EnFEM2026 – User Manual

A Finite Element Analysis Program

by

Daqing Xu
Luis G. Vasquez
Shin-Tower Wang
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

CHAPTER 1. Introduction

1.1 General Description

EnFEM is a versatile finite element analysis software designed for simulating two-dimensional mechanical behavior of solid materials under various loading conditions. It allows for plane-strain, plane-stress, and axi-symmetric analyses, focusing primarily on geotechnical engineering challenges. EnFEM offers a comprehensive range of linear and non-linear material models, boundary conditions, and element types, equipping users with the tools needed to solve both straightforward and complex geotechnical problems.

1.2 Key Features of the Program

EnFEM provides several features aimed at empowering engineers to achieve meaningful results.

1. A user-friendly CAD-like interface for creating and modifying geometries, setting analysis parameters, and conducting post-processing.
2. A rich material library that includes various models such as linear elasticity, Tresca plasticity, Von Mises plasticity, Mohr-Coulomb plasticity, Druker-Prager plasticity, Duncan-Chang hyperbolic elasticity, and Modified Cam-Clay.
3. Support for multi-stage analysis, enabling the simulation of sequentially applied loads, initial geostatic soil stresses, slope stability, and excavation scenarios.
4. Advanced soil-structure interaction capabilities, featuring computational contact analysis to model material separation, rebounding, and finite sliding.
5. A zero-thickness interface approach for simulating retaining walls or culverts as beam elements
6. A robust mesh generator that creates first or second-order, quadrilateral or triangular finite elements in structured, unstructured, or combined mesh formats.
7. A powerful nonlinear direct solver optimized for parallel processing.
8. Quick and precise contour plotting thanks to OpenGL-based graphics technology.
9. A variety of output options, including contour plots, vector plots, XY-plots, cutline plots, and structural curve plots, encompassing numerous categories and variables such as integrations (moments) and interactions along user-defined node paths.
10. Comprehensive example files and an accompanying manual that assist users in swiftly grasping the software through common geotechnical engineering problems.

1.3 Documentations of EnFEM

The documentation for EnFEM includes three volumes. The User’s Manual provides extensive information on operating the program on personal computers, covering installation, general usage, capabilities, and features. The manual is distributed in the standard Adobe© PDF format.

The Technical Manual offers insights into the theoretical foundations and constitutive models used in EnFEM, along with various technical details. It also includes useful references related to the concepts applied within the EnFEM program. Conversely, the Example Manual provides a comprehensive, step-by-step walkthrough of several common problems in geotechnical engineering. The examples outlined in the Example Manual are included with the software installation for user reference. This manual is distributed as a standard Adobe© PDF file.

Both manuals are installed alongside the EnFEM program and can be accessed through any of the following methods: i) through the Windows Start Menu, ii) via the Ensoft tile in Windows 10, and 11, or iii) from the “Help” menu within the EnFEM software.

1.4 Contents of the EnFEM Package

The standard package sent with EnFEM consists of the following items:

1. One USB Key (sometimes known as hardware key or dongle)
2. One USB Memory Stick with installation programs for all Ensoft software products (this may be missing in future releases, since the full program is available for downloading from the www.ensoftinc.com website

The user-selected installation directory (default directory is .*Program Files**Ensoft**EnFEM2026*) contains the main Windows-program module (*EnFEM.exe*), the program engine for direct computations (*EnFEMSolver.exe, mesh2D.dll*), and other support files. The User’s, Technical, and Example Manuals are also included in the installation directory. Several modeling examples with input files (files with name/extension of the type *filename.sep*) are installed on a separate *(Root Drive)**Ensoft**EnFEM2026-Examples* subdirectory in the root Windows drive (usually *c:*\). The installation directory for Example files can be changed by the user if desired.

1.5 Hardware Requirements

To use EnFEM, the user needs the following minimum hardware configuration:

1. Any personal computer with multi-core processor.
2. A hard disk with at least 5GB of free space (large meshes may require more).
3. 64bit Microsoft Windows 10 or Windows11
4. Minimum of 4GB of free RAM memory (8 GB recommended).
5. Open (available) USB port (v2 or newer).
6. Graphics card compatible to the Microsoft Windows™ operating system and compatible with OpenGL.
7. A three-button mouse or similar pointing device.
8. Optional: Any windows-compatible printer.

1.6 Technical Support

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

1.6.1 Preferred Methods of Software Support

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

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

Users are strongly encouraged to utilize electronic support via email. In all technical support requests via email, please include the following information:

1. Full software version/release/update (obtained from the **“Help->About”** dialog),
2. Description of the user’s problem or concern,
3. Attach a copy of the input-data file that is associated with the issue/concern (files with name/extension of the type *filename.sep*), and
4. Name and telephone number of the contact person and of the licensed user (or name and office location of the licensed company site and/or serial number of the USB Key).

Technical help by means of direct calls to our local telephone number, (512) 244-6464, is available, but is limited to the business hours of 9 a.m. to 5 p.m. (US central time zone, UTC −6:00). The current policy of Ensoft is that all telephone calls for software support will be answered free of charge if the user has a valid maintenance contract.

1.6.2 Upgrade Verification and Internet Site

EnFEM v2024 provides options for the user to check the most recent maintenance release through an internet connection by selecting **“Help->Check for Updates”** from within the software. This command starts with the default internet browser and will display the user’s maintenance expiration date, the user’s software release number and the most recent release number that is available for downloading.

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

1.6.3 Renewal of Program Maintenance

The cost to renew program maintenance will depend on the length of time for which the program maintenance has been expired. There are small price increases with time after expiration. The pricing policy for renewing maintenance that has not expired can be found on the Ensoft website at http://www.ensoftinc.com

1.6.4 Changes of Support Policy

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

CHAPTER 2. Installation & Getting Started

2.1 Installation Procedures

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

Users with standard single-user licenses can check the following link to a PDF with Installation notes: https://www.ensoftinc.com/doc/Ensoft_Single-User_License_Installation_Booklet.pdf

Users with local network licenses can check the following link to a PDF with Network Installation Notes: https://www.ensoftinc.com/doc/Ensoft_Network_License_Installation_Booklet.pdf

2.1.1 Installation of Single-User Version

This version of EnFEM has been tested to be compatible with the following versions of the Microsoft Windows® operating systems: Windows 10 and 11 in 64-bit releases.

The following guidelines are recommended during the installation process of EnFEM for single-user licenses.

1. Plug the supplied USB Key into one of the available USB ports in your computer. The USB Key is plug-and-play compatible so the operating system will recognize the USB Key automatically and a small but solid green light should appear at the end of the USB Key (a flickering green light or no light indicate problems with the standard windows driver or with the USB Key).
2. If the user installs from a distribution USB Memory Stick and the main installation program does not start automatically upon insertion of the Memory Stick then click on the **"Windows Start Menu"** button and select **"Run"**. On the command line, type *d:\setup.exe* or *e:\setup.exe*, where *d:* or *e:* represents the drive that contains the distribution Memory Stick. Click **"OK"** to execute the command and start the main installation program for ENSOFT's software. A screen similar to the one in Figure2-1 should appear.
3. If the user installs from a downloaded file, please run the downloaded file (double click) and go to instruction #5.
4. Click anywhere on the **"EnFEM 2026"** icon and then click on the **"Install Standard"** button to start the installation of EnFEM.
5. The user should read the license agreement shown in Figure2-2. Users can review the License Agreement online in the following link:
   https://www.ensoftinc.com/doc/Ensoft%20License%20and%20Disclaimer.pdf
6. The installer will place the same file (*Ensoft License and Disclaimer.pdf)* in the installation directory. Please click **"Yes"** if you agree and would like to proceed.

7. Select **"Single-User License"** in Figure2-3 then click **"Next"**. For network installations please contact Ensoft support (support@ensoftinc.com).

8. The user will be provided with an option to select a drive and directory for the installation of EnFEM (see Figure 2-4); the default is *c:\Program Files\Ensoft\EnFEM2026*. If the desired directory does not exist, the installation program will automatically create a new directory in the chosen hard drive. Most distribution files will be copied to the installation directory(either the default or the one selected by the user).Example files are copied to the following subdirectory under the root Windows drive (unless selected differently by the user during installation):

(Root Drive)\Ensoft\EnFEM2026-Examples

9. During the installation the user will be asked to set the file extension association for opening EnFEM 2024 input data files (see Figure 2-5). If the user agrees (leaves the default check mark) then double clicking (or running) any input data file with extensions of the type *filename.sep* will start the installed *EnFEM 2026* software.

10. The user will be prompted to confirm the shortcut directory name that will be created in the **"Windows Start Menu"** (See Figure 2-6). The default is *Start Menu/Programs/Ensoft/EnFEM2026*. Windows 8 and 10 will automatically create an *Ensoft* tile with the same shortcuts

Figure2-1: Main installation screen for ENSOFT software (may change with time)

After the installation is finished, it is not necessary to reboot Windows for the program to run. The user may run the program by selecting **“EnFEM v2026”** from the standard links installed in the Microsoft Windows® Start Menu: **"Start Menu->All Programs->Ensoft->EnFEM2026”**

Figure2-2: Installation screen with License Agreement (may change with time)

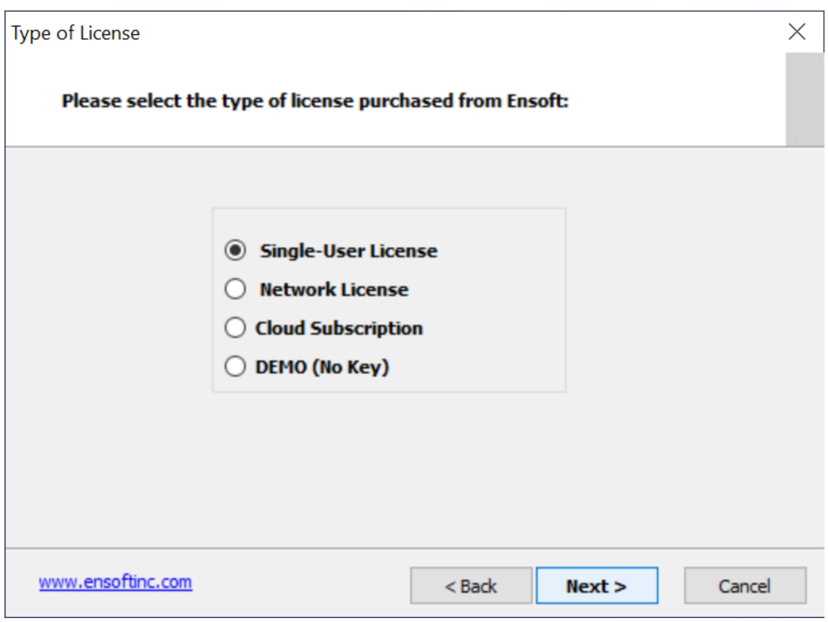


Figure2-3: Selection of Single-User License (may change with time)

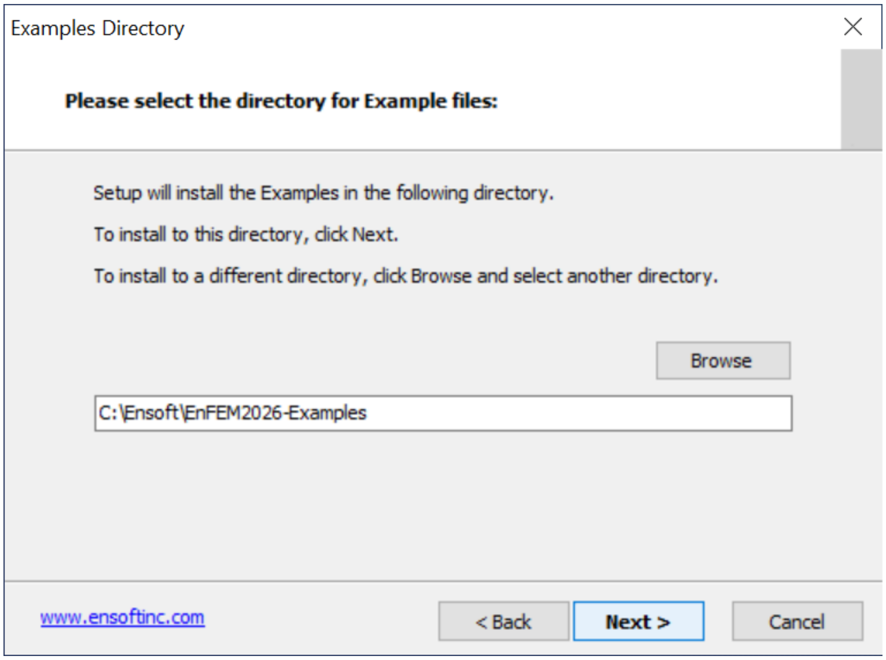


Figure 2-4: Default Installation Directory for Program Files (may change with time)

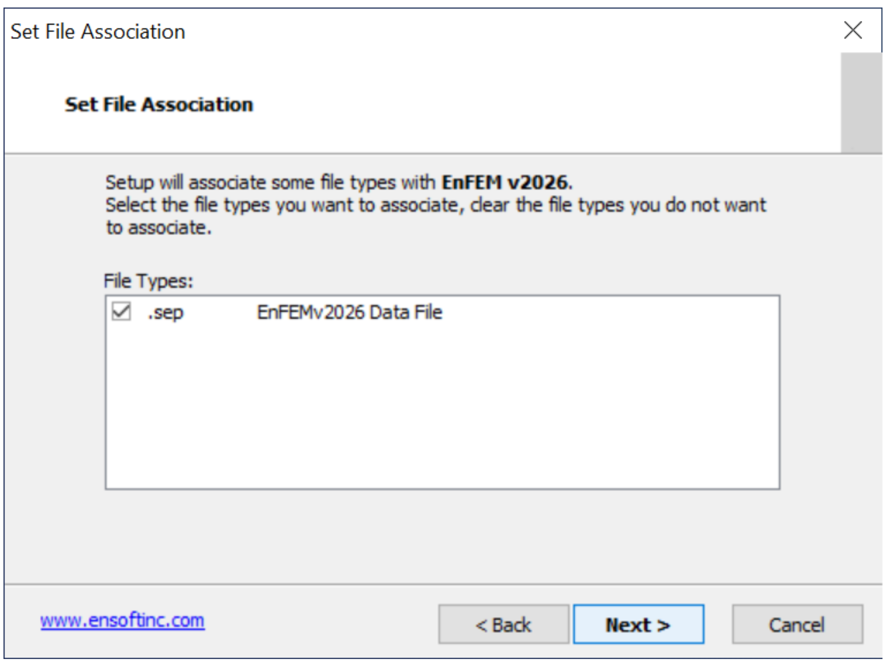


Figure 2-5: File Extension Association for EnFEM 2026 Data Files (may change with time)

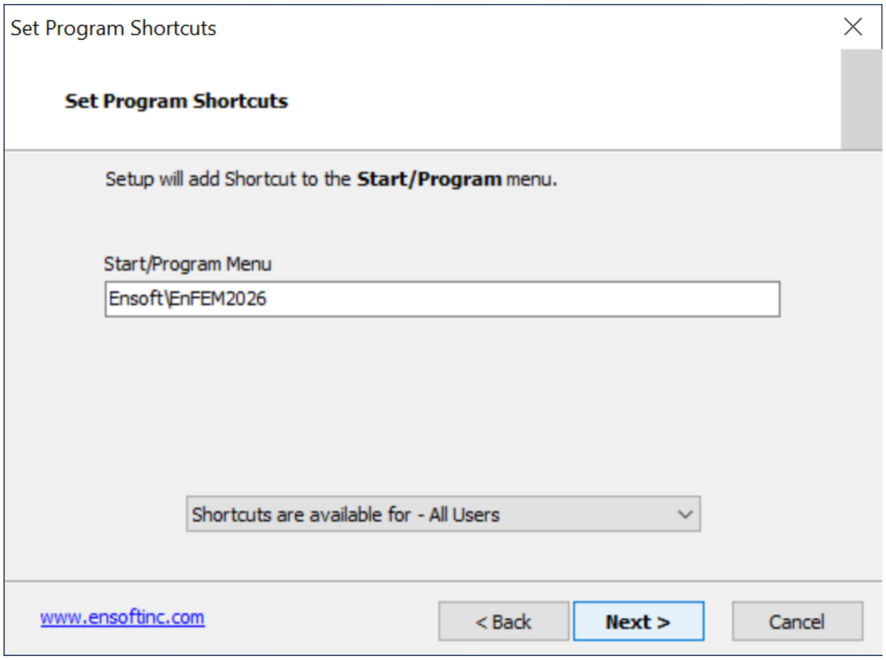


Figure 2-6: Default Shortcut Folder in Windows Start Menu (may change with time)

2.1.2 Introduction of Network Version

Special network licenses and USB hardware keys (network keys) are available for users who desire to operate EnFEM on a Windows network. The network version is limited to users with two or more licenses at the same physical site. Discounted rates apply for users purchasing multiple licenses for the same site.

Network versions of EnFEM have special subroutines written for installations in “software servers” and for installations of “individual clients”. The “software server” or “license server” is the computer that will be carrying the Net USB Key provided by ENSOFT, INC. The software server is not necessarily the same as the existing network server. Any computer in the existing Windows network may be designated software server for EnFEM as long as the network key is attached to an available USB port (or through an USB hub) and the “server” version of the software is installed on its local drive. Software “clients” may be all computers of the licensed site that have the program installed as client. Software clients do not need any hardware key attached to their local ports. The program installed in software clients will be allowed to run if their IP are within the allowable (licensed) IP ranges and if the computer designated as software server is accessible on the network with the proper operating system and with its network key secured in place.

Users of the network version of EnFEM are allowed to have the software installed on multiple computers within the licensed IP ranges of their local Windows network. However, only a number of users equal to the total number of purchased licenses will be able to operate the program at the same time.

2.1.2.1 Installation of Network Version

Installers of network licenses should refer to a separate booklet with installation instructions for the Network version of this product. The document can be downloaded from the Ensoft web site:

https://www.ensoftinc.com/doc/Ensoft_Network_License_Installation_Booklet.pdf

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

2.1.2.2 Silent Installations on Client Computers

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

https://www.ensoftinc.com/doc/Silent%20Install%20on%20Client%20Computers.pdf

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

2.1.3 Backup of Original Software

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

2.1.4 Software Updates on the Internet

Occasionally, ENSOFT will produce software improvements and/or fixes and place the latest software programs on ENSOFT's internet site. Users can check for available updates by selecting **"Help->Check for Updates"** from the EnFEM menu. Software users may freely download the latest program update from the *"PRODUCTS->Downloads"* link in the following site: *http://www.ensoftinc.com*

2.1.5 Demonstration Mode

If the USB hardware lock is not plugged into the computer when the program is opened, then EnFEM will operate in *"Demonstration Mode"*. This mode is generally used for prospective users to perform simple testing in order to evaluate the program, as well as view the included example files (containing both input and output data). Demonstration mode has several limitations as noted below:

1. Model files *cannot* be saved.
2. Existing model files can be opened in demonstration mode, but the file is *"read-only"*. Input data, mesh details, and results are visible, but changes *cannot* be saved.
3. Analysis is limited to 500-nodes.

2.2 Getting Started

2.2.1 User Interface

The EnFEM user interface is illustrated in Figure2-7. It features a "Menu Bar" with drop-down options, alongside a "Toolbar" that provides multiple shortcut icons. Each icon corresponds to a command and is accessible through the relevant drop-down menu. Beneath the toolbar lies the "Working Zone," where users can create geometric entities, make selections, view the mesh, and plot various results. Below the Working Zone is the "Information Panel," which offers interactive information and prompts. During command execution, users will view and enter data in different sections of this panel. It includes:

- Command Information: displayed when the mouse hovers over a shortcut icon.
- Command Prompt: provides messages for executing command operations
- Coordinate Input Box: enter vertex coordinates during specific commands
- Cursor Coordinates: show the current position of the mouse cursor
- Selection Information: displays the count of selected geometric entities or finite elements

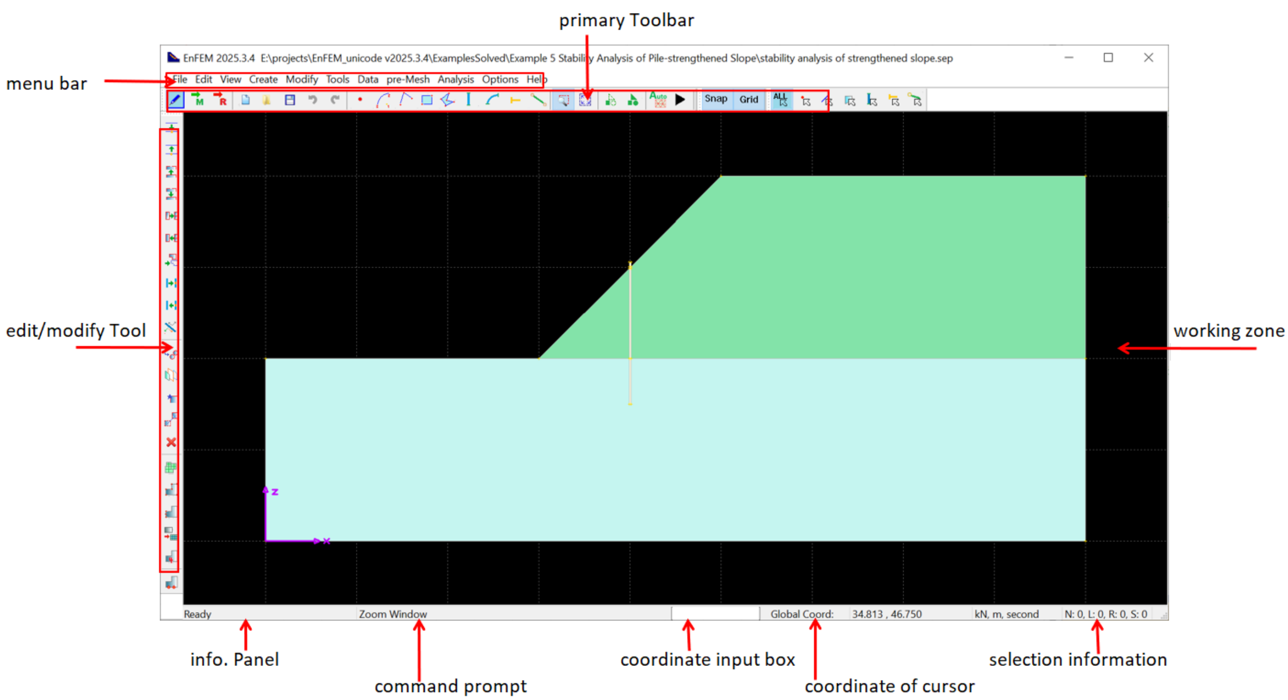


Figure2-7: User interface of EnFEM

Typically, commands in EnFEM are carried out in this manner:

1. choose the command operation from either the drop-down menu or the shortcut icon.
2. Follow the command prompt to make the necessary selections or input the required numerical values in the coordinate input box.

3. RIGHT-CLICK with the mouse to follow any additional prompts to complete the command operation.

2.2.2 Working Modes in EnFEM

EnFEM operates in three essential modes: "CAD-MODE," "MESH-MODE," and "RESULT-MODE." In "CAD-MODE," users can create and modify geometric entities, set analysis input parameters such as stages, boundary conditions, material properties, interactions, and solver settings, define mesh parameters, and generate the mesh. "MESH-MODE" enables users to inspect, assess, and interact with the generated finite element mesh. Once the analysis is complete, "RESULT-MODE" facilitates post-processing tasks and visualizations of different contours or x-y curves. Each mode in EnFEM features its own distinct menu bar and toolbar icons.

2.2.3 Menu Bar Items

Primary menu items used in EnFEM are displayed in Figure2-8. Same commands may change slightly in different working modes.

