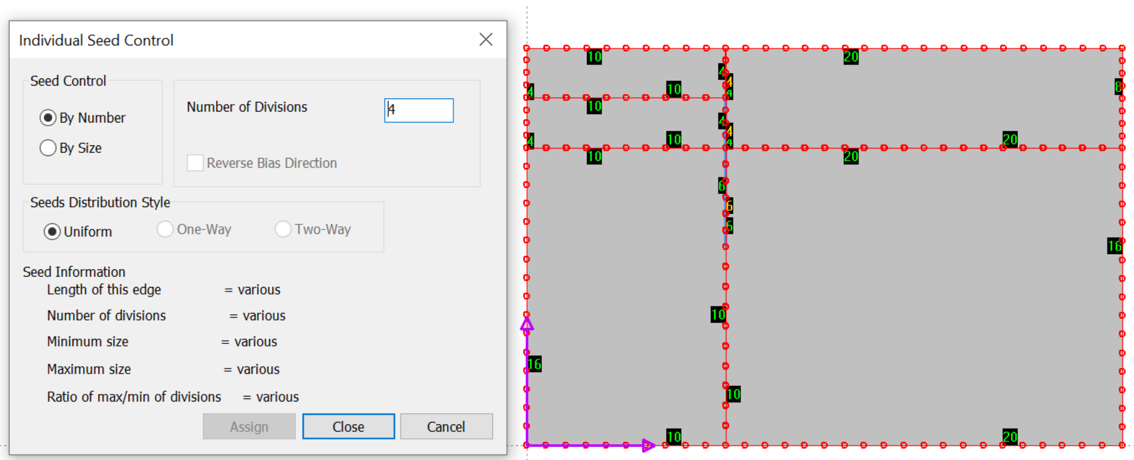


Figure 2.15-15: local seed control and Show edge division and seeds in example 15

Click the menu “**pre-Mesh->Show Edge Division**” and “**pre-Mesh->Show Seeds**” again to turn off display of division and seeds.

Click the menu “**pre-Mesh->Run Mesh Generation**” to generate the mesh.

If the mesh mode is not automatically switched, click the icon  to switch to mesh mode. Since edges of solid elements and beam elements are overlapped on the interaction interface, EnFEM provides a useful tool to check element connection for each element independently. Click the menu “**Mesh->Probe Element**”. Click to activate solid element selection icon . Click one solid element adjacent to the beam. Element connections and the element number will be displayed for selected element only (Figure 2.15-16). Deactivate the solid element selection and activate the beam element selection icon , users can check the element connection for a beam element.

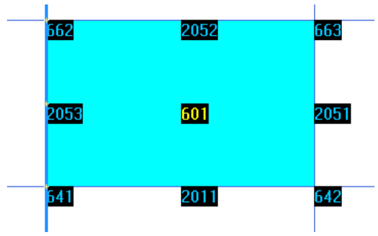


Figure 2.15-16: Probe element connection on overlapped edges

Step 9: Analysis and result output

Click the menu "**Analysis->Analysis Setting**". Default settings work well for this model. Click menu "**Analysis->Run Analysis**". In the control dialog, check all three stages (Figure 2.15-17). Click "**Run**" to submit the job to the solver.

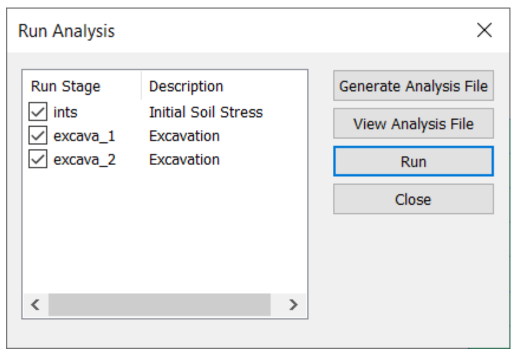


Figure 2.15-17: Dialog of "Run analysis" for example 15

After calculation is successfully completed or terminated due to errors, the message file (.log) will be opened automatically. It is recommended to review this file to check convergence and warning/error messages.

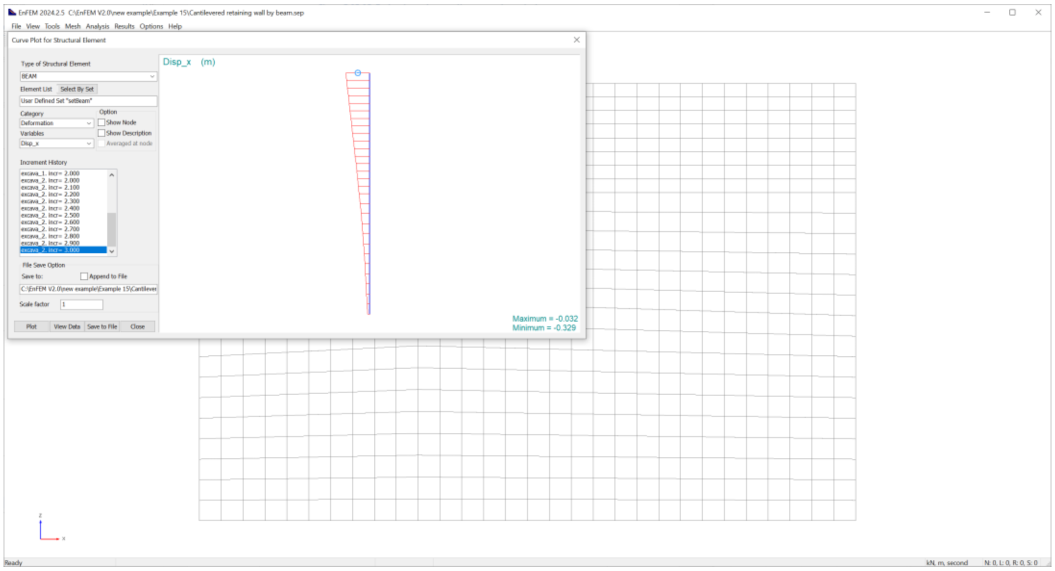


Figure 2.15-18: Horizontal displacement of retaining wall

Click to switch to "RESULT-MODE" if necessary. In this example, result output will be focused on behavior of the retaining wall. Click the menu "**Results->Curve Plot for Structural Elements**", or click on the toolbar panel. In the popping up dialog, select "BEAM" as the type of structural element. Click "**Select By Set**". In the set selection dialog, select the beam set "setBeam" which is defined in step 2. If users need new beam set for extra plots, users can switch to "CAD-MODE", and define desired new beam sets. Go back to the result mode, click "**Select By Set**" again. New sets will be available in the set selection list.

Since element list is determined from geometric entity set, "User Defined Set" will replace real element list. Select "Deformation" from category, then select "disp_x" from variables. Click the last load increment in the increment history. Click "**Plot**" to plot the result curve on the right window (Figure 2.15-18). If "Show Description" is checked, explanation of sign convention will also be plotted under curve title.

Change category selection to "Internal Forces". Select "Moment" and click "**Plot**" again. The moment diagram of the retaining wall will be plotted as shown in Figure 2.15-19. If the scale factor is too small, users may modify it and replot the curve again. Curve amplitude will be enlarged when Scale factor is larger than 1, or the curve amplitude will be reduced if the scale factor is smaller than 1.

Now change the category to "Interface". In the drop list under variables, normal pressure, tangent shear, normal status and tangent status will be available for both side of the beam. Select "Normal Pressure L" and click "**Plot**". The distribution of the soil pressure on the back of the wall is plotted as in Figure 2.15-20. The "*L*" or "*R*" represent left or right side when someone faces toward the beam direction. In this case, the beam direction (indicated by a red arrow) points downward. The left side is the back side of the retaining wall. In Figure 2.15-20, the green dashed line represents the side to be interested. If the result value is positive, the curve point will be plotted at the same side as the green dashed. Otherwise, it will be plotted on the opposite side.

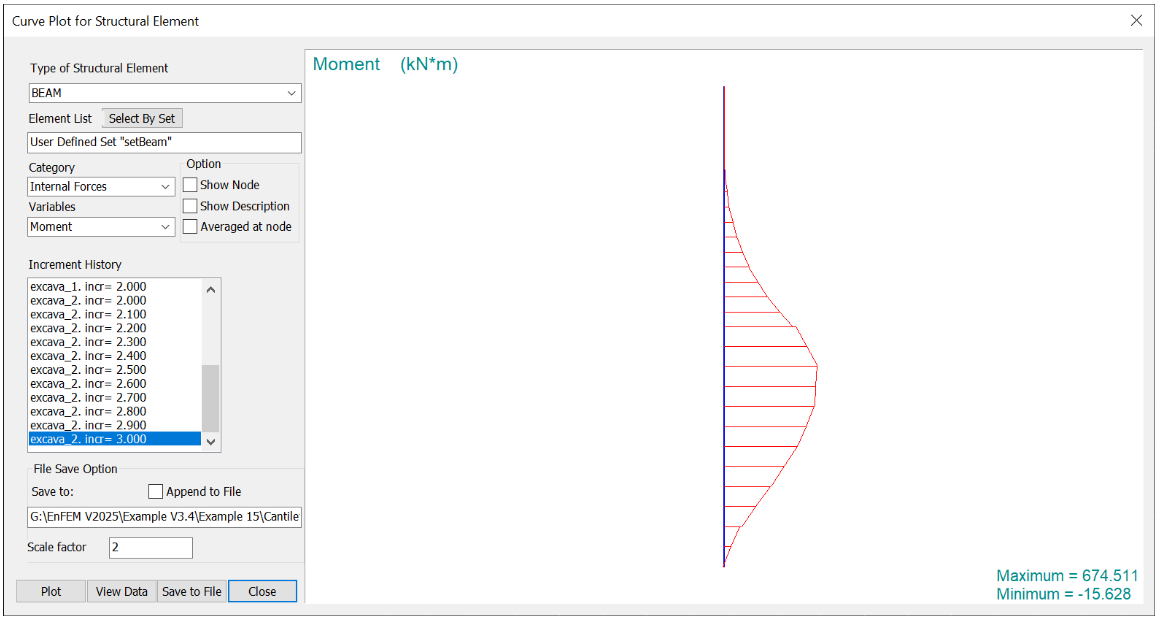


Figure 2.15-19: Moment diagram of retaining wall

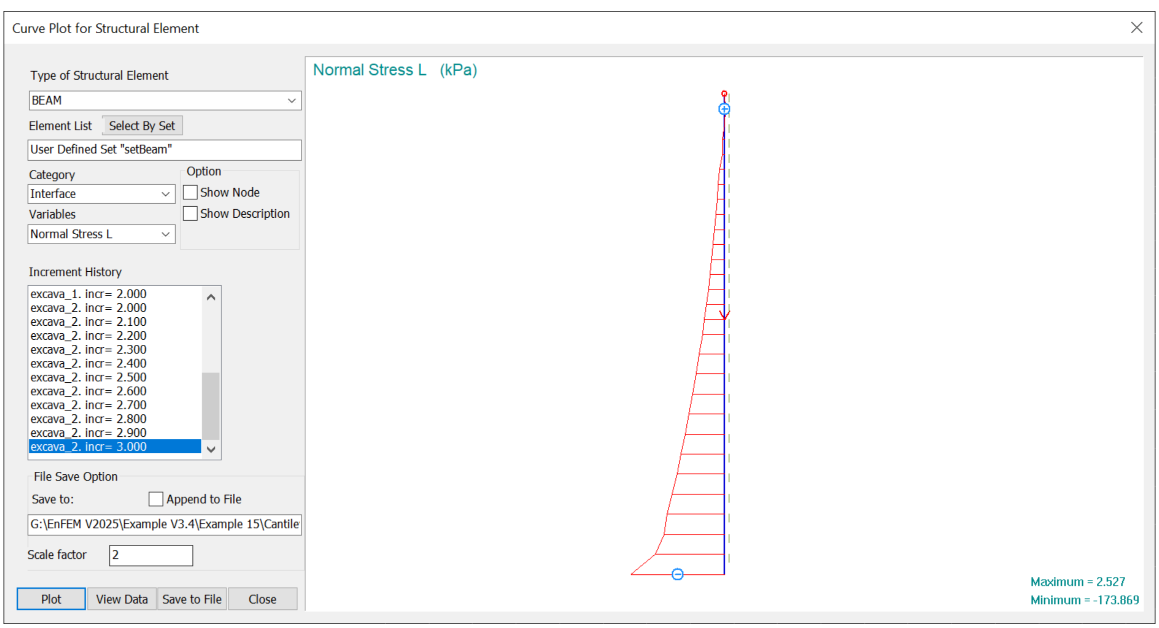


Figure 2.15-20: Distribution of soil pressure along the back of retaining wall

2.16 Example 16: Excavation and struts supported retaining wall subjected to unsymmetric surcharge load

This example simulates another excavation in geotechnical engineering. Struts are used to support retaining walls during excavation. Since surcharges are not symmetric, full model is developed. Geometry of this model is shown in Figure 2.16-1. Units are “kN,m, second”.

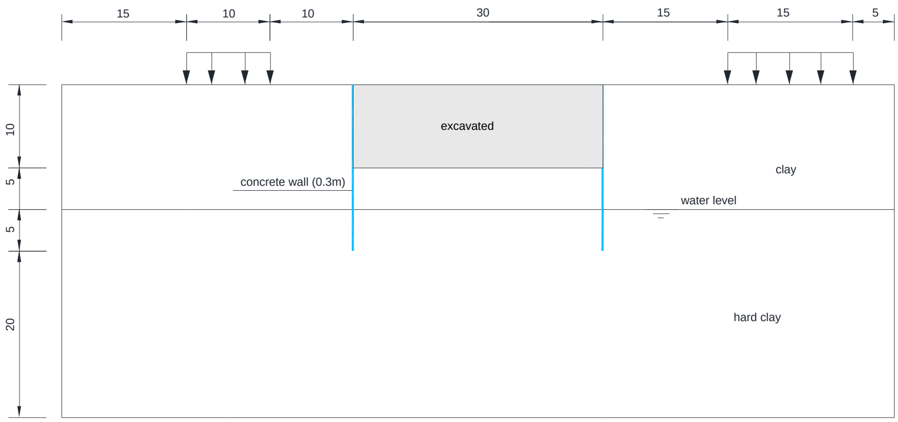


Figure 2.16-1: Geometry of excavation and strut supported retaining wall

Material property:

<u>Diaphragm wall:</u>

- Unit weight:.. $\gamma = 0$
- Young’s modulus:.. $E = 2.8 \times 10^7\ kPa$
- Section area: .... $A = 0.3\ m^2$
- Poisson's ratio:.. $\nu = 0.2$
- Beam spacing:.. $1\ m$

<u>Clay:</u>

- Unit weight:.. γ=18kN/m^3
- Slope of the normal compression line: .... lambda λ =0.1
- Slope of the swelling line:.............................. kappa κ =0.025
- Slope of critical line:.................................... M =1.2
- Specific volume number:.................................. N =3
- Poisson's ratio:.. υ =0.35
- Over-consolidation ratio:................................ *OCR=1.0*
- Material model: modified cam-clay
- Drained

<u>Hard Clay:</u>

- Unit weight:.. γ=8kN/m^3(submerged)

- Slope of the normal compression line: .... lambda λ =0.06
- Slope of the swelling line:............................... kappa κ =0.014
- Slope of critical line:.. M =1.2
- Specific volume number:................................. N =2.5
- Poisson's ratio:.. υ =0.3
- Over-consolidation ratio:................................ *OCR=1.1*
- Material model: modified cam-clay
- Drained

Boundary conditions:

- Displacement constraint on vertical soil boundaries.
- Displacement constraint on bottom edge of the soil region.
- Gravity load
- Surcharge: $2\ kN/m^2$
- Zero-thickness interface between walls and soil.

Zero-thickness interface for soil-structure-interaction:

- Normal stiffness:..use default
- Tangent stiffness:..use default
- Friction angle:.. $\theta = 20\,°$
- Dilation angle:.. $\varphi = 0\,°$
- Cohesion:.. $c = 5\ kPa$

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM, or click **"File->New"** to start a new project. Control data for this example are shown in Figure 2.16-2. Select analysis type as “Plane Strain”. Select preselected unit “kN, m, second”.

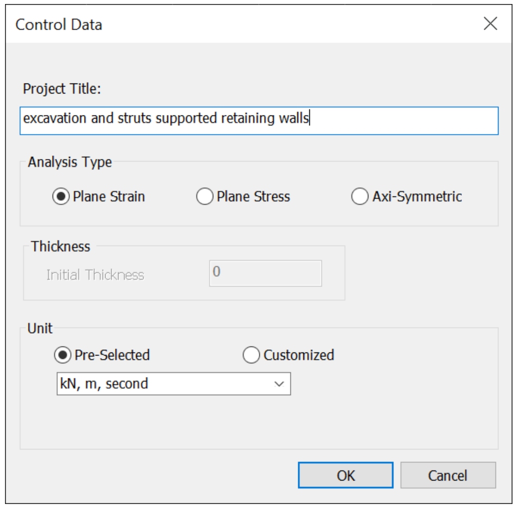


Figure 2.16-2: Control data in Example 16

Click **"Options->Drawing space"** and setup the drawing space like that shown in Figure 2.16-3. Click **"OK"** to exit.

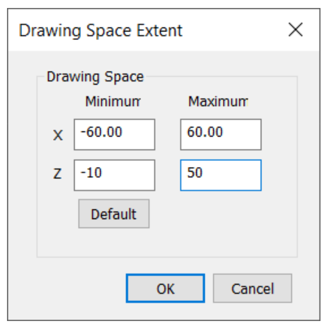


Figure 2.16-3: Setup drawing space for Example 16

Click to activate the rectangle creation tool. This model has 6 rectangles (Figure 2.16-4). Since rectangles 2 and 6 have surface load applied, these two regions are created as polygons. Key in "*-50,0*" in the coordinate input box, press RETURN to create the left-bottom polygon vertex. Key in "*50,25*" to create the first rectangle. Click and snap to the second and the third polygon vertexes (-50,25), (-15,25). Key in the relative coordinates "*[0,5]*" continuously to create next three polygon vertexes. Key in "*[-15,0]*" and "*[-10,0]*" I to create the boundary points for the left surface load. Key in "*-50,40*". Click RIGHT-CLICK and select "**Close Polygon**". The region "rectangle 2" is finished. Click again and snap to vertex (-15,25), key in "*[30,5]*", to create the rectangle 5. Repeat the same procedures to create remaining rectangles and polygon.

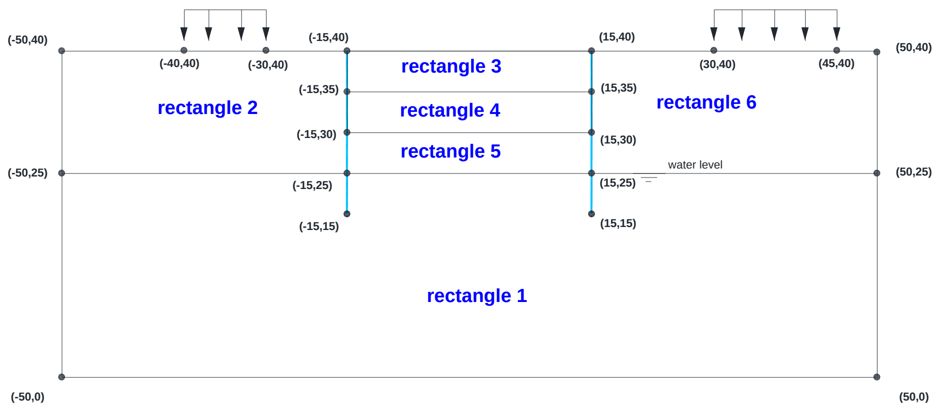


Figure 2.16-4: Create rectangles and polygon for example 16

Click to activate straight beam creation. Key in Hold CTRL to snap to vertex (-15,40) and LEFT-CLICK to create the start beam vertex. Key in *"-15,15"* to create the second vertex of the left beam. Right lick and select "End Creation" to finish the first beam. Similarly create the right beam with vertexes (15,40) and (15,15). Now all region objects have been created. EnFEM will insert/divide region edge or beam segments to keep geometry compatibility automatically.

Click to activate strut creation. Hold CTRL and snap to the left-top vertex of excavated soil (x=-15, z=40), LEFT-CLICK to create the strut start vertex. "snap" to vertex (15,40) and LEFT-CLICK again to finish the first strut. Similarly create the second strut from vertex (-15,35) to (15,35).

Step 2: define region set and beam set

Click the menu "**Tools->Set Management**" to open the set management window. Select the option "**region**" and click "**New**" to create a new region set. Key in the set name "*setExca_1*". Click "**Add Obj.**" and select rectangle 1 in Figure 2.16-4, RIGHT-CLICK and select "**End Selection**". Click "**Save & Exit**" to return to the set management dialog. Similarly create a new set "*setExca_2*" which includes rectangle 2 in Figure 2.16-4. Create a region set "*setClayTop*" for rectangle 2~6. Create a region set "*setHardClay*" for rectangle 1 (Figure 2.16-5).

Select the option "Beam" in Figure 2.16-5 and create three new beam sets "*setWall*", "*setWallLeft*" and "*setWallRight*". The set "wall" contains all beam segments in this model. Set "*setWallLeft*" and "*setWallRight*" have only the left four or the right four beam segments respectively.