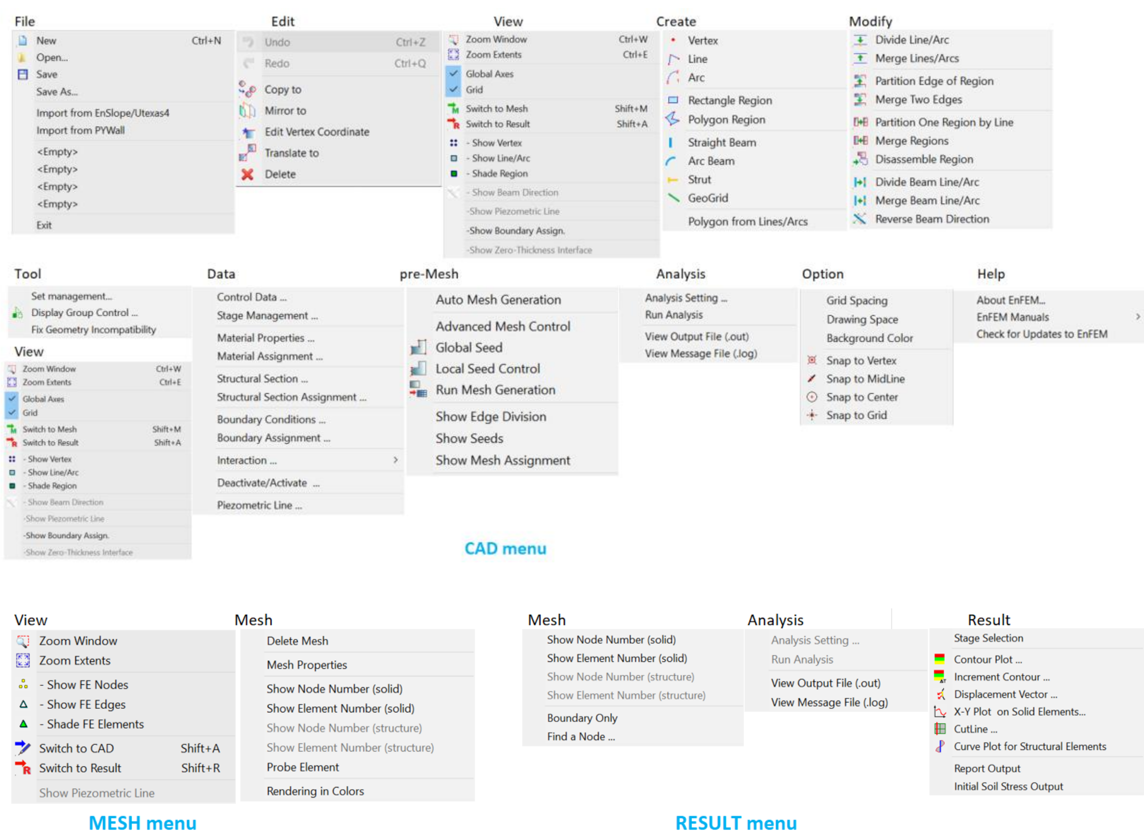


Figure2-8: Menu bar system in EnFEM

2.2.4 Toolbar Shortcut Icons

2.2.4.1 "CAD-MODE" Toolbar Icons

Figure 2-9 displays the icons found on the "CAD-MODE" toolbar. This toolbar is divided into two sub-toolbars: the first focuses on file operations, the creation of geometric entities, and zoom functionalities, while the second is dedicated to entity editing, modification, and mesh generation.

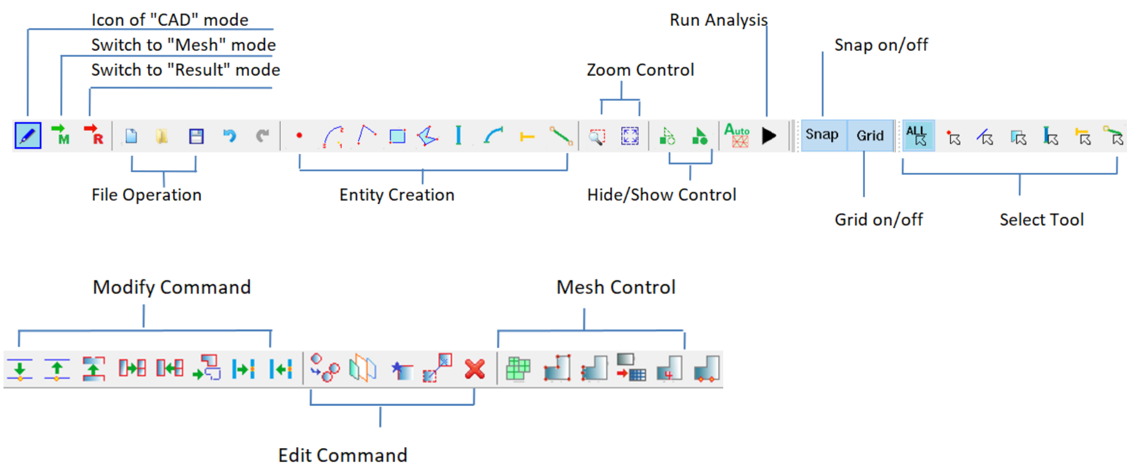


Figure 2-9: "CAD-MODE" toolbar

2.2.4.2 "MESH-MODE" Toolbar Icons

Figure 2-10 displays the icons from the "MESH-MODE" toolbar. It's important to note that these icons differ from those found in "CAD-MODE," and some of them may be inactive.

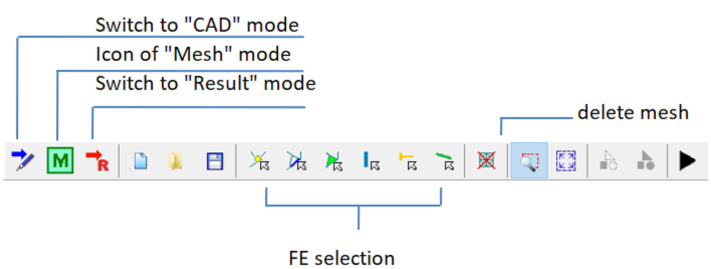


Figure 2-10: "MESH-MODE" toolbar

2.2.4.3 "RESULT-MODE" Toolbar Icons

Figure 2-11 displays the icons found in the "RESULT-MODE" toolbar. It's important to note that these icons differ from those in "CAD-MODE" and "MESH-MODE," with some icons being disabled. The result window features the same icons for "Zoom control," "File operation," and "Display control" as the preceding two toolbars.

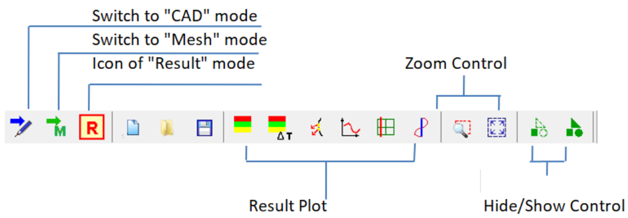


Figure 2-11: "RESULT-MODE" toolbar

2.2.5 Icons

All icons used in EnFEM are listed as follows, along with their action. Note the differences between *"inactive"*, and *"active"* icons.

	Switch to "CAD-MODE"		"CAD-MODE" activated
	Switch to "MESH-MODE"		"MESH-MODE" activated
	Switch to "RESULT-MODE"		"RESULT-MODE" activated
	Create a vertex		Create a vertex (operation active)
	Create a straight line		Create a straight line (operation active)
	Create an arc		Create an arc (operation active)
	Create a rectangle		Create a rectangle (operation active)
	Create a polygon		Create a polygon (operation active)
	Create a beam line		Create a beam line (operation active)
	Create a beam arc		Create a beam arc (operation active)
	Create a strut		Create a strut (operation active)
	Create a geogrid		Create a geogrid (operation active)
	"Show" vertices		
	"Show" region edges		
	"Show" shading of regions		
	"Show" beam direction		
	Select all entity type		Select all types (operation active)
	Select vertex only		Select vertex only (operation active)
	Select line/arc only		Select line/arc only (operation active)
	Select region only		Select region only (operation active)
	Select beam only		Select beam only (operation active)

	Select strut only		Select strut only (operation active)
	Select geogrid only		Select geogrid only (operation active)
	Divide a line/arc		
	Merge two lines/arcs		
	Partition edges of regions		
	Merge two edges		
	Partition a region		
	Merge regions		
	Dissemble regions		
	Divide a beam		
	Merge two beams		
	Reverse direction of a beam		
	Copy		
	Mirror		
	Edit vertex		
	Translate entities		
	Delete		
	Automatic mesh generation		
	Global seed assignment		
	Local seed assignment		
	Generate mesh		
	Show edge divisions		
	Show seeds on edge		
	“Show” FE nodes		
	“Show” FE edges		
	“Show” shading of FE elements		
	Select FE nodes		Select FE nodes (selection active)
	Select FE edges		Select FE edges (selection active)

	Select FE elements		Select FE elements (selection active)
	Delete mesh		
	Contour plots		
	Contour between two increments		
	Vector plots		
	X-Y plots		
	X-Y cutline-plot		
	Plot curve of a structural member		
Snap	On/Off of snap		
Grid	On/Off of grid plot		
	Zoom-window		Zoom-window (operation active)
	Zoom to extents		
	Display control		
	Show all entities		
Auto	Auto mesh generation		
	Run analysis		
	Snap to vertex		Snap to vertex (selection active)
	Snap to midpoint		Snap to midpoint (selection active)
	Snap to center (circle/polygon/rectangle)		Snap to center (selection active)
	Snap to grid		Snap to grid (selection active)
	New project		
	Open existing project		
	Save project		

2.2.6 File System

When an analysis is created and executed in EnFEM, several files are generated. These files, categorized by their extensions and briefly described below, include:

1. *.sep: Model file containing geometry definitions, materials, boundary conditions, mesh, analysis controls, and additional data.
2. *.fem: Neutral mesh file.
3. *.gsp: Results at Gaussian integration points, stored in binary format.
4. *.log: Log file that captures information during the analysis process.
5. *.dat: Input file to nonlinear solver.
6. *.out: Results output file in ASCII format.
7. *.inr: History of load increments.
8. *.ctb: Output of contact analysis in binary format.
9. *.obm: Results of structural members and zero-thickness interface elements in binary format

2.2.7 General Finite Element Analysis Workflow in EnFEM

In general, the following list outlines the workflow for creating and performing a FE-analysis in EnFEM:

1. In "CAD-MODE"
    a. Set up working zone preferences (drawing space extents, grid size, background color, etc.)
    b. Create/modify geometric entities
    c. Define set-groups (selection sets for assignments or display)
    d. Define control data
    e. Define stage data
    f. Define and assign material properties
    g. Define and assign boundary conditions
    h. Define and assign interaction properties (if necessary)
    i. Define mesh controls
        i. General controls (element shape, element type, mesh type)
        ii. Assign global seed control
        iii. Assign local seed control (if necessary)
    j. Generate mesh
    k. Setup solver controls (optional)
    l. Submit analysis to the solver
2. In "RESULT-MODE"
    a. Use post-processing tools to visualize desired output (contours, X-Y Plots, etc.)
3. If necessary, modify model and repeat analysis as needed.

CHAPTER 3. Creating & Editing Geometric Entities

3.1 Global Coordinate System

EnFEM operates within a Cartesian coordinate system where the horizontal axis aligns with the X-axis and the vertical axis aligns with the Z-axis, as illustrated in Figure3-1. The positive direction of the Y-axis extends 'into' the screen.

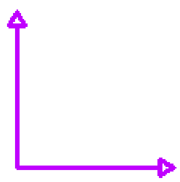


Figure3-1: Global coordinate system in EnFEM

When the analysis type is set to "axi-symmetric," a dashed orange vertical line will appear on the screen, indicating the axis of symmetry, as seen in Figure 3-2. In this case, the horizontal axis represents the radius direction.

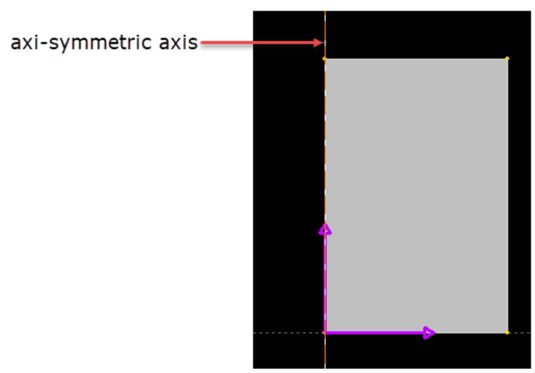


Figure 3-2: Axi-symmetric axis in EnFEM

To toggle the visibility of both the global axes and the axi-symmetric axis (if applicable), navigate to the menu bar and select "**View->Global Axes**."

3.2 Creating Geometric Entities

EnFEM is a two-dimensional finite element analysis software, which means that its geometric entities are confined to two-dimensional objects (regions), one-dimensional objects (such as lines, arcs, beams, struts, or geogrids), and zero-dimensional objects (vertices). Users can directly create simple 2D shapes, like rectangles or straight-edge polygons. More complex shapes, such as polygons with curved edges, can be formed indirectly by converting them from a series of closed lines or arcs.

When creating geometric entities in EnFEM, it's crucial to differentiate between independent and dependent entities—vertices, lines, and arcs. The classification hinges on their associativity with a geometric region. For instance, the commands "create a vertex," "create a line," and "create an arc" generate independent entities that lack direct associativity with a region. Independent lines and arcs are generally used for constructing a polygon indirectly or for creating auxiliary lines that aid in editing and modification. These independent entities are visually represented in deep blue.

In contrast, commands like "create a rectangle" and "create a polygon" directly produce regions that contain dependent entities. Specifically, vertices, lines, and arcs are termed dependent when they are part of a region entity (i.e., they are "children" of a "parent" region). Dependent lines and arcs are displayed in grey. It's important to note that boundary conditions and mesh controls in EnFEM can only be applied to these dependent entities.

Beams, struts, and geogrids are classified as independent entities, just like regions.

To maximize efficiency, users should proactively consider the types of boundary conditions that will be implemented, the necessary mesh refinement, and the expected outcomes. For example, an additional vertex may be needed along a region edge to apply point displacements at a specific location, or modeling the centerline of a region may facilitate easier result definitions at that point. Material properties and boundary conditions are assigned specifically to regions and the edges/vertices of regions, respectively. Solid finite elements are exclusively generated on 2D region objects. Consequently, a valid model should consist of 2D regions, beams, struts, or geogrids. Independent vertices, lines, and arcs will be automatically removed when the finite element mesh is created.

3.2.1 Vertex

Vertices are essential zero-dimensional entities that serve various purposes in EnFEM, such as acting as endpoints for lines or arcs, corners for regions, and endpoints for beams or struts. Independent vertices are rarely needed since EnFEM automatically generates vertices when creating 1D and 2D objects. Any independent vertices in the model will be removed during the mesh generation process. If an independent vertex coincides with the position of a dependent vertex belonging to a region, it can lead to errors when applying boundary conditions, potentially causing them to be mistakenly applied to the independent vertex instead. It is important to note that in EnFEM, vertices are distinct from finite element nodes.

There are two methods for creating a vertex in EnFEM:

3.2.1.1 Creation with Mouse

Start EnFEM, or click  to switch to *"CAD-MODE"*. Activate the vertex creation tool by clicking the icon on the toolbar. The information panel will then display the command prompt "Input new vertex." Position the mouse cursor at your desired location within the working area and LEFT-CLICK to create an independent vertex, which will be highlighted in yellow.

For more precise vertex placement using the mouse, you can utilize the "snap" technique. To do this, hold down the CTRL key to activate "snap" mode, then move the cursor to an existing vertex. Once you're aligned with the desired point, LEFT-CLICK to create the new vertex. The snapping point can be an existing vertex of a region, beam, strut, or geogrid, as well as an intersection of grid lines, the midpoint of a line, or the center of an arc. Further details about the "snap" technique can be found in Section 3.2.7 of this manual, as well as in the Example Manual.

<u>NOTE:</u> when using the "snap" technique, it is important to avoid creating redundant vertices, as highlighted in Section 3.2.1.

3.2.1.2 Creation with Coordinate Input Box

Alternatively, you can create a vertex by directly inputting its coordinates. In "CAD-MODE," click on the coordinate input box located beneath the working area and enter the X and Z coordinates as demonstrated in Figure 3-3. Remember to use a comma to separate the X and Z coordinates. Press ENTER to place the vertex at the specified location.

Figure 3-3: Using coordinate input box to create a vertex

3.2.2 Line

Lines in EnFEM are one-dimensional geometric entities characterized by two end vertices, and they are always straight. Independent lines are shown in a deep blue color, while dependent lines appear in a lighter grey, slightly less intense than that of their parent region entities.

To create a line in EnFEM, you can choose from two available methods:

3.2.2.1 Creation with Mouse

To start creating lines in EnFEM, open the program or switch to "CAD-MODE" by clicking the icon . Next, click the line-creation tool icon  on the toolbar. The information panel will display the prompt “Input start vertex of line.” Move your mouse cursor to the desired location within the working zone and LEFT-CLICK to set the starting vertex of the line. A dashed line will appear as a preview, displayed in cyan if the working zone has a black background, or in red if the background is white.

Once you place the start vertex, the prompt will change to “Input end vertex of line.” Move the cursor to where you want the endpoint of the line, then LEFT-CLICK again. The created line will now be shown in blue. You can continue creating additional lines by either clicking or entering coordinates, or you can right-click and select "**End Creation**" to stop the process.

<u>Note</u>: Line creation in EnFEM is continuous until you explicitly terminate it using the "**End Creation**" option from the right-click menu. If the current line creation is interrupted, the next line will begin from a new point, not from the previous line's end vertex. You can also press the ESC key to exit the line creation tool.

For more precise line placement using the mouse, utilize the “snap” feature. By pressing and holding the CTRL key, you can "snap" to specific points (as configured in the “**Options->snap to vertex**” menu). Release the CTRL key, then LEFT-CLICK to position the line vertex. Additional details about the “snap” technique can be found in Section 3.2.7 of this manual and the Example Manual.

3.2.2.2 Creation with Coordinate Input Box

In "CAD-MODE," click on the icon  in the toolbar to initiate the line creation process. Next, select the coordinate input box beneath the working zone and enter the X and Z coordinates as depicted in Figure 3-4. Press ENTER to set the start vertex of the line. If you move the mouse cursor away from the coordinate input box after entering the coordinates, a dashed cyan line will appear on the screen, similar to the first method.

At this point, the command prompt will update to show “Input end vertex of line.” Enter the desired coordinates for the end vertex and press ENTER. To finalize the line creation, you can right-click and select "**End Creation**," or simply press ESC to exit the command.

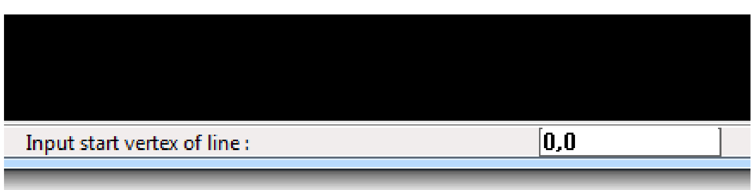


Figure 3-4: Using absolute coordinates to create start vertex of a new line

NOTE: When entering the coordinates for the endpoint vertex, you have two options: you can use absolute coordinates or relative coordinates. The format for absolute X and Z coordinates is to separate them with a comma, as illustrated in Figure 3-4. In contrast, relative coordinates are represented differently, as shown in Figure 3-5, where the X and Z coordinates are enclosed in square brackets **[x,z]**. This format indicates that the endpoint will be positioned relative to the starting point of the line. In the example provided in Figure3-5, the endpoint is set to be '*100 units*' away from the starting vertex in both positive X and positive Z directions.

Figure3-5: Using relative coordinates to create line end-vertex

Both methods of line creation can be combined freely to create multiple lines.

3.2.3 Arc

An arc is a one-dimensional geometric entity characterized by three vertices: the center, the starting point, and the endpoint of the arc. In EnFEM, arcs are always defined in this specific order and are constructed in a counterclockwise direction. The color coding for arcs parallels that of lines: an independent arc is depicted in blue, while a dependent arc that is part of a 2D region is shown in grey.

There are two methods available for creating an arc in EnFEM:

3.2.3.1 Creation with Mouse

To begin, start EnFEM or switch to “CAD-MODE” by clicking the appropriate icon . Next, click the arc creation tool icon  on the toolbar. The information panel will display the command prompt: “Input center vertex of arc.” Position your mouse cursor at the desired location in the working zone and LEFT-CLICK to set the center vertex of the arc. A dashed line will appear in cyan (if the working zone background is black) or in red (if the background is white), indicating the preview for the arc start vertex. The command prompt will now change to “Input start vertex of arc” in the information panel. Move your mouse to select the desired location and LEFT-CLICK to define the arc start vertex. A dashed arc will then be visible, similar to the middle image in Figure 3-6.

Next, the information panel will prompt you to “Input end vertex of arc”. Move your mouse in a counterclockwise direction and LEFT-CLICK again to set the arc end vertex, completing the arc creation. Once the arc is successfully created, the arc creation tool remains active, allowing you to create additional arcs if needed. To exit the arc creation tool, simply press the ESC key.

During the arc creation process, you can also utilize the "snap" technique to accurately capture the center or start vertex. By pressing and holding the CTRL key while moving the mouse, you can "snap" to an existing vertex. After snapping, LEFT-CLICK to define the center or arc start of your new arc. For detailed information about the "snap" technique, please refer to Section 3.2.7 of this manual, as well as the Example Manual.

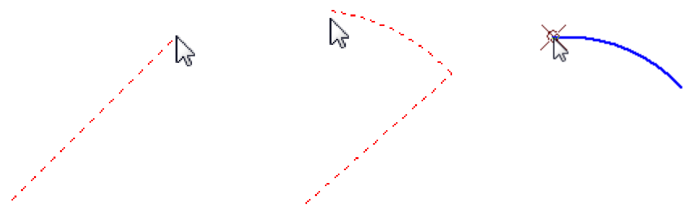


Figure 3-6: LEFT-CLICK three times to create an arc with the mouse

3.2.3.2 Creation with Coordinate Input Box

Launch EnFEM or switch to “CAD-MODE” by clicking the icon . Next, select the icon on the toolbar to activate the arc creation tool. Click in the coordinate input box located under the working zone and enter the X and Z coordinates for the arc's center, as illustrated in Figure 3-7. After entering the coordinates, press ENTER to define the arc center. You’ll see the command prompt change to display, “Input start vertex of arc”.

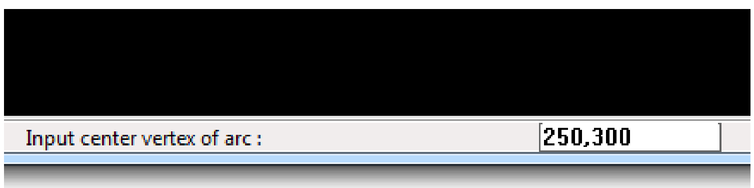


Figure 3-7: Key-in absolute coordinates of an arc center

Now, enter the absolute coordinates for the arc's start vertex and press ENTER. The command prompt will then instruct you to “Input end vertex of arc”. Lastly, enter the absolute coordinates for the arc's end vertex and press ENTER to complete the arc creation.

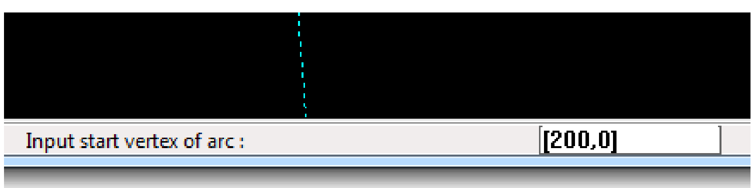


Figure 3-8: Key in relative coordinates to create the arc start vertex

Alternatively, you can use relative coordinates to define the arc's start vertex, as shown in Figure 3-8. For instance, entering `[200, 0]` indicates that the start vertex will be placed 200 units away from the arc center in the positive X-direction.

Similarly, you can define the end vertex using relative coordinates, as demonstrated in Figure 3-9. It's important to note that the relative coordinate definition for the end vertex differs from that of the start vertex—entering `[200, 45]` specifies an arc radius of 200 units and an arc angle of 45 degrees, measured counterclockwise.

Figure 3-9: Key in arc angle to create arc end vertex

You can freely combine both methods of arc creation. Here are some helpful tips to keep in mind:

1. Arcs are always drawn in counterclockwise direction.
2. You can use the snap-method to set the arc's end vertex at an existing vertex of another entity. However, if the existing vertex doesn't lie precisely on the arc's path, the end vertex might not appear where you expect, but rather along the arc's path.
3. If you press the ESC key after the first click, it will cancel the current arc creation operation. Pressing ESC after the second click will cancel the selection of the arc's start vertex and prompt you for a new input for it.
4. If the arc creation tool is waiting for you to input the arc's center, pressing ESC will terminate the arc creation tool entirely.
5. Note that EnFEM does not support arcs with a full 360° (a complete circle). To create a closed loop, you should use two or more arcs.

3.2.4 Rectangle

A rectangle is a two-dimensional shape defined by two diagonal vertices, as modeled in EnFEM. There are two methods available for creating a rectangle:

3.2.4.1 Creation with Mouse

To start creating a rectangle in EnFEM, first launch the application or switch to "CAD-MODE" by clicking . Then, select the rectangle-creation tool  from the toolbar. You'll see the command prompt "Input start vertex of rect." in the information panel. Position the mouse cursor where you want the first vertex of the rectangle to be, and LEFT-CLICK to place it. You'll notice dashed lines appearing in cyan (on a default black background) or in red (on a white background) to preview the rectangle, as shown in Figure 3-10. The command prompt in the information panel will now update to "Input end vertex of rect." Move the mouse cursor to the desired location of the opposite corner of the rectangle and LEFT-CLICK again to complete the rectangle.

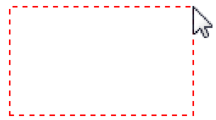


Figure 3-10: Create a rectangle with the mouse (white background)

3.2.4.2 Creation with Coordinate Input Box

In "CAD-MODE," start by clicking the icon on the toolbar. Next, click on the coordinate input box below the working area and enter the desired X and Z coordinates, such as "`0,0`". Press ENTER to establish the first diagonal vertex. You will see the command prompt update to say "Input end vertex of rect." Now, input a second set of X and Z coordinates, for example, "`200,100`," and press ENTER. This action will create a rectangle using absolute coordinates.

NOTE: You can also use relative coordinates for the second vertex by enclosing the input coordinates in square brackets `[x, z]`. This approach means the second vertex will be set relative to the first, based on the rectangle's width and height. Both methods of defining the vertices can be freely combined. To exit the rectangle creation tool, simply press the ESC key.