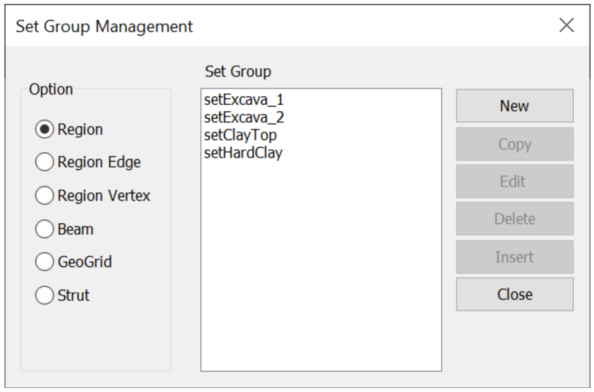


Figure 2.16-5: Create region sets

Select the option "Strut" in Figure 2.16-5. Create two new strut sets "*setStrutTop*" and "*setStrutBtm*". Add the upper strut to set "*setStrutTop*" and add the lower strut to set "*setStrutBtm*".

Beam segments in the same entity set should be head-toe connected and consistent direction. To view beam directions, click the menu "**View->Show Direction**". The tangential direction (in red color) is defined from the beam start vertex to the end vertex. The normal direction (in green color) is counter clockwise 90° from the tangential direction. If some beams have inverted directions, users may click the menu "**Modify->Reverse Beam Direction**" and select these beams to reverse them.

Step 3: define soil material and beam section

Click the menu "**Data->Stage Management**". Click "**New**" to create a new stage "*ints*" with type of "Initial Soil Stress". Click "**Initial Soil Stress Option**", keep the default option as "K0 method", click "**Add**" and select all soil regions. Click "**Input Data**" to open a new window for initial soil stress interpolation. Click "**Add Row**" to add a new data line. Input the vertical coordinates of top and bottom soil body, input the component of vertical stress and lateral stress factor as in Figure 2.16-6. Click "**Save & Exit**" to exit to initial stress option dialog. Then click "**OK**" to exit to stage management window.

Sequentially create five new stages "*install_strut_top*", "*excava_1*", "*install_strut_btm*", "*excava_2*" and "*excava_3*". Set stages of installation of strut as "General Static". Set initial and Set three excavation stages as "Excavation". The initial increment and the maximum increment size for strut installation stages are set to "1". Set the initial increment size for excavation stages to "*0.01*". as shown in Figure 2.16-7.

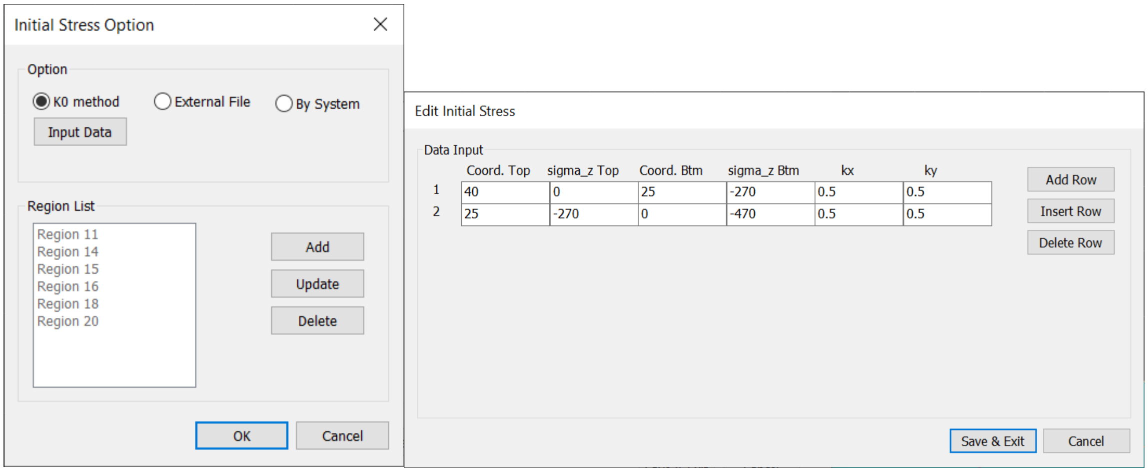


Figure 2.16-6: Define initial soil stress for example 16

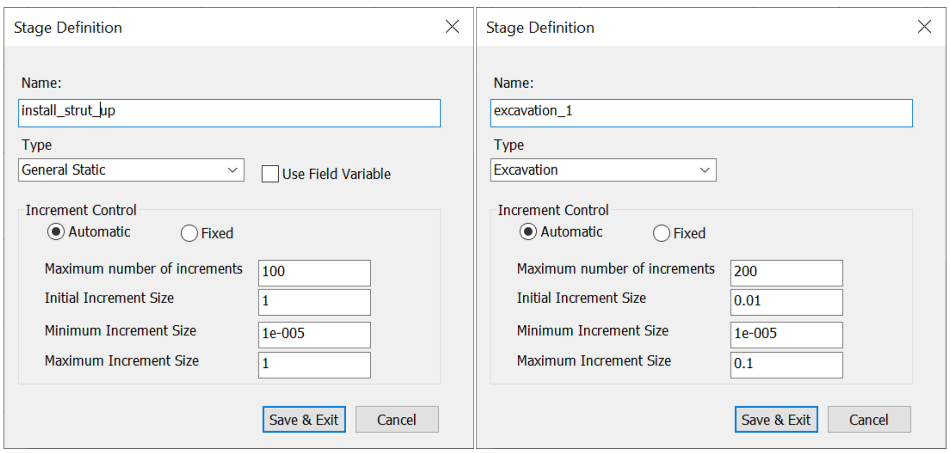


Figure 2.16-7: Define struts installation and excavation stages in example 16

Step 4: define soil material and beam section

Click the menu "**Data->Material Properties**" to open the material management window. Click "**New**" to define two new materials as shown in Figure 2.16-8. Click the menu "**Data->Material Assignment**". Check the option "By Set", select the set "setClay_up" from the set list, select "clay" from the material group list, click "**Assign**" if regions have no material assignment, or click "**Update**" if regions do have previously assigned materials. If regions in the same set have different material assignments, listed material group on the right of the set list will be displayed as "mixed". Next, assign material "clay_hard" to region set "setClay_hard".

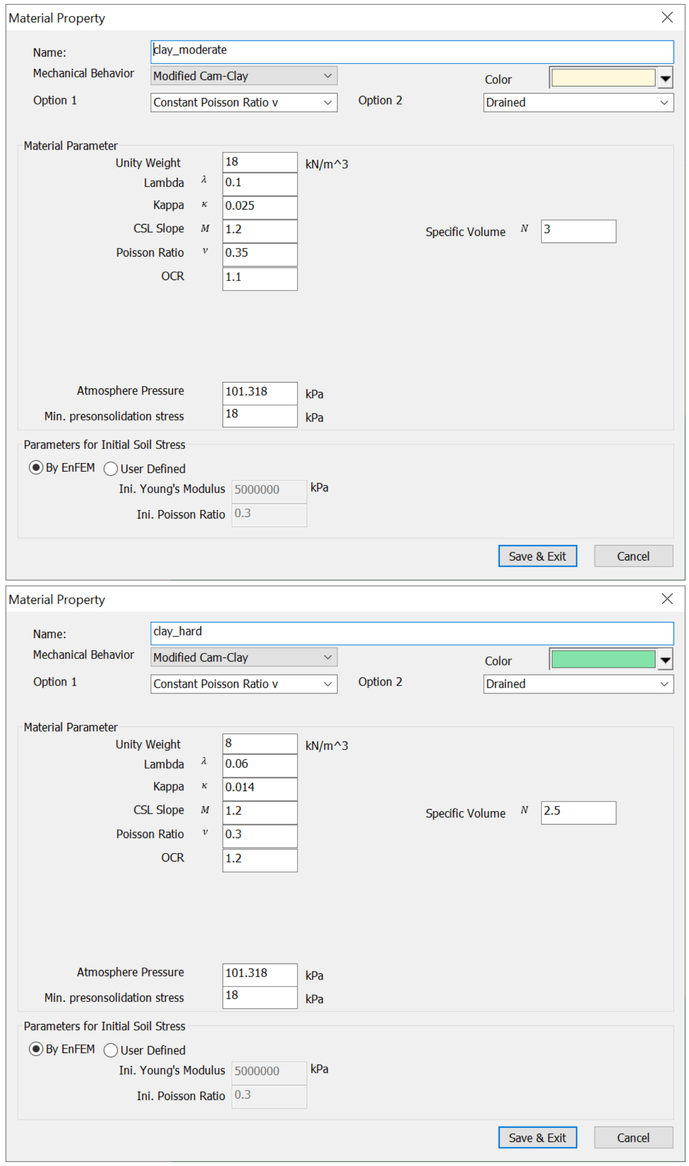


Figure 2.16-8: Define two modified CAM-Clay materials

Click the menu "**Data->Structural Section**" to open the structural section management window. Click "**New**" to define a new beam section "`secWall`" and a new strut section "`secStrut`" as shown in Figure 2.16-9. Click "**Save & Exit**" to save the new beam section into the model database. Click the menu "**Data->Structural Section Assignment**". Assign section "secWall" to the beam set "setWall". Assign the strut section "secStrut" to both strut sets "setStrutTop" and "setStrutBtm".

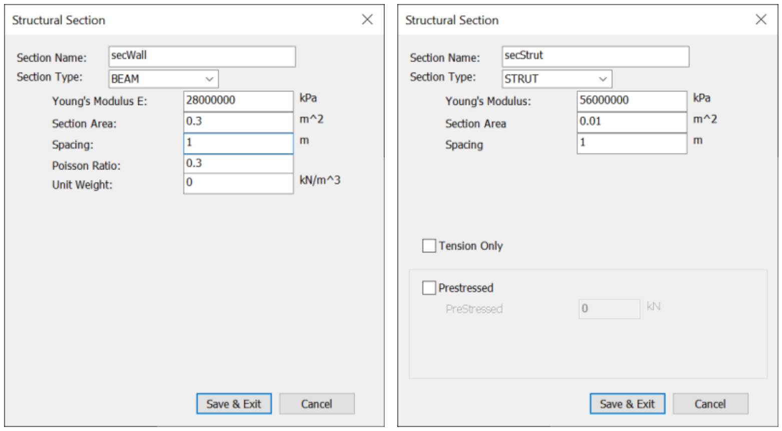


Figure 2.16-9: Define wall and strut sections

Step 5: define and assign boundary conditions

Click the menu "**Data->Boundary Conditions**" to open the boundary condition management window. Click "**New**" to create two displacement BC, one gravity load BC and one surface load BC as in Figure 2.16-10.

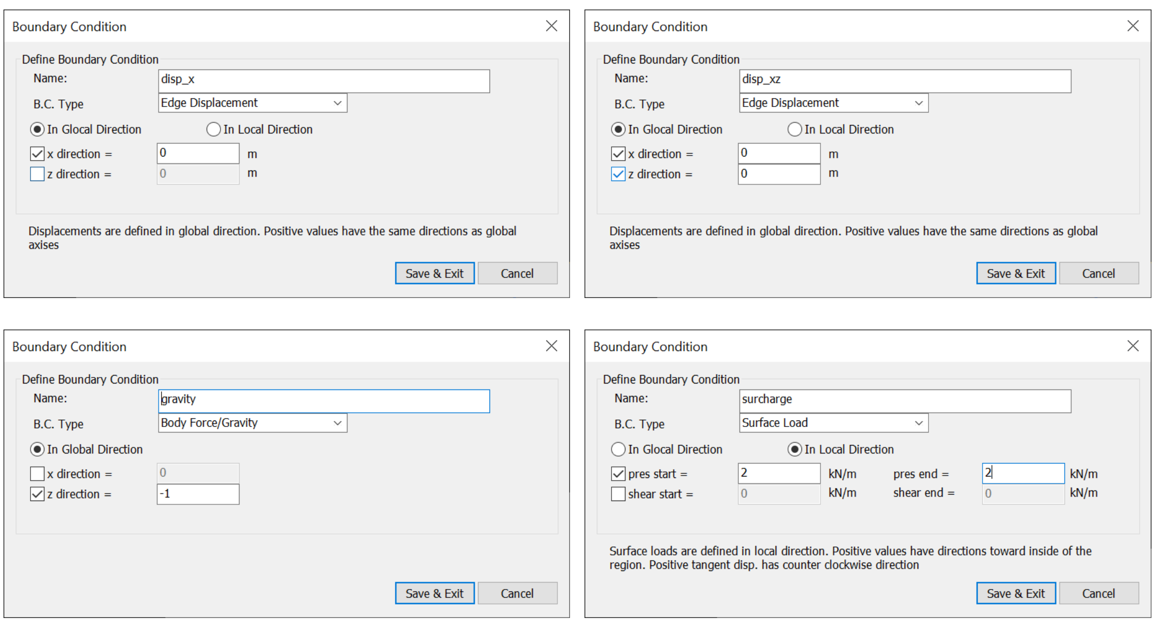


Figure 2.16-10: Define displacement and load boundary conditions

Click the menu "**Data->Boundary Assignment**". In the popping up dialog, click the BC "disp_x", select the first stage "ints" from the drop list of "*B*.C. Starts from Stage". Click "**Assign**", select the left and right edges of the whole soil part, RIGHT-CLICK and select "**End Selection**" to finish this BC assignment. Similarly, assign BC "disp_xz" to the bottom edges of the whole soil body, and assign "gravity" to all eight soil regions. Select the BC "surcharge" and assign it to the left and right surface edges as shown in Figure 2.16-11. Note that the start stage for BC "surcharge" will start from "install_strut_top".

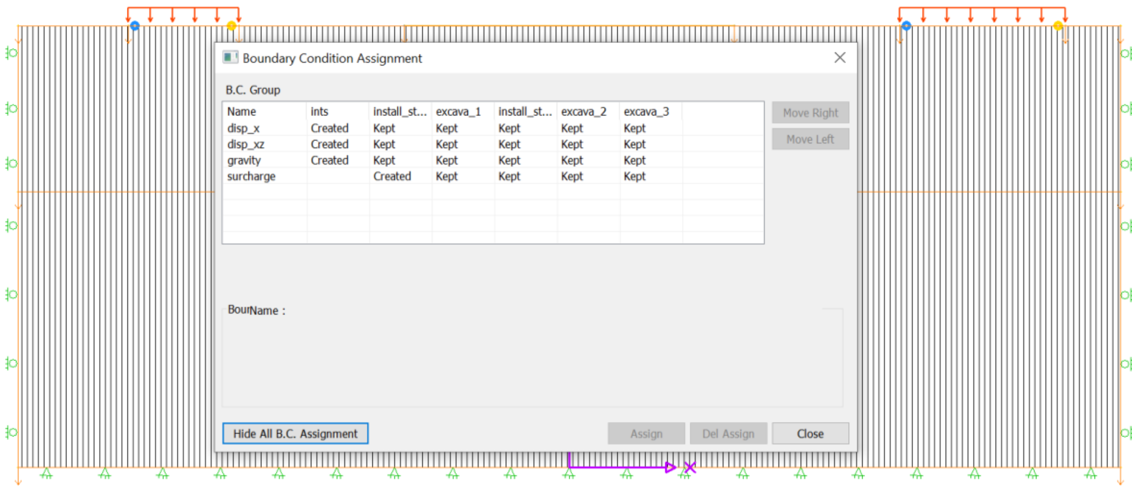


Figure 2.16-11: Assign surcharge BC in example 16

Click any BC group, corresponding boundary condition symbols will be displayed adjacent to selected geometric entities. Users may also click "**Show All Assignment**" to review all BC assignment simultaneously.

Step 6: Define soil-structure-interaction interface

Click the menu "**Data->Interaction->Zero Thickness Interface**" to open interface management dialog. Click "**New**" to create a new interface group. Keep the default group name, then input interface parameters as in Figure 2.16-12. The dilation angle is recommended to use the default value or input as "*0*". Click "**Add Beams**". Click or use a selection window to select right four beam segments. RIGHT-CLICK and select "**End Selection**" from the popup menu to finish adding beams to the interface. Click "**Add Regions**". Click or use a crossing window to select soil regions as highlighted in Figure 2.16-12. RIGHT-CLICK and select "**End Selection**" from the popup menu. Click "**Create Interface**". If the interface is successfully identified by EnFEM, two deep green dash lines will be added adjacent to the beam segments. Click "**Save & Exit**" to return to interface group management window. Similarly create the second interface group as shown in Figure 2.16-13.

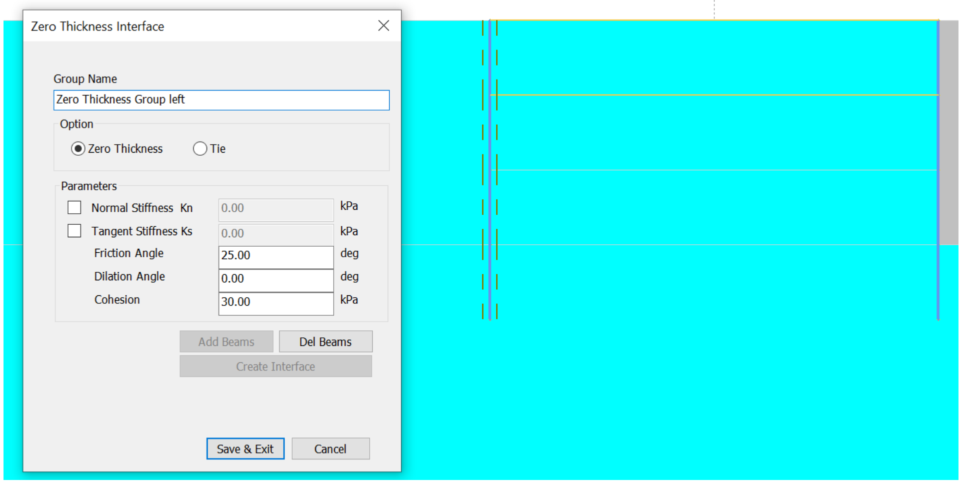


Figure 2.16-12: Define zero-thickness interface group for right wall

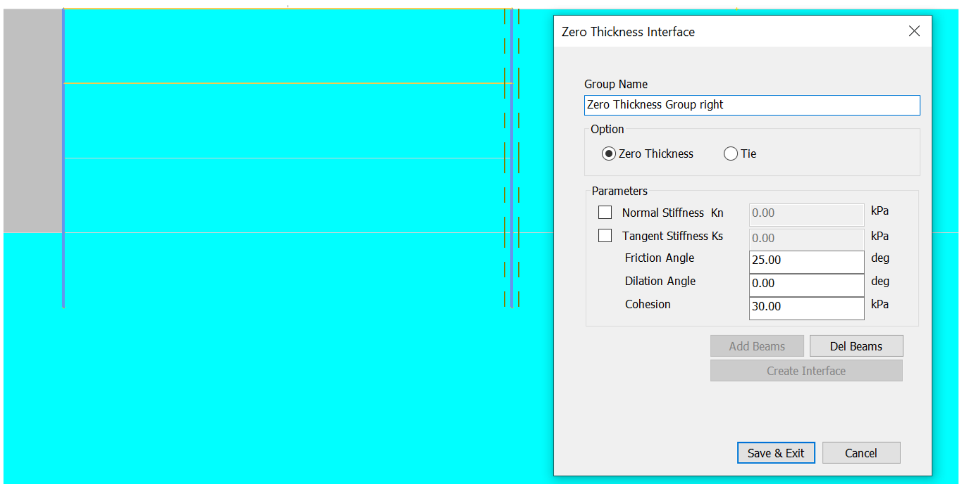


Figure 2.16-13: Define zero-thickness interface group for left wall

Step 7: Define deactivation/activation for excavation construction

Click the menu "**Data->Deactivation/Activation**". Click the tab "Region". In the stage column, all three defined stages should be listed. Double-click the cell locating on the "Set Deactivated" column and stage "excava_1" row, select the set "exca_1" (left image of Figure 2.15-13). Similarly set the region set "exca_2" to be deactivated in the stage "excava_2". Click the tab "Beam". Set the beam set "setBeam" to be activated in the initial soil stress stage "ints" (right image of Figure 2.16-14).