3.2.5 Simple Polygon

A simple polygon is a two-dimensional shape formed by connecting a series of vertices in sequential order with straight lines. The least number of edges required to create a simple polygon is three, corresponding to a triangle, while there is no upper limit on the number of edges.

In EnFEM, you can create a simple polygon using two methods:

3.2.5.1 Creation with Mouse

Launch EnFEM and switch to "CAD-MODE" by clicking . Next, activate the polygon creation tool by clicking the corresponding icon on the toolbar. The information panel will display the prompt "Input start vertex of polygon." Move your mouse to the desired location in the working area and LEFT-CLICK to establish the first vertex of the polygon. A dashed line will appear in cyan (on the default black background) or in red (on a white background) to provide a preview of the polygon, as shown in Figure 3-11. Once you set the first vertex, the command prompt in the information panel will change to "Input next vertex of polygon". Continue this process by LEFT-CLICKING to add as many additional vertices as desired (at least three). To complete the polygon, RIGHT-CLICK and select "**Close Polygon**," or you can simply LEFT-CLICK on the first vertex again.

For more precise polygon creation, you can use a "snap" technique. Hold down the CTRL key to snap to a specific point (as indicated by toolbar icons) and then LEFT-CLICK to place the polygon vertex. Further details about this snapping technique can be found in Section 3.2.7 of this manual, along with examples in the Example Manual.

Note: If the first vertex is created by snapping to an existing vertex or grid point, you can close the polygon by snapping back to the first vertex. If you press the ESC key before creating the second vertex, the current polygon creation will be canceled. Pressing the ESC key after completing the first edge will remove the last vertex.

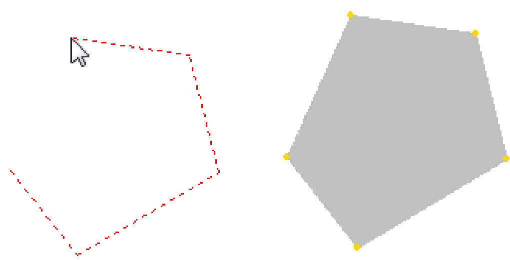


Figure 3-11: Create a polygon with the mouse

If you press the ESC key before creating the second vertex, the polygon creation operation will be canceled. Conversely, if you press the ESC key after completing the first edge of the polygon, the most recent vertex will be deleted.

3.2.5.2 Creation with Coordinate Input Box

In "CAD-MODE," click the icon  on the toolbar. Next, select the coordinate input box located beneath the working area and enter the desired X and Z coordinates (for example, "`0,0`"). Press ENTER to establish the first vertex of the polygon. You will see the command prompt change to display "Input next vertex of polygon". Now, enter a second set of X and Z coordinates, using either absolute or relative format `[x,z]`, and press ENTER to create the second vertex. Continue this process until you have created the desired number of vertices. Once all vertices are established, you can close the polygon by either re-entering the exact coordinates of the first vertex or by right-clicking and selecting "**Close Polygon**".

You can freely mix mouse clicks and coordinate inputs while creating polygon vertices.

3.2.6 General Polygon

General polygons differ from simple polygons due to their ability to incorporate curved edges, such as arcs. Examples of general polygons are illustrated in Figure 3-12.

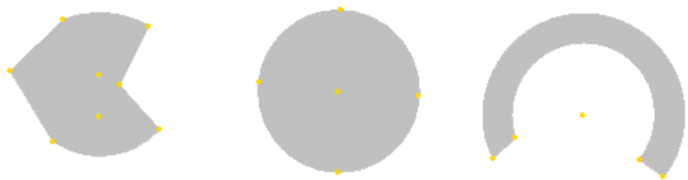


Figure 3-12: Examples of general polygons

Unlike the methods used to create simple polygons, general polygons are constructed indirectly. They consist of a series of connected lines and arcs created in a ‘head-to-toe’ manner, which can later be transformed into a region with a straightforward operation. For demonstration, a fan-shaped general polygon featuring two arcs and two straight lines is used, as depicted in Figure 3-13.

The following steps outline the basic procedure, referencing previous object creation methods. For further details, consult the full tutorial example in Example 1 - Thick Cylinder Subject to Internal Pressure found in the Example Manual.

1. Create two-arcs – an inner and outer – with a shared center. The length of the arcs can be chosen at your discretion.
2. Complete the arcs by drawing two lines that connect their endpoints, as shown in Figure 3-13.
3. From the menu bar, select **"Create->Polygon from Lines/Arcs"**. The line selection icon  should be automatically activated
4. Holding down the SHIFT key, LEFT-CLICK to select all four lines/arcs, and release the SHIFT key when complete. RIGHT-CLICK and select **“End Selection”** from the popup menu.
5. Once complete a closed region will have been created as shown in Figure 3-13.

Figure 3-13: Convert arcs and lines to a general polygon

Some useful tips for creating a general polygon, are as follows:

1. If the selected arcs and lines do not create a closed region, you will receive an error message when executing the command. In this case, carefully check the intersections of the lines and arcs, and if necessary, delete and recreate them.

2. Independent lines and arcs are displayed in blue. Once these components are converted into a general polygon, they become integral parts of that polygon and are no longer independent, with their colors changed to grey to indicate this dependence. Dependent lines and arcs cannot be used to form another general polygon. When two general polygons share an edge, they should be created using the following steps:

- Create edge segments of the first polygon
- Select these segments and convert them to the first general polygon
- For the second polygon, create **ALL** edges, even if some overlap with the edges of the first polygon.
- Select the newly created lines/arcs and convert them into the second general polygon (only independent lines and arcs will be recognized during this conversion process).

3. The *minimum* number of edges for common general polygon shapes is as follows

- Triangular region – three lines,
- Pie-shaped region - one arc and two lines,
- Circular region – two arcs

4. By default, the region entity will display its edges and shaded internal area. In the case of a curved region, arcs will be rendered as short line segments on the screen. If viewed closely, there may be small gaps between the shaded area and the curved boundary, which is completely normal.

5. Lines and arcs must be connected in a 'head-to-toe' manner. Intersecting line/arc segments is not permitted, as this could lead to invalid and entangled geometry. Examples of such invalid configurations can be found in Figure 3-14.

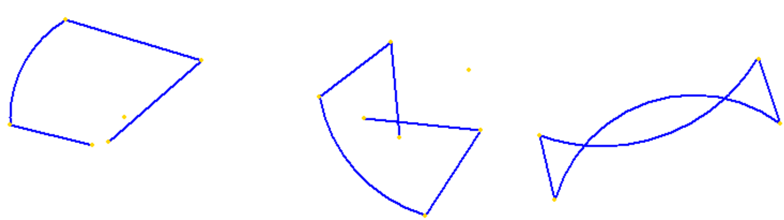


Figure 3-14: Invalid geometries for a general polygon

3.2.7 Straight Beam

Straight beams are one-dimensional geometric entities defined by two endpoints. They can be utilized to represent features like retaining walls or culverts with straight edges. Beams are visually represented in a sky-blue color and function as independent entities.

In EnFEM, there are two methods to create a straight beam:

3.2.7.1 Creation with Mouse

The process of creating a beam is similar to that of creating a line. Begin by launching EnFEM or click  to "CAD-MODE." To activate the straight beam creation tool, click the icon  on the toolbar. The information panel will display the prompt: “Input start vertex of straight beam.” Position your mouse cursor at the desired location in the working area and LEFT-CLICK to establish the starting vertex of the beam. A cyan dashed line (on a black background) or a red dashed line (on a white background) will be shown as a preview, similar to line creation.

Next, the information panel will change to prompt: “Input end vertex of straight beam.” Move the mouse cursor to the desired endpoint and LEFT-CLICK again. The new straight beam will now appear on the screen in sky blue. To finish the beam creation process, right-click and select "**End Creation**."

If the new beam coincides with an existing region edge or intersects with an existing region, the region will automatically be partitioned to ensure geometric compatibility. Additionally, the beam will be partitioned at the edge intersections.

NOTE: You can create multiple straight beams in succession. To begin a new beam, right-click and choose “**End Creation**,” or press the ESC key to exit the current beam creation tool. The snap technique for creating straight beams is consistent with previous methods. Hold down the CTRL key to “snap” to a specific point, then LEFT-CLICK to define either the start or end vertex of the beam. Further information about the “snap” technique can be found in Section 3.2.7 of this manual, as well as in the Example Manual.

3.2.7.2 Creation with Coordinate Input Box

In "CAD MODE," begin by clicking the icon on the toolbar. Next, select the coordinate input box located beneath the working area and enter the X and Z coordinates as demonstrated in Figure 3-4. After inputting the coordinates, press ENTER to establish the starting vertex of the beam. To initiate a new beam line, RIGHT-CLICK and choose "**End Creation**," or simply press ESC to exit the current beam creation tool.

NOTE: When entering the coordinates for the end vertex, you have two options: you can input them as absolute coordinates or relative coordinates. For absolute coordinates, format them as the desired values separated by a comma, as illustrated in Figure 3-4.The format of a *relative coordinate* input is displayed in Figure3-5, where the X and Z-coordinates are wrapped by a pair of square brackets ***[x, z]***. This input specifies that the beam end vertex will be placed at a location ‘relative’ to the beam start vertex.

Both methods for creating straight beams can be used interchangeably. You can press ESC at any time to exit the beam creation process.

3.2.8 Arc Beam

An arc beam is a one-dimensional geometric entity frequently employed to model a curved culvert. In EnFEM, an arc beam is defined by three vertices: the center, the arc start, and the arc end, which must be created in that specific order. Furthermore, arc beams are constructed in a counterclockwise direction. In the interface, arc beams are represented by a sky-blue color.

To create an arc beam in EnFEM, there are two available methods:

3.2.8.1 Creation with Mouse

To create an arc beam in EnFEM, first, launch the software or switch to "CAD-MODE". Then, click the icon  on the toolbar to enable the arc beam creation tool. The information panel will display the command prompt: "Input center vertex of arc beam." Position your mouse cursor at the desired location in the working area and LEFT-CLICK to establish the center vertex of the arc beam. A dashed line will appear in cyan on a default black background or in red on a white background, indicating the position of the arc's start vertex.

The command prompt in the information panel will then change to: "Input start vertex of arc beam." Move your mouse cursor to select the desired location for the start vertex and LEFT-CLICK to place it. A dashed arc will now be visible, as shown in the middle image of Figure 3-6.

Next, the information panel will prompt: "Input end vertex of arc beam." Move your mouse in a counterclockwise direction and perform a LEFT-CLICK again to create the end vertex, completing the arc definition.

For greater precision when creating an arc beam with the mouse, you can employ the snap technique. By holding down the CTRL key, you can snap to specific points (controlled by toolbar icons) and then LEFT-CLICK to establish the arc beam's vertices. More details on the snap technique can be found in Section 3.2.7 of this manual, as well as in the Example Manual.

3.2.8.2 Creation with Coordinate Input Box

In *"CAD-MODE"*, click the  icon on the toolbar. Click the coordinate input box under the working zone and key in the X and Z-coordinates as shown in Figure 3-7.

Press ENTER to set this point as the center vertex of the arc beam. You will notice that the command prompt now prompts you with the message, "Input start vertex of arc beam." Next, enter the desired absolute coordinates for the start vertex of the arc and press ENTER again. The command prompt will then change to say, "Input end vertex of arc beam." To complete the arc beam creation, enter the absolute coordinates for the end vertex and press ENTER.

Alternatively, you can use relative coordinates to define the start vertex of the arc beam, as illustrated in Figure 3-8. Similarly, relative coordinates can also be applied to create the end vertex, as shown in Figure 3-9. Keep in mind that the definition for relative coordinates differs for the end vertex compared to the start vertex. For instance, using `[200, 45]` denotes an arc radius of 200 units and an arc angle of 45 degrees in a counterclockwise direction.

You can freely combine both methods of creating arc beams. Here are some helpful tips to keep in mind:

1. Arc beams are always created in the counterclockwise direction.
2. The snap-method can be used to set the end vertex at an existing vertex of another entity. Note, however, that if the existing vertex does not align perfectly with the arc path, the end vertex may not be placed where you expect it; instead, it will be positioned on the arc path.
3. Pressing the ESC key after the first click will cancel the current creation operation. If you press ESC after the second click, it will cancel the start vertex of the arc and prompt for a new input.
4. To create a closed circular beam, combine two or more arc segments.

3.2.9 Strut

Struts are linear geometric entities characterized by two endpoints. In EnFEM, struts can be utilized to model horizontal supports for retaining walls or tension cables in tie-back structures. In the EnFEM interface, struts are represented in a sandy brown color.

There are two methods available for creating a strut in EnFEM:

3.2.9.1 Creation with Mouse

Launch the program or switch to "CAD-MODE" by clicking . Activate the strut creation tool by clicking  on the toolbar. The information panel will display the command prompt “Input start vertex of strut.” Position your mouse cursor at the desired location within the working zone and LEFT-CLICK to set the start vertex of the strut. You will see a dashed line in cyan (on the default black background) or in red (if using a white background) following your cursor as a preview for the strut's placement.

Next, the information panel will change to show the command prompt “Input end vertex of strut.” Move the cursor to the intended endpoint and LEFT-CLICK again. A new strut will appear on the screen in a sandy brown color. Remember, a strut consists of only one segment, so there's no need to use the right-click menu to finish it. To exit the strut creation tool at any time, simply press the ESC key.

Additionally, the snap technique can be utilized during strut creation. For detailed information regarding the snap technique, refer to Section 3.2.7 of this manual, as well as in the Example Manual.

3.2.9.2 Creation with Coordinate Input Box

In "CAD-MODE," click on the toolbar to begin. Navigate to the coordinate input box below the working area and enter the X and Z coordinates as illustrated in Figure 3-4. After entering the values, press ENTER to set the start vertex of the strut. When you move the mouse cursor away from the coordinate input box, a dashed cyan line will appear on the screen, similar to the first method.

Now, the command prompt will display the message "Input end vertex of strut." Enter the desired coordinates for the end vertex and press ENTER. A strut has two vertices only. It is not continuous.

NOTE: You can input the end vertex coordinates in two ways, just like with line creation: you can use absolute coordinates or relative coordinates. Absolute X and Z coordinates should be formatted using a comma to separate the values, as shown in Figure 3-4. For relative coordinates, format the input as shown in Figure3-5, where the X and Z coordinates are enclosed in square brackets *[x, z]*. This format indicates that the end vertex will be positioned relative to the start vertex of the strut.

To exit the strut creation tool at any time, simply press the ESC key.

3.2.10 Geogrid

Geogrids are linear, one-dimensional geometric entities characterized by two end vertices. Within EnFEM, geogrids can be utilized to model tieback anchors (deadmen) or synthetic reinforcements in soil. They are always fully bonded to soil parcels and are represented in EnFEM with a green color.

There are two methods available in EnFEM for creating a geogrid:

3.2.10.1 Creation with Mouse

Launching the program or switching to "CAD-MODE." Then, click to activate the beam-line creation tool. The information panel will display the command prompt “Input start vertex of geogrid.” Position your mouse cursor at your desired location in the working zone and perform a LEFT-CLICK to establish the starting vertex of the geogrid.

A dashed line will appear in cyan (if using the default black background) or in red (if the background is white), following your cursor as a preview of the line being created. Once you see the message “Input end vertex of geogrid” in the information panel, move the cursor to your desired endpoint and LEFT-CLICK again. The newly created geogrid will now be displayed in green.

Since each geogrid consists of a single segment, there is no need to right-click and select "**End Creation**" to complete the process. Additionally, the snap technique can be utilized during the geogrid creation. For further details on the snap technique, please refer to Section 3.2.7 of this manual, as well as in the Example Manual.

3.2.10.2 Creation with Coordinate Input Box

In "CAD-MODE," activate the geogrid creation tool by clicking  on the toolbar. Next, click the coordinate input box located beneath the working zone and enter the desired X and Z coordinates as illustrated in Figure 3-4. Press ENTER to establish the geogrid vertex. If you move the mouse cursor outside of the coordinate input box after entering the values, a dashed line will appear on the screen, similar to the first method.

The command prompt will now indicate “Input end vertex of geogrid.” You can input the desired coordinates for the end vertex and press ENTER.

NOTE: There are two methods for entering the end vertex coordinates, similar to the line creation process: you can provide <u>absolute</u> coordinates or <u>relative</u> coordinates. For absolute coordinates, enter the X and Z values separated by a comma, as shown in Figure 3-4. For relative coordinates, format the input as shown in Figure3-5, wrapping the X and Z values in square brackets *[x, z]*. This method defines the beam's end vertex based on a location that is relative to the start vertex. To exit the current geogrid creation tool, simply press the ESC key.

3.2.11 Snap-Methods

“Object snap” is a built-in feature in EnFEM designed to assist users in creating geometry with precision. This functionality enables users to accurately capture an existing vertex when establishing a new vertex for a geometric entity. The available “Snap-to” options in EnFEM include the following, which can be activated or deactivated via the menu “**Options->Snap to**”:

1.  : snap to the end vertices of an existing region, line, or beam
2.  : snap to midpoint of a line or beam (which can be either straight or curved)
3.  : snap to the center of an arc/ arc beam
4.  : snap to grid intersections

By default, all four snap modes are enabled, but users have the option to deactivate any of them. To use the snapping feature, hold down the CTRL key while moving the mouse cursor to the desired location until the snap icon appears. Once the snap icon is visible, click the left mouse button to create the vertex at that precise location. Figure 3-15 illustrates the snap icon during the creation of a new line, where the starting vertex of the new line is snapped to the corner vertex of an existing rectangle.

Figure 3-15: Snap to the corner of a rectangle when create a new line

This manual, as well as the EnFEM Example Manual will go on to refer to this technique throughout this document, as the 'snap method'. Alternatively, it also may mention 'snap to …' with some geometric reference, as in 'snap to the bottom-right vertex'.

3.2.12 Adjustment of view in creation procedure

When creating geometric entities, users may need to draw a line or polygon that extends from the current view to another area outside the working zone. To facilitate this, users can utilize the mouse wheel to zoom in and out, as well as the middle mouse button to pan the view, all without interrupting the ongoing line or polygon creation process. Additionally, the "zoom extents" and "zoom window" functions can be employed to change the working zone while maintaining the current line or polygon creation

3.3 Selecting Geometric Entities

3.3.1 Selection Type

Many operations involve selecting geometric entities. EnFEM have six selection filters: vertex filter , line/arc filter , region filter , beam filter , strut filter and geogrid filter . These six filters are mutually exclusive. At any time, the user can only activate one of them. To select a geometric entity, the user must first activate the appropriate selection icon. For instance, to select a region edge, click on the line/arc selection icon and ensure that the other filters are turned off. If not, an unintended entity, such as a vertex, may be selected, leading to errors.

Many editing and modifying commands in EnFEM will automatically activate the correct selection icons. However, users may occasionally need to manually activate or deactivate selection icons as needed.

For cases where multiple entity types need to be selected, EnFEM offers a "select all" option. When this option is enabled , users can select any type of geometric entity based on the cursor's position. The priority for selection is as follows: **vertex** > **line/arc** > **region**. For example, if the cursor is positioned near a vertex, a region edge, and within a region, only the vertex will be selected.

The highlights for selected geometric entities are as follows (see Figure 3-16 for reference):

- Vertex: highlighted with a red circle around it.
- Line/arc: rendered in red.
- Region: rendered in cyan.
- Beam lines/arcs: rendered in red
- Strut: rendered in red
- Geogrid: rendered in red

NOTE: To deselect currently selected entities, activate the corresponding selection icon or the "select all" option , then LEFT-CLICK on a blank area of the screen.

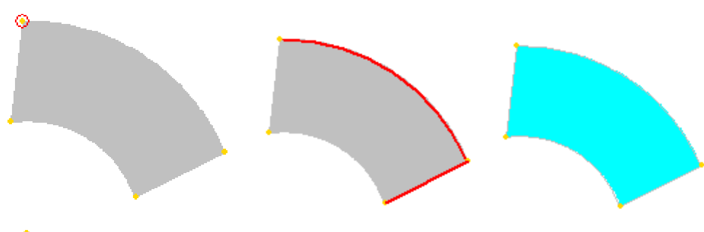


Figure 3-16: Selection of geometry entities: vertex, line/arc, and region

3.3.2 Methods of Selection

There are two methods for selecting geometric entities in EnFEM: "click to select", or use a selection window. Both methods are detailed below.

3.3.2.1 Click to Select

The first selection method involves a simple Left-Click on the desired entity (such as a vertex or line) to select it. If multiple entities need to be selected, hold down the SHIFT key while Left-Clicking on each additional geometric entity in succession. If you accidentally select an undesired entity while holding the SHIFT key, you can simply click on that entity again to remove it from the selection set. This method is particularly convenient for selecting a small number of geometric entities.

3.3.2.2 Selection Windows

The second method, which is often used, is utilizing a selection window. There are two types of selection windows available: “contained” and “crossing.” For illustrative purposes, refer to the "fan-shaped" geometry in Figure 3-16 when reviewing the following sections.

3.3.2.2.1 Contained Selection Window

For a “contained” selection, begin by positioning the cursor in the bottom left corner of the geometry, click and hold the left mouse button, then drag the cursor toward the top right corner of the display. During this action, a green selection window with solid edges will appear on the screen. Segments will be included within the selection box. For example, once the selection window encloses all four line/arc entities in Figure 3-16, release the mouse button to complete the operation.

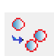
3.3.2.2.2 Crossing Selection Window

For a “crossing” selection, start by moving the cursor to the upper right corner of the geometry. With the desired selection icon active, click and hold the left mouse button, then drag the cursor diagonally toward the lower left. This movement will create a crossing selection window, which is indicated by yellow dashed edges. Upon releasing the mouse button, all entities that intersect with the crossing window—as well as those entirely within it—will be selected.

3.4 Editing Geometric Entities

The following editing tools are provided by EnFEM: "Copy", "Mirror", "Edit Vertex Coordinate", "Translate", and" Delete". Details of these editing tools can also be found in the context of full tutorial examples from the included Example Manual.

3.4.1 Copy Entity

The *"copy to"*  tool is used to generate identical geometric entities at a different position. Source objects should be independent entities such as independent vertices, independent lines/arcs, regions, beams, struts or geogrids. Dependent geometric entities cannot be copied. Copy of a region has the same material assignment as the source region. However, boundary conditions and contact/interaction definitions are not copied to the new region entity.

Procedures to perform the operation are as follows:

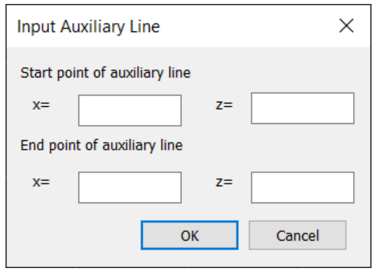


Figure 3-17: Definition of auxiliary line as the copy reference

1. Click to switch to *"CAD-MODE"* if necessary.
2. On the menu bar, click **"Edit->Copy to"**, or click the icon from the toolbar
   a. Turn on one selection filter
   b. Note that the following command prompt is displayed -**"Select geometry entities to copy"** on the information panel which locates below the working zone window
3. Select the desired *independent* entities to copy. Dependent entities, such as vertices of a line, or edges of a region, cannot be copied.
4. RIGHT-CLICK and select **"End Selection"** from the popup menu.
   a. NOTE: Clicking" **Cancel**" will terminate the current *"copy to"* operation.
5. A dialog box will open for defining an auxiliary line as the copy reference **Error! Reference source not found.**(Figure 3-17). Key in the start and end coordinates of the auxiliary line.
   a. Since only the *difference* of the X and Z-coordinates is of interest, inputs can be arbitrary values. For example, enter in a start point of *"0,0"* and an ending point of *"0,100"*. This copies the object, *100-units* in the positive Z-direction

6. Click **"OK"** to close the dialog and complete the copy operation. The location of copied objects is the position of the source objects plus the shift of the auxiliary line.
   a. If **"Cancel"** is clicked instead of *"**Ok**"*, current copy operation will be terminated.
7. Repeat steps 3 ~ 6 to copy other geometric entities or RIGHT-CLICK and select "**Cancel Command**" to terminate "**Edit->Copy to**" command

3.4.2 Mirror Entity

The *"mirror to"* tool is used to copy geometric entities *'mirrored'* about a reference line. Source objects should be independent entities such as independent vertices, independent lines/arcs, beams, struts, geogrids, or regions. Dependent geometric entities cannot be mirrored. Mirrored regions have the same materials as the source regions. However, boundary conditions and contact/interaction definitions are not copied.

Procedures to perform the operation are as follows:

1. Click to switch to *"CAD-MODE"* if necessary
2. On the menu bar, click **"Edit->Mirror to"** or click the icon from the toolbar
    a. Turn on one selection filter
    b. On the bottom toolbar, below the working zone, the following command prompt is displayed - **"Select geometry entities to mirror"**.
3. Select the desired independent entities to mirror
4. RIGHT-CLICK and select **"End Selection"** from the popup menu.
   NOTE: Clicking *"***Cancel***"* will terminate the current *"mirror to"* operation.
5. A dialog box will open for defining an auxiliary line as the mirror axis (Figure 3-17**Error! Reference source not found.**). Key in the start and end coordinates of the auxiliary line.
6. Click **"OK"** to close the dialog and complete the mirror operation. Source and mirrored objects are symmetrical about the auxiliary line.
    a. If **"Cancel"** is clicked instead of *"***Ok***"*, current mirror operation will be terminated.
7. Repeat step 3 ~6 to mirror copy other geometric entities, or RIGHT-CLICK and select "**Cancel Command**" to terminate the "**Edit->Mirror** *to*" command.

3.4.3 Edit Vertex Coordinate

This tool is used to relocate one or more independent or dependent vertices to new positions by means of modifying their coordinates directly. This operation does not change the material assignment, boundary conditions, or contact interface definitions assigned to the region.

Procedures to perform the operation are as follows:

1. Click to switch to *"CAD- MODE"* if necessary
2. On the menu bar, click **"Edit->Edit Vertex Coordinate"** or click the icon from the toolbar
3. Make sure that only the vertex selection icon is turned on.
4. On the bottom toolbar, below the working zone, the following command prompt is displayed - **"Select vertex entities to edit coordinates"**.
5. Select one vertex to edit.
6. RIGHT-CLICK and select **"End Selection"** from the popup menu.
    a. NOTE: Clicking *"***Cancel***"* will terminate the current *"Edit Vertex"* operation.

7. A dialog window will open, as shown in Figure 3-18. Check the appropriate option **"Apply to X"** and/or **"Apply to Z"**, and input the desired new coordinates in the input box(es). Old coordinates will be displayed in the input box as a reference.

8. Click **"OK"** to close the dialog and complete vertex relocation.
    a. If **"Cancel"** is clicked instead of *"***Ok***"*, current mirror operation will be terminated.

9. Repeat step 5 ~ 8 to edit other vertices coordinates, or RIGHT-CLICK and select "**Cancel Command**" to terminate current "**Edit->Edit Vertex Coordinate**" command.

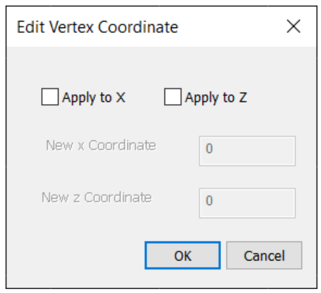


Figure 3-18: Edit vertex coordinates

NOTE: This operation relocates selected vertices and changes the region topology. Sometimes this operation may result in an inverted region direction or entangled topology. In EnFEM, all region objects use a counterclockwise direction for boundary condition assignment and mesh-seed control. If the direction of a region is inverted, switching to clockwise, issues will be encountered during boundary condition assignment and mesh generation. Users should be aware of the possibility of inverted region direction when editing vertices. Figure 3-19 shows two instances which should be avoided – crossing geometry and clockwise directions – when editing vertices.

Additionally, relocating more than one selected vertex may generate redundant vertices and zero-length line segments if both options "Apply to X" and "Apply to Z" are checked, and also should be avoided.

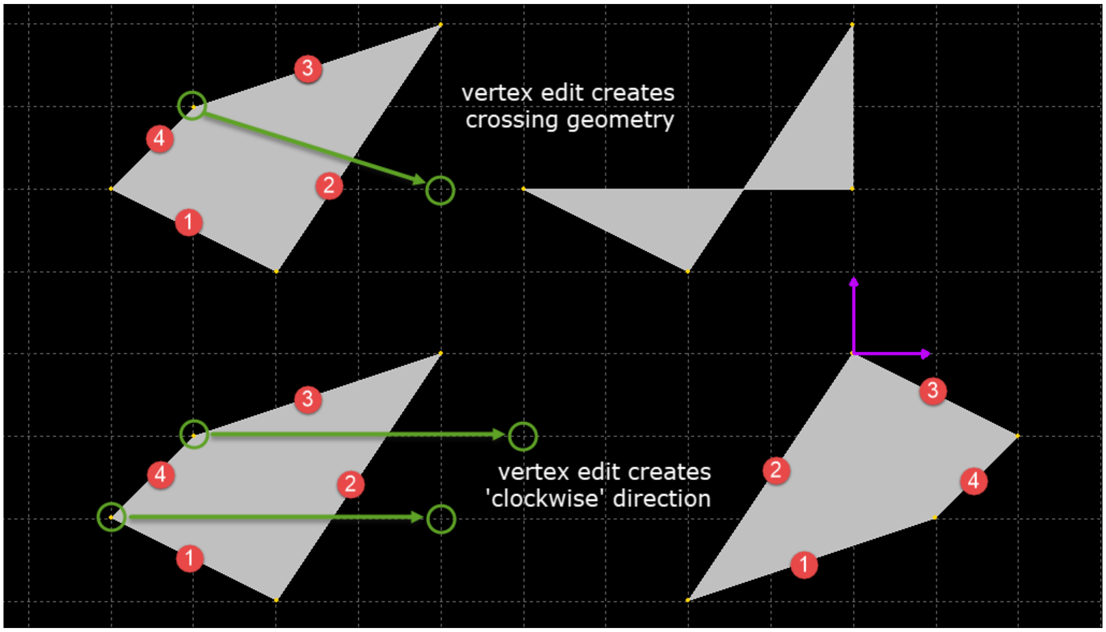


Figure 3-19: Cases to avoid when editing vertex coordinates

3.4.4 Translate Entity

This tool is used to move geometric entities (independent vertices, independent lines/arcs, regions, beams, struts, or geogrids) by the distance provided by the user. The operation does not change the material assignment, boundary conditions, or contact interface definitions assigned to the region.

Procedures to perform the operation are as follows:

1. Click to switch to *"CAD- MODE"* if necessary
2. On the menu bar, click **"Edit->Translate to"** or click the icon from the toolbar
   a. Turn on one selection filter
   b. On the bottom toolbar, below the working zone, the following command prompt is displayed - **"***Select geometry entities to translate***"**
3. Select the desired independent entities to copy.
4. RIGHT-CLICK and select **"End Selection"** from the popup menu.
   a. NOTE: Clicking *"***Cancel***"* will terminate the current *"translate to"* operation.
5. A dialog window will open, as shown in Figure 3-20. Key in the desired translation values for the X and Z-directions.
6. Click **"OK"** to close the dialog and complete the translation operation.
   a. If **"Cancel"** is clicked instead of *"***Ok***"*, current translation operation will be terminated.
7. Repeat step 3 ~ 6 to translate other geometric entities, or RIGHT-CLICK and select "**Cancel Command**" to terminate the "**Edit->Translate to**" command

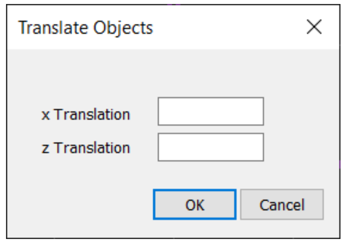


Figure 3-20: Translate objects

3.4.5 Delete Entity

This tool is used to delete independent vertices, independent lines/arcs, regions, beams, struts, or geogrids. Vertices of independent lines/arcs, and dependent lines/arcs of a region cannot be deleted.

If one or more regions are deleted, this operation may have an influence on material assignments, boundary condition assignments, contact interface definitions, and mesh generation, among other input data. Users should check all relevant assignments and definitions carefully to maintain data integrity before running the analysis.

If one dependent vertex of a region is deleted, its parent region will be updated based on two different situations:

1. If two arcs intersect at the deleted vertex and these two arcs belong to the same circle, a new arc will replace these two arcs. Following the counterclockwise direction, the new arc begins from the start vertex of the front arc and ends at the end vertex of the back arc.
2. Lines/arcs intersecting at the deleted vertex will be replaced by a new line. This line will connect the ends of the deleted lines/arcs directly.

If there are two vertices at a common corner of two adjacent regions, it is recommended to hide one region before performing the delete operation on the desired vertex. In this scenario it is also recommended to referenceSection3.6 for valid geometric configurations.

Procedures to perform the operation are as follows:

1. Click to switch to *"CAD- MODE"* if necessary
2. On the menu bar, click **"Edit->Delete"** or click the icon from the toolbar
    a. Turn on one selection filter .
    b. On the bottom toolbar, below the working zone, the following command prompt is displayed - "Select geometry entities to delete".

c. Additionally, a warning message will appear, as shown in Figure 3-21**Error! Reference source not found.**. User's may check the *"Do not display…"* option if desired. Click **"Close"** to close the warning message window.

3. Select the desired entities to delete.

4. RIGHT-CLICK and select **"End Selection"** from the popup menu to complete the operation.

a. NOTE: Clicking *"***Cancel***"* will terminate the current "delete*"* operation.

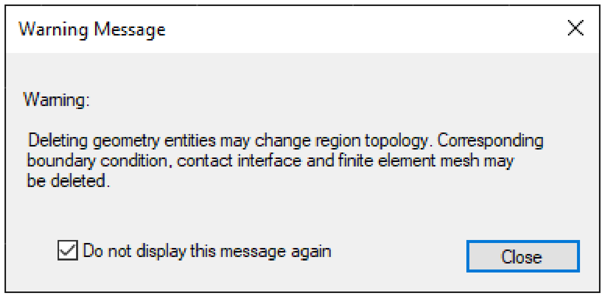


Figure 3-21: Warning message for deleting operation

11. Repeat step 3 ~ 4 to delete other geometric entities, or RIGHT-CLICK and select "**Cancel Command**" to terminate current "**Edit->Delete**" command.

3.4.6 Undo/Redo

The "Undo" function allows users to easily reverse their most recent actions involving creation, editing, or modification. The "Redo" function serves as the counterpart to "Undo," enabling users to restore any actions that were undone. However, following restrictions are applied:

1. By default, *"*Undo*"* feature is limited to 30 operations

12. Actions performed under the "Data," "Mesh," and "Analysis" drop-down menus do not support undo/redo functionality.

13. When creating continuous lines or beams, it is advisable not to use "Undo" to cancel a previous line segment.

14. Be aware that after saving the current model, all undo/redo data will be cleared

To perform these operations, follow these steps:

1. Click to switch to *"*CAD-MODE*"* if necessary
2. On the menu bar, click **"Edit->Undo"** or click the icon from the toolbar
   a) For the *"*Redo*"* command, select **"Edit->Redo"** or click the icon from the toolbar

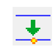
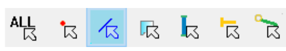
3.5 Entity Modification

In EnFEM, a variety of modification tools are available, including "divide," "merge," "partition," and "disassemble." These tools allow users to manipulate existing independent geometric entities. The following section provides details on how to use these tools.

3.5.1 Divide Independent Line/Arc

This tool is specifically designed to divide two intersecting lines, two arcs, or a combination of one line and one arc. It is important to ensure that the selected lines or arcs are independent entities, identifiable by their blue color.

Procedures to perform the operation are as follows:

1. Click  to switch to *"CAD- MODE"* if necessary.
2. On the menu bar, click **"Modify->Divide Line/Arc by Intersection",** or click the icon  from the toolbar. The line selection icon is automatically activated as shown as .
3. Select the desired lines or arcs. Make sure that selected lines/arcs are intersected.
4. RIGHT-CLICK and select **"End Selection"** from the popup menu to complete the operation.

Important Notes:

- Clicking "**Cancel**" will abort the ongoing "divide" operation.

- If you RIGHT-CLICK and select "**Cancel Command**," it will terminate the "**Modify->Divide Line/Arc by Intersection**" command entirely.

The procedure is illustrated in Figure 3-22. The left image displays the original two source lines, the middle image shows the selected lines, and the right image confirms the division operation. In this case, two independent lines are divided into four separate segments at their intersection point.

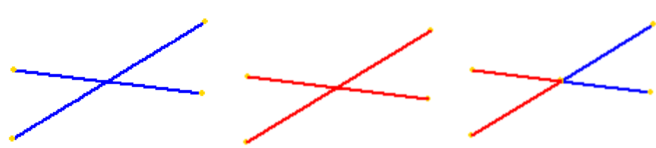


Figure 3-22: Divide independent lines/arcs by intersection

Please note that this dividing operation is only applicable to lines or arcs that genuinely intersect. Configurations resembling a "T" shape do not qualify as intersecting and therefore cannot be processed using this tool.

3.5.2 Merge Independent Lines/Arcs

The “merge” operation is used to combine two line segments or two arc segments into a single entity. It is essential that the line or arc segments involved are independent entities. When merging two lines, a new line segment will be created that connects the outer vertices of the original lines. If two arcs are selected for merging, they must both belong to the same circle.

Procedures to merge two lines/arcs:

1. Click to switch to *“CAD- MODE”* if necessary
2. On the menu bar, click **"Modify->Merge Lines/Arcs"** or click the icon from the toolbar. The line selection icon is automatically activated as shown as .
3. Select geometric entities you wish to merge.
4. RIGHT-CLICK and select **"End Selection"** from the context menu to complete the operation.

Important Notes:

- Clicking "**Cancel**" will terminate the ongoing “merge” operation.

- If you RIGHT-CLICK and select "**Cancel Command**," it will abort the entire "**Modify->Merge Lines/Arcs**" command.

This procedure is illustrated in Figure 3-23. The left image displays three line segments, with the two lines on the left designated for merging. The merged line segment is depicted in the right image of Figure 3-23.

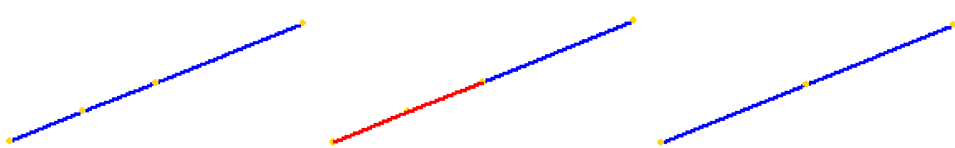


Figure 3-23: Merge line segments

<u>NOTE</u>: This operation is applicable for merging two lines or two arcs and is valid for independent only. The lines or arcs to be merged should be connected "head-to-toe”.

3.5.3 Partition Edge

This operation allows users to partition (divide) one or more edges of a region using an auxiliary line. When the auxiliary line intersects with the region edges, additional vertices will be created at these intersection points. Before starting, users need to create an independent line entity to serve as the auxiliary line. It is important to note that the auxiliary line will not be deleted automatically and must be removed manually if no longer needed.

Steps to Perform the Partition Operation:

1. Click  to switch to *"CAD- MODE"* if necessary.
2. Create an auxiliary line which intersects with the desired region edge.
   a. On the menu bar, click **"Modify->Partition Edge of Region"** or click the icon  from the toolbar. Make sure the "Select All" icon is activated 
3. Select the auxiliary line and region-edge.
4. To complete the operation, RIGHT-CLICK and select **"End Selection"** from the popup menu. If a beam or geogrid is also selected and intersected with the auxiliary line, this beam/geogrid will be partitioned as well.

Important Notes:

- Clicking "**Cancel**" will abort the current partition operation.

- If no auxiliary line is provided, a prompt will appear, allowing users to enter the coordinates for the auxiliary line.

- After partitioning, check to ensure that the source edges have been appropriately divided.

- Remember that the auxiliary line will NOT be deleted automatically; users need to delete it manually if desired.

- If you RIGHT-CLICK and select "**Cancel Command**," the active edge partition command will be terminated

This process is visually represented in Figure 3-24.

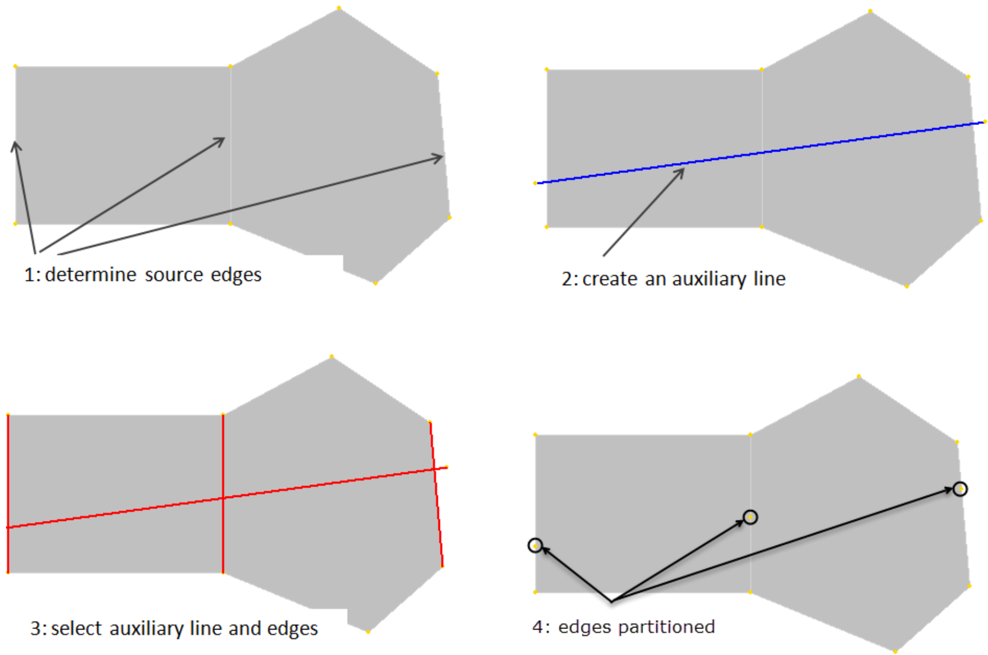


Figure 3-24: Partition Edge of Region

If a beam or geogrid overlaps with the partitioned edge, it will be automatically divided to ensure geometric compatibility.

3.5.4 Merge Edges

This operation allows you to merge two edges within the same region, which can be either two lines or two arcs. It's important to note that when merging two arcs, they must be positioned on the same circle. Edges from different regions cannot be merged.

To execute this operation, follow these steps:

1. Click to switch to *"CAD- MODE"* if necessary.
   a. On the menu bar, click "**Modify->Merge Two Edges"** or click the icon from the toolbar. You can use the "Select All" option or use the line/arc filter icon to select region edges
2. Use the mouse to select two edges that belong to the same region.
3. To finalize the operation, RIGHT-CLICK and select **"End Selection"** from the popup menu.
4. If you wish to terminate the merging operation, RIGHT-CLICK and select "**Cancel Command**"

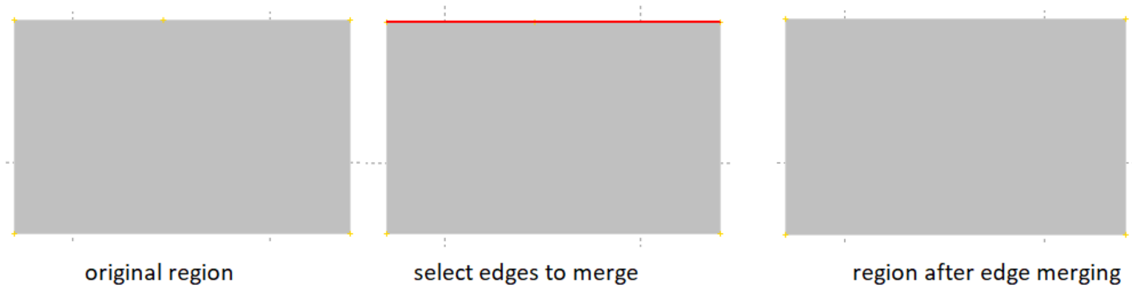


Figure 3-25: Merge two edges of the same region

Additionally, when beams overlap on the selected edges, those beams will also be merged to ensure geometric compatibility.

3.5.5 Partition Region

This operation allows you to partition (or divide) a single region into two distinct regions using an auxiliary line. The auxiliary line can either have its endpoints located on the edges of the original region or traverse the region entirely. It is important to note that each newly created region must consist of at least three edges (forming a triangle) after the partition.

To perform this operation, follow these steps:

1. Click to switch to *"CAD- MODE"* if necessary.
2. Create an auxiliary line by selecting two vertices on the region, or by drawing one that passes through the region.
3. On the menu bar, click "**Modify->Partition One Region by Line"** or click the icon from the toolbar. Ensure the "Select All" icon is activated
4. Select the auxiliary line and the region you wish to partition. Please remember that only one region can be partitioned in a single operation.
15. RIGHT-CLICK and select "**End Selection**" from the popup menu to finalize the operation.

<u>NOTE</u>: Clicking "**Cancel**" will terminate the current partition operation. Make sure to verify that the original region has been partitioned correctly. The auxiliary line will be automatically deleted for clarity.

If you right-click and select "**Cancel Command**," the "**Modify -> Partition One Region by Line**" operation will also be terminated.

Refer to Figure 3-26 for visual guidance on these steps.

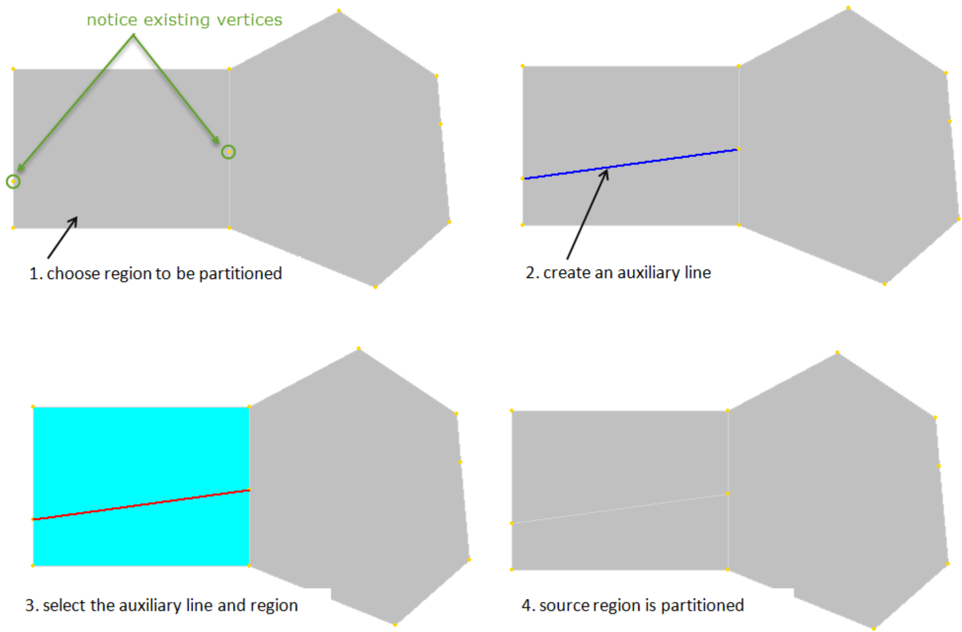


Figure 3-26: Partition one region by an auxiliary line

Beams that overlap the edges of the region will be automatically partitioned to ensure compatibility between geometry entities.

3.5.6 Merge Regions

This operation facilitates the merging of two or more adjacent regions, requiring that the source regions share at least one common edge (with matched dependent lines and vertices – refer to Figure 3-27). After "merge" operation, new regions will not retain the original material assignments. However, boundary conditions and contact interfaces assigned to edges other than the common edge will be transferred to the new regions.

To execute this operation, follow these steps:

1. Click to switch to *"CAD- MODE"* if necessary.
   a. On the menu bar, click "**Modify->Merge Regions"** or click the icon from the toolbar. En sure that the region selection icon , is activated.
2. Select the desired regions; multiple selections are allowed.
3. To complete the selection, RIGHT-CLICK and choose "**End Selection**" from the context men.
   a. <u>NOTE:</u> Clicking *"***Cancel***"* will terminate the current *"merge"* operation.
4. Confirm that the source regions have been merged correctly.

5. If RIGHT-CLICK and select "**Cancel Command**", current 'Merge" operation will be terminated**.**

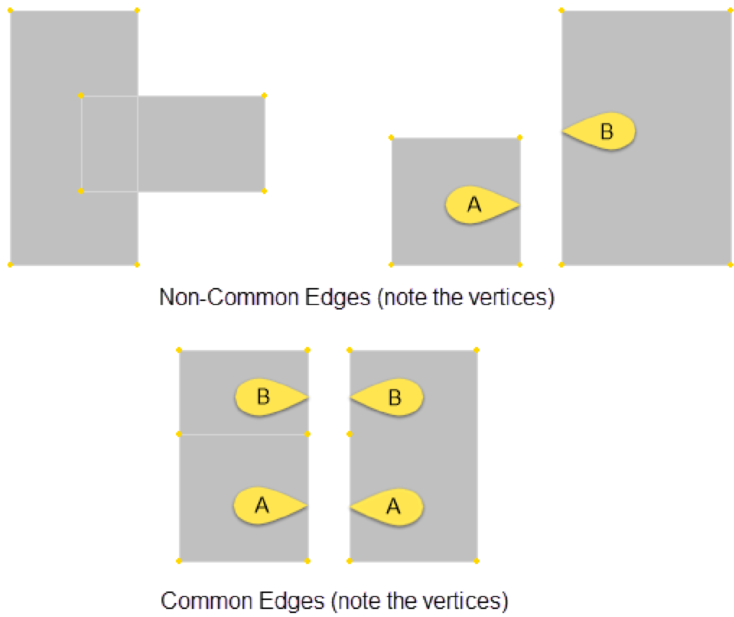


Figure 3-27: Examples of Common Edges for Merge Region

3.5.7 Divide Beam

This tool allows users to divide a beam using an auxiliary line. If the beam intersects a region edge, it is recommended to use the "Partition Region Edge" option to partition both the region edge and the beam at the same time.