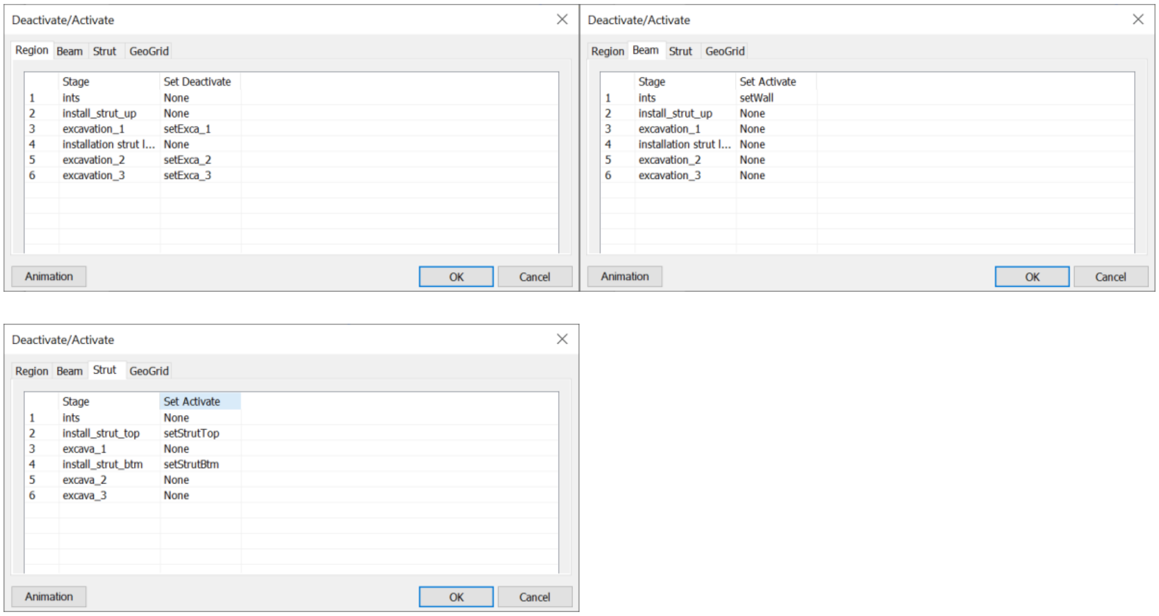


Figure 2.16-14: Set deactivation and activation in example 16

Click "**Animation**" to verify excavated bodies and construction sequence to be correctly defined.

Step 8: Mesh generation

Click the menu "**pre-Mesh->Advanced Mesh Control**". Click rectangle 3, then RIGHT-CLICK and select "**End Selection**". Set mesh property on this region as "Quadrilateral", "Structured" and "Quadratic Q8" with reduced integration rule (Figure 2.16-15Figure 2.16-15: Mesh property on excavated soil region). Set the same mesh property on remaining two excavated soil regions. Mesh properties on other regions are set to "Quadrilateral", "Unstructured" and "Quadratic Q8" with reduced integration rule. Click "**pre-Mesh->Global Seed Control**", and input element size =2. Click "**pre-Mesh->Local Seed Control**" to set element size on all beam segments to be "1". Use local seed control to define element size on edges connected with beam segments to be transited from "2" to "1". to Click the menu "**pre-Mesh->Run Mesh Generation**".

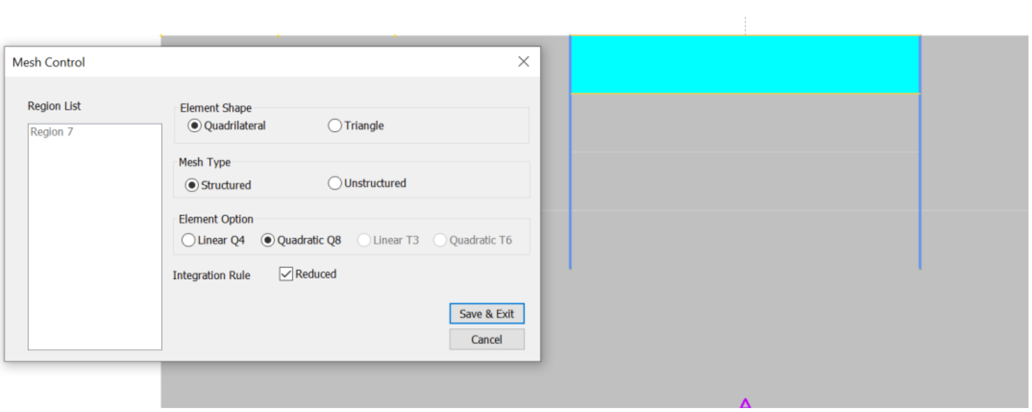


Figure 2.16-15: Mesh property on excavated soil region

The new mesh looks like that shown in Figure 2.16-6.

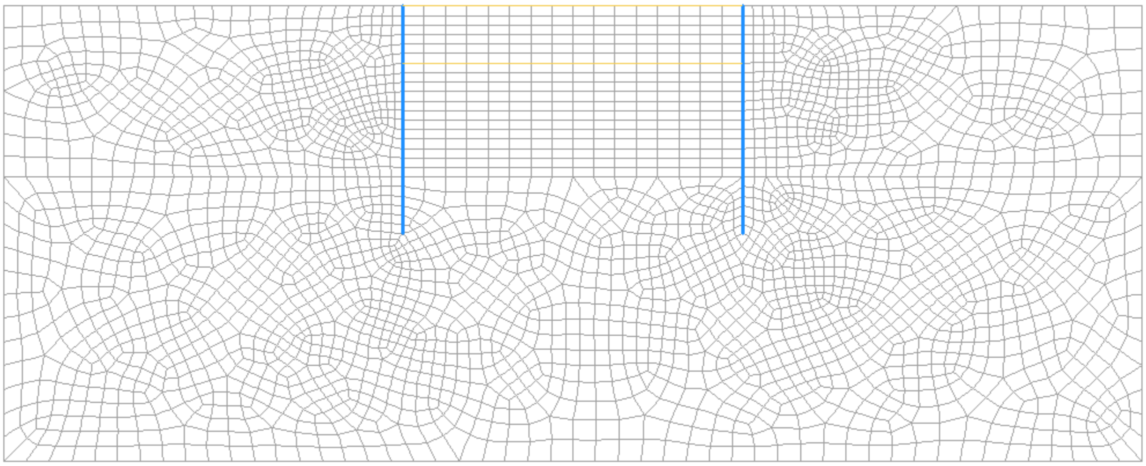


Figure 2.16-16: Final discretization on Example 16

If the mesh mode is not automatically switched, click  to switch to mesh mode. Since edges of solid elements and beam elements are overlapped on the interaction interface, users may use "probe" tool to investigate element connections. Click the menu "**Mesh->Probe Element**". Click to activate solid element selection icon . Click one solid element adjacent to the beam. Element connections and the element number will be displayed for selected element only. Deactivate the solid element selection and activate the beam element selection icon , users can check the element connection for a beam element.

Step 9: Analysis and result output

Click the menu "**Analysis->Analysis Setting**". Modify settings as necessary. Click "**OK**" to exit. Click menu "**Analysis->Run Analysis**". In the control dialog, check all three stages (Figure 2.16-17). Click "**Run**" to submit the job to the solver.

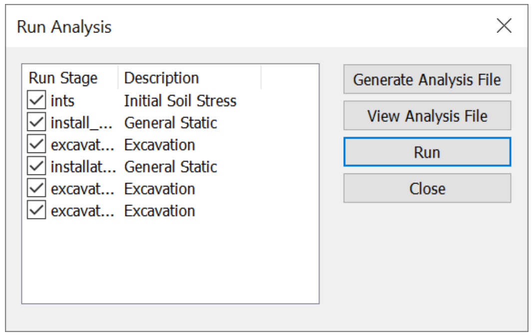


Figure 2.16-17: Dialog of "Run analysis" for example 16

After calculation is successfully completed or terminated due to errors, the message file (.log) will be opened automatically. It is recommended to review this file to check warning/error messages to update the model.

Click to switch to "RESULT MODE". In this example, result output will be focused on behavior of the retaining wall. Click the menu "**Results->Curve Plot for Structural Elements**", or click on the toolbar panel. In the popping up dialog, select "BEAM" as the type of structural element. Click "**Select By Set**". In the set selection dialog, select the beam set "setWall_left". Since element list is determined from geometric entity set, "User Defined Set" will replace real element list. Select "Internal Forces" from category, then select "Shear Force" from variables. Click the last load increment in the increment history. Click "**Plot**" to plot the result curve on the right window (Figure 2.16-18). If "Show Description" is checked, explanation of sign convention will also be plotted under curve title. Positive shear force will make a micro beam segment to rotate in counter clockwise direction.

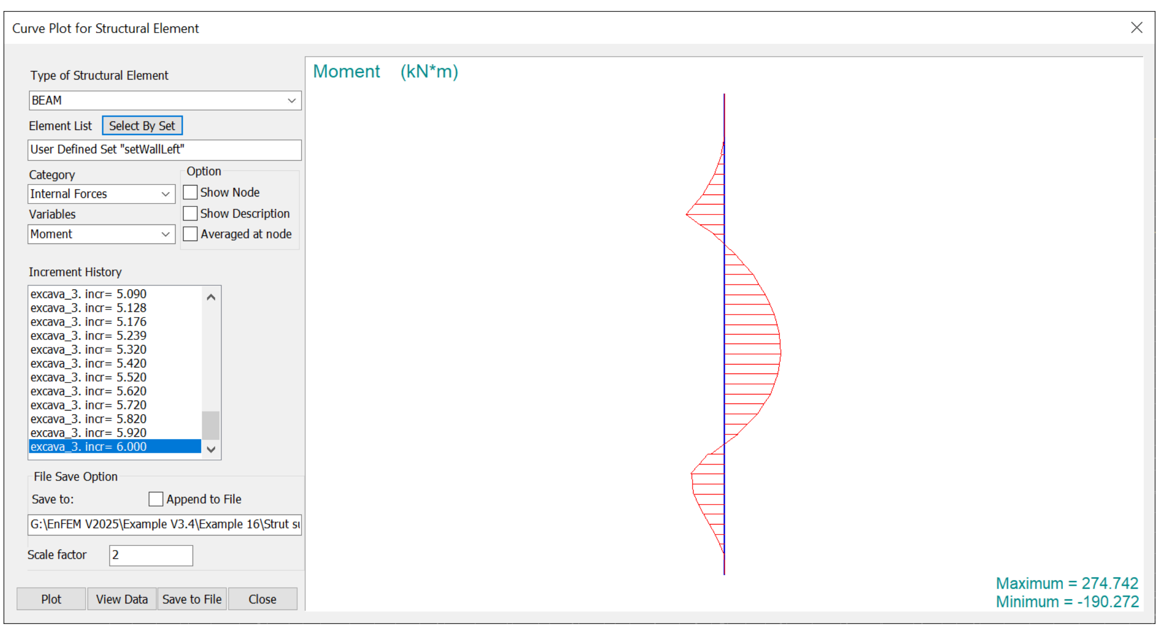


Figure 2.16-18: Shear forces on right retaining wall after excavation

Now change the category to "Interface". In the drop list under variables, normal pressure, tangent shear, normal status and tangent status will be available for both side of the beam. These output variables are distinguished as "*L*" or "*R*" which represent the left or right size along beam direction. Select "Tangential Status L" and click "**Plot**". Tangential statuses on the back of the right retaining wall is plotted as in Figure 2.16-19. Status "*0*" represent loss of contact (separation). Status "*1*" represents "sticking". Status "*2*" represents "sliding".

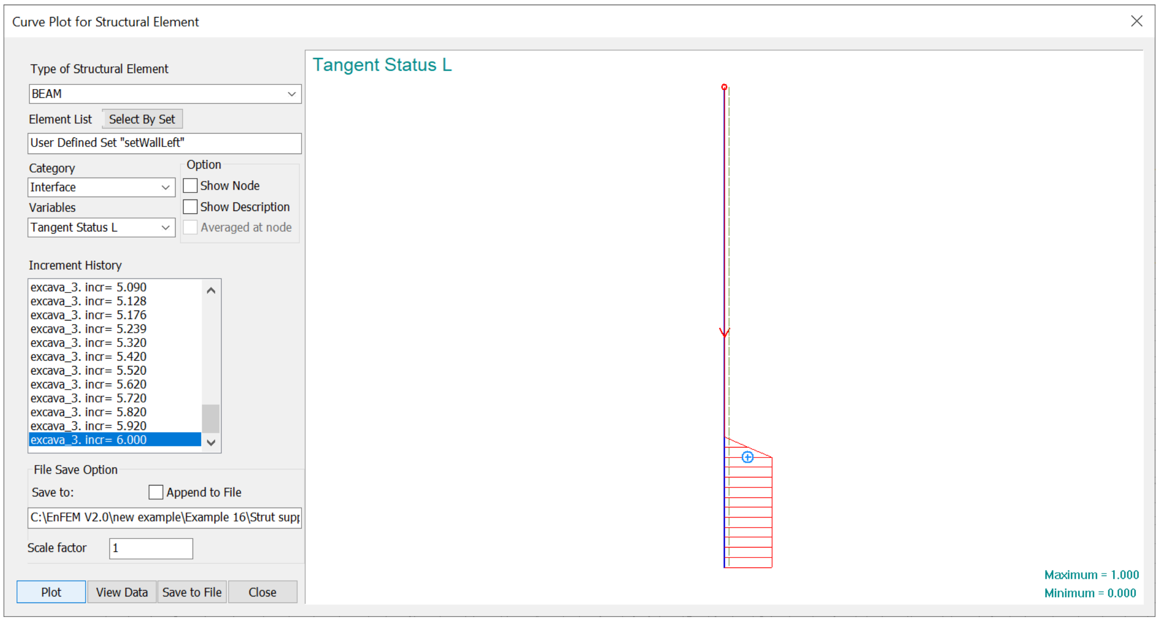


Figure 2.16-19: tangential status on back of right retaining wall

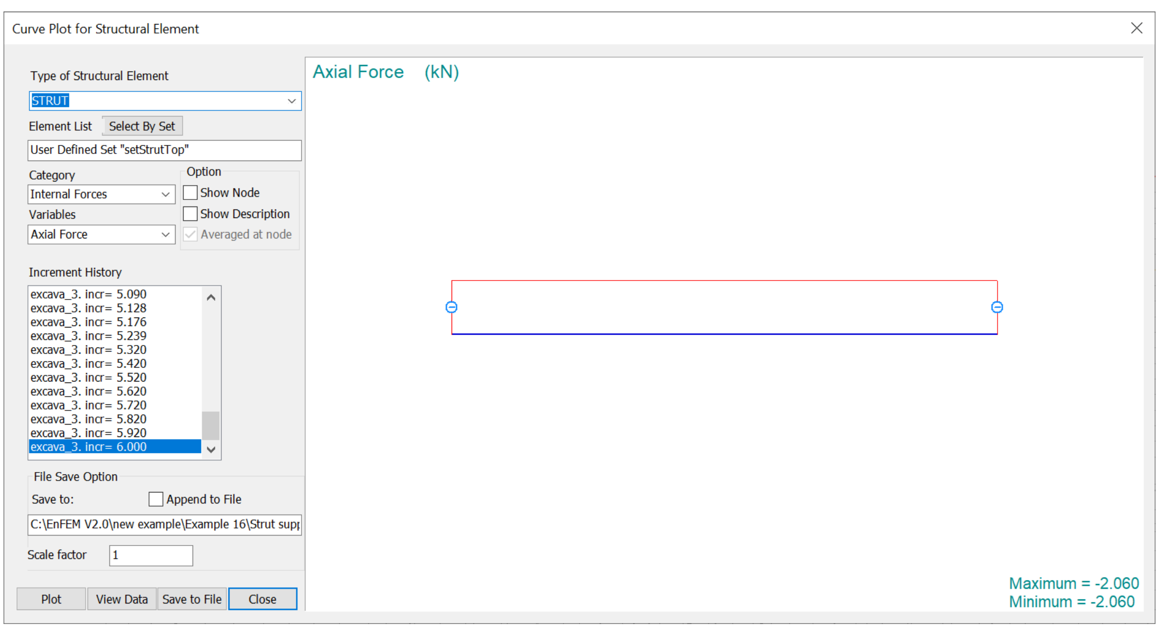


Figure 2.16-20: tangential status on back of right retaining wall

Change the type of structural element to “STRUT”. Click “**Select By Set**” and select the set “*setStrutTop*”. Select “Internal Forces” as the category. Then “Internal Forces” will be automatically because this is the only output for a strut structure. Select the last increment. Click “**Plot**” to view the internal compression force in this strut.

2.17 Example 17: Analysis of buried culvert using beam elements

The analysis of a buried culvert has been simulated in the example 4, where the culvert is discretized as solid elements and interface behavior is calculated by means of contact constrained method between solid elements. Same problem will be analyzed in this example. Beam elements are used for the culvert. Soil-structure interaction is simulated by zero-thickness interface elements. Since the culvert has 8 in thickness, geometry will be adjusted to let beams locate on the center of the culvert (Figure 2.17-1). Units in this example are "lbf, in, second".

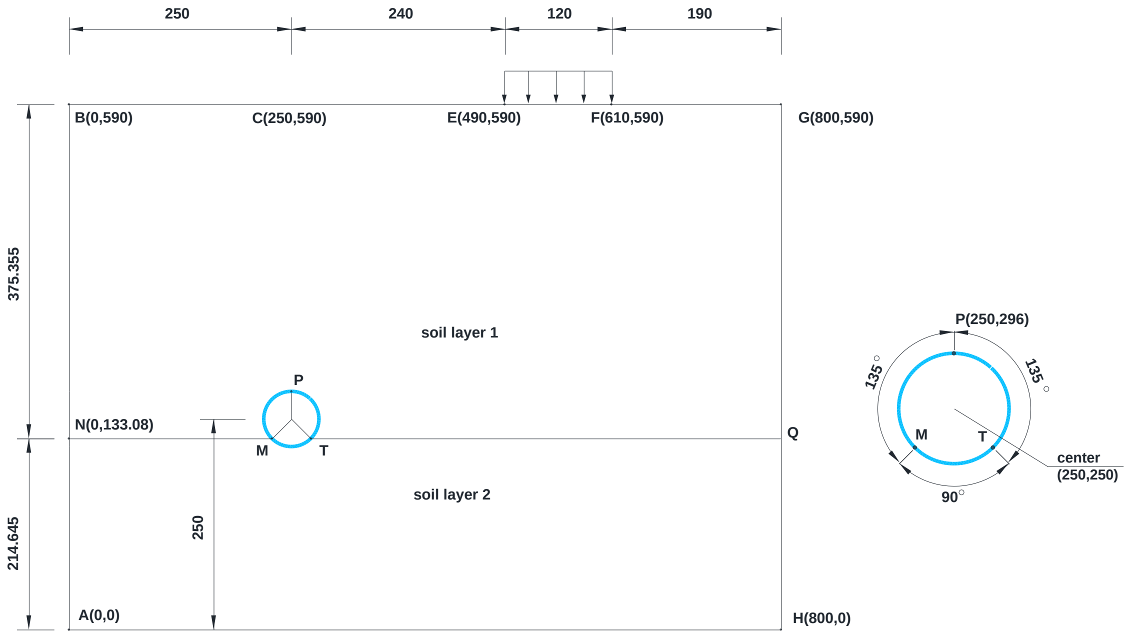


Figure 2.17-1: Geometry of buried culvert using beam elements

Material property:

<u>Culvert:</u>

- Unit weight:.................................$\gamma = 0.09\ pci$
- Young's Modules:.........................$E = 4.06 \cdot 10^6\ psi$
- Poisson's ratio:............................$\nu = 0.3$
- Section area:.................................$A = 8\ in^2$
- Spacing:...$1\ in$

<u>Top soil layer:</u>

- Unit weight:.................................$\gamma = 0.074 pci$
- Young's Modules:.........................$E = 2000\ psi$
- Poisson's ratio: ....$\nu = 0.45$
- Cohesive strength:.......................$c = 5\ psi$

- Friction angle:............................... $\phi = 30°$
- Dilation angle:................................ $\psi = 30°$
- Material model : .... Mohr-Coulomb plasticity

Bottom soil layer:

- Unit weight:.................................. $\gamma = 0.074\ pci$
- Young's Modulus:......................... $E = 8000\ psi$
- Poisson's ratio:............................ $\nu = 0.35$
- Cohesive strength:...................... $c = 10\ psi$
- Friction angle:.............................. $\phi = 35°$
- Dilation angle:.............................. $\psi = 35°$
- Material model : ....Mohr-Coulomb plasticity

Boundary Conditions:

- Displacement constraints on vertical external soil edges
- Displacement constraints on bottom edge of bottom soil layer
- Gravity load
- Ground surcharge of $70\ psi$
- Zero-thickness interface between wall and soil body

Zero-thickness interface for soil-structure-interaction:

- Normal stiffness: ....use default
- Tangent stiffness: .... use default
- Friction angle:............................. $\theta = 10\ °$
- Dilation angle:............................. $\varphi = 0\ °$
- Cohesion:...................................... $c = 5\ kPa$

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM, or click **"File->New"** to start a new project. Keep the default "Plane Strain" option. Select unit "lbf, in, second" as shown in Figure 2.17-2.

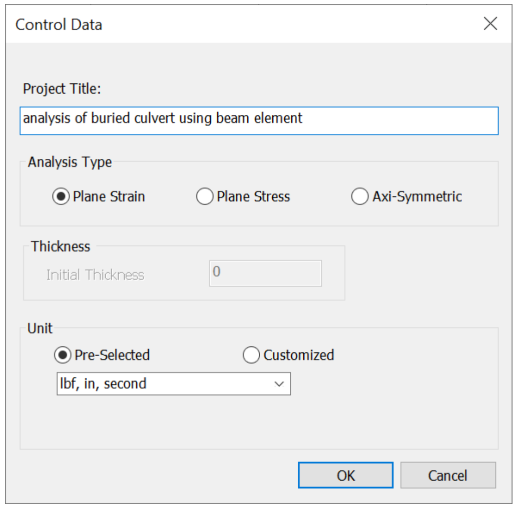


Figure 2.17-2: Control data in Example 17

Click **"Options->Drawing space"** and setup the drawing space (Figure 2.17-3). Click **"OK"** to exit.