To perform this operation, follow these steps:

5. Click to switch to *"*CAD- MODE*"* if necessary.
6. On the menu bar, click **"Modify->Divide Beam Line/Arc",** or click the icon from the toolbar. The line selection icon is automatically activated .
7. Select the desired beam line or beam arc and the auxiliary line. Make sure that selected beam lines/arcs and auxiliary line are intersected.
8. RIGHT-CLICK and select **"End Selection"** from the popup menu to complete the operation.
9. If RIGHT-CLICK and select "**Cancel Command**", current "**Modify->Divide Beam Line/Arc**" command will be terminated.

3.5.8 Merge Beams

This operation allows you to merge two adjacent beam segments. If the beams overlap region edges, it is advisable to use "**Modify->Merge Two Edges**" to simultaneously merge both the region edges and the beams.

Follow these steps to perform the operation:

1. Click to switch to *"CAD- MODE"* if necessary.
2. On the menu bar, click **"Modify->Merge Beam Line/Arc",** or click the icon from the toolbar. The line selection icon is automatically activated .
3. Select the beam lines or beam arcs you want to merge
4. RIGHT-CLICK and select **"End Selection"** from the popup menu to complete the operation.
5. If RIGHT-CLICK and select "**Cancel Command**", current "**Modify->Merge Beam Line/Arc**" command will be terminated.

3.5.9 Disassemble Regions

This operation breaks down one or more regions into a series of independent lines and arcs, effectively reversing the process of creating a general polygon from lines and arcs. This disassembly provides users with enhanced flexibility for modifying geometric entities based on existing analysis models. However, it's important to note that once the region is disassembled, all previously assigned material properties, boundary conditions, and contact interface definitions associated with the disassembled region will be removed.

NOTE: When disassembling two adjacent regions that share a common edge, this edge comprises two dependent line or arc segments, each associated with its respective parent region. Consequently, disassembling one region will result in one dependent line or arc segment (remaining intact) and one independent line or arc segment (from the disassembled region) along the common edge.

To perform the disassembly operation, follow these steps:

1. Click to switch to *"CAD- MODE"* if necessary.
2. On the menu bar, click "**Modify->Disassemble Region"** or click the icon from the toolbar. Ensure that the region selection icon , is activated
3. Select the regions you want to disassemble.
4. RIGHT-CLICK and select **"End Selection"** from the popup menu to complete the operation
   a. NOTE: Clicking *"***Cancel***"* will terminate the current *"disassemble"* operation.

5. Source regions are disassembled into a series of independent lines and arcs.
6. If RIGHT-CLICK and select "**Cancel Command**", current "**Modify->Disassemble Region"** operation will be terminated**.**

3.6 Valid and Invalid Geometric Configurations

Finite element analysis in geotechnical engineering predominantly examines the strength and deformation characteristics of soil materials, the interactions between soil and structures, and the strength of structural components. Figure 3-28 depicts a common scenario in this domain—a retaining wall. In this example, three distinct materials are present: the concrete of the retaining wall and two different types of soil corresponding to each layer. For the sake of simplicity, this section does not address the contact interface (for details, see Section 3.8), and it is assumed that the retaining wall is perfectly bonded to the soil materials.

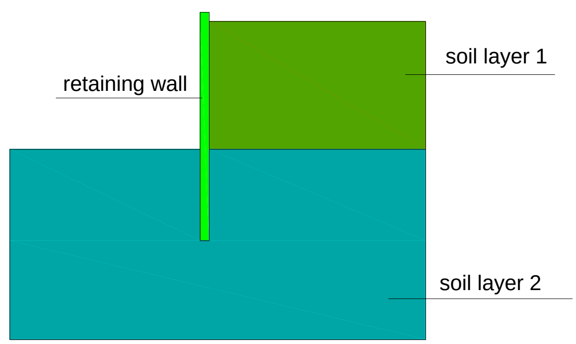


Figure 3-28: A typical problem in geotechnical engineering-retaining wall analysis

In EnFEM, the physical components of the soil and retaining wall are represented by three two-dimensional regions. The first region, labeled “Soil Layer 1,” is a rectangular area, while the regions for the “Retaining Wall” and “Soil Layer 2” are defined as general polygons. As there is no contact interface between these regions and they are seamlessly bonded through the mesh, the model consists of a single geometry set (a term that will be elaborated on in Section 3.8 below).

When developing geometric entities in EnFEM, it is essential to follow certain guidelines:

1. Material Property Assignment: Each region object must be associated with a single material property. If a region is assigned materials more than once, the latest assignment will replace any previous ones. Therefore, for the geometric configuration illustrated in Figure 3-28, which features three distinct material properties, three separate regions must be defined.
2. Shared Edges and Vertices Adjacent regions should share line or arc segments as well as their vertices along common edges. Refer to Figure 3-27, and Figure 3-30 for specifics regarding the retaining wall in this context.

3. Region Compatibility: When creating a new region, EnFEM automatically checks for compatibility with existing regions or beams. If edges overlap but differ in length, an additional vertex will be automatically inserted to ensure compatibility between the regions or between regions and beams.
4. Region Configuration: Regions cannot overlap or be connected by just a single vertex, as demonstrated in Figure 3-29. This limitation is crucial to avoid errors during mesh generation. If regions are separated by gaps, they will be treated as independent, necessitating the application of appropriate boundary conditions to ensure stability.
5. Beam and Region Compatibility: When beams are present in the model, it's crucial for beam vertices to align with the vertices of adjacent region objects at the same location. EnFEM automatically checks and maintains this compatibility between beams and regions.

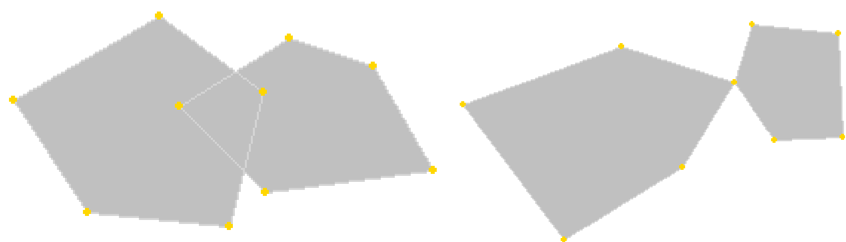


Figure 3-29: Invalid geometric configurations

Taking these fundamental guidelines into account, let's re-evaluate the geometry of the retaining wall. Figure 3-30 illustrates this geometry segmented into its basic geometric entities, with red crosses marking the vertices of each region object. Despite the retaining wall's straightforward rectangular shape, it is essential to construct it as a six-edge polygon to ensure compatibility with the other two region entities. Failure to adhere to this compatibility requirement may lead to errors during mesh generation or yield unpredictable outcomes during analysis.

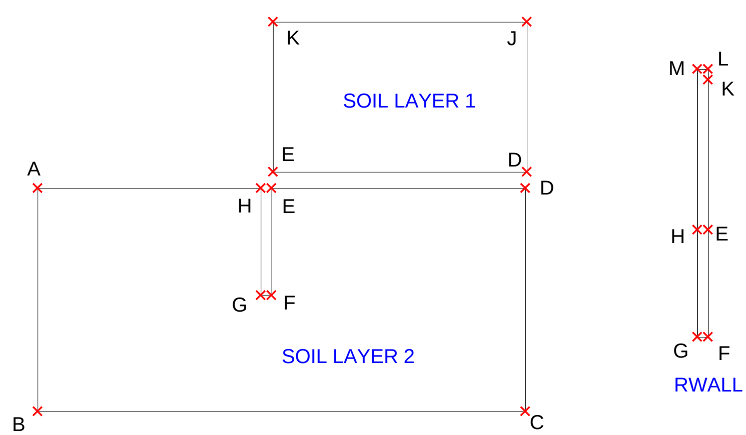


Figure 3-30: Valid regions for retaining wall example

Figure 3-30 illustrates the minimum number of region entities required to simulate the retaining wall problem in EnFEM, given the involvement of three different materials. Experienced users frequently break down the retaining wall into multiple rectangular sub-regions. By dividing a polygon into several quadrilateral regions, it becomes possible to create a structured mesh, which is recognized for its accuracy and efficiency. Additionally, it may be beneficial to model a defined "centerline" along the retaining wall to facilitate the selection of specific results. Details regarding mesh generation will be covered in subsequent chapters of this manual.

Figure 3-31 depicts an excavation wall scenario simulated using a beam and a zero-thickness interface. This example illustrates how the soil-structure interaction (SSI) interface is represented through a zero-thickness approach, with the retaining wall modeled as a beam. Since beams are one-dimensional entities, they inherently lack thickness. The right boundary of the excavated soil coincides with the retaining wall, while the left boundary aligns with the remaining soil, as shown in the first image. The soil component consists of five rectangular regions. It is essential for the beam to maintain vertex compatibility with the soil regions, specifically requiring the explicit creation of vertices B1, B2, and B3. An exception to this is the tip of the retaining wall, for which EnFEM can automatically establish the zero-thickness interface. Additional information regarding model development can be found in the example manual

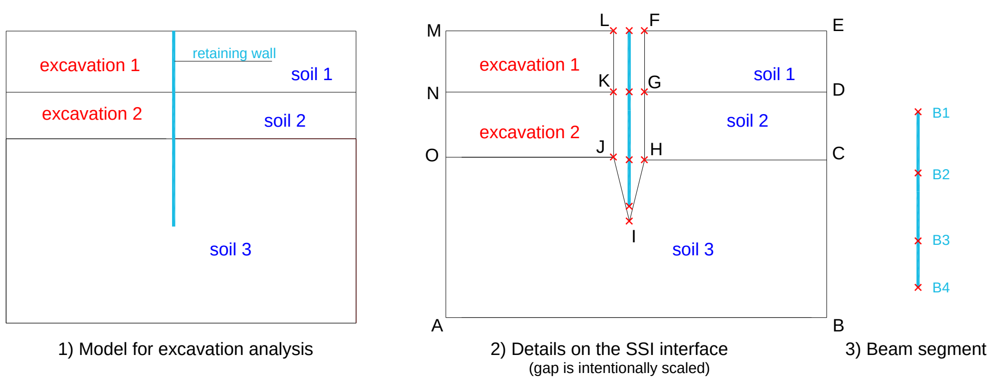


Figure 3-31: Excavation wall example simulated by beam and zero-thickness interface

3.7 Simulation of Voids

Voids (holes) in soil are commonly encountered in geotechnical engineering, particularly when modeling structures like tunnels or culverts. In EnFEM, an indirect method is utilized to create these voids. For instance, if a culvert is made from concrete, its thickness usually warrants the use of solid elements for simulation. As depicted in Figure 3-32, the closed culvert is represented by two ring regions, with the internal void being generated automatically. Typically, a culvert is constructed on stiffer soil, allowing for the possibility of splitting the culvert along this soil interface. The upper and lower soil layers are represented as general polygons. Even if both soil layers are identical, it is essential for users to divide the soil around the culvert using a line to form two distinct polygons that encapsulate the culvert. Contact interfaces are then defined between the culvert and the surrounding soil components. For further information on this modeling technique, refer to Example 4 - Interaction (contact) analysis of buried culvert in the Example Manual.

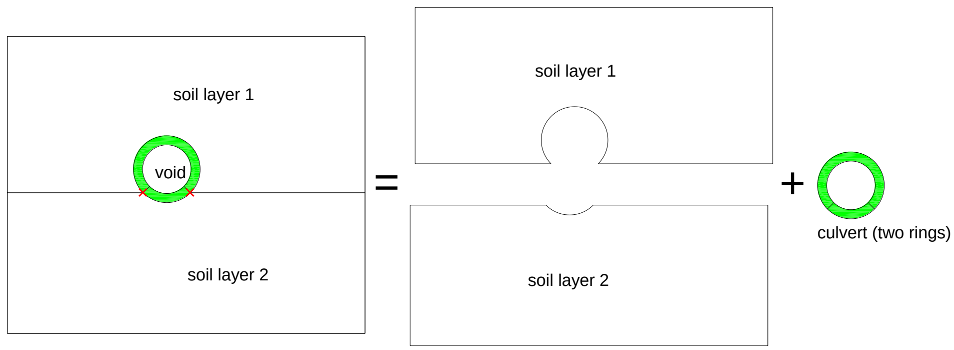


Figure 3-32: Culvert by regions and void simulation

If the culvert is constructed from steel, it will be considerably thinner than the one illustrated in Figure 3-32. Using solid elements for such a thin structure can lead to challenges in generating a quality mesh. Instead, the thin culvert can be effectively modeled using two curved beam segments (as shown in Figure 3-33). In EnFEM, a circular beam must consist of at least two arc segments, hence the culvert will be divided at the soil interface and represented as two arc beam segments, as depicted in Figure 3-33. The void within the culvert is automatically created. The soil layers can be represented with the same polygon shapes as in Figure 3-32. Zero-thickness interfaces are employed between the soil regions and the culvert beams. For further details on this modeling technique, please refer to Example.

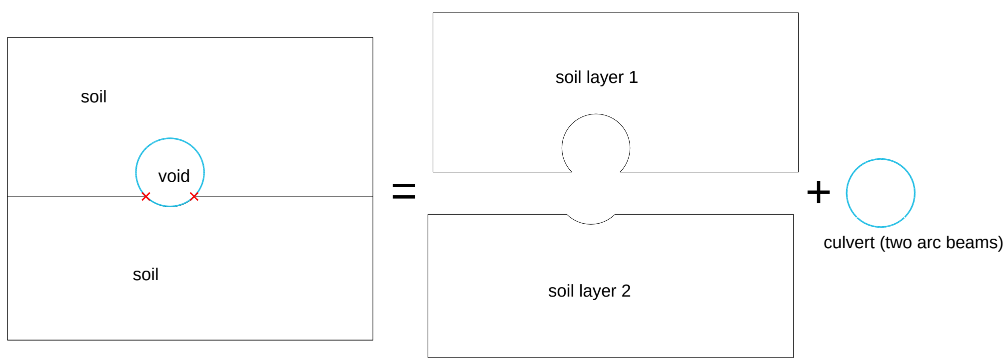


Figure 3-33: Culvert by beams and void simulation

3.8 Geometry-Sets for Contact Analysis

In the previous examples illustrated in Figure 3-28 and Figure 3-32, no interaction (contact) definitions were introduced. Consequently, the common edges between adjacent regions are assumed to be perfectly bonded (via the mesh). The finite element nodes are shared across elements from neighboring regions, even if these regions have different material properties. In such scenarios, there is only one independent geometry set created, which includes the entire combined geometry. It is important to note that this perfectly bonded condition, where nodes between adjacent bodies are shared along the common edges, is the default setting during mesh creation. Only when an interaction boundary condition is applied (as discussed later) can treat adjacent bodies independently.

However, to better mimic real-world behavior and incorporate some interaction (contact) definitions, it is critical to recognize the need for independent geometry sets. To accurately model soil-structure interactions, a 'contact interface' should be established between the structural elements and the soil. By implementing this interaction in the context of the aforementioned examples (Figure 3-28 and Figure 3-32), the models will feature two independent 'geometry-sets': one for the structure and another for the soil. The interaction between these sets is defined by the 'contact' boundary condition.

In EnFEM, a model can include multiple independent configurations if required, and each independent 'geometry-set' can encompass one or more regions. With the introduction of this boundary condition, the finite elements on either side of the contact interface (common edges) will not share finite element nodes. The model can simulate closure or separation in the normal direction, as well as sticking or sliding in the tangential direction along the contact interface, once an interaction boundary condition is activated. More comprehensive information regarding interaction (contact) definitions is available in CHAPTER 8 of this manual, along with detailed examples in the Example Manual.

CHAPTER 4. Control Data and Stages

4.1 Introduction

This chapter presents the definition of control parameters and stage controls, which can be accessed under the “CAD-MODE” in the "Data" drop-down menu.

An overview of materials for solid entities and structural sections will be provided in CHAPTER 5, while boundary conditions will be discussed in CHAPTER 6.

Definitions of interaction (contact constraint method or zero-thickness interface) will be covered in CHAPTER 8.

The sub-sections in this manual are organized to align with typical workflow procedures for developing a model in EnFEM. It is advisable to adhere to this workflow when working on a new model.

4.2 Control Data

Control Data includes essential project information such as the project title, analysis type selection, thickness definition (if applicable), and unit settings. To access this, navigate to “**Data -> Control Data**” in “CAD MODE.” The default Control Data window is illustrated in Figure 4-1.

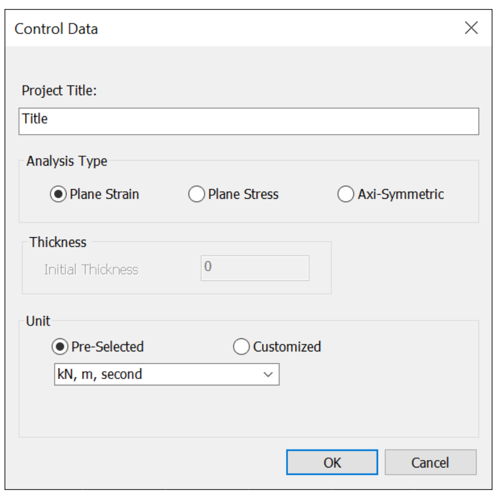


Figure 4-1: Input dialog box of Control Data

This dialog allows users to define the following parameters:

1. Project Title: A description of the project (optional)
2. Analysis Type: Options include plane strain, plane stress, or axisymmetric analysis. If plane stress is chosen, the thickness must be specified
3. Units: Users can select from predefined units or create custom units. Available predefined units include “kN, m,” “N, mm,” “lbf, in,” “kip, in,” and “kip, foot.” For additional unit

types, users can click the “**Customized**” option and choose the appropriate units as shown in Figure 4-2.

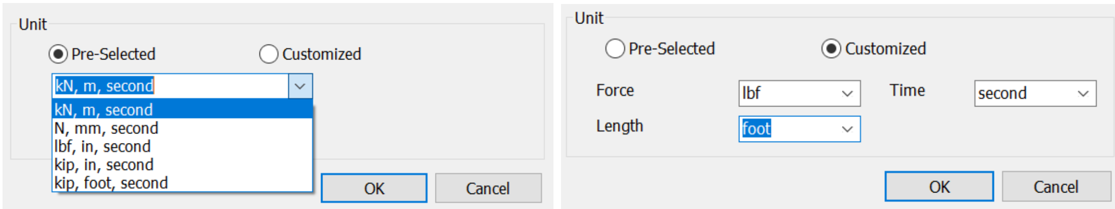


Figure 4-2: Predefined unit and customized unit

<u>NOTE</u>: When the analysis type is configured as "axi-symmetric," the horizontal coordinate is identified as ‘radius, r,’ while the vertical coordinate is referred to as ‘depth, z’. An orange dashed line will be shown on the screen to mark the position of the axi-symmetric axis, as illustrated in Figure 4-3. Furthermore, within “CAD-MODE,” users can click on "**View -> Global Axes**" from the menu bar to toggle the visibility of the global axis and the axi-symmetric axis.

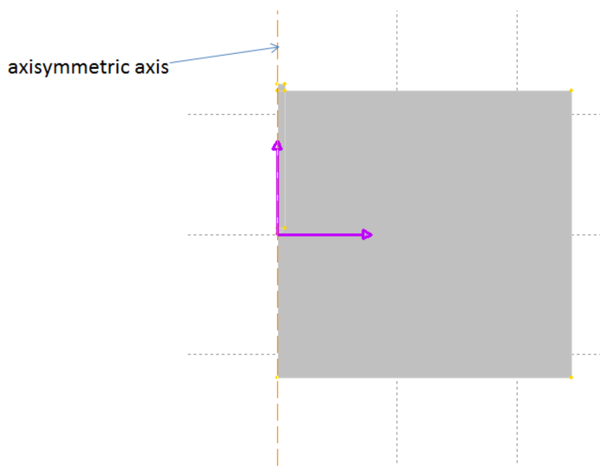


Figure 4-3: Axis layout for axi-symmetric analysis type

4.3 Stage Management

In geotechnical engineering, construction practices are typically segmented into various stages, such as tie-back installation, excavation, post-excavation surcharge, and backfill soil. Each stage may involve distinct loads and boundary conditions. As a result, finite element analysis often requires the simulation of multiple stages. EnFEM is equipped to handle multi-stage simulations effectively. To access this feature, navigate to the menu bar and select "**Data -> Stage Management**" to open the “Stage Group Management” dialog, as illustrated in Figure 4-4.

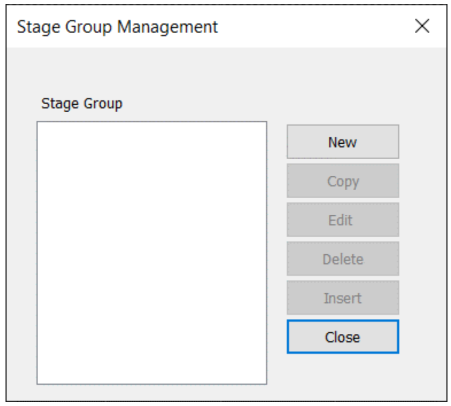


Figure 4-4: Dialog of Stage Group Management

Stages are organized in the Stage Group window according to the sequence in which they will be created. When adding a new stage, it is placed at the bottom of the list. The analysis proceeds in the order presented in the Stage Group window.

To create a new stage definition, click on "**New**." To duplicate an existing stage, select it and click "**Copy**." The new stage will be named by appending "_copy#" to the original stage name, and it will be positioned directly after the source stage, with the "#" representing a default number. To modify an existing stage, select it from the Stage Group list and click "**Edit**" to open the stage definition dialog. If you wish to remove a stage, choose it from the list and click "**Delete**." Additionally, if you want to insert a new stage after a selected one, click "**Insert**.".

4.3.1 Create A New Stage

The user-interface to define a new stage is shown in Figure 4-5. First, the user must input the desired stage name. Next, they select the stage type from the drop-down menu labeled "Type." EnFEM offers four stage types: *Initial Soil Stress, General Static, Excavation, and Stability (Strength Reduction Method).* If "Initial Soil Stress" is chosen, EnFEM will automatically manage increment data. For the other stage types, users can either utilize the default settings or specify a different increment size based on their experience. The initial increment size applies to the first increment of the current stage. For problems that exhibit high nonlinearity, it is advisable to use a smaller size for the initial increment. During computations, EnFEM automatically adjusts the increment size according to convergence difficulty. If a calculation fails to converge, EnFEM will reduce the increment size to improve convergence performance. However, if the increment size drops below the minimum threshold after reduction, EnFEM will terminate the current calculation job. Typically, the default increment settings suffice for most scenarios.

Click **"Save & Exit"** to close the stage definition window. Click **"Cancel"** to cancel current stage definition.

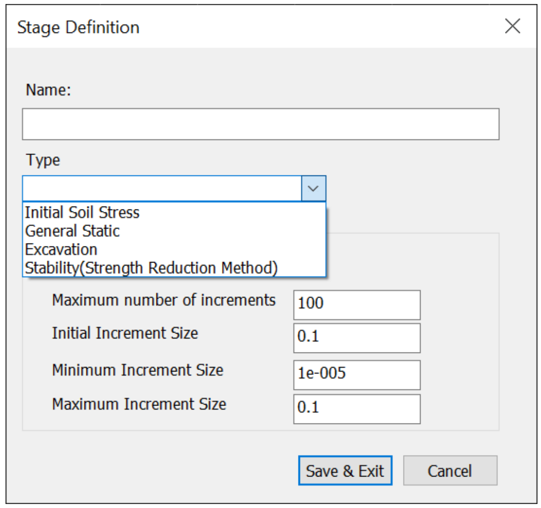


Figure 4-5: Setup of a stage

4.3.1.1 Initial Soil Stress Stage

"Initial soil stress" refers to a specific condition in geotechnical engineering, often serving as the starting point for many analyses. In this state, soil particles experience non-zero stresses that maintain equilibrium with their self-weight while remaining deformation-free. Moreover, certain constitutive models, such as the Modified Cam-Clay model, require correctly defined initial stresses. The initial stress state consists of three components: vertical stress σ_z, and two lateral stresses, σ_x and σ_y, which typically represent the principal soil stresses. EnFEM can estimate the initial stress conditions for normally consolidated or lightly consolidated soils, but the estimation is not applicable for heavily consolidated soils in this version.

There are two scenarios in which defining "Initial Soil Stress" is essential in EnFEM:

1. CASE 1: modeling actual geometric dimensions and working conditions, such as a shallow foundation on soil, a retaining wall embedded in soil, or the stability analysis of a slope. In this case, the "Initial Soil Stress" stage should be the first stage, and the gravitational load must be applied during this stage.
2. CASE 2: simulating a laboratory soil test, where the soil sample is assumed to have uniform initial stresses that equilibrate with the external pressure applied on its surfaces. Importantly, gravitational load should not be applied in this situation, and the "Initial Soil Stress" stage should also be the first stage.

When a stage is classified as "Initial Soil Stress" (as shown in Figure 4-6), the "Initial Stress Option" button becomes active. Clicking this button opens the "Initial Stress Option" dialog (depicted in Figure 4-7. EnFEM offers three methods to estimate initial soil stresses: K_0 method, importing data from an external file, or directly estimating gravitational load.

To create a new initial soil stress stage, users must define the regions for which soil stresses need to be estimated. This involves clicking "**Add**," selecting the appropriate regions, and then right-clicking to choose "**End Selection**." The selected regions will then be added to the "region list." The "**Clear**" button allows users to delete the contents of the region list, while the "**Update**" button refreshes the list after reselecting regions.

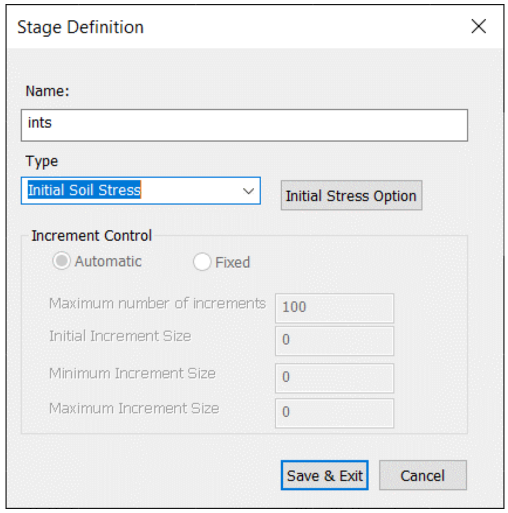


Figure 4-6: Initial soil stress stage definition

Three methods to generate initial soil stresses are explained here. More technical information on these topics can be found in the EnFEM Technical Manual.

4.3.1.1.1 K_0 Method

This method can accurately estimate initial soil stresses when the ground is flat, and the soil layers are horizontally distributed. To input soil layer information and the coefficient of lateral stress, click the "Input Data" button (refer to Figure 4-8). Use the "**Add Line"** option to append a data row to the end of the list. To insert a data row at the cursor's current location, select "**Insert Line**". If you need to remove a row, use the "**Delete Line**" option, which will eliminate the row in which the cursor is currently positioned.

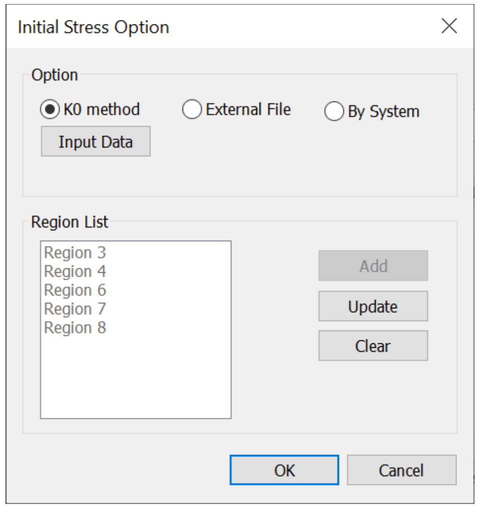


Figure 4-7: Initial soil stress options

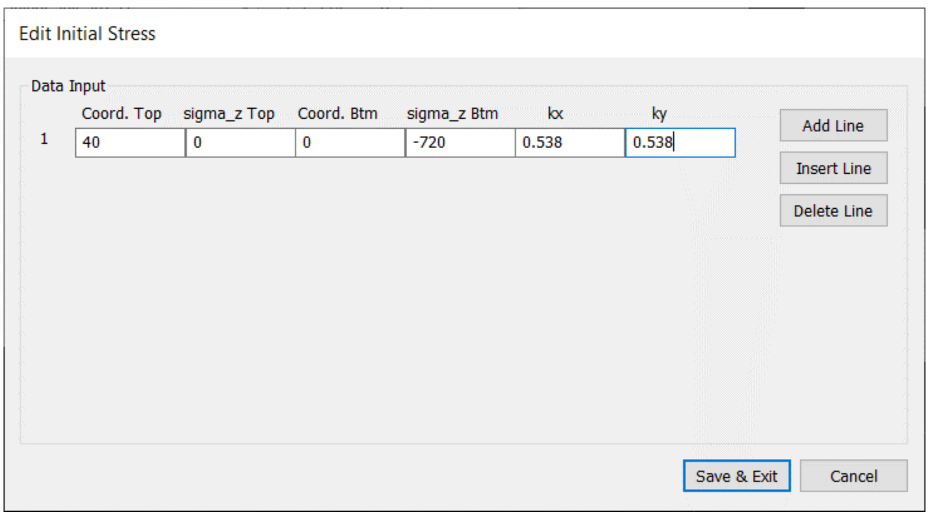


Figure 4-8: Parameters Input for K_0 method

4.3.1.1.2 External File Method

When the ground is uneven or the soil layers are inclined, the K0 method cannot be utilized. In this case, you can select the second option shown in Figure 4-7, which allows you to read initial stresses from an external stress file (refer to Figure 4-9). To create this stress file in EnFEM, you'll typically need to perform a separate "initial stress-only analysis" first. After completing this analysis in the secondary model, you can generate a stress report that serves as the source for the external stress file. Subsequently, a second analysis is conducted, using this file for the "Initial Stress Option." This method is explained in more detail in *Example 11 – Excavation on Inclined Soil*, in the Example Manual.

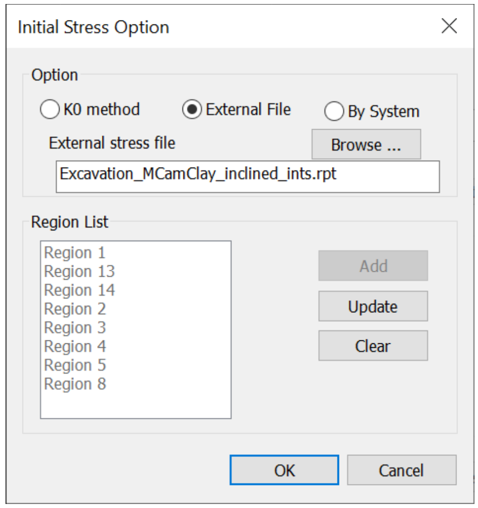


Figure 4-9: Use an external stress file to generate initial soil stress

4.3.1.1.3 'By-System' Method

When the ground is significantly uneven or when analyzing slope stability, the first two methods for establishing initial soil stress may be challenging to implement correctly. EnFEM offers a third option that utilizes actual material models and real contact interfaces to estimate initial stress. However, this approach may encounter convergence issues due to the high shear stresses present in zero-thickness interface elements. It is advisable to use this method only if the first two methods prove unsuccessful. The deformation data generated from the gravity load can either be retained or disregarded for later analysis stages, while stress and strain data are consistently preserved to ensure accurate stress-strain history.

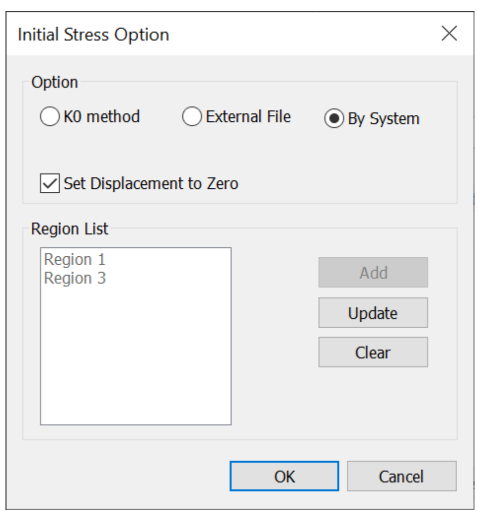


Figure 4-10: Use an external stress file to generate initial soil stress

4.3.1.2 General Static Stage

This option applies to stages that are not categorized as "Initial Soil Stress," "Excavation," or "Slope Stability Analysis." Users will need to enter a "stage name" and choose from two increment control options: "Automatic" or "Fixed." The "Automatic" option, illustrated in Figure 4-11, is generally the preferred choice as it is suitable for most analyses. The default increment settings usually suffice. The "Fixed" option, on the other hand, requires only the maximum increment value to be specified. It is strongly recommended to use the "Automatic" option, as EnFEM can dynamically adjust the increment size to enhance convergence performance. In contrast, the "Fixed" option adopts uniform increment size without adjustments. If convergence issues arise during calculations, computation will be aborted.

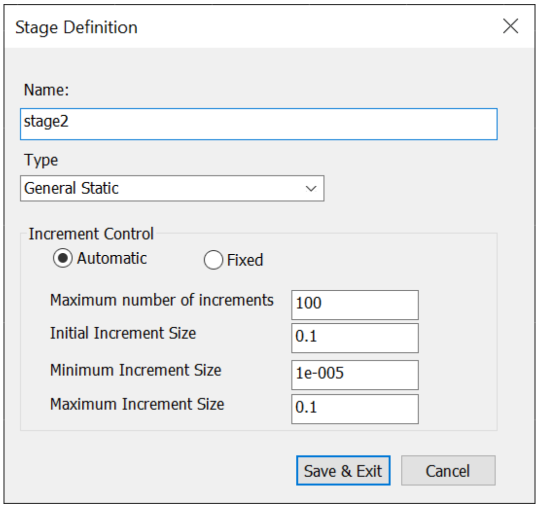


Figure 4-11: Automatic control of General Static Stage

4.3.1.3 Excavation Stage

This stage type is specifically designated for soil excavations. The control of increment size is identical to that used in the "General Static" type. During the initial increment of this stage, the finite elements corresponding to the excavated soil will be deactivated. If any contact or zero-thickness interfaces are established on the excavated finite elements, these soil-structure interfaces will also be deactivated automatically.

Once the excavation stage is defined in the model, it is essential to specify the appropriate region set for the excavated soil body. Users can do this by navigating to "**Tools -> Set Management**" in the "CAD MODE”. Additionally, it is necessary to define the excavated soil sets using the "**Data -> Deactivate/Activate**" command.

4.3.1.4 Slope Stability Analysis using SRM (Strength Reduction Method)

This stage is designed for assessing the safety factor of a slope using the Strength Reduction Method (SRM). It is important to note that the soil constitutive models utilized must be either elastic or of the Mohr-Coulomb type. The default safety factor range is set between 0.5 and 2. Users can click on "**Define SRM**" to modify this default range (see Figure 4-12).

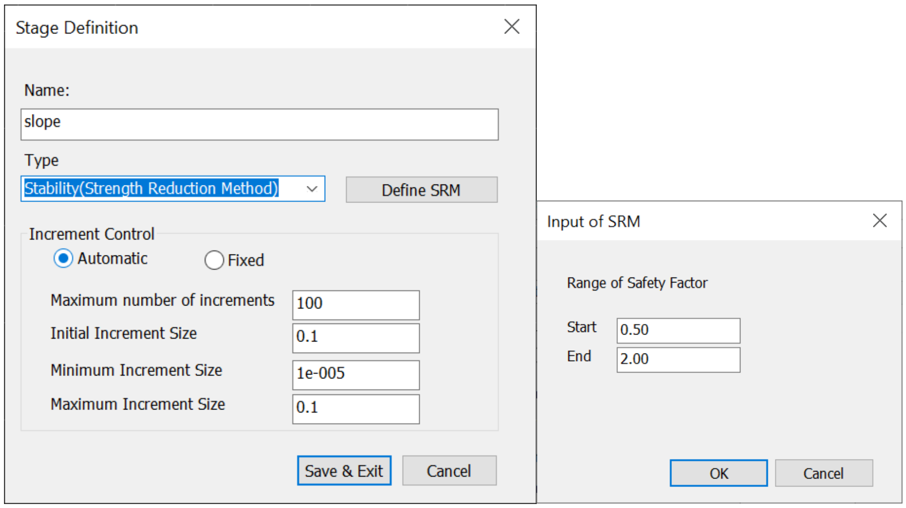


Figure 4-12: Stability stage using strength reduction method

4.3.1.5 Control of Load Increments

EnFEM is employed for nonlinear analyses in the field of geotechnical engineering. Typically, loads are applied incrementally, and the convergence for each increment is achieved through iterative processes. EnFEM offers two options for increment control: automatic and fixed. The "Automatic" option allows the software to determine the size of each increment (Figure 4-11). Users can set general limits for the maximum and minimum increment sizes, as well as specify the maximum number of increments. In contrast, the "Fixed" option maintains a constant increment size, as illustrated in Figure 4-13. Here, all increments have equal size. If convergence is not achieved during a particular increment, computation will be aborted. For the fixed increment control, the only required input is the total number of increments. The analysis utilizes uniform load increments, where each increment is calculated as 1 divided by the total number of increments (resulting in an increment of 0.01 in this example). If the computation fails to achieve convergence within the maximum allowed iterations, or if it experiences bifurcation during the current increment, the process will be automatically terminated.

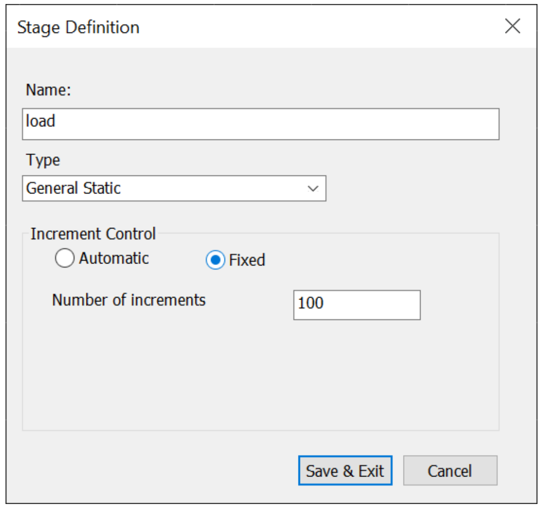


Figure 4-13: Fixed load increment control

In EnFEM, the total load factor for each stage is set to *"1"*. Each loading increment will apply a fraction of the total loads which are applied in the current stage. Loads applied in the previous stages will remain until the end of the calculation. Assume a model has three stages named "gravity", "surcharge", and "backfill". The range of load increments for each stage will be assigned as follows: 0 → 1 for "gravity", 1 → 2 for *"surcharge"*, and 2 → 3 for "backfill". When the increment is equal to 1, this indicates the end of the first stage and the start of the second stage.

The following explains the parameters for the *"Automatic"* option:

4.3.1.5.1 Maximum number of increments:

This parameter sets the maximum allowable load increments for one stage. The default value is 100. When the computation has difficulty to converge within a given load increment, the load increment is reduced and the solution will be attempted again. This may result in the total number of load increments greater than the default value of 100, as the total increment number is influenced by the size of a load increment. Users may define a new increment limit larger than 100 if necessary.

4.3.1.5.2 Initial increment size:

The initial increment size is the first increment size used by the current stage. A value of 5% to 10% is appropriate for most calculations. Occasionally a smaller initial size is necessary when the problem is highly nonlinear.

4.3.1.5.3 Maximum increment size and minimum increment size:

EnFEM uses a smart algorithm to control the increment size automatically. If convergence is easily reached, the size for the next increment may be increased. However, the increased increment size will not exceed the maximum size defined here. If the computation cannot reach convergence, or if the result bifurcates, the program will redo the computation using continuously smaller increment sizes until the minimum increment size is reached, or convergences occurs. If convergence still cannot be attained once reaching the minimum increment size, the computation will be terminated automatically.

4.3.2 Copy Stage

This operation is to duplicate a stage when the new stage has similar parameters as an existing one. Select a stage from the *stage group list* and click "**Copy**" to complete the duplicate procedure as shown in Figure 4-14. The default name of the duplicated stage is the source stage name plus "_copy#" where # represents a number. Users may change the default name by "**Edit**" operation.

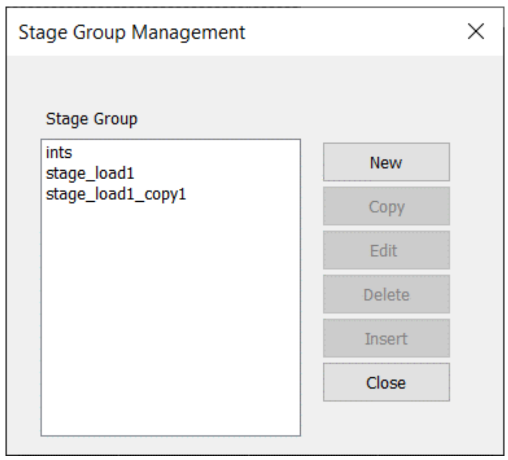


Figure 4-14: Copy a stage

Since one model should include only one "initial soil stress" stage, users should avoid to duplicate an "initial soil stress" which will trigger a model error.

4.3.3 Edit Stage

This operation allows the user to modify an existing stage definition. Select a stage from the "Stage Group" window (Figure 4-4) and click **"Edit"** to open the "Stage Definition*" dialog*. Users can then modify the stage as desired. Note, that if stage 'type' is changed, other parameters (boundary conditions) may also be affected. Click **"Save &Exit"** to save changes to the model and close the "Stage Definition*"* dialog. Clicking **"Cancel"** will exit without saving changes.

4.3.4 Delete Stage

This operation allows the user to remove existing stage definitions from the model. Select an existing stage from the "Stage Group" window (Figure 4-4), and click **"Delete"** to remove it from the model. There is no "undo" option to recover deleted stages, so caution should be used.

NOTE: If existing boundary conditions are related with a deleted stage, users should update the stage data for these boundary conditions.

4.3.5 Insert Stage

This operation is to insert a new stage in front of a selected stage. Select an existing stage from the “Stage Group” window (Figure 4-4), click “**Insert**” to pop up the new stage dialog. After finishing parameters input, click **"Save &Exit"** to exit to the stage management window. The inserted stage will locate in front of the source stage.

If a model includes the “**Initial Soil Stress**” stage, users should avoid inserting any stage in front of it, because the “**Initial Soil Stress”** must be the first stage in the stage sequence.

CHAPTER 5. Materials for Soils & Structures

5.1 Soil Materials

EnFEM provides seven material models for solid finite elements: *linear elasticity*, *Tresca plasticity*, *von Mises plasticity*, *Mohr-Coulomb plasticity*, *Druker-Prager plasticity*, *Duncan-Chang hyperbolic elasticity*, and *Modified Cam-Clay*. For more technical details on each material model, users may refer to the EnFEM Technical Manual.

From the menu bar, click **“Data->Material Properties”** to open the “Material Group Management” window, where users can create, edit, or delete material property definitions (Figure 5-1).

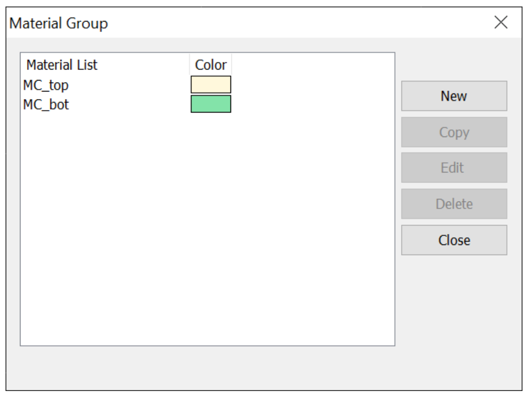


Figure 5-1: Material Group Management dialog

Clicking **"New"** will open the “Material Property” window (Figure 5-2), and allow users to create a new material definition. Selecting a material from the “Material group” list and then clicking **"Edit"**, will open the “Material Property” window, and allow users to edit an existing material. Selecting a material from the “Material group” list and then clicking **"Delete"**, will remove it from the model.

NOTE: Input parameters will change based on the selected material. Additionally, users should define a unique name for each material definition in a model. Once the material parameters are defined in the “Material Property” window, the user should select **“Save & Exit”** to store the new material property to the model.

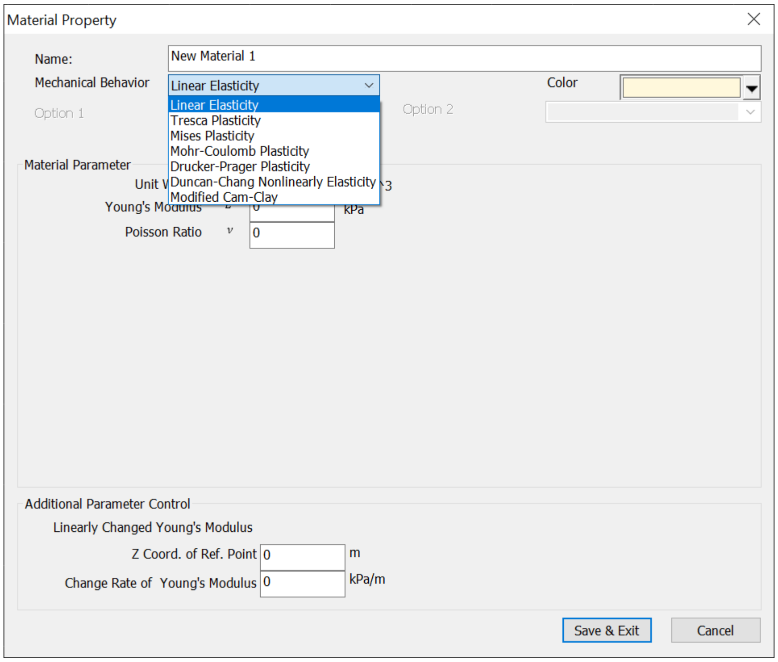


Figure 5-2: Material property dialog for linear elasticity model

5.1.1 Linear Elasticity

Material parameters for the linear elasticity model include unit weight, Young's Modulus, and Poisson's Ratio as shown in Figure 5-2. Young's Modulus and Poisson's Ratio should be defined appropriately. If gravity load is not considered, the unit weight can be ignored.

The linear elasticity material is able to simulate linearly changed Young's modulus with depth. Users need to define the reference z coordinate and change rate of Young's.

The behavior of soil materials is rarely close to that of linear materials except when the load is very small. However, structural elements such as retaining walls, culverts, or piles are typically made up of concrete or steel, and exhibit linear elastic behavior to a certain degree. In most cases these structural elements can be modeled using the linear elastic material.

5.1.2 Tresca Plasticity

Tresca plasticity assumes that yielding occurs when the maximum shear stress reaches a critical value. This yielding function is defined in the equation below:

$$\Phi = \tau_y - \frac{1}{2}(\sigma_{max} - \sigma_{min}) \tag{5.1-1}$$

Where σ_{max} and σ_{min} are the maximum and minimum principal stresses.

Figure5-3 shows the input parameters required for the Tresca plasticity model. Young's Modulus, Poisson's Ratio, and yielding stress should be defined appropriately. If gravity loading is not considered, the unit weight can be ignored.

The Tresca yielding criterion is pressure insensitive which means that hydrostatic pressure does not change the yielding stress. Therefore, Tresca plasticity is usually used for studying behavior of metal materials. It is seldom used to simulate soil behavior in geotechnical engineering.

The Young's modulus and yielding stress can be defined as being linearly changed with depth as shown in Figure5-3.

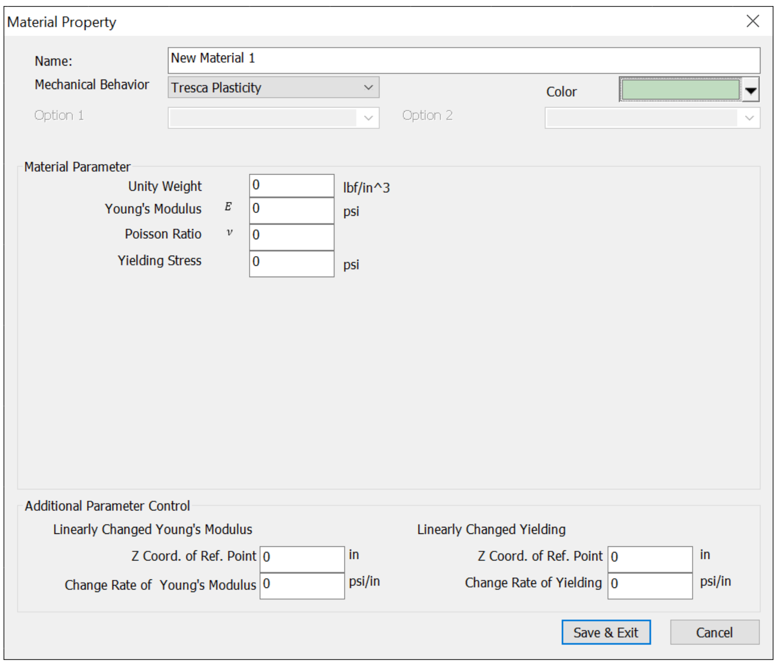


Figure5-3: Material property dialog for Tresca plasticity model

5.1.3 von Mises Plasticity

The von Mises plasticity model assumes that yielding occurs when the equivalent tensile stress, or equivalent von Mises stress,σ_v, reaches the yield stress,σ_y. This yielding function is defined in the equation below:

$$\Phi = \sigma_y - \sqrt{\frac{3}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

(5.1-2)

Where, σ_1, σ_2 and σ_3 are principal stresses. The material parameters are displayed in Figure 5-4.