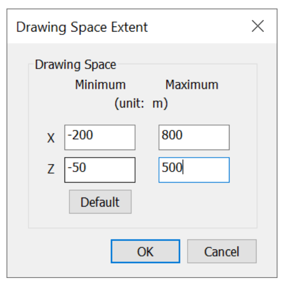


Figure 2.17-3: Setup drawing space for Example 17

Arc edges of soil regions are created first. Click to activate arc creation. In the coordinate input box, key in the coordinate of the arc center as "`250,250`", press RETURN to create the arc center vertex. Since the culvert radius is 46 in, key in the relative coordinate of the arc start vertex "*P*" as "`[0,46`]". Next, key in "`[46,135]` " where the second number "*135*" represents the arc angle 135°. Press RETURN to complete the first arc. Press and hold CTRL, "snap" to the arc center. Snap to the end vertex of the first arc (point "*M*"). Key in "`[46,90]`" and press RETURN. The second arc with 90° arc angle is generated. Hold CTRL, "snap" to the arc center, "snap" to the end vertex of the second arc (point "*T*"), then complete the last arc by "snap" to point "*P*" of the first arc to finish the whole culvert.

Click on the toolbar panel to for line creation. Key in "`0,0`" to generate the line start vertex. Key in the absolute coordinate "`0,590`" to create the left soil boundary. The point "*N*" line can be determined by line dividing method. Hold CTRL, "snap" to point "*M*". LEFT-CLICK to create the line start vertex. Key in the end vertex such as "`[-300,0]`" which should be far enough to intersect with the line AB. RIGHT-CLICK and select "**End Selection**" to terminate continuous line creation. Click the menu "**Modify->Divide Line/Arc**". Select the intersected two lines. RIGHT-CLICK and select "**End Selection**". Previously selected two lines are divided into four line segments. Delete unneeded lines. Using the same method to create soil boundary line and horizontal soil interface line on the right of the culvert.

Click  if necessary. Hold CTRL and "snap" to point "*B*". Then continuously create line segments using relative coordinates "*[250,0]*", "*[240,0]*", "*[120,0]*", "*[190,0]*". Using "snap" method to create the line AH.

Click the menu "**Create->Polygon from Lines/Arcs**". Select line and arc components of soil layer 2. RIGHT-CLICK and select "**End Selection**". Recreate the line NM and TQ. Then create the polygon for the soil layer 1 using the same method.

Click the menu "**Create->Arc Beam**" or click the icon  to create arc beam. Using "snap" method to create three arc beam segments overlapping on soil boundary arcs created previously.

Step 2: Define region set and beam set

In "CAD MODE", click the menu "**Tools->Set Management**" to open the set management window. Select the option "Region", click "**New**" to create a new region set. Key in the set name as "*setSoil_top*". Click "**Add Obj.**", select soil layer 1 in Figure 2.17-1, RIGHT-CLICK and select "**End Selection**". Click "**Save & Exit**" to return to the set management window. Similarly create a new set "*setSoil_bottom*" for soil layer 2. Create a region set "*setSoil_all*" for both soil layers (Figure 2.17-4).

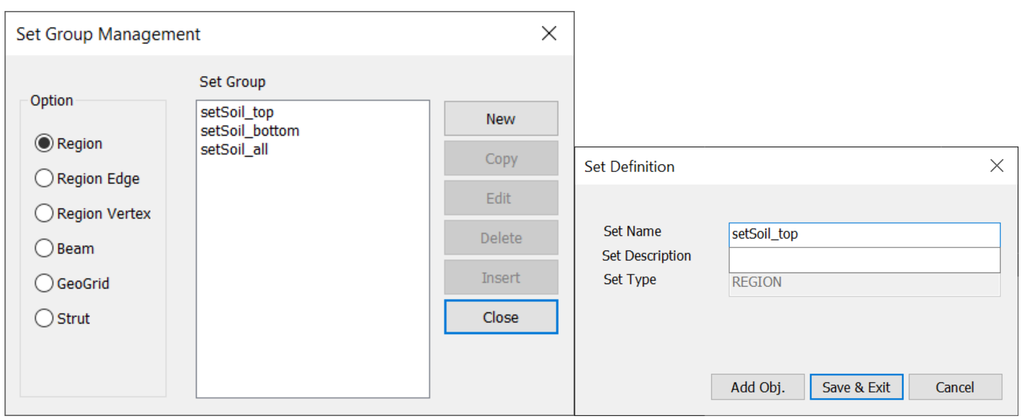


Figure 2.17-4: Set management in example 17

Select the option "Beam" in Figure 2.17-4 and create one new beam set "*setCulvert*" including all culvert beam arcs.

Beam segments for the same structure entity should be head-toe connected and have consistent directions. EnFEM provides a tool to view beam directions. Click the menu "**View->Show Direction**" to turn on display of beam directions. The tangential direction (red color) is defined from the beam start vertex to the end vertex. The normal direction (green color) is counter clockwise 90° from the tangential direction. When straight beams are created, beams may have inconsistent directions. Users can click the menu "**Modify->Reverse Beam Direction**" and select individual beam to reverse its direction.

Step 3: Define stage data

Click the menu "**Data->Stage Management**". Click "**New**" to create a new stage "*ints*" in type of "**Initial Soil Stress**". Click "**Initial Soil Stress Option**", keep the default option as "**K0 method**" to estimate initial soil stresses, click "**Add**", select both soil regions, RIGHT-CLICK, then select "**End Selection**" to add soil regions. Click "**Input Data**" to open the window for initial soil stress interpolation. In the new initial stress window, click "**Add Row**" to add two new data rows. Input the vertical coordinates of top and bottom soil body. These lateral stress factors are estimated from Poisson's ratio. Then input values of vertical stress and lateral stress factor k_x, k_y as shown in Figure 2.17-5. Click "**Save & Exit**" to exit to initial stress option dialog. Then click "**OK**" to exit to stage management window.

Create a new stage "*surcharge*" in type of "**General Static**". Keep the default settings for increment option and increment size.

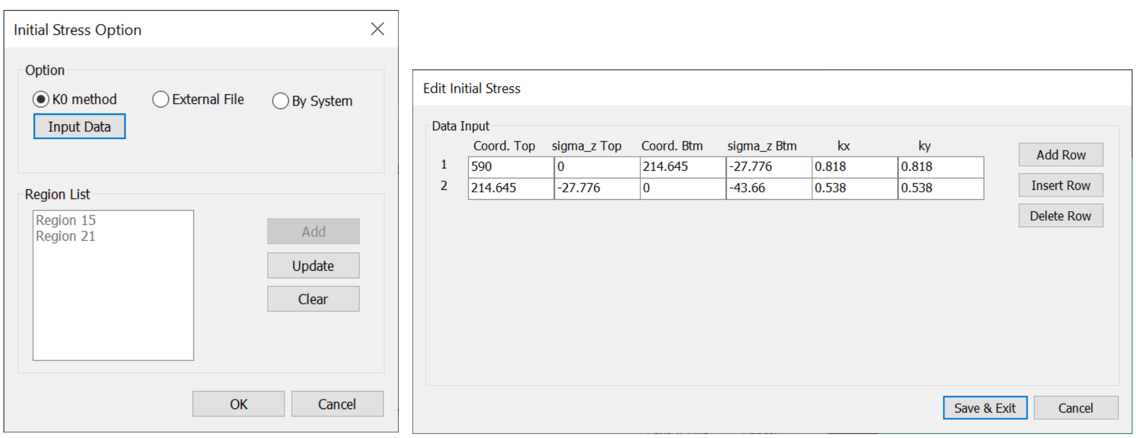
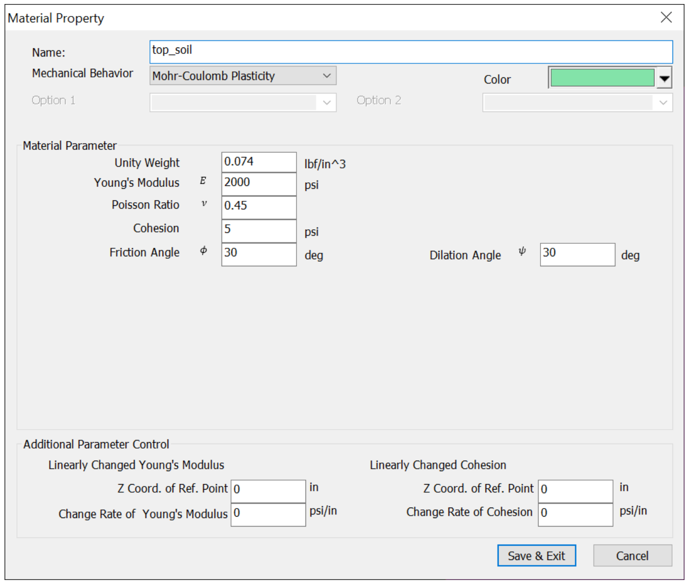


Figure 2.17-5: Define initial soil stress for example 15

Step 4: Define soil material and beam section

Click the menu "**Data->Material Properties**" to open the material management window. Click "**New**" to define two new materials as shown in Figure 2.17-6. Click the menu "**Data->Material Assignment**". Check the option "**By Set**", select the set "*setSoil_top*" from the set list, select "*top_soil*" from the material group list, click "**Assign**". Similarly, assign material "*btm_soil*" to the region set "*setSoil_bottom*".



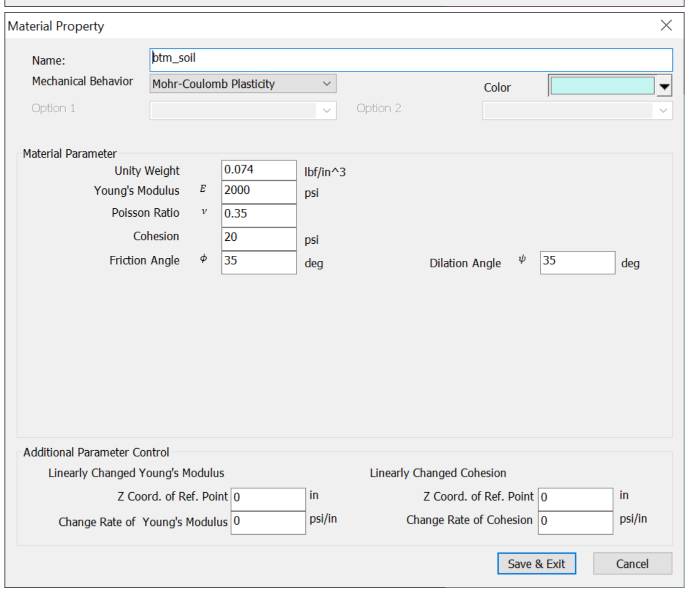


Figure 2.17-6: Define and assign Mohr-Coulomb in Example 17

Click the menu "**Data->Structural Section**" to open the structural section management window. Click "**New**" to define a new beam section as shown in Figure 2.17-7. Click "**Save & Exit**" to save the new beam section into the model database. Click the menu "**Data->Structural Section Assignment**". The default option is "Beam". Users may select assign by "Set" or by "Geometric Entity". Click the set name or the geometric entity, select "secCulvert" from the section type list, then click "**Assign**". If selected structure already has a section, click "**Update**" to refresh section assignment.

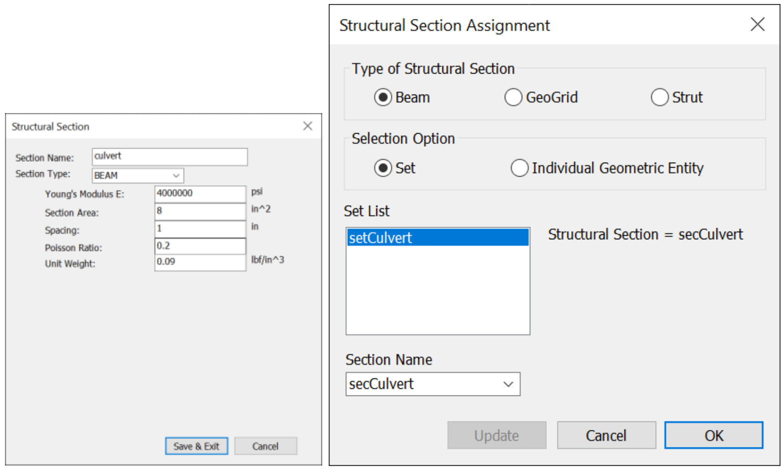


Figure 2.17-7: Define and assign beam section to culvert

Step 5: Define and assign boundary conditions

Click the menu "**Data->Boundary Conditions**. Click "**New**" to create two displacement boundary conditions, one gravity load boundary and one surface load as shown in Figure 2.17-8.

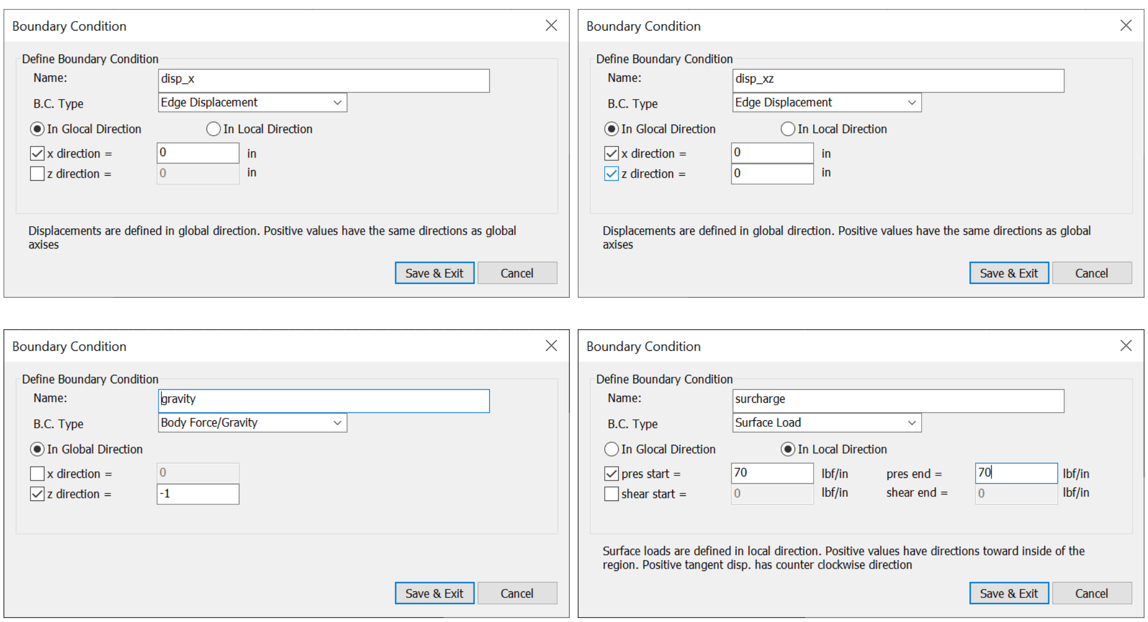
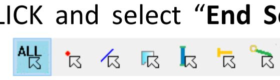


Figure 2.17-8: Define displacement and load boundary conditions

Click the menu "**Data->Boundary Assignment**", click the BC "disp_x", click "**Assign**", select all left and right vertical edges of soil layers, RIGHT-CLICK and select "**End Selection**" to finish this BC assignment. Similarly, assign BC "disp_xz" to the bottom edge of the soil body. Select BC "gravity". Click "**Assign**", select all regions and beams. RIGHT-CLICK and select "**End Selection**" (Figure 2.17-9). Note the selection filter is set to"Select All"  because region entities and structure entities will be selected simultaneously here.

Since the surcharge will be applied during the second stage, click “**Created**” in the “surcharge” row, then click “**Move Right**” to change the start stage for this surcharge load. Select the second column of the surcharge BC “surcharge”, click “**Move Right**” to let this BC start from the second stage “surcharge”. Click “**Assign**” to assign this BC to line “*EF*” in Figure 2.17-1.

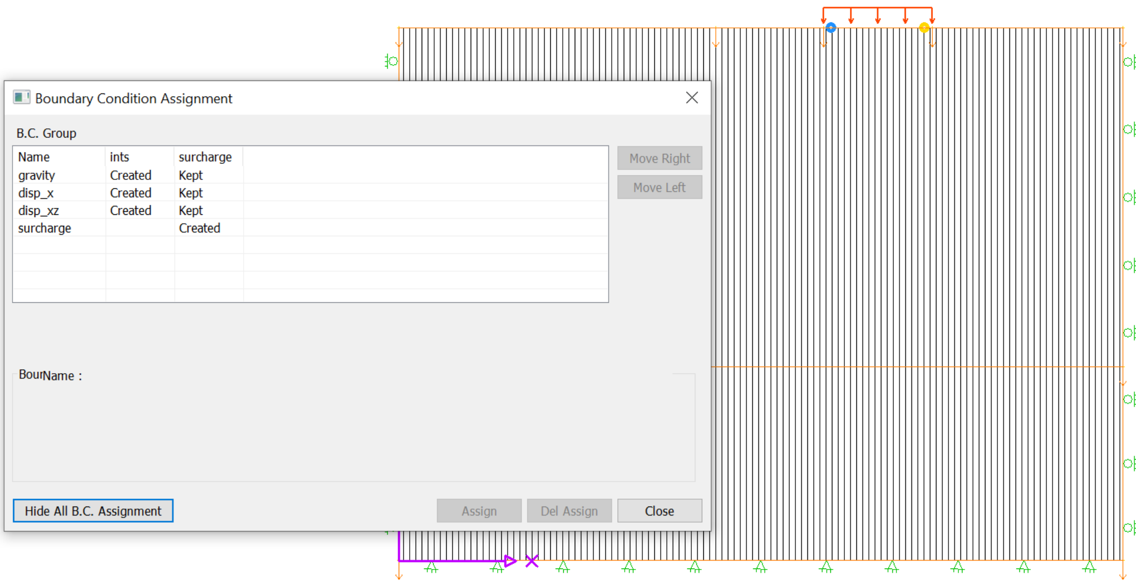


Figure 2.17-9: View all boundary conditions in Example 17

Step 6: Define soil-structure-interaction interface

Click the menu “**Data->Interaction->Zero Thickness Interface**” to open interface management window. Click “**New**” to create a new interface group. Input interface parameters as in Figure 2.17-10. Change the default friction angle to 30, and keep the default value for the dilation angle. Click “**Add Beams**”. Click or using a selection window to select all three beam segments. RIGHT-CLICK and select “**End Selection**” to finish adding beams to the interface. Click “**Create Interface**”. If the interface is successfully created, deep green dash lines will be added adjacent to the beam segments (Figure 2.17-10). Click “**Save & Exit**” to return to interface group management window. Click “**Close**” to exit.

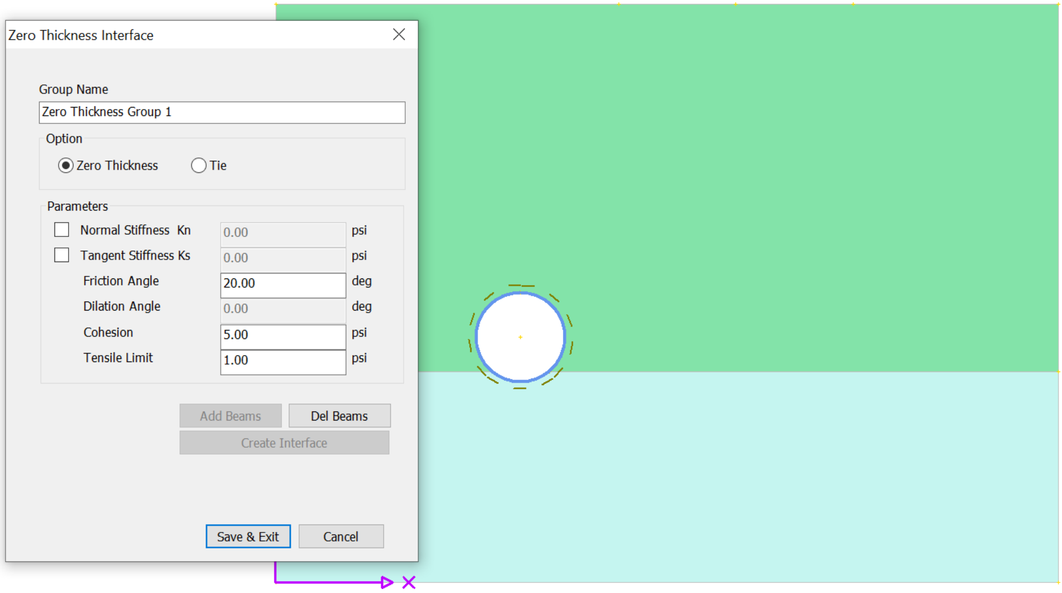


Figure 2.17-10: Define zero-thickness interface in example 17

Step 7: Define deactivation/activation for excavation construction

Click the menu "**Data->Deactivation/Activation**". Since there are no regions excavated in this analysis, keep region set deactivated as "None". Click the tab "Beam". Set the beam set "setCulvert" to be activated in the initial soil stress stage "ints" (Figure 2.17-11).