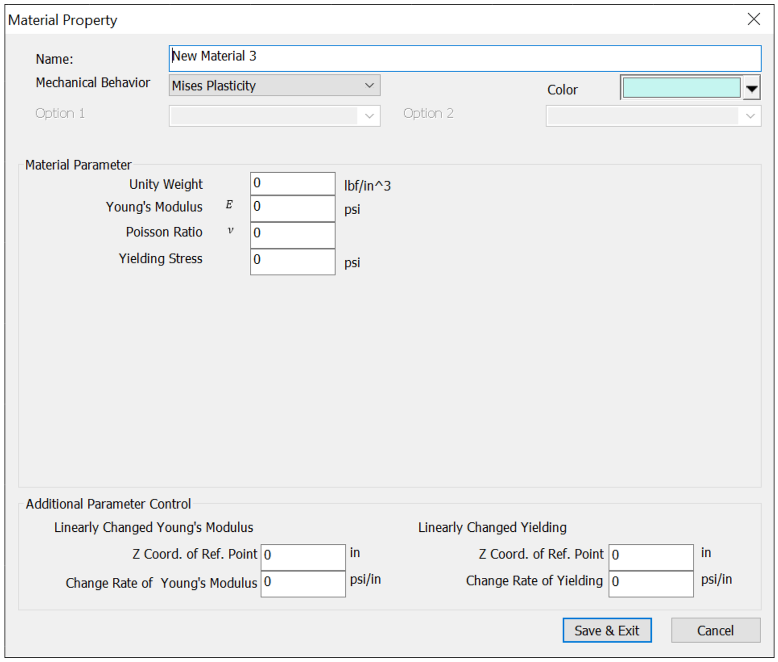


Figure 5-4: Material property dialog for von Mises plasticity model

Young's Modulus, Poisson's Ratio, and yielding stress should be defined appropriately. If gravity loading is not considered, the unit weight can be zero.

Similar to the Tresca plasticity model, the von Mises yielding criterion is also pressure insensitive. Consequently, it is recommended that this plasticity model should only be used for metal materials. Occasionally, the behavior of undrained saturated clay can be properly simulated by the von Mises plasticity model.

The Young's modulus and yielding stress can be defined as linearly changed with depth as shown in Figure 5-4.

5.1.4 Mohr-Coulomb Plasticity

The Mohr-Coulomb plasticity model assumes that yielding is the result of frictional sliding between material particles. The yielding criterion of Mohr-Coulomb plasticity is as follows:

$$\Phi = (\sigma_{max} - \sigma_{min}) + (\sigma_{max} + \sigma_{min}) \sin \phi - 2c \cos \phi$$

(5.1-3)

Where, σ_{max} and σ_{min} are the maximum and minimum principal stresses, ϕ is the friction angle and c is the cohesive strength.

Material parameters for the Mohr-Coulomb model are shown in Figure 5-5.

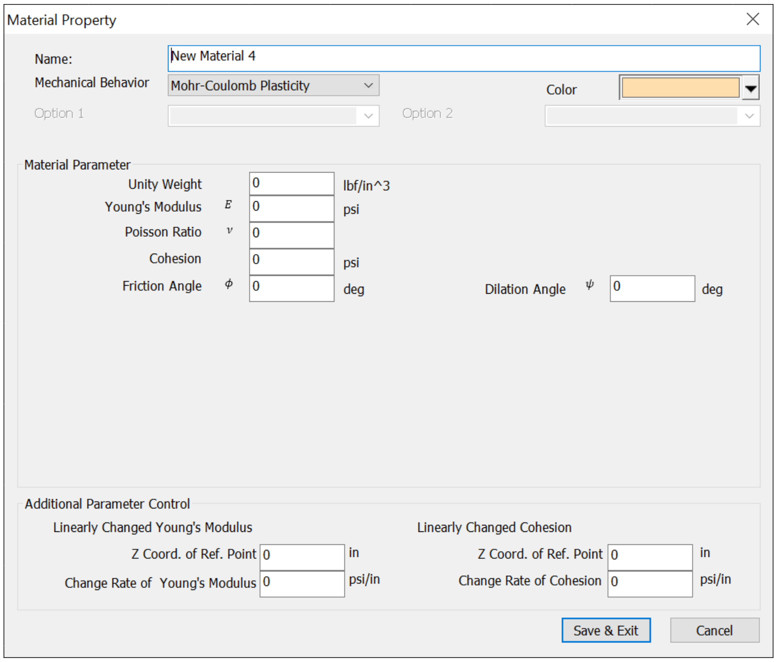


Figure 5-5: Material property dialog for Mohr-Coulomb plasticity model

Young's modulus, Poisson's Ratio, cohesion, friction angle, and dilation angle are required for this model. Both angles have units of degrees. If gravity loading is not considered, the unit weight can be zero.

The dilation angle can be equal to or different from the friction angle. If both angles are equal, an associated flow rule is applied. Otherwise, a non-associated flow rule will be used.

Since the Mohr-Coulomb model is pressure sensitive and parameters are easy to obtain from the laboratory tests, this model is suitable to soil materials and is widely used in geotechnical engineering.

The Young's modulus and cohesion stress can be defined as being linearly changed with depth as shown in Figure 5-5.

5.1.5 Drucker-Prager Plasticity

The Drucker-Prager plasticity model is proposed to smooth the flow rule and can be regarded as a smoothed approximation of the Mohr-Coulomb model. The yield criterion is defined as:

$$\Phi = \alpha I_1 + \sqrt{J_2} - k \tag{5.1-4}$$

where I_1 is the first invariant of the stress tensor, andJ_2 is the second invariant of the deviatoric stress tensor. Parameters α and k are determined from criterion matching with the Mohr-Coulomb plasticity model.

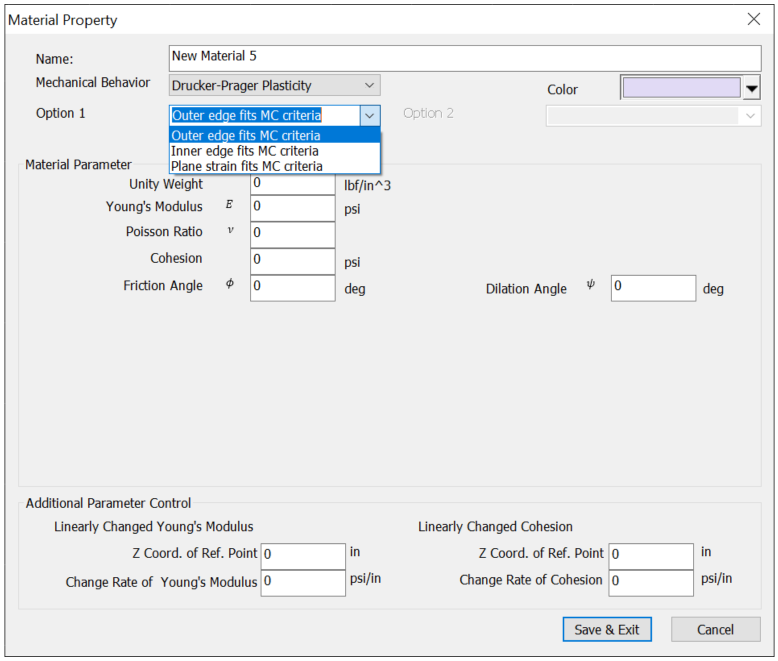


Figure 5-6: Material property dialog for Druker-Prager plasticity

Parameters for the Druker-Prager plasticity model are shown in Figure 5-6. These parameters are similar with those used by the Mohr-Coulomb plasticity model (Young's modulus, Poisson's Ratio, cohesion, friction angle, and dilation angle). The Druker-Prager plasticity model should match the Mohr-Coulomb model criteria based on how the following parameters α and k are determined:

- Match the outer apex of the yielding surface on the π plane of the Mohr-Coulomb criteria

$$\alpha = \frac{2\sin\phi}{\sqrt{3}(3-\sin\phi)},$$

(5.1-5)

$$k = \frac{6c\cos\phi}{\sqrt{3}(3-\sin\phi)}.$$

(5.1-6)

- Match the inner apex of the yielding surface of the Mohr-Coulomb criteria:

$$\alpha = \frac{2\sin\phi}{\sqrt{3}(3+\sin\phi)},$$

(5.1-7)

$$k = \frac{6c\cos\phi}{\sqrt{3}(3+\sin\phi)}.$$

(5.1-8)

- Plane strain matching:

$$\alpha = \frac{\tan\phi}{\sqrt{9+12\tan^2\phi}},$$

(5.1-9)

$$k = \frac{3c}{\sqrt{9+12\tan^2\phi}}.$$

(5.1-10)

Where,ϕ is the friction angle and c is the cohesive strength.

5.1.6 Duncan-Chang Hyperbolic Elasticity

This model assumes that the stress-strain relationship is nonlinear. This nonlinear curve can be accurately approximated by a hyperbola. Figure 5-7shows the parameter input window for the Duncan-Chang hyperbolic elastic model. Users have an option to choose "E-υ model" (control Poisson ratio) or "E-B model" (control bulk modulus). The tangent modulus E_t is calculated from following equation:

$$E_t = \left[1 - R_f S\right]^2 K P_a \left(\frac{\sigma_3}{P_a}\right)^n$$

(5.1-11)

where S is the stress level given by:

$$S = \frac{(1-\sin\phi)(\sigma_1-\sigma_3)}{2c\cos\phi + 2\sigma_3\sin\phi}.$$

(5.1-12)

And,

K = modulus number
n = modulus exponent number
R_f = failure ratio
c = cohesive strength

ϕ = friction angle

σ_1, σ_3 = major and minor principal stresses

P_a = atmosphere pressure

When defining parameters for the Duncan-Chang model, $\sigma_1 \text{and} \sigma_3$ should be compressive and input as positive numbers. If nonlinearity of the friction angle is considered, it can be determined from the following equation:

$$\boldsymbol{\phi = \phi_0 - d\phi \cdot \log\left(\frac{\sigma_1}{P_a}\right)}$$

(5.1-13)

The failure ratio is given by the following relationship:

$$\boldsymbol{R_f = \frac{(\sigma_1 - \sigma_3)_f}{(\sigma_1 - \sigma_3)_u}}$$

(5.1-14)

Where,

$(\sigma_1 - \sigma_3)_f$ = stress difference at failure

$(\sigma_1 - \sigma_3)_u$ = asymptotic value of stress difference.

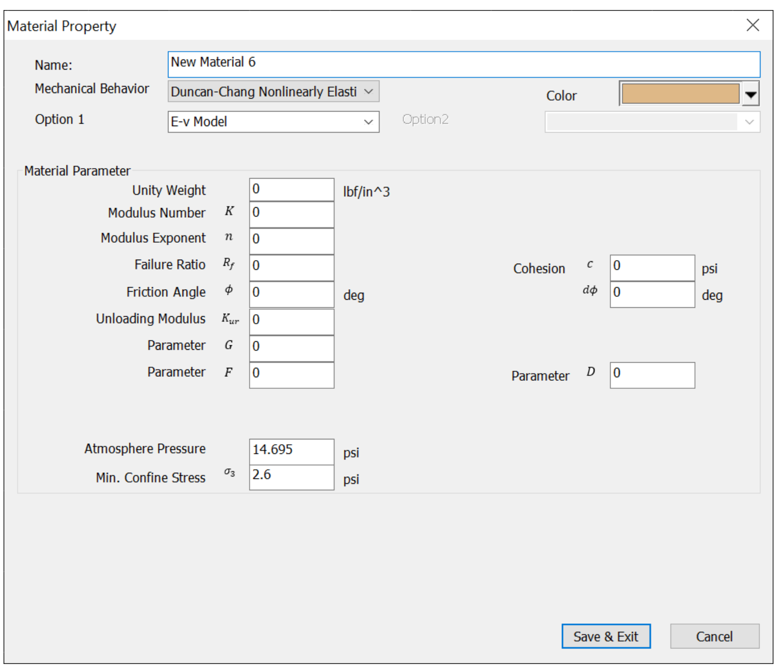


Figure 5-7: Material property dialog for Duncan-Chang hyperbolic elasticity model

In the case of unloading, the tangent modulus is replaced by the unloading modulus:

$$E_{ur} = K_{ur} P_a \left(\frac{\sigma_3}{P_a}\right)^n$$

(5.1-15)

If *"E-υ model"* is selected, the tangent Poisson ratio is determined from the following:

$$\upsilon = \frac{G - F\lg\left(\frac{\sigma_3}{P_a}\right)}{1 - \frac{D(\sigma_1 - \sigma_3)}{KP_a\left(\frac{\sigma_3}{P_a}\right)^n (1 - R_f S)}}$$

(5.1-16)

Otherwise, if *"E-B model"* is selected, tangent bulk modulus is computed as:

$$K_t = K_b P_a \left(\frac{\sigma_3}{P_a}\right)^m$$

(5.1-17)

Where, K_b is the modulus number for the bulk modulus and m is the modulus exponent of K_b (used to define the initial modulus).

The parameter σ_3 represents the minor pre-consolidation stress in the stress history. This parameter must be bigger than 0.0 to start computation.

5.1.7 Modified Cam-Clay

The modified cam-clay model is a plastic strain-hardening model based on critical state theory. The mean pressure has a logarithmic relationship with the void ratio. Figure 5-8 shows parameters required by the modified cam-clay plasticity model in EnFEM. This model allows users to select "Constant Poisson Ratio υ" or "Constant Shear Modulus G" from *Option 1*, and to select "Drained" or "Undrained" from *Option 2*. Other parameters are defined as follows:

1. Unit Weight, γ
2. Lambda, λ = slope of normal consolidation line
3. Kappa, κ = slope of swelling line
4. CSL Slope, M= slope of critical state line (p-q line)
5. Specific Vol, N = specific volume at unit consolidation stress.
6. ν= Poisson ratio
7. OCR =over consolidation ratio.
    a. NOTE: Normally consolidated or lightly over-consolidated soil is supported in EnFEM.

In Figure 5-8, additional options are shown for the initial soil stress. The user may use the "Use Default" setting or set the two extra parameters required: Initial Young's Modulus and Initial Poisson's Ratio. These two parameters are only used for setting the initial soil stress. Based on the assumed mean pressure,p, and the over-consolidation ratio, the Initial Young's Modulus can be estimated with the following steps:

1. Calculate the specific volume corresponding with the assumed mean pressure p

$$\Gamma = N - \lambda \log(p_c) - \kappa \log\left(\frac{p}{p_c}\right)$$

(5.1-18)

2. Calculate the bulk modulus

$$K = \frac{\Gamma}{\kappa} p$$

(5.1-19)

3. Calculate the Young's modulus

$$E = 3K(1 - 2\nu)$$

(5.1-20)

The Initial Poisson's Ratio has another function to control lateral stresses for normally consolidated or lightly over-consolidated soil. The lateral stress factor K_0 has the following relationship with the initial Poisson's ratio:

$$K_0 = \frac{\nu}{1 - \nu}$$

(5.1-21)

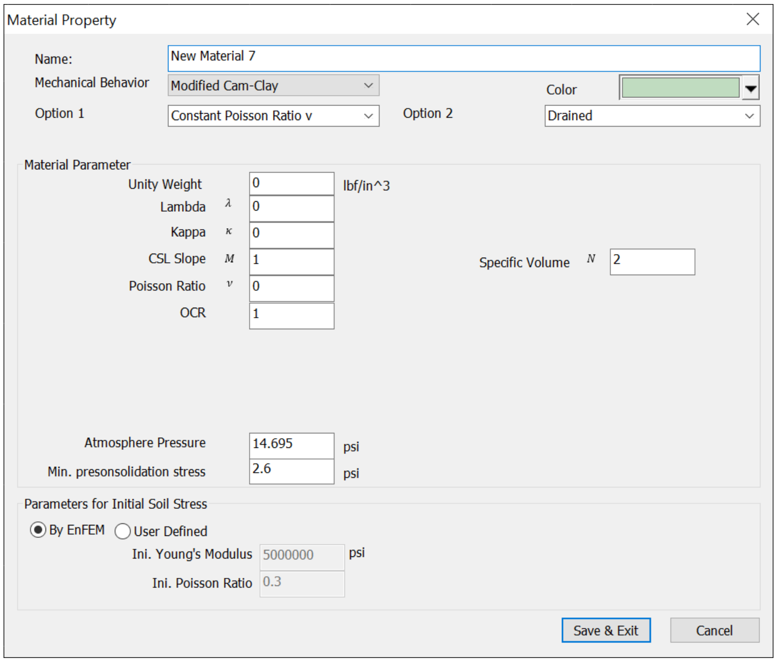


Figure 5-8: Material property dialog for modified cam-clay plasticity model

5.2 Assignment of Materials

After material properties have been defined, the next step is to associate these materials with regions. From the menu bar, click **"Data->Material Assignment"** to open the "Material Assignment" dialog as shown in Figure 5-9.

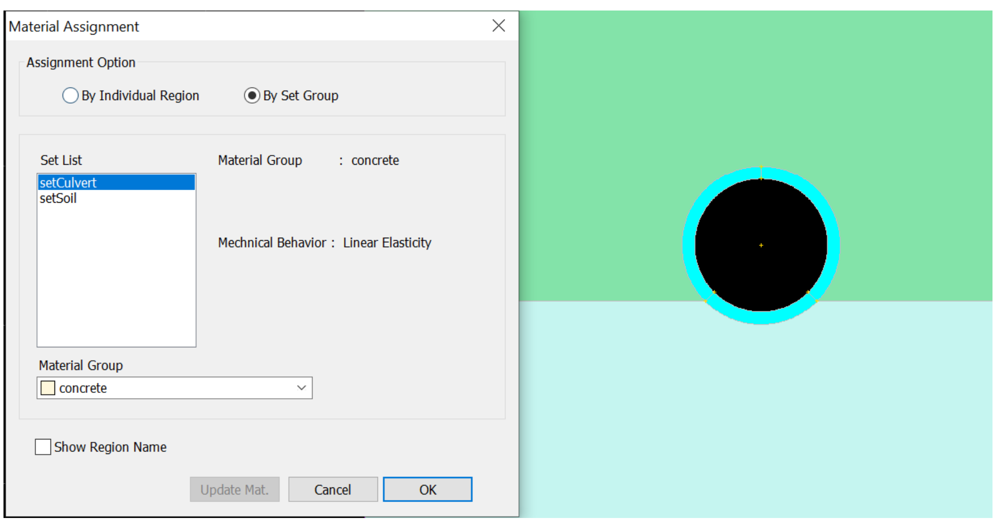


Figure 5-9: Window to assign material property to region objects

Two options are available to assign materials to region entities: "By Individual Region" and "By Set". Note that if using the "By Set" option, a region selection-set must be previously defined. More information on defining selection-sets can be found CHAPTER 7, of this manual.

5.2.1 Assignment as "By Individual Region"

If "By individual Region" is selected, a list of all available regions within the model are shown in the "Region List" window. The regions are listed with a numerical identity ("Region#"). The following procedures outline this assignment process:

1. Select the desired region from the "Region List"
    a. Once selected the region will be highlighted in Cyan color
2. Select the material from the "Material Group" drop down list.
3. Click **"Assign".**
    a. After assignment, the selected regions will be highlighted in purple color.
    b. Note that information about the current material is displayed next to the "Region List".

5.2.2 Assignment as "By Set"

If "By Set" is selected, a list of all available region-sets within the model are shown in the "Set List" window. Note that a region selection-set must be previously defined. The following procedures outline this assignment process:

1. Select the desired region from the "Set List"
    a. Once selected the whole set will be highlighted in Cyan color
2. Select a previously defined material from the "Material Group" drop down list.
3. Click **"Assign".**
    a. Once assigned the whole set will be highlighted in purple color.
    b. Note that information of current material is displayed next to the "Set List".

5.3 Structural Sections

In EnFEM, structural sections are essential for structure elements such as beams, reinforcements, and tie-backs. The software offers the following section definitions: BEAM, STRUT, and GEOGRID. Both beam and strut sections in EnFEM are assumed to be linear elastic, while geogrids can exhibit either elastic or plastic behavior. To manage structural sections, users can navigate to the menu bar and select "**Data -> Structural Sections**" to access the "Structural Section Group Management" dialog. Here, users can create, edit, or delete structural section definitions (see Figure 5-10).

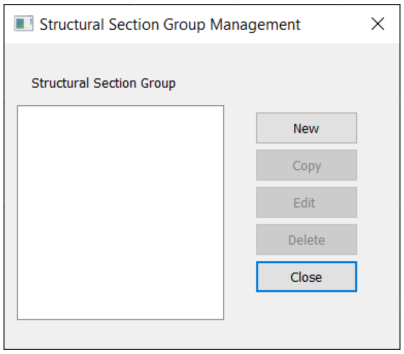


Figure 5-10: Structural Section Group Management dialog

Clicking "**New**" will bring up the “Structural Section” dialog, enabling users to create a new structural section definition. To duplicate a section definition, select a section and click “**Copy**.” If you want to edit an existing definition, select a section from the “Structural Section Group” and click "**Edit**" to open the “Structural Section” dialog. To remove a section from the model, simply select the desired section and click "**Delete**." It is important to ensure that each structural section has a unique name. Once you have defined the section parameters, click “**Save & Exit**” to store the new parameters in the model database and return to the section group management window (see Figure 5-10).

5.3.1 Beam section

In the structural section definition dialog (Figure 5-11), click the dropping list box of "Section Type" and select the type "BEAM" to switch to the beam section property. Explanations of each parameter are listed here:

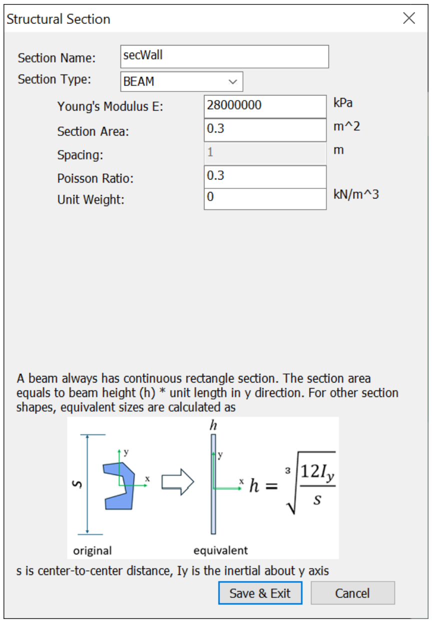


Figure 5-11: Structural section dialog for a beam

1. Beam Young's modulus E
2. Section area: It can be the real section area or equivalent area
3. spacing: This parameter is unchangeable. It is set to 1 length unit. Beams are always assumed to be continuous in the y direction.
4. Poisson Ratio: Poisson ratio of beam material
5. Unit Weight: Weight per unit length. This parameter is optional.

If unit weight is zero, the weight of beam is not considered in the analysis.

After finishing all parameters, click "**Save & Exit**" to save data to the section library and exit to the section group management window.

5.3.2 Strut section

The behavior of a strut is similar as a truss. Only axial deformation and axial forces are available from a strut. In the structural section definition dialog (Figure 5-12), click the combo box of "Section Type" and select the type "**STRUT**" to switch to the strut section property. A strut section has two parameters:

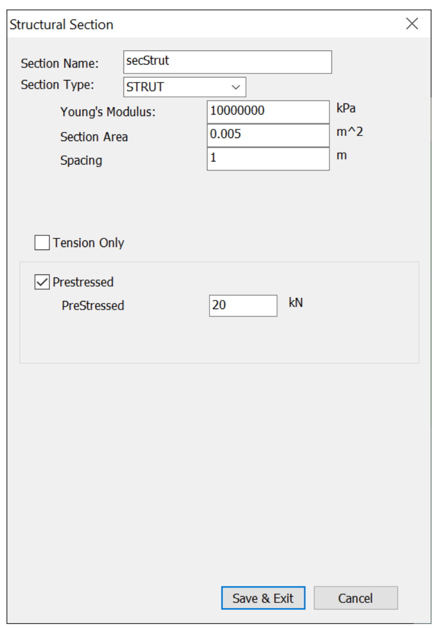


Figure 5-12: Structural section dialog for a strut

1. Young's modulus E
2. Section area: real section area or equivalent section area
3. Spacing: interval space between adjacent struts
4. Option for tension only
5. Prestressed force of "*single*" strut

If the spacing parameter is not equal to *"1"*, equivalent section area will be calculated by EnFEM automatically.

After finishing all parameters, click "**Save & Exit**" to save data to the section library and exit to the section group management window.

5.3.3 Geogrid section

Geogrid is usually to be used to simulate tie-back anchor or soil reinforcements. The behavior of a geogrid is similar as a truss. Yielding behavior is available. In the structural section definition dialog (Figure 5-13), click the combo box of "Section Type" and select the type "**GEOGRID**" to switch to the geogrid section property. A geogrid section has following parameters:

1. Young's modulus E
2. Section area: real section area or equivalent section area
3. Spacing: interval space between adjacent struts
4. Option of tension only
5. Plastic parameter: yielding tension stress and ratio of plastic modular over elastic modular Et/E

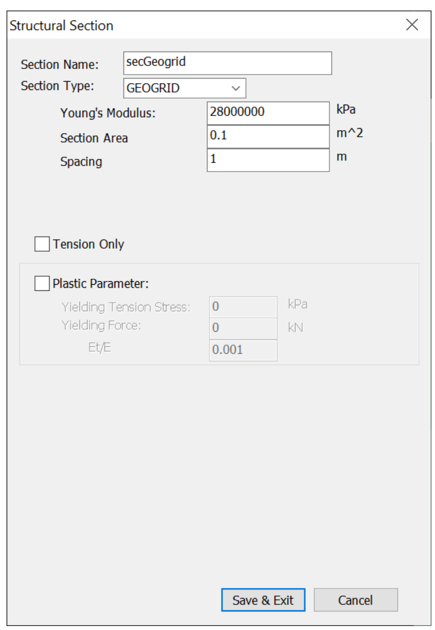


Figure 5-13: Structural section dialog for a geogrid

The default spacing parameter is *1*. If the spacing parameter is not *1,* equivalent section area will be calculated by EnFEM automatically.