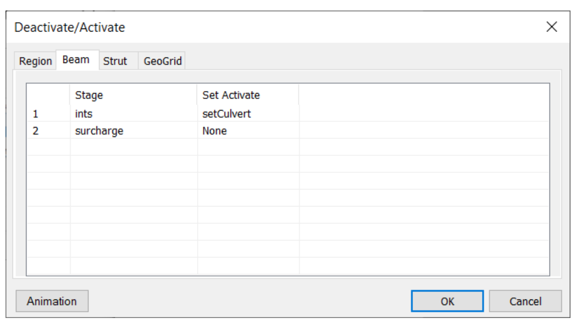


Figure 2.17-11: Set activated beams in Example 17

Step 8: Mesh generation

Click the menu "**pre-Mesh->Advanced Mesh Control**". Select both soil regions. RIGHT-CLICK and select "**End Selection**" from the popup menu. The mesh control dialog will be opened. This model still uses the quadrilateral Q8 element for its excellent performance. Set the element shape to "Quadrilateral", mesh type to be "Unstructured". Element option is set to Q8 with reduced integration rule (Figure 2.17-12). Since beam elements in EnFEM are three-node with quadratic interpolation functions, paired solid elements should be quadratic too (Q8 or T6). Click "**Save & Exit**" to save mesh data to the model database.

Click the menu "**pre-Mesh ->Global Seed Control**". Input number "*30*" as the global element size. This element size will generate coarse element seeds on external boundaries. Click "**Assign**" then click "**Close**" to exit. The culvert part should be refined and lines adjacent to the culvert should be adjusted to obtain a high-quality mesh. Click the menu "**pre-Mesh ->Show Edge Division**" and "**pre-Mesh ->Show Seeds**" to turn on display of edge discretization and seed location. Click the menu "**pre-Mesh ->Local Seed Control**". Use a crossing window to select the left arc beam and overlapped region boundary. RIGHT-CLICK and select "**End Selection**". In the local seed control dialog, set the division number to "*12*" as in Figure 2.17-13. Click "**Assign**", then click "**Close**". Similarly set division "*12*" and "*8*" to the right and bottom culvert segments and overlapped region edges.

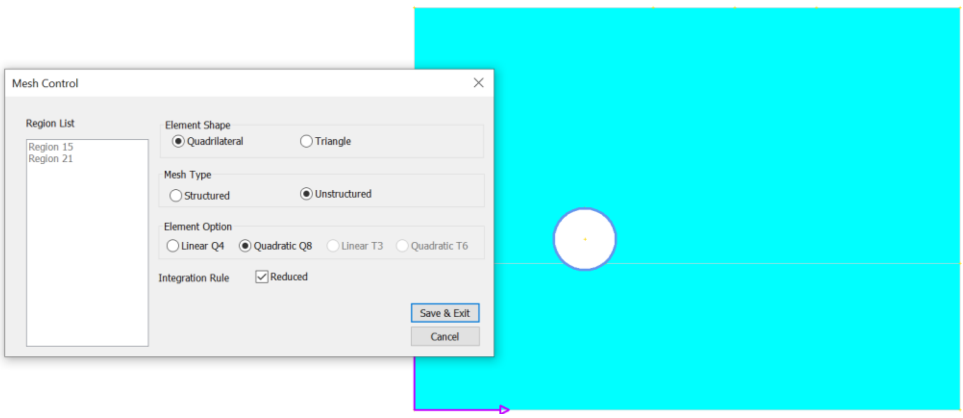


Figure 2.17-12: Mesh control for regions in example 17

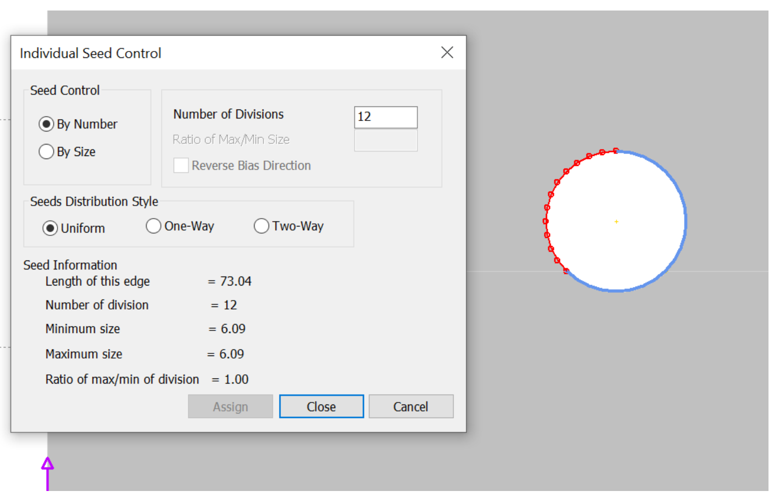


Figure 2.17-13: Local mesh refinement for culvert segment

Click the menu "**Mesh->Local Seed Control**". Select the horizontal line to the left of the culvert. RIGHT-CLICK and select "**End Selection**". Change the option of seed control to "By Size", distribution style to "One-Way". If the seed concentration is inverted, check "Reverse Bias Direction". Change the minimum and maximum size to "*5*" and "*30*" respectively. Click "**Assign**" then click "**Close**" to exit. Following same procedures to modify the right adjacent line to concentrate toward the culvert with sizes varying from "*5*" to "*30*".

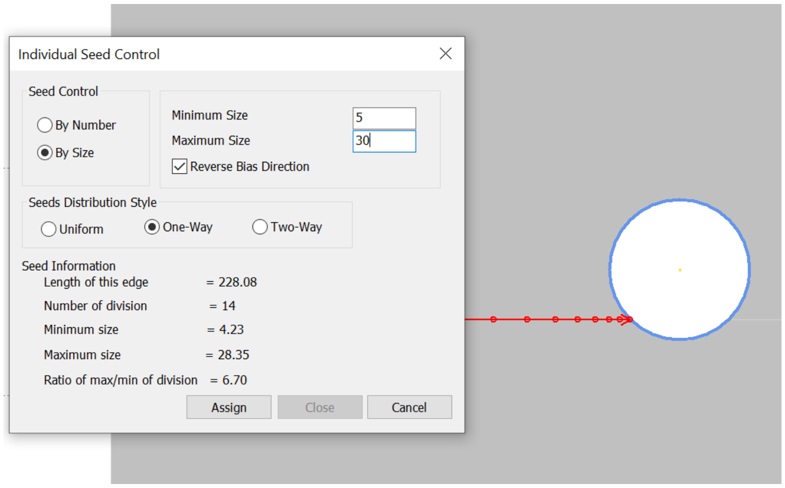


Figure 2.17-14: Local mesh refinement for culvert segment

Click the menu "**Mesh->Run Mesh Generation**" to finish mesh generation.

Step 9: Analysis and result output

Default analysis settings are appropriate for this model. Click menu "**Analysis->Run Analysis**". In the control dialog, check all stages (Figure 2.17-15). Click "**Run**" to submit the job to the solver.

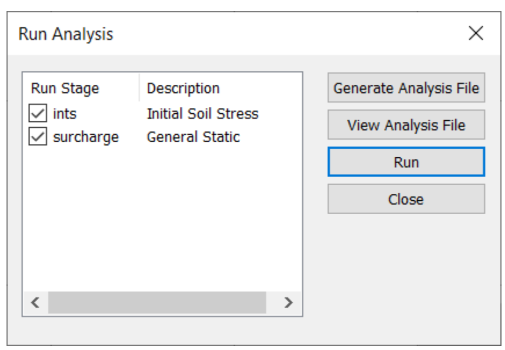
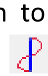


Figure 2.17-15: Run analysis in example 17

After calculation is successfully completed or terminated due to errors, the message file (.log) will be opened automatically. It is recommended to review this file to check convergence or warning/error messages.

Click  to switch to "RESULT MODE". Click the menu "**Results->Curve Plot for Structural Elements**" or click  on the toolbar panel to open the window of curve plot for structural element. By default, "BEAM" is selected as the type of structural element. If beam sets are predefined, moment curve will be plotted automatically by default (Figure 2.17-16). Users may click "**Select By Set**" to select other beam sets. If shear force is expected, select "Internal Forces" from category, click "Shear" from variables, then click "**Plot**" to plot other internal forces along the culvert.

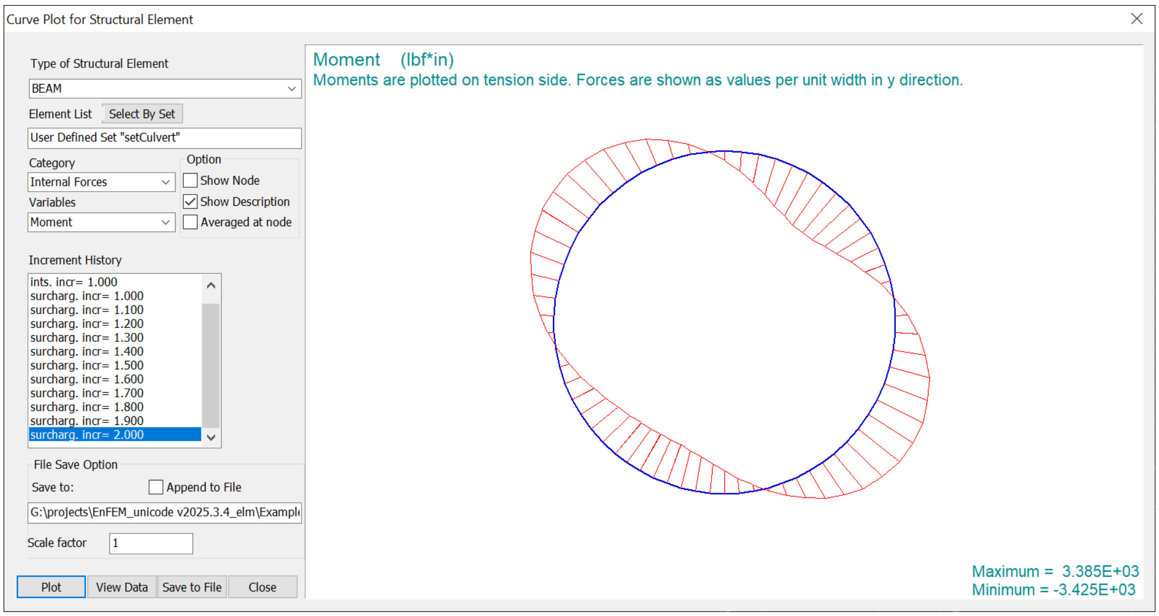


Figure 2.17-16: Moment plot for culvert

Change to "Interface" in category, select "Normal Pressure R". Click "**Plot**" to plot the normal pressure on the interface. Since the culvert has only one side contacting with soil, the left side has no interface (Figure 2.17-17).

The culvert can also be discretized as solid elements as in example 4. Soil-structure-interaction can be simulated using "contact constrained method". Comparison between these two approaches is presented in example 4.

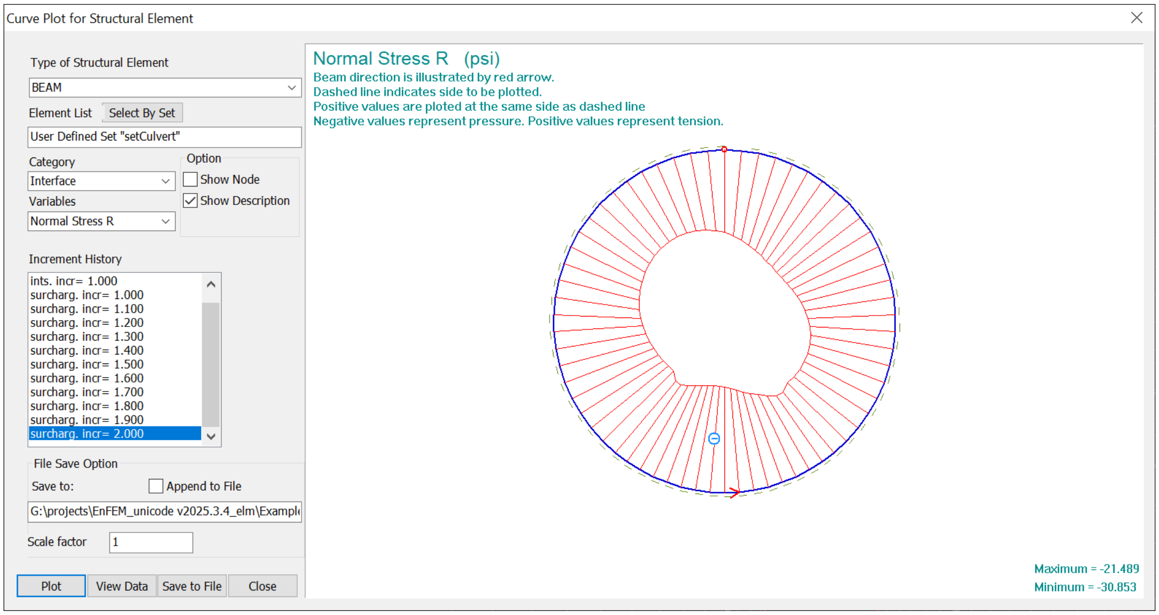


Figure 2.17-17: Normal pressure on interface

2.18 Example 18: Excavation and tie-backs supported retaining wall

Tie-backs supported retaining walls are common in excavations or slope protection. In this example, how to use tie-backs in EnFEM is illustrated. This example also introduced how to define prestressed forces in the tie-backs. Units in this example are “kN, m, second”.

Material property:

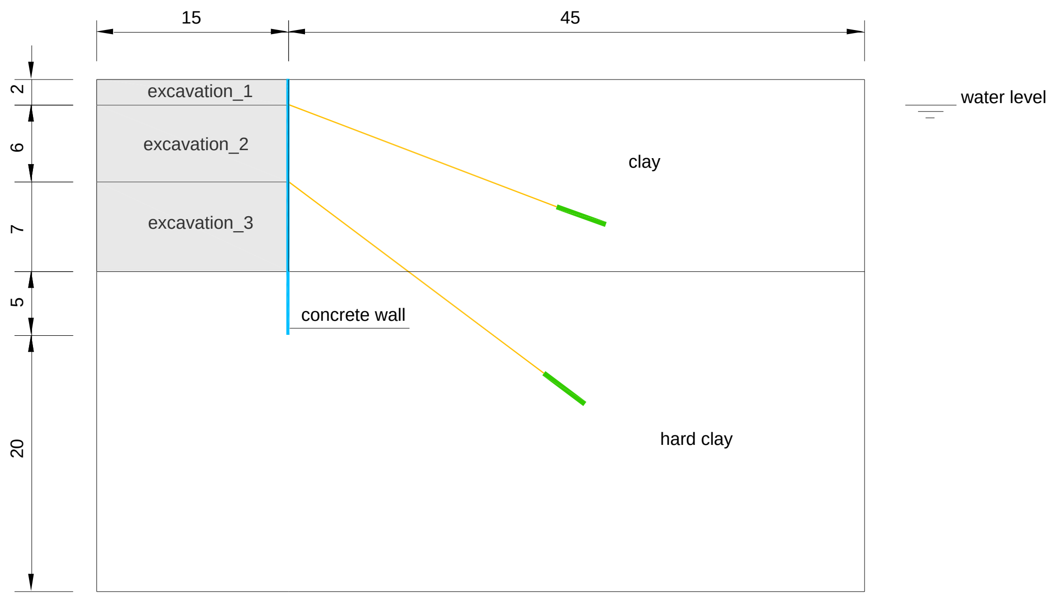


Figure 2.18-1: Geometry of tie-back supported retaining wall

Diaphragm wall:

- Unit weight: .... $\gamma = 0$
- Poisson's ratio: .... $\nu = 0.2$
- Young’s modulus: .... $E = 2.8 \cdot 10^7\ kPa$
- Section area: .... $A = 0.3\ m^2$
- Spacing: .... $1\ m$

Strut:

- Young’s modulus: .... $E = 1.0 \cdot 10^7\ kPa$
- Section area: .... $A = 0.005\ m^2$
- Spacing: .... $1m$

Geogrid:

- Young’s modulus: .... $E = 8.4 \cdot 10^{10}\ kPa$
- Section area: .... $A = 0.001\ m^2$
- Spacing: 1m

Clay:

- Unit weight:.. γ=18kN/m^3, (γ=8kN/m^3 for submerged)
- Slope of the normal compression line:.................. lambda λ =0.1
- Slope of the swelling line: .... kappa κ =0.025
- Slope of critical line: .... M =1.2
- Specific volume number:.. N =3
- Poisson's ratio:.. υ =0.35
- Over-consolidation ratio:.. *OCR=1.1*
- Drained

Hard Clay:

- Unit weight:.. γ=8kN/m^3(submerged)
- Slope of the normal compression line: .... lambda λ =0.07
- Slope of the swelling line:.................................. kappa κ =0.009
- Slope of critical line:.. M =1.1
- Specific volume number:.. N =2.5
- Poisson's ratio:.. υ =0.3
- Over-consolidation ratio:.. *OCR=1.2*
- Drained

Boundary conditions:

- Displacement constraint on vertical soil boundaries.
- Displacement constraint on bottom edge of the soil region.
- Gravity load
- Zero-thickness interface between wall and soil body.

Zero-thickness interface for soil-structure-interaction:

- Normal stiffness:.. use default
- Tangent stiffness: .... use default
- Friction angle: .... $\theta = 20°$
- Dilation angle: .... $\varphi = 0°$
- Cohesion: .... $c = 5\ kPa$

Model development and analysis steps:

Step 1: Create geometric entities

Start EnFEM, or click **"File->New"** to start a new project. Input title, select appropriate analysis type as "Plane Strain", and select preselected unit as "kN, m, second" (Figure 2.18-2).

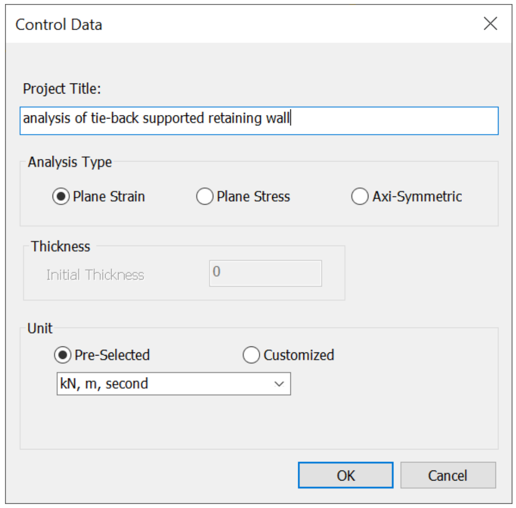


Figure 2.18-2: Control data in Example 18

Click **"Options->Drawing space"** and set the drawing space like that shown in Figure 2.18-3. Click **"OK"** to exit.

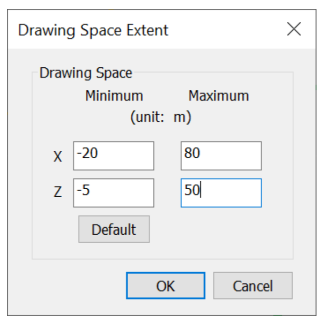
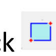


Figure 2.18-3: Setup drawing space for Example 18

The soil part can be divided into seven four-side rectangle regions (Figure 2.18-4). Click  to activate the rectangle creation tool. Key in the coordinate "0,0" as the left-bottom corner of rectangle 5 in the coordinate input box, press RETURN to create it. Key in relative coordinate "[15,25]" as the diagonal vertex using relative coordinate. Similarly create the rectangle 6 using diagonal vertexes (x=15, z=25) and (x=60, z=0). Using the same method to create remaining rectangles.

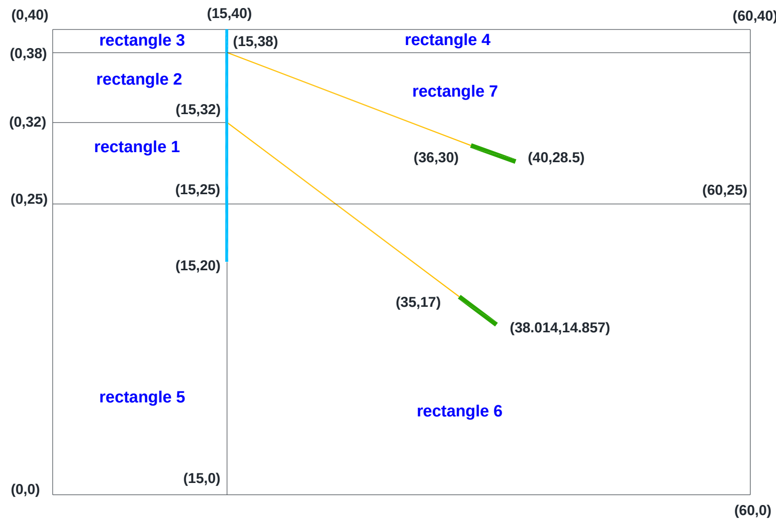


Figure 2.18-4: Regions and coordinates of control points

Click to create straight beams. Hold CTRL to snap to vertex (15, 40) and key in "*15,20*" as the end vertex. RIGHT-CLICK and select "**End Creation**". Note that EnFEM automatically inserts essential vertexes on region edges to keep geometry compatibility.

Click  to create geogrids. Key in coordinates to create a geogrid from start vertex (x=36, z=30) to vertex (x=40, z=28.5). Similarly create the second geogrid from coordinate (x=35, z=17) to (x=38.014, z=14.857). Note that each geogrid had only one segment. EnFEM don't support continuous geogrids with more than one segment.

Click  to create struts. Hold CTRL and snap to the first beam vertex at coordinate (x=15, z=38), then "snap" to the geogrid vertex (36,30). Similarly create another strut from beam vertex (x=15, z=32) to the geogrid vertex (x=35, z=17).

Step 2: Define region set and beam set

Click the menu "**Tools->Set Management**" to open the set management window. Select the option "**Region**" and click "**New**" to create a new region set "*setExcava_1*". Click "**Add Obj.**" and select rectangle 3 shown in Figure 2.18-4, RIGHT-CLICK and select "**End Selection**". Click "**Save & Exit**" to return to the set management window. Similarly create a new set "*setExcava_2*" for region rectangle 2 and a set "*setExcava_3*" for rectangle 1. Create a region set "*setClay_dry*" from rectangle 3 and 4. Create a region set "*setClay_hard*" from rectangle 5 and 6. Create a region set "*setClay_submerged*" from rectangle 1, 2 and 7.

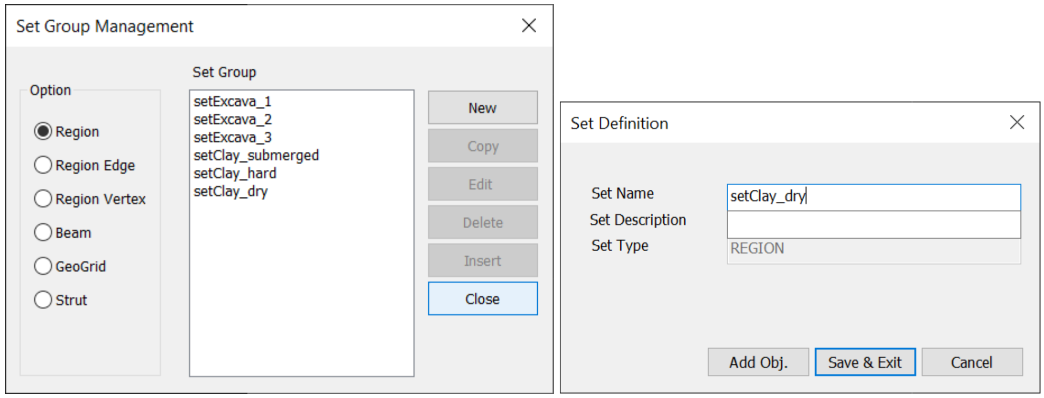


Figure 2.18-5: Create region sets

Select the option "Beam" and create one new beam set "`setWall`" which contains all four beam segments in this model.

Select the option "Geogrid". Create two new strut sets "`setGeogrid_up`" and "`setGeogrid_low`". These two new sets contain the upper geogrid and lower geogrid entity respectively.

Select the option "Strut". Create two new strut sets "`setStrut_up`" and "`setStrut_low`". These two new sets have the upper strut and lower strut entity respectively.

Beam segments should be head-toe connected and follow consistent directions. To view beam directions, click the menu "**View->Show Direction**". The tangential direction (red color) is defined from the beam start vertex to the end vertex. The normal direction (green color) is counter clockwise 90° from the tangential direction. If some beams have inverted directions, users may click the menu "**Modify->Reverse Beam Direction**" to reverse them.

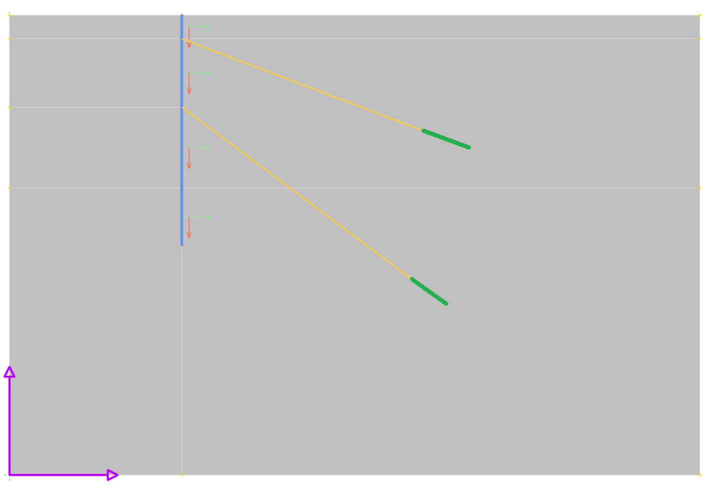


Figure 2.18-6: Consistent beam directions in EnFEM

Step 3: Define stage data

Click the menu "**Data->Stage Management**". Click "**New**" to create a new stage "*ints*" in type of "Initial Soil Stress". Click the button "**Initial Soil Stress Option**" to open a new window to define initial soil stresses. Keep the default option as "K0 method", click "**Add**" to add all soil regions. Click "**Input Data**" to open a dialog for initial soil stress interpolation. Click "**Add Row**" to add a new data line. Input the vertical coordinates of top and bottom soil body, input the component of vertical stress and lateral stress factor as in Figure 2.18-7. Click "**Save & Exit**" to exit to initial stress option dialog. Then click "**OK**" to exit to stage management dialog.

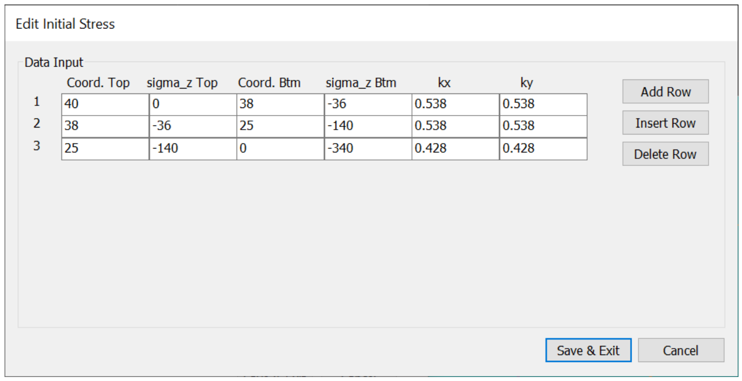


Figure 2.18-7: Create data table for initial soils stress interpolation

Sequentially create four new stages "*excava_1*", "*tieback_up*", "*excava_2*", "*tieback_bottom*" and "*excava_3*". These three excavation stages are in type of "Excavation". Keep the default settings for increment option and increment size for these excavation stages. The two tieback stages are in type of "General Static". Keep the option "Automatic" for increment control. Change the initial and maximum increment size to 1 for tie–back installation stages.

Step 4: Define soil material and beam section

Click the menu "**Data->Material Properties**" to open the material management dialog. Click "**New**" to define three new materials as shown in Figure 2.18-8. Click the menu "**Data->Material Assignment**". Check the option "By Set", select the set "setClay_dry" from the set list, select "clay_dry" from the material group list, click "**Assign**" to complete assignment, or click "**Update**" if regions do have previously assigned materials. Similarly, assign the material "clay_submg" to region set "setClay_submerged", and the material "clay_hard" to the set "setClay_hard".

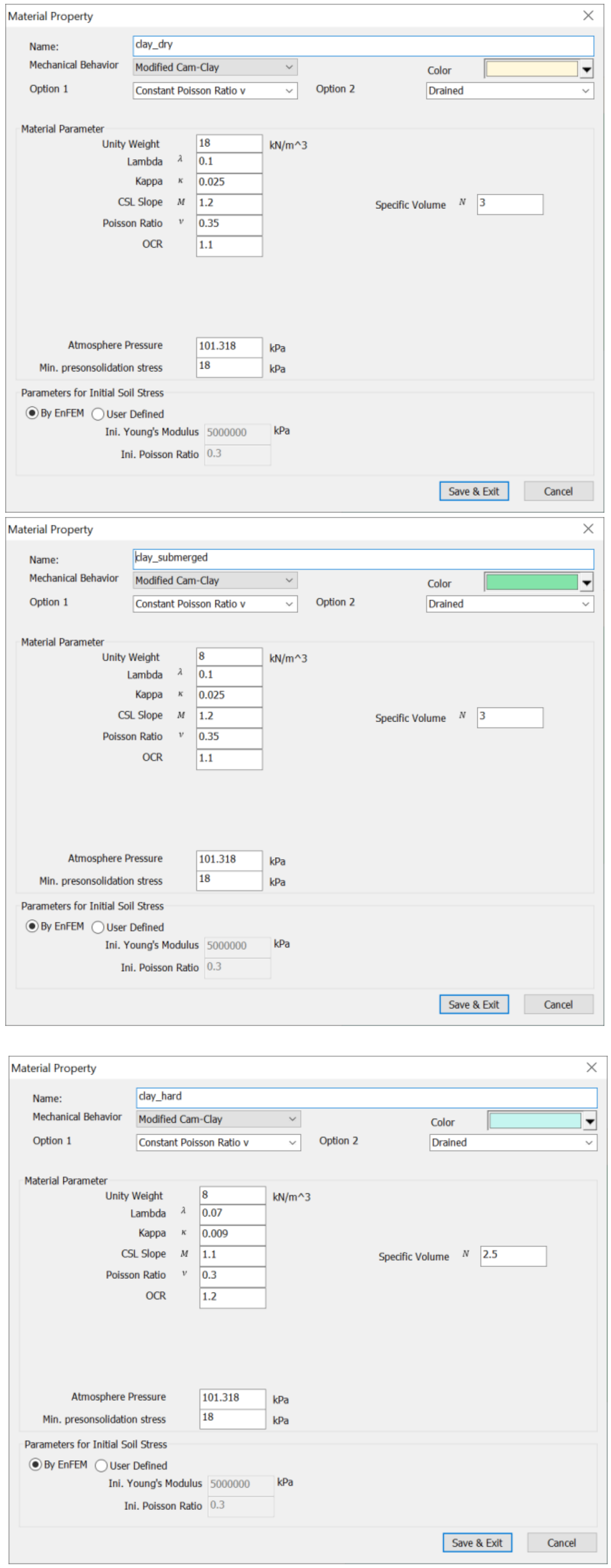


Figure 2.18-8: Material definition for example 18

Click the menu "**Data->Structural Section**" to open the structural section management dialog. Click "**New**" to define a new beam section "`secWall`", a new strut section "`secStrut`", and a new geogrid section "`secGeogrid`" as shown in Figure 2.18-9. Click "**Save & Exit**" to save the new beam section into the model database. Click the menu "**Data->Structural Section Assignment**". Assign section "*secWall*" to the beam set "*setWall*". Assign the strut section "*secStrut*" to both strut sets "*setStrut_up*" and "*setStrut_low*". Assign the geogrid section "*secgeogrid*" to geogrid sets "*setGeogrid_up*" and "*setGeogrid_low*".

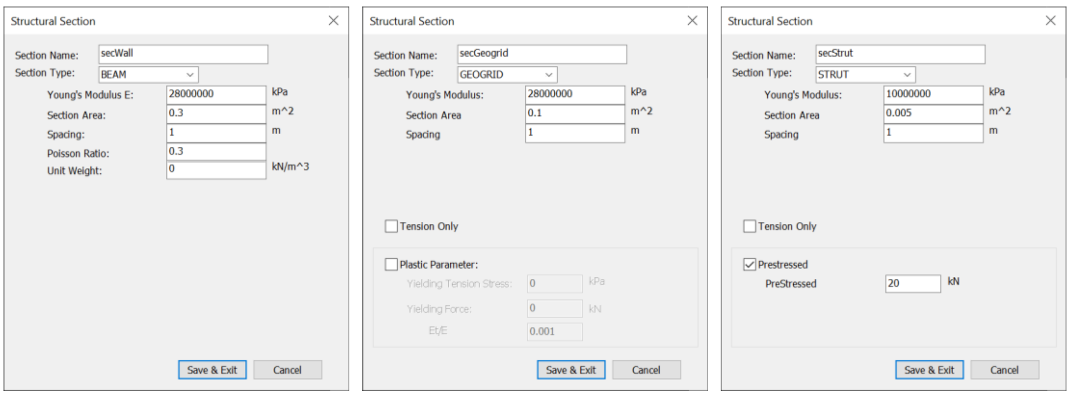


Figure 2.18-9: Define structural sections for tie back and retaining wall

Step 5: Define boundary conditions

Click the menu "**Data->Boundary Conditions**" to open the boundary condition management dialog. Click "**New**" to create two displacement BCs and one gravity load BC as shown in Figure 2.18-10.

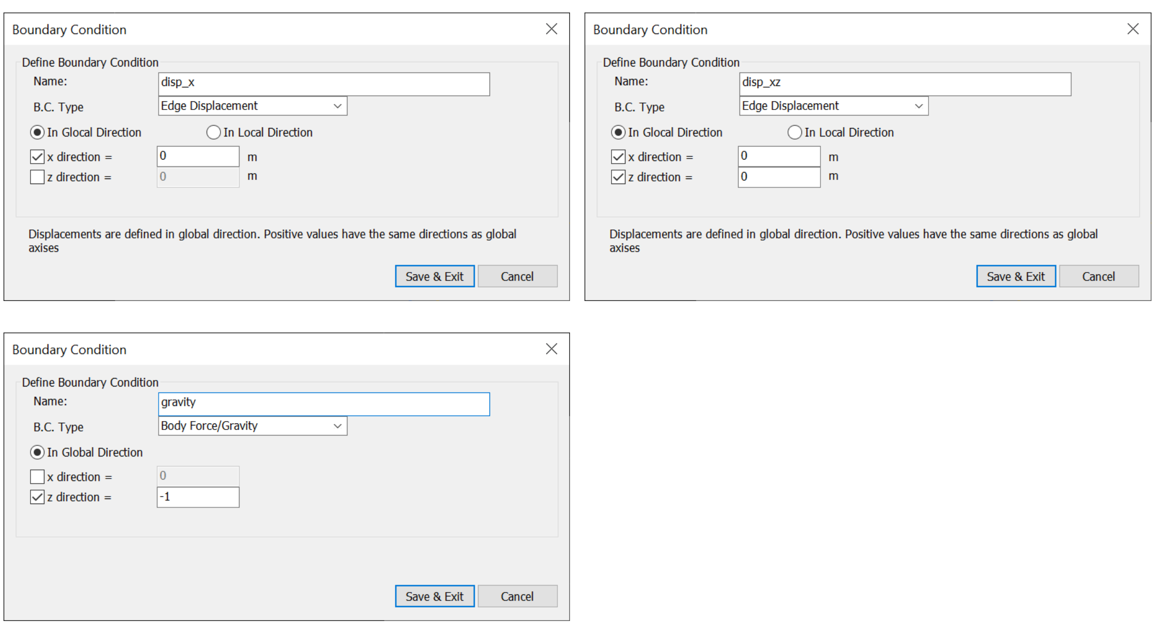


Figure 2.18-10: Define displacement and load boundary conditions

Click the menu "**Data->Boundary Assignment**". In the popping up dialog, click the BC "disp_x", click "**Assign**", select the left and right vertical edges of all soil regions, RIGHT-CLICK and select "**End Selection**". It will be convenient to use a "contained" window. Press and hold SHIFT for multiple selection operation.

Similarly, assign BC "disp_xz" to the bottom edges of the whole soil body, and assign "gravity" to all soil regions. In this example, the self-weight of the wall is ignored.

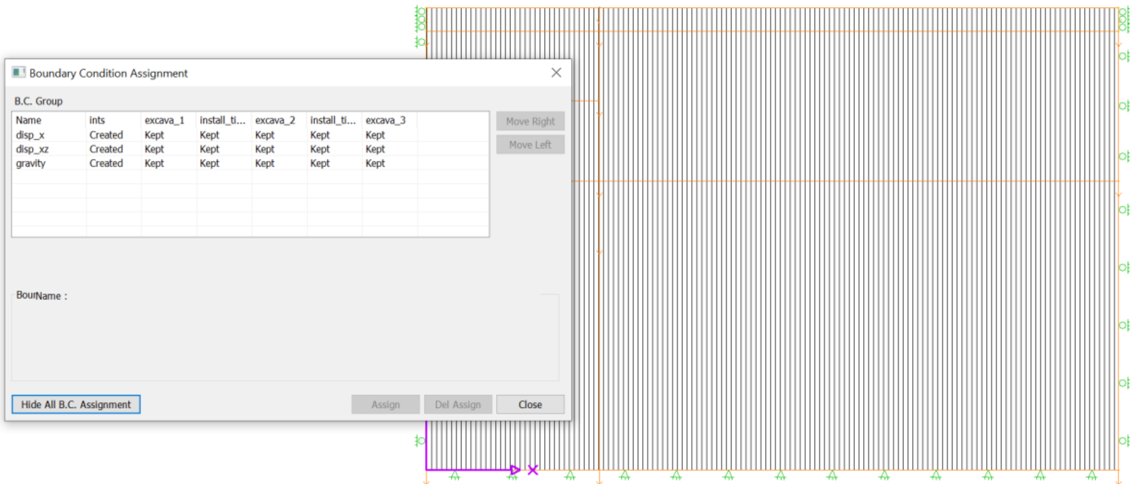


Figure 2.18-11: Assign boundary condition in example 18

After assignment, click B.C. group individually to verify assignment to be correct. Corresponding boundary condition symbols will be displayed adjacent to selected geometric entities. Users may also click "**Show All Assignment**" to review all B.C. assignment simultaneously.

Step 6: Define soil-structure-interaction interface

Click the menu "**Data->Interaction->Zero Thickness Interface**" to open interface management dialog. Click "**New**" to create a new interface group. Input interface parameters as shown in Figure 2.18-12. Click "**Add Beams**". Click or use a selection window to select all four beam segments. RIGHT-CLICK and select "**End Selection**" from the popup menu to finish adding beams to the interface. Click "**Create Interface**". If interfaces are successfully created, two deep green dash lines will be plotted adjacent to the beam segments. Click "**Save & Exit**" to return to interface group management dialog.

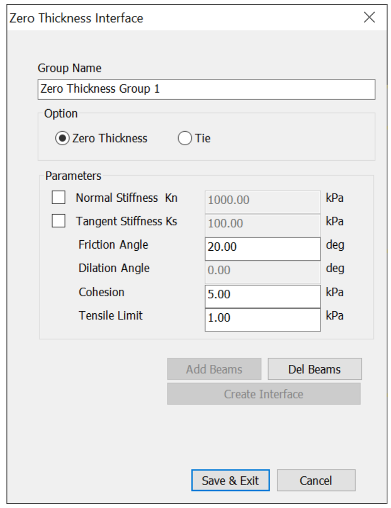


Figure 2.18-12: Define zero-thickness interface group for retaining wall

Step 7: Define deactivation/activation for excavation construction

Click the menu "**Data->Deactivation/Activation**". Click the tab "Region". Double click the cell locating on the "Set Deactivated" column and stage "excava_1" row, select the set name "setExca_1". Similarly set the region set "setExca_2" to be deactivated in the stage "excava_2". Click the tab "Beam". Set the beam set "setWall" to be activated in the initial soil stress stage "ints". Click the tab "Strut". Set the strut set "setStrut_up" to be activated in the stage "tieback_up", and set the strut set "setStrut_low" to be activated in the stage "tieback_low" (Figure 2.18-13).

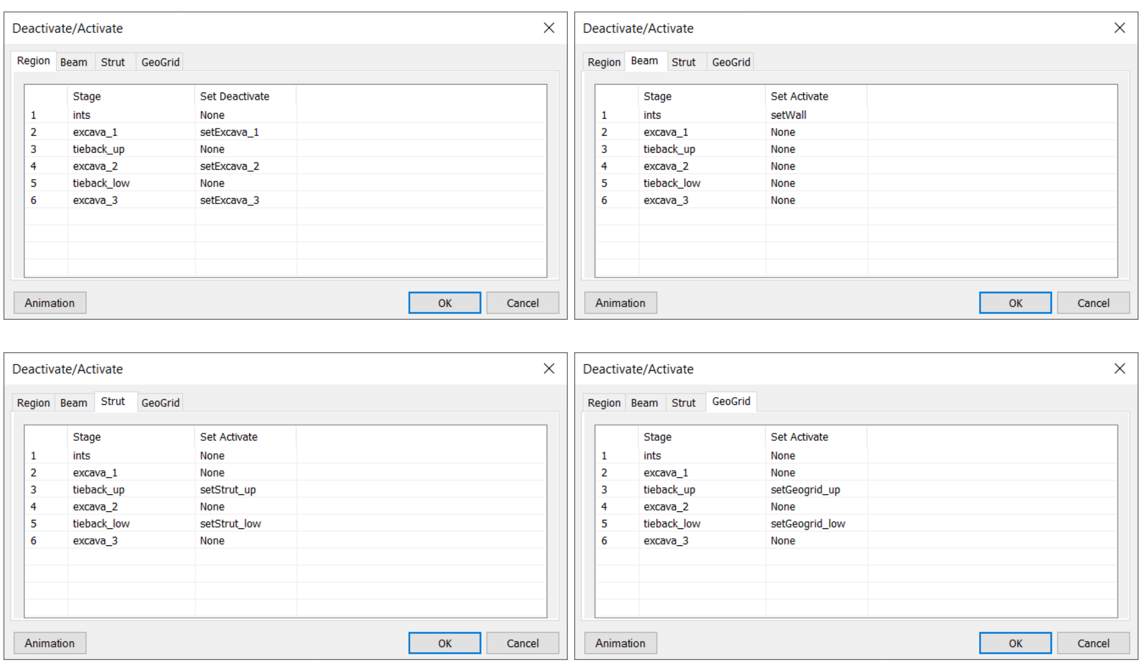


Figure 2.18-13: Set deactivation and activation in example 18

Click "**Animation**" to verify geometric entities to be deactivated or activated correctly.