After finishing all parameters, click "**Save & Exit**" to save data to the section library and exit to the section group management window.

Geogrid elements share nodes with adjacent soil finite elements. They can be regarded to be bonded with soil solid elements perfectly.

5.4 Assignment of Structural Section

Structural section assignment should be executed after completing definition of structural sections. If a structure section will be assigned to multiple structures, it is recommended to define a set of structure entities first. Click menu "**Data->Structural Section Assignment**" to popup the dialog as shown in Figure 5-14. Users can assign a structural section to structural entities if there no previous assignment, or update current section assignment.

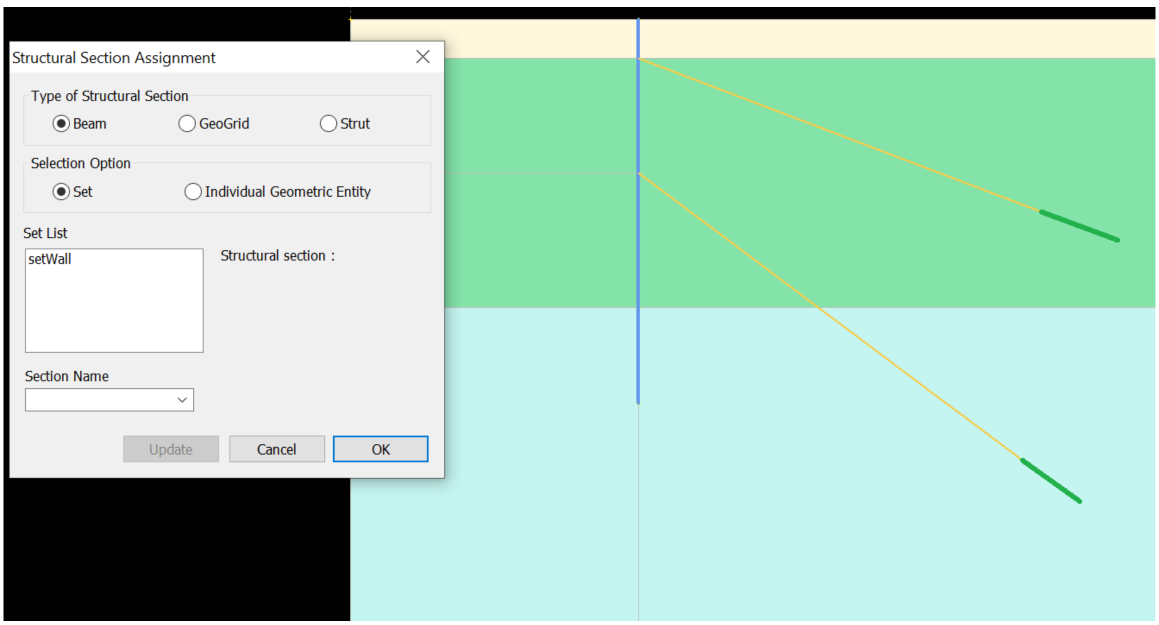


Figure 5-14: Assign structural section to structural entities

EnFEM provides three types of structures: beam, strut and geogrid. Click to select one type. Corresponding structure entities will be displayed in the list window as shown in Figure 5-14.

This option provides users to assign structural sections based on set group definition, or pick one single geometric entity from the list. If option "Set" is checked, all sets corresponding with selected section type will be displayed in the *object list* window. If option "Individual Geometric Entity" is selected, all structural entities with selected section type will be displayed in this window.

Click to pick one set or one structural entity. Note that all members in the selected set group or the selected structural entity will be marked in red. If this set/entity has been assigned a section before, section name will be displayed to the right of the list window. In the dropping list of section type, only matching section types will be displayed. Click to select one section definition. If this is a new assignment, button "**Assign**" will be enabled. Otherwise, button "**Update**" will be enabled. Click the button "**Assign**" or "**Update**" to finish this assignment operation. For a new assignment, selected structural entities will be marked as purple after successful assign operation. If the structural entity or set has been assigned a section, it will be displayed in a red color as shown in Figure 5-14. Click button "OK" to exit from this dialog.

CHAPTER 6. Boundary Conditions

6.1 Boundary Conditions

Boundary conditions in EnFEM include displacement conditions and stress/force conditions. A specialty interaction (contact) condition is also provided for modeling soil-structure interaction, but it is discussed separately in CHAPTER 8. From the menu bar, click **“Data->Boundary Condition”** to open the Boundary Condition Group Management dialog as shown in Figure 6-1.

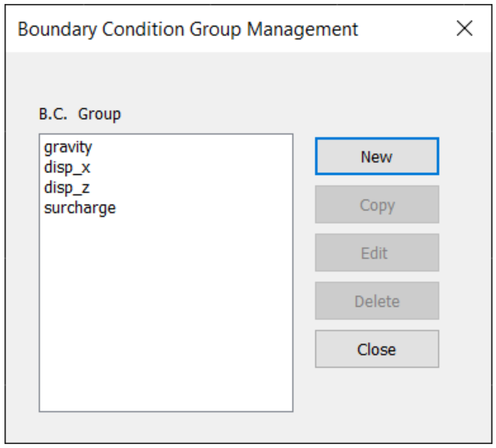


Figure 6-1: Boundary Condition Group Management dialog

Four operations are available:

1. **"New"**: create a new boundary condition definition.
2. **"Copy"**: duplicate a boundary condition definition.
3. **"Edit"**: modify an existing boundary condition.
4. **"Delete":** remove the unwanted boundary condition from the model.

Clicking **"New"** will open the specific Boundary Condition dialog, as seen in Figure 6-2. Define a unique name for the B.C, select the specific *“*B.C Type*”*, and enter in the associated input parameters. Clicking **“Save & Exit”** will finish the definition.

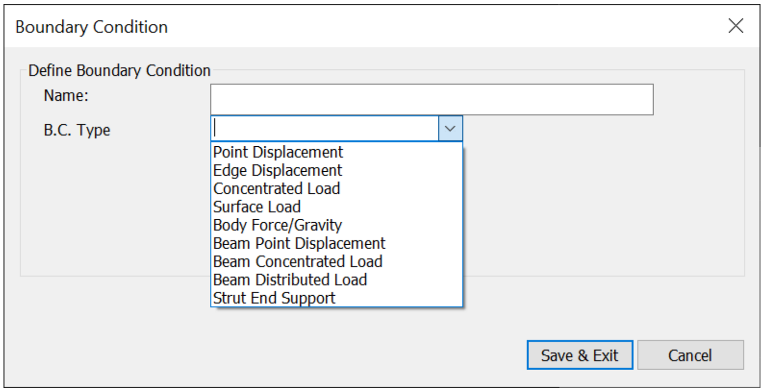


Figure 6-2: Boundary Condition dialog.

For a FE analysis in geotechnical engineering, various boundary conditions are essential, such as gravity loads, external loading along geometry edges, and/or displacement constraints on edges or vertices. In EnFEM, 2D geometric object (region) and its dependent entities (vertices, lines, arcs) are available for boundary condition assignments. Structure entities usually involve soil-structure interactions. They are rarely assigned displacement constraints. The drop box with the label “B.C. Type” includes ten predefined boundary conditions, which will be covered in the following sections.

6.1.1 Point Displacement

The point displacement condition is a displacement constraint applied to region vertices, in the global X and/or Z-directions.

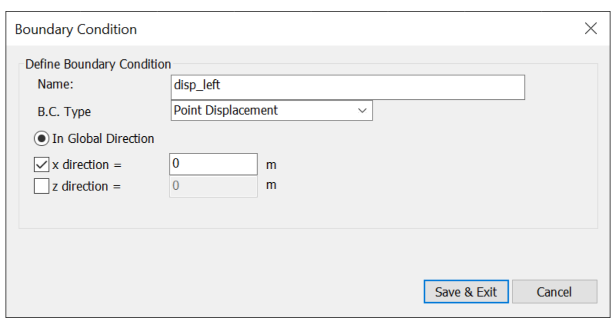


Figure 6-3: Point displacement B.C options

The constraint is activated if the associated option is checked as shown in Figure 6-3. Once active, the user may enter in the desired value for the constraint. Values can be zero or non-zero. Positive X-values are to the right, and positive Z-values are upward.

6.1.2 Edge Displacement

The edge displacement condition is a displacement constraint applied to a region edge. This boundary condition has two options on direction: in global direction, or in local direction. If "In Global Direction" is selected, positive displacement has the same direction as the system axis: positive x toward right and positive z toward upper (Figure 6-4). If "In Local Direction" is selected, directions are defined as "normal to edge" or "tangent to edge". For most analyses the "In Global Direction" should be used. If an edge is inclined, the option "In Local Direction" may be necessary.

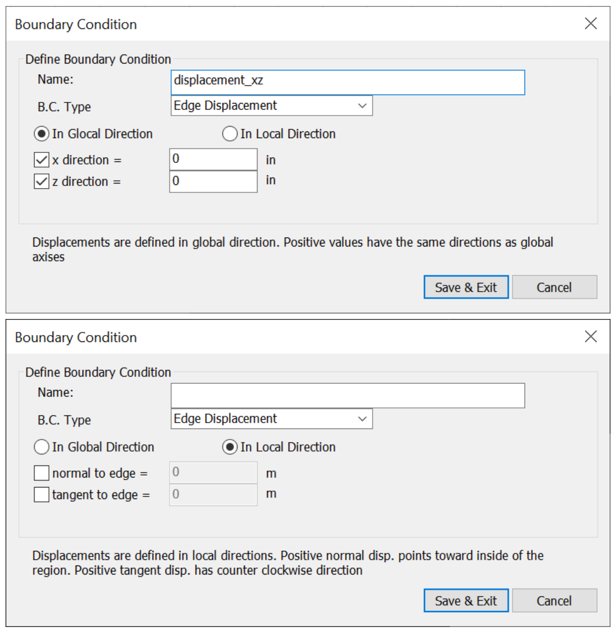


Figure 6-4: Boundary condition of edge displacement constraints

The constraint is activated if associated option is checked. For example, if a displacement boundary is defined as "In Local Direction", and only "normal to edge" is checked, this B.C. is equivalent to a roller, constraining the displacement in the direction normal to the edge. If both "normal" and "tangent" are checked, this B.C. will be a "pin" to constrain movements in both directions. Non-zero displacements pointing toward the inside of the region represent positive values for the normal direction. While non-zero displacements in a counterclockwise direction, with respect to the region center, represent positive values for the tangent direction. While available for use, the latter constraint (tangent) is rare in engineering practice

6.1.3 Concentrated Force

The concentrated force boundary condition is a force constraint (loading) applied to one or several region vertices.

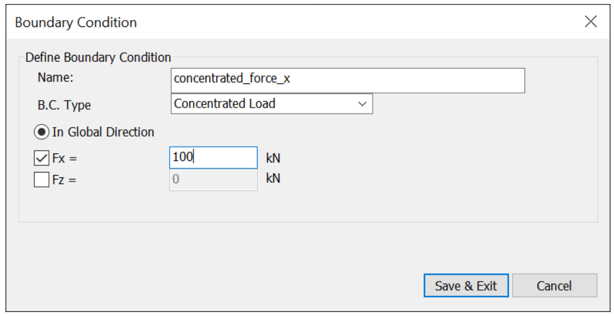


Figure 6-5: Concentrated force B.C. options

The constraint is activated if associated option is checked - as shown in Figure 6-5. Once active, the user may enter in the desired numerical value for the constraint. Note that the “Fx” or “Fz” represents the force applied in the global axis direction. The force value should be non-zero if it is activated. Positive X-values are to the right, and positive Z-values are upward.

Concentrated forces are always applied to region vertices. They cannot be assigned to finite element nodes directly.

6.1.4 Surface Load

A surface load applies a traction to a region edge with two directional options:

1. **In Local Direction** (default): Defined by two components—normal (pressure) and tangent (shear)—as shown in Figure 6-6. The positive pressure direction points inward toward the region, while the positive tangent direction is counterclockwise about the region center (Figure 6.8).

2. **In Global Direction**: Forces are limited to uniform distribution, with positive **x** loads pointing right and positive **z** loads pointing upward (Figure 6-7).

The load units are in Force/Sq. Unit Length. For "In Local Direction," pressure and shear inputs require *start* and *end* values, visualized with a yellow circle (start) and a blue circle (end) (Figure 6-8). If different values are set, the load varies linearly. <u>Note</u>: Start-to-end transitions from positive to negative or vice versa are not supported.

Guidelines for Surface Loads

- Surface loads are typically applied as “In Local Direction” to external edges, such as a surcharge on the ground.

- For internal edges, using “In Local Direction” may cause issues identifying the load’s target region. In such cases, use “In Global Direction”, e.g., for self-weight on a concrete slab at the bottom of an excavation pit.

For more details, refer to Example 10 in the example manual.

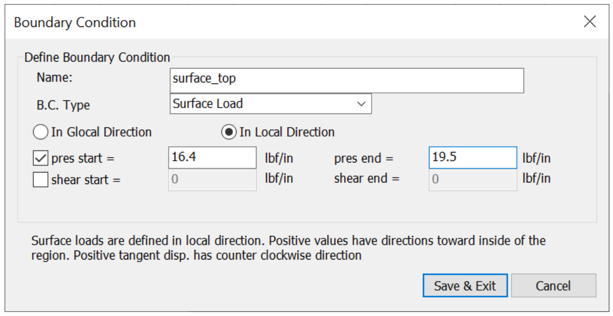


Figure 6-6: Surface load defined as "In Local Direction"

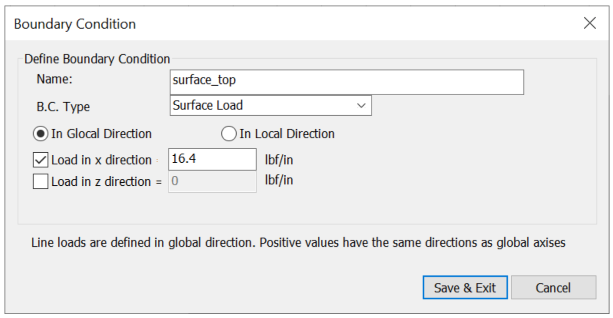


Figure 6-7: Surface load defined as "In Global Direction"

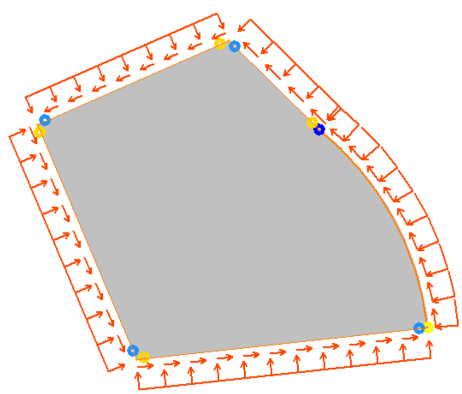


Figure 6-8: Positive directions of surface load B.C.

Pressure and shear surface loads require two inputs: *start* and *end*. These positions are visually indicated:

- **Start**: Marked by a yellow circle.
- **End**: Marked by a blue circle (Figure 6-8).

6.1.5 Body Force/Gravity

The body force/gravity condition is a force constraint (loading) applied to a region entity in the global x and/or z-directions (Figure 6-9).

- To activate, check the associated option and input the desired numerical value.
- Positive x values apply force to the right, while positive x values apply force upward.

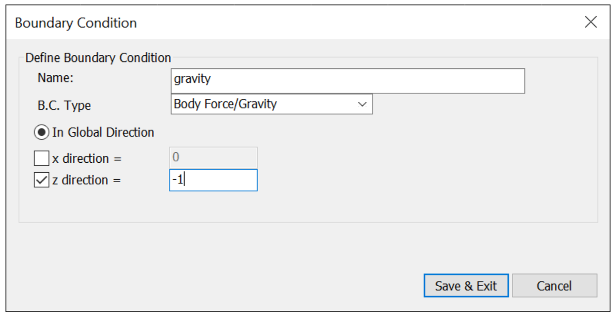


Figure 6-9: Body force/gravity B.C. definition

A common application of this boundary condition is to apply gravity, or self-weight, to soil regions. In this case, users should input "-1" as the gravity load in the Z-direction (Figure 6-9). It represents gravity force in the negative Z-direction (downward).

6.1.6 Beam Point Displacement

Parameters of the beam point displacement are shown in Figure 6-10. This boundary condition includes displacement constraint in the global X direction, in the global Z direction and an option for rotation constraint. Positive values in x direction point to the right and positive values in z direction point to the upward.

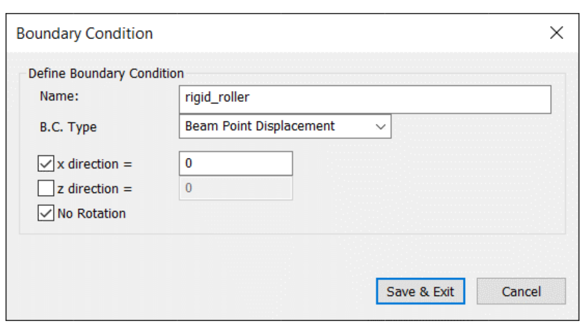


Figure 6-10: Point displacement of beams

The beam point displacement can have several behaviors based on various selections. It can be a "roller" if only the constraint in the "*x*" or "*z*" direction is activated, or a "pin" if constraints in both "*x*" or "*z*" directions are activated. An "embedded" behavior is obtained if all three options are activated. A special "rigid roller" constraint is created if "**No Rotation**" is combined with one translation constraint in the "*x*" or "*z*" direction.

Note: The "**No Rotation"** option must be used with at least one translation simultaneously.

6.1.7 Beam Concentrated load

The beam concentrated load boundary includes three components: the horizontal force, the vertical force and the moment. Positive forces in the x direction point to the right, and positive forces in the z direction point to upward. A positive moment is always in counter clockwise direction (Figure 6-11).

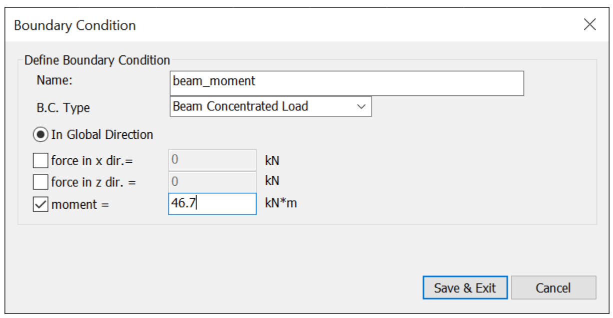


Figure 6-11: Concentrated loads of beams

Activated load options should be paired with non-zero values.

6.1.8 Beam Distributed Load

Parameters of a distributed load on beams is shown in Figure 6-12. Note that distributed beam loads are related to beam directions. Positive distributed loads are always along the positive beam direction. Users may click "**View->Show Beam Direction**" in "CAD MODE" to display the beam direction. If current beam direction is incorrect, click "**Modify->Reverse Beam Direction**" from the CAD menu to reverse it. The normal direction is perpendicular to the beam axis, and the tangent direction is parallel to the beam axis. Alternatively, click icon on the toolbar can turn on the beam direction. Two examples of positive distributed beam loads are shown in Figure 6-13.

Distributed beam loads have values at the "start" and "end" location. At least, one value at the "start" or "end" is non-zero. The "start" value can be different from the "end" value. However, transition from positive to negative or vice versa is not supported.

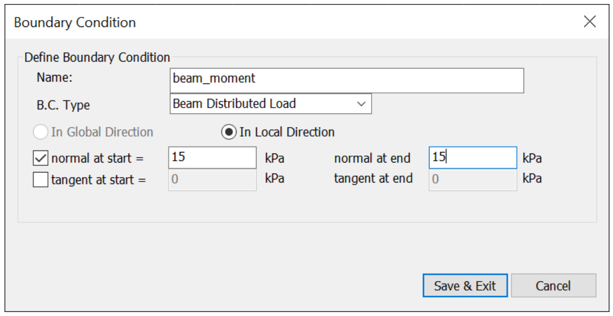


Figure 6-12: Distributed loads on beams

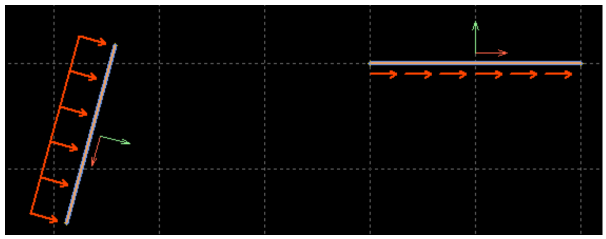


Figure 6-13: Positive values of distributed loads on beams

6.1.9 Strut End Support

The strut end support is specifically used for "pinned" end of a horizontal strut to support a retaining wall (Figure 6-14). It does not have input parameters. This boundary is used for a half part of a symmetric model where struts are used as the horizontal support of retaining walls.

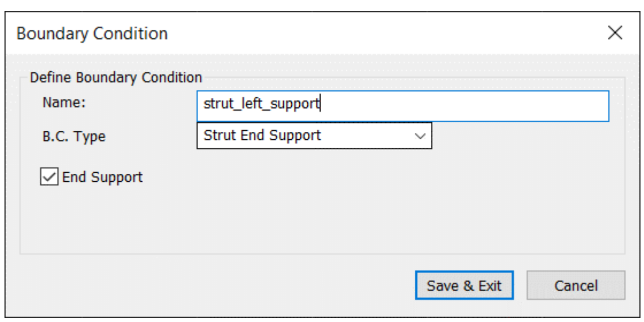


Figure 6-14: End support of struts

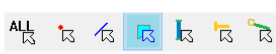
6.1.10 Restriction to "Modify" operation

As mentioned previously, users can modify previously defined boundary conditions from the Boundary Condition Group Management dialog, as shown in Figure 6-1. Select an existing definition from the *B.C Group* list and click **"Edit"** to modify it. However, it should be noted, that only the name, values, and active directions (i.e. X or Z) can be modified. Switching to another boundary condition type is not supported and will result in a model error. If it is desired to define a new type of boundary condition, the user should create a new boundary condition and delete the unused one.

6.2 Applying Boundary Conditions to Geometric Entities

Boundary conditions must be associated with the appropriate geometry after stages are defined, as boundary conditions are strictly linked to stages.

To assign boundary conditions:

1. **Open Boundary Assignment**
    - Navigate to “**Data → Boundary Assignment”** from the menu bar to open the Boundary Condition Assignment dialog (Figure 6-15).
    - All boundary conditions appear under the "*Name*" column, and stages are displayed as the first row.
    - If the group window has no adequate width to show all stages, use mouse to expand the window in the horizontal/vertical directions.
2. **Select and Configure the Boundary Condition**
    - Choose a boundary condition from the "*B.C. Group*" list.
    - By default, the boundary condition starts from the first stage. Adjust the start stage by selecting the corresponding cell and clicking “**Move Right”** or “**Move Left”**. The boundary condition remains active until the final stage.
3. **Assign Boundary to Geometry**
    - Click “**Assign”**. EnFEM automatically activates the appropriate filter icon based on the selected boundary condition (e.g., the region filter for gravity loads ).
    - Follow the command prompt at the bottom of the working zone to select entities (vertex, edge, region, or beam). Use **SHIFT + LEFT-CLICK** or a selection window for multiple selections.
4. **Finalize the Assignment**
    - **RIGHT-CLICK** and select “**End Selection”** to complete the assignment.
    - The assigned boundary condition details will be displayed below the boundary condition group window.

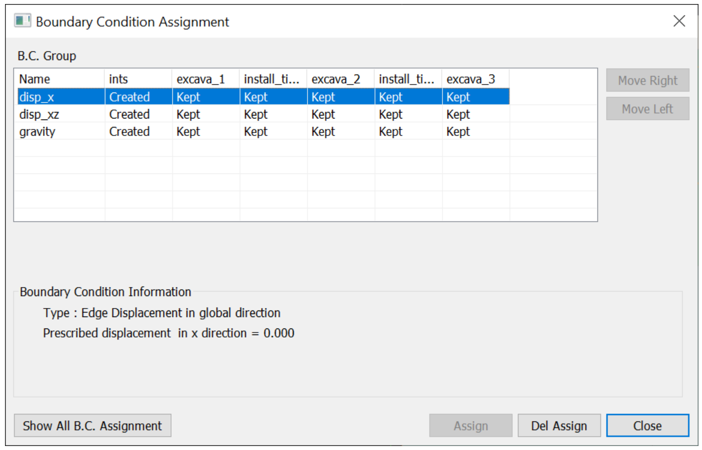
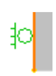
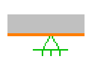
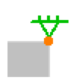
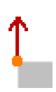
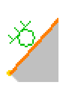
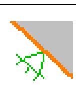
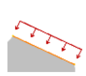
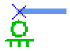
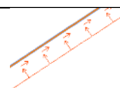


Figure 6-15: Boundary condition assignment dialog

To modify an existing boundary condition assignment, select the desired condition from the "B.C. Group" window and click “**Del. Assign”** to clear the current associations with geometric entities. Once cleared, the boundary condition can be re-assigned by following the steps described previously.

To verify an assignment, select the boundary condition from the "B.C. Group" list. The