Step 8: Mesh generation

Click the menu "pre-**Mesh->Advanced Mesh Control**". Select all soil regions. RIGHT-CLICK and select "**End Selection**" from the popup menu. Set element shape to "Triangle", mesh type to "Unstructured", element option to "Quadratic T6". Click "**Save & Exit**".

Click the menu "**Mesh->Global Seed Control**". Input number "`1.5`" as the global element size. This size is applicable to both 2D regions and 1D beam entities. Click "**Assign**" then click "**Close**" to exit. Click the menu "**pre-Mesh->Show Edge Division**" and "**pre-Mesh->Show Seeds**" to turn on display of edge discretization and seed locations

Click the menu "**Mesh->Run Mesh Generation**" to generate the mesh.

If the mesh mode is not automatically switched, click to switch to "MESH MODE". Since edges of solid elements and beam elements are overlapped on the interaction interface, users may use "probe" tool to investigate element connections. Click the menu "**Mesh->Probe Element**". Click to activate solid element selection icon . Click one solid element adjacent to the beam. Element connections and the element number will be displayed for selected element only. Deactivate the solid element selection and activate the beam element selection icon , users can check the element connection for a beam element.

Step 9: Analysis and result output

Click the menu "**Analysis->Analysis Setting**". Modify settings as necessary. Click "**OK**" to exit. Click menu "**Analysis->Run Analysis**". In the control dialog, set check marks before all stages (Figure 2.18-14). Click "**Run**" to submit the job to the solver.

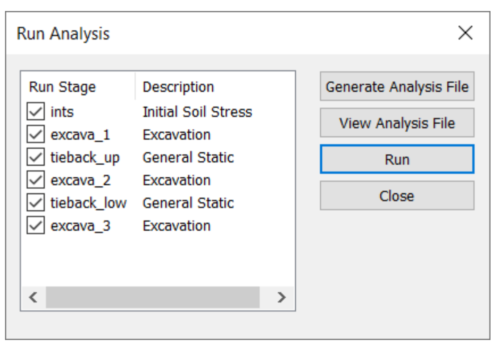


Figure 2.18-14: Dialog of "Run analysis" for example 18

After calculation is successfully completed or terminated due to errors, the message file (.log) will be opened automatically. It is recommended to review this file to check warning/error messages to update the model.

Click to switch to "*RESULT MODE*". Here horizontal displacement of the point on the ground after the retaining wall is investigated. Turn on node number and find the node to connect the lower strut to the wall (node 3084). Click  to open X-Y plot window. Input the node number "`3084`" under "Label of single node". Click "**Plot**" to plot historic horizontal displacement for soil after the wall as shown in Figure 2.18-15.

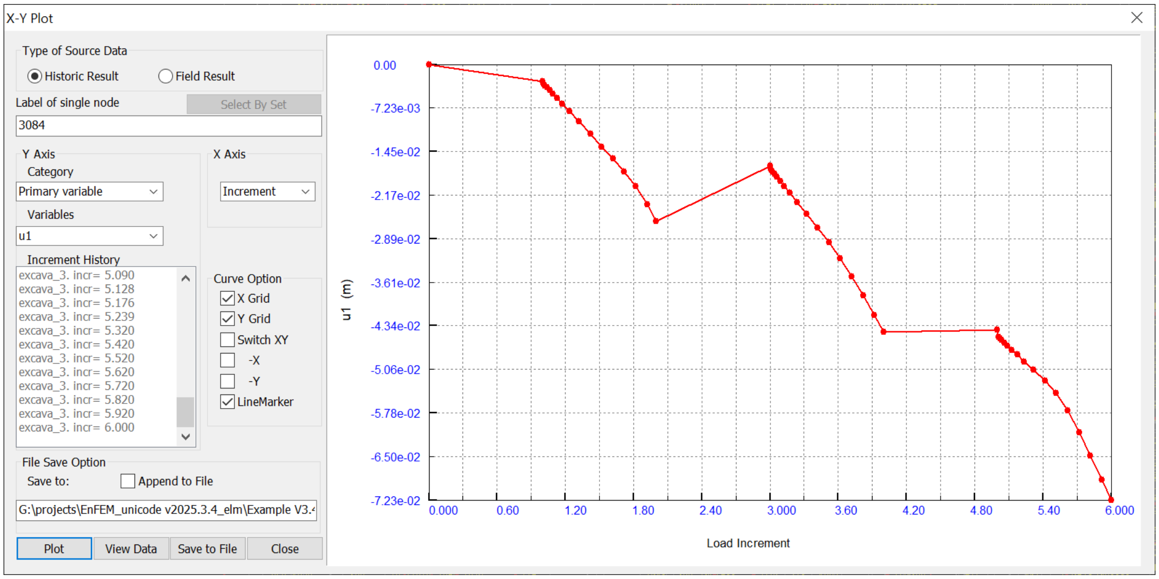
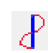


Figure 2.18-15: Displacement history at lower tieback-wall connection in Example 18

Click the menu "**Results->Curve Plot for Structural Elements**" or click the icon  on the toolbar panel. In the popping up dialog, default plot is "BEAM" type. If a valid beam set is found, EnFEM will plot moment curve automatically. Users may click "**Select By Set**" to select other sets, or select "Internal Forces" from category, then select "Shear Force" or "Axial Force" for other plots. Click the last load increment in the increment history. Click "**Plot**" to plot the result curve on the right window (Figure 2.18-16). If "Show Description" is checked, explanation of sign convention will also be plotted under curve title. Positive shear force will make a beam section to rotate in counter clockwise direction.

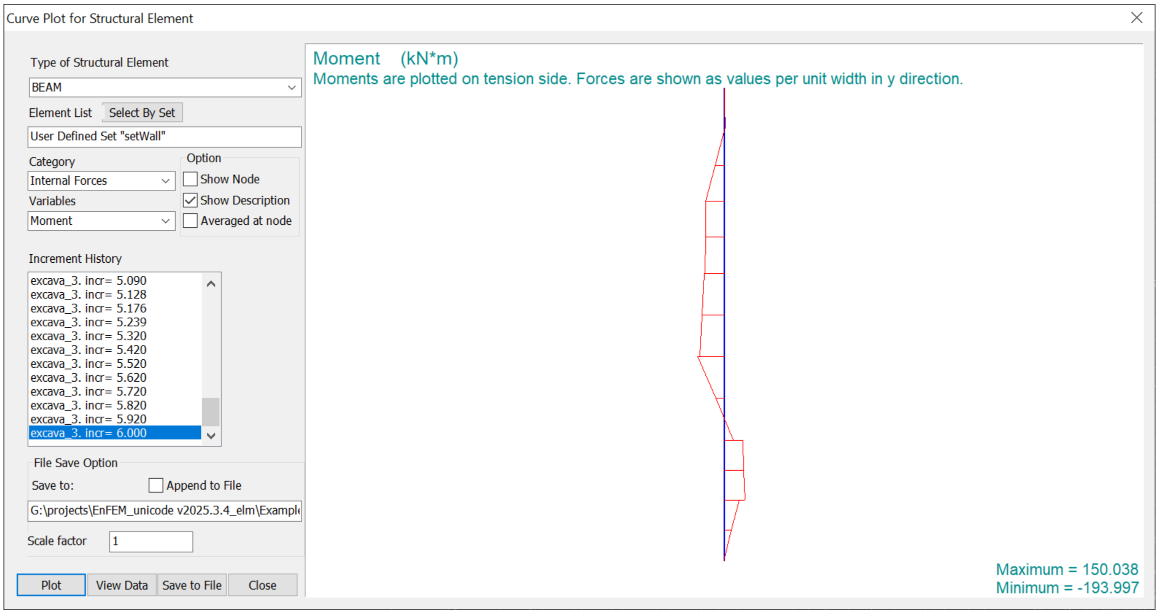


Figure 2.18-16: Axial force of retaining wall after all excavation

Change “STRUT” as the type of structural element. Click “**Select By Set**”. In the set selection dialog, select the strut set “setStrut_low”. Select “Internal Forces” from category. Strut output has only one internal force, “Axial Force” from variables is automatically selected. Click the last load increment in the increment history. Click “**Plot**” to plot the result curve on the right window (Figure 2.18-17).

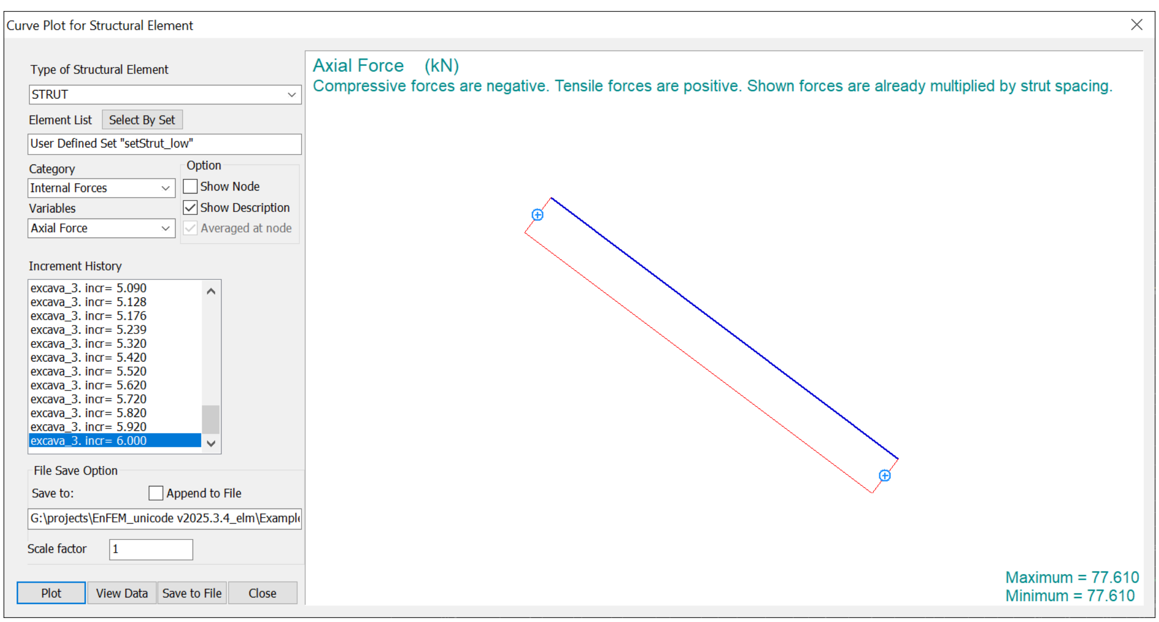


Figure 2.18-17: Axial force of bottom tie-back

Change "GEOGRID" as the type of structural element. Click "**Select By Set**". Select the strut set "setGeogrid_up". Select "Internal Forces" from category. Strut output has only one internal force, "Axial Force" from variables is automatically selected. Click the last load increment in the increment history. Click "**Plot**" to plot the result curve on the right window (Figure 2.18-18).

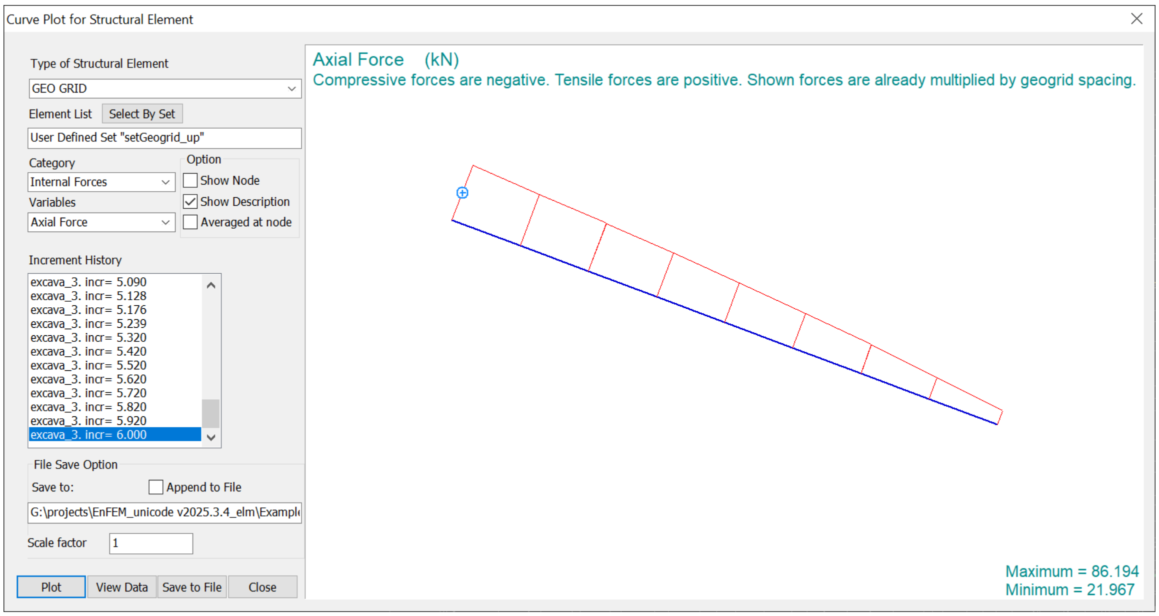


Figure 2.18-18: Axial force of upper geogrid

2.19 Example 19: Import a model from EnSlope

This example presents how to import an EnSlope model for slope stability analysis.

Import EnSlope model:

Start EnFEM. In the control Data window, select the unit option “Customized”. Then set the force unit as “lbf” and length unit as “ft” (Figure 2.19-1). If the EnSlope model is developed in SI unit, the force unit here should be “kN” and length unit should be “m”. Click “OK” to exit from the control data window. Click the menu “**File->Import from EnSlope/UTEXAS4**”. The import window is shown as in Figure 2.19-2. The default coordinate tolerance is set to 0.2 ft or 0.05 m. Two vertexes with distance smaller than the tolerance will be merged as one vertex. The default anchor length is 3 ft or 1 m.

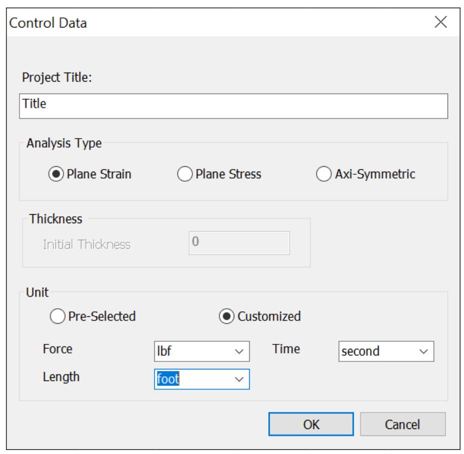


Figure 2.19-1: Setup units before importing

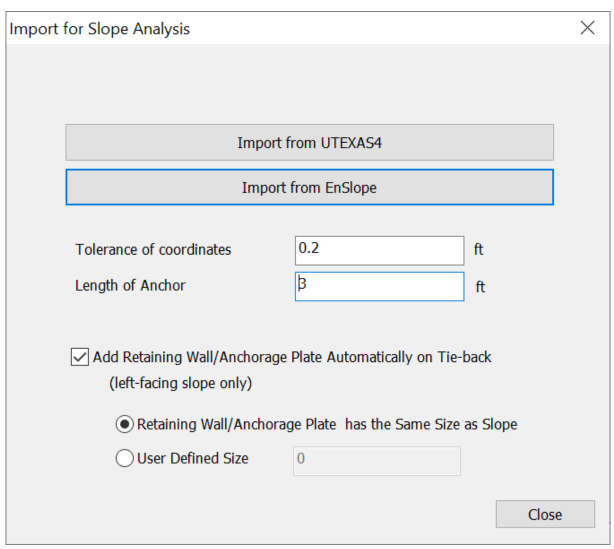


Figure 2.19-2: Dialog to import an EnSlope model

Since EnSlope does not define retaining walls at the slope side to connect with a tie-back, the retaining wall will be supplemented automatically by EnFEM.

Click the button "**Import from EnSlope**". Select the model file "*ENSLOPE EX1 Surcharge.slpd*" from the directory of example 19. The imported model is shown in Figure 2.19-3. Imported model has two predefined stages: initial soil stress (geostatic) stage and stability analysis stage. Since the ground is inclined, the option to estimate initial soil stress is set to "By System".

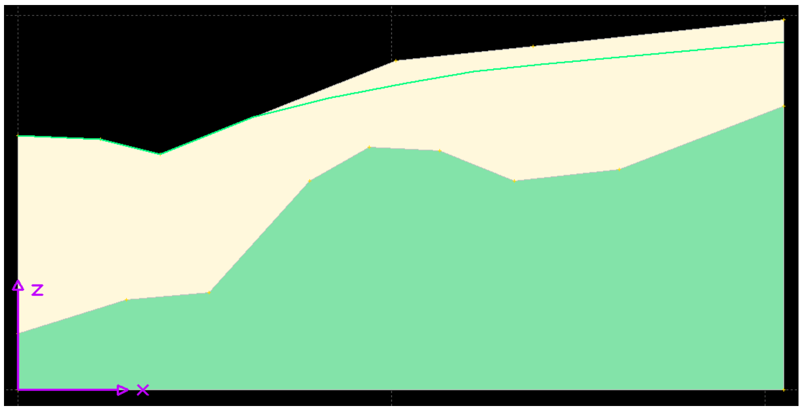


Figure 2.19-3: Imported model shown in EnFEM

Click the second stage "Stability (SRM)" from the stage management window, then click the button "**Edit**" to review parameters predefined for this stage (Figure 2.19-4).

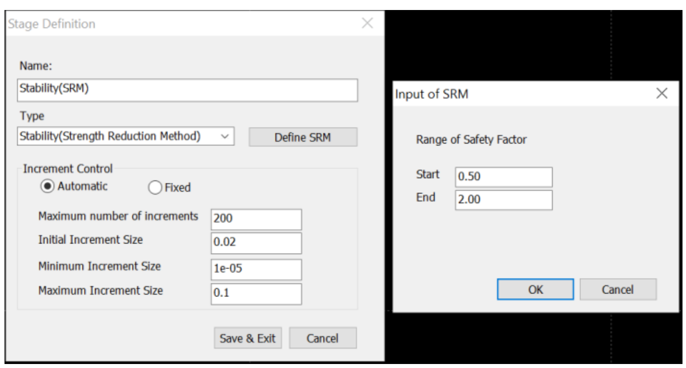


Figure 2.19-4: Stage parameters for the stability stage

Click the menu "**View->Show Boundary Assign**." to check boundary conditions to be assigned correctly as in Figure 2.19-5.

Click the menu "**pre-mesh->Auto Mesh generation**" to create the finite element mesh. By default, quadratic triangle element T6 will be applied to discretize the regions.

Click the menu "**Analysis->Run Analysis**" to complete analysis of this slope stability.

The contour of the horizontal displacement u1 is shown in Figure 2.19-6. The safety factor at the failure of convergence is about SF=1.651.

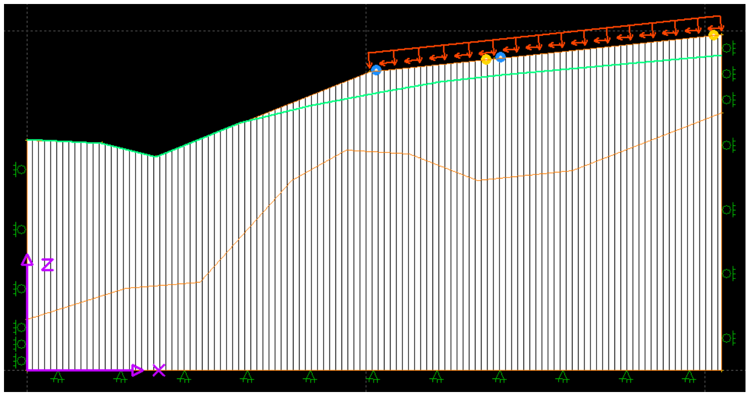


Figure 2.19-5: Boundary conditions in imported model

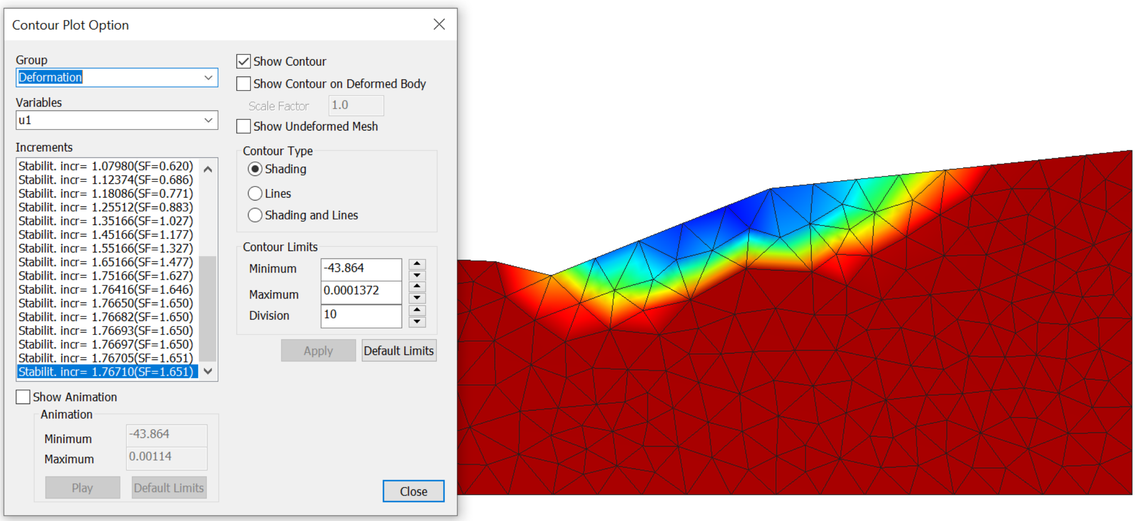


Figure 2.19-6: Contour of horizontal displacement u1

2.20 Example 20: Import a model from PYWall

This example illustrates how to import a PYWall model for retaining wall analysis.

Import PYWall model:

The model in PYWall looks like that in Figure 2.20-1. Start EnFEM. In the control Data, set the force unit as “lbf” and length unit as “in”. If the PYWall model uses SI unit, “kN” for force unit and “m” for length unit should be selected.

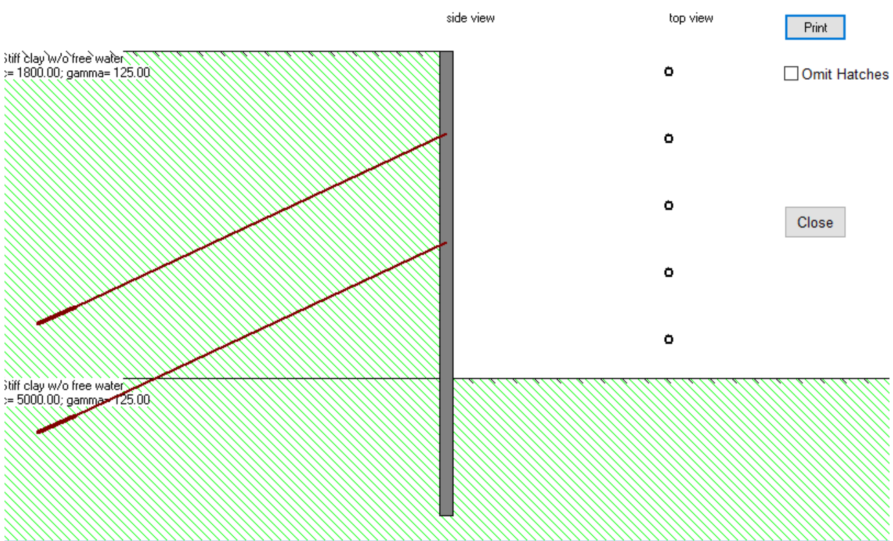


Figure 2.20-1: Model of a retaining wall supported by two tie-backs in PYWall 2025

Click the menu “**File->Import from PYWall**”. The importing window will pop up as shown in Figure 2.20-2.

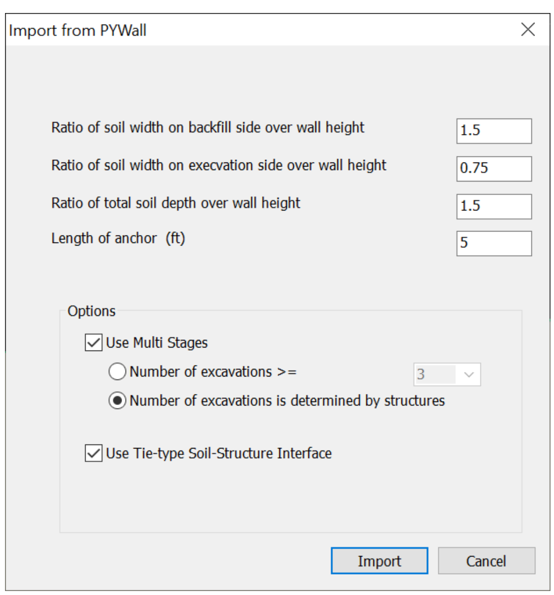


Figure 2.20-2: Parameter window for importing a PYWall model

Since the PYWall model has two tie-backs, keep the default "Number of excavations is determined by structures". Keep the default option "Use Tie-type Soil-Structure Interface". Click the button "**Import**" and select the PYWall model file "*Example 6a – Anchored Timber-Lagging Wall in Stiff Clay.py8d*". After importing, the message file will be opened automatically as shown in Figure 2.20-3. It is recommended to read the contents of importing information to check any error or warning message.

```
****** Start to Import model from PYWall ******

Geometry objects have been imported successfully

Walls have been created successfully

Structures have been created successfully

Construction stages have been created successfully

Boundary conditions have been created successfully

****** End of importing model from PYWall ******
```

Figure 2.20-3: Information about model importing from PYWall

The model in EnFEM looks like that in Figure 2.20-4.

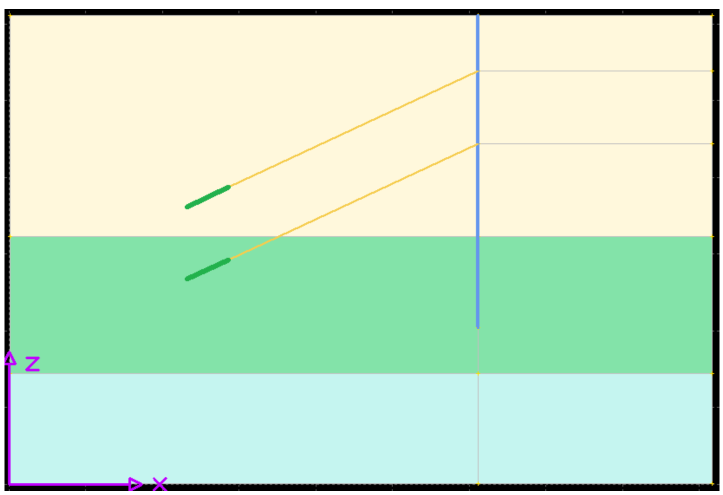


Figure 2.20-4: model of retaining wall analysis imported from PYWall

Since materials, boundary conditions and soil-structure interactions are created automatically by EnFEM, users only need to check these parameters to make sure to be correctly created. Click the menu "**pre-Mesh->Auto mesh generation**" to generate the mesh. The default mesh is always unstructured mesh using quadratic triangle elements. As shown in Figure 2.20-5, the mesh around the wall is refined than the mesh near the boundaries.

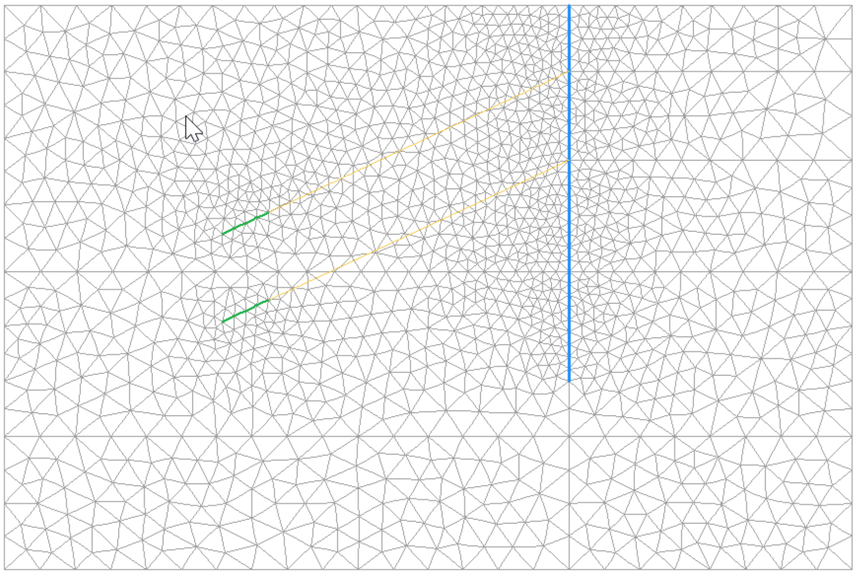


Figure 2.20-5: Non-uniform mesh generated for imported PYWall model

Click the menu "**Analysis->Run Analysis**" to submit the job to the solver. After the analysis is completed, click the icon to go to post processor. Click the icon to check beam internal forces (Figure 2.20-6). The default output is the beam moment. Select "Shear force" from the "variable" drop box. The shear force at the last increment is shown in Figure 2.20-7

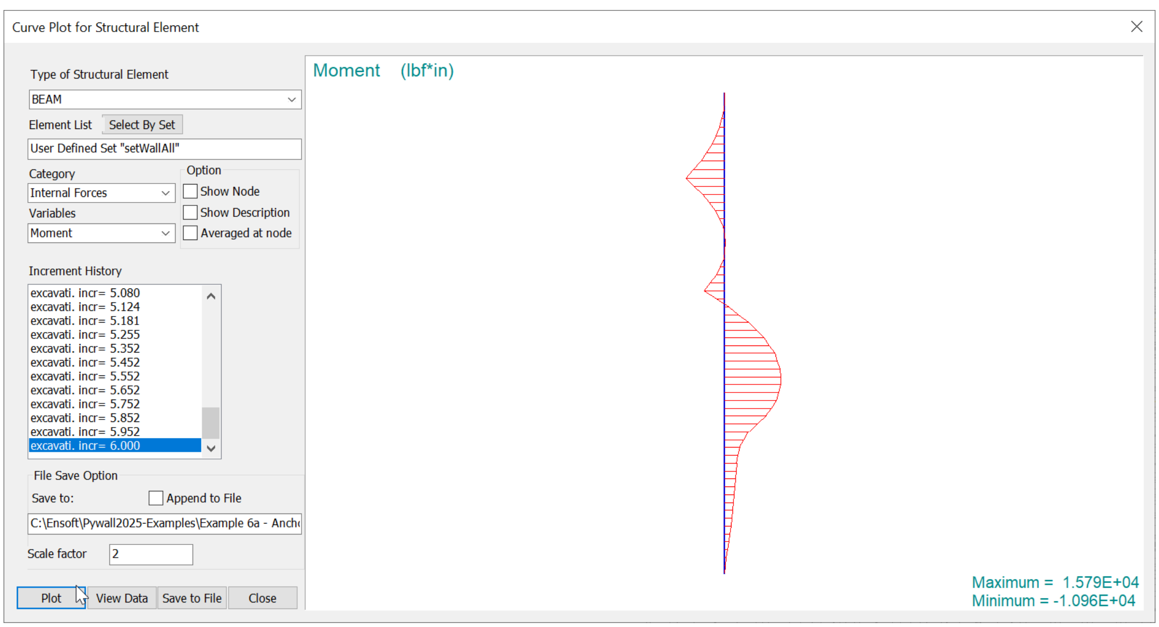


Figure 2.20-6: Moment of wall for imported PYWall model

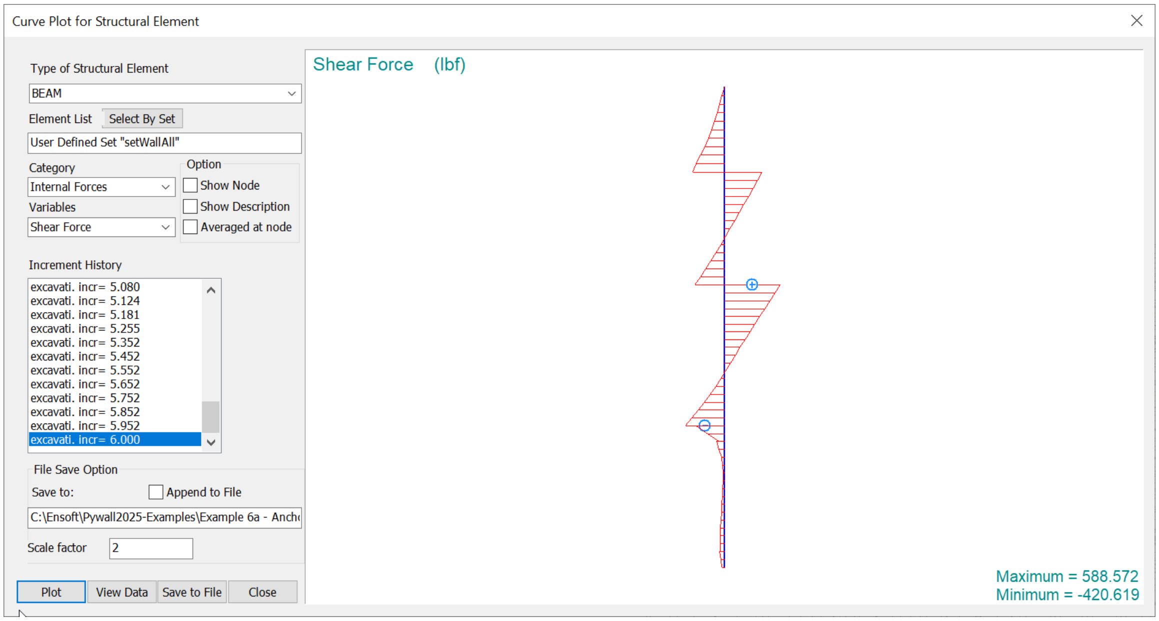


Figure 2.20-7: Shear force of wall for imported PYWall model

List of Technical References

1. J.M. Duncan and C.Y. Chang, "Nonlinear analysis of stress and strain in soils", *Journal of the soil mechanics and foundations division*, 1970.96(SM5):1629-1653.
2. Kyung Kim, Chai H. Yoo, "*Design Loading on Deeply Buried Culverts*", Journal of Geotechnical and Geoenvironmental Engineering, ASCE, January 2005, P20~P27

License Agreement & Disclaimer

PROGRAM LICENSE AGREEMENT & DISCLAIMER

This Network License strictly prohibits the PRODUCT to be used in or from computers located in office locations that are different than the licensed SITE(S) or outside the registered CLIENT SUBNETS. Users in physical office locations other than the registered SITE(S) with ENSOFT are required to purchase additional licenses of the PRODUCT, even if the COMPANY is the same and/or if the additional offices are located in the same city.

2. OTHER RIGHTS AND LIMITATIONS

2.1 EDUCATIONAL VERSION

If the PRODUCT was licensed with educational discounts, the use of the PRODUCT is strictly limited to academic or research activities of educators and/or students. Any other usage voids all the rights under this LICENSE. If in doubt, please contact ENSOFT to determine if you have a PRODUCT version that was licensed for educational purposes.

2.2 LIMITATIONS ON REVERSE ENGINEERING, DECOMPILATION, AND DISASSEMBLY

You may not reverse engineer, decompile, or disassemble the PRODUCT, except and only to the extent that such activity is expressly permitted by applicable law notwithstanding this limitation.

2.3 RENTAL

You may not rent, lease or lend the PRODUCT to third parties.

2.4 TRADEMARKS

This LICENSE does not grant you any rights in connection with any trademarks or service marks of ENSOFT.

2.5 SUPPORT SERVICES

ENSOFT shall provide you with limited support and maintenance services (“SUPPORT”) related to the PRODUCT during the initial year after purchase and for any additional SUPPORT term that the COMPANY elects to purchase for PRODUCT after the initial warranty period expires.

Use of SUPPORT is limited to the ENSOFT policies and programs described in the PRODUCT’s User’s Manual, in documentation on the ENSOFT web site, and/or in other materials provided by ENSOFT.

Any supplemental software code that may be provided to you as part of the SUPPORT or that is downloaded from the ENSOFT web site shall be considered part of the PRODUCT and subject to the terms and conditions of this LICENSE. With respect to technical information you provide to ENSOFT as part of the SUPPORT, ENSOFT may use such information for its business purposes, including for product support, development or advertisements. However, ENSOFT will not utilize such technical information in a form that personally identifies the USER.

2.6 SOFTWARE TRANSFER

The initial user of the PRODUCT may make a permanent transfer of this LICENSE and PRODUCT and only directly to an end user. This transfer must include all of the PRODUCT (including all component parts, the media and printed materials, any upgrades, this LICENSE, and the PRODUCT’s appropriate hardware KEY). Such transfer may not be by way of consignment or any other indirect transfer. The transferee of such one-time transfer must agree to comply with the terms of this LICENSE, including the obligation not to further transfer this LICENSE and PRODUCT.

2.7 TERMINATION

Without prejudice to any other rights, ENSOFT may terminate this LICENSE immediately if you fail to comply with the terms and conditions of this LICENSE. In such event, you must destroy all copies of the

PRODUCT and all of its component parts and provide evidence of such destruction in writing to ENSOFT within 3 business days.

2.8 INDEMNIFICATION

Within and during the terms of this LICENSE, ENSOFT warrants that the use of the initially provided PRODUCT does not infringe any patent, copyright, or trademark in the United States, and ENSOFT shall indemnify and hold USER harmless against any and all losses, damages and expenses, (including attorney's fees) which USER may sustain or incur as a result of a breach of this warranty.

2.9 CONFIDENTIAL INFORMATION

ENSOFT and the USER may have access to certain proprietary information and materials of the other that are confidential and of substantial value to the respective party, which value would be impaired if such information were disclosed to third parties ("CONFIDENTIAL INFORMATION"). ENSOFT and the USER agree that neither shall disclose any CONFIDENTIAL INFORMATION to any third party and shall take every reasonable precaution to protect CONFIDENTIAL INFORMATION.

The provisions of this section shall not apply to any information which (i) is or becomes available to the public other than by breach of the LICENSE agreement by the receiving party, (ii) is rightfully received by receiving party from a third party without confidential limitations, (iii) is independently developed by receiving party's employees without access to CONFIDENTIAL INFORMATION, or (iv) is known to the receiving party without any restriction on its use or disclosure prior to first receipt of it from the disclosing party.

3. COPYRIGHT

All title and intellectual property rights in and to the PRODUCT (including but not limited to any images, photographs, animations, video, audio, music, and text that may be incorporated into the PRODUCT), the accompanying printed materials, and any copies of the PRODUCT are owned by ENSOFT. All title and intellectual property rights in and to the content which may be accessed through use of the PRODUCT is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This LICENSE grants you no rights to use such content. All rights not expressly granted are reserved by ENSOFT. This PRODUCT is protected by the United States Copyright Law and International Copyright Treaty.

4. SOFTWARE DISCLAIMER

Although the PRODUCT has been used with apparent success in many analyses, new information is developed continuously and new or updated PRODUCT releases may be written from time to time. All users are requested to inform ENSOFT immediately if any errors are found in the PRODUCT. As modifications, updates, or new versions are produced, the latest codes are posted on ENSOFT's web site and made available to all visitors for free downloading.

No warranty, expressed or implied, is offered as to the accuracy of results from ENSOFT's PRODUCT. The PRODUCT 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 documentation that is distributed with the digital media. Users are assumed to recognize that the input parameters, e.g., soil properties, increment length, tolerance on solution convergence, and many others, can have a significant effect on the solution and must be chosen carefully. Users should have a thorough understanding of the relevant theoretical criteria (appropriate references are suggested in the software documentation).

5. GOVERNING LAW

This LICENSE is governed by the laws of the State of Texas and laws and treaties of the United States of America.

6. CONTACT INFORMATION

Should you have any questions concerning this LICENSE or if you desire to contact ENSOFT for any reason, please use the following:

Ensoft Incorporated
3003 West Howard Lane
Austin, Texas 78728
United States of America

NO OTHER WARRANTIES

TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, ENSOFT DISCLAIMS ALL OTHER WARRANTIES AND CONDITIONS, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OR CONDITIONS OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND TITLE, WITH REGARD TO THE PRODUCT, AND THE PROVISION OF SUPPORT SERVICES. THIS LIMITED WARRANTY GIVES YOU SPECIFIC LEGAL RIGHTS. YOU MAY HAVE OTHERS, WHICH VARY FROM STATE/JURISDICTION TO STATE/JURISDICTION.

LIMITATION OF LIABILITY

TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL ENSOFT BE LIABLE FOR ANY SPECIAL, INCIDENTAL, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, OR ANY OTHER PECUNIARY LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE THE SOFTWARE PRODUCT OR THE FAILURE TO PROVIDE SUPPORT, EVEN IF ENSOFT HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN ANY CASE, ENSOFT’S ENTIRE LIABILITY UNDER ANY PROVISION OF THIS LICENSE AGREEMENT SHALL BE LIMITED TO THE GREATER OF THE AMOUNT ACTUALLY PAID BY YOU FOR THE PRODUCT OR U.S.$1.00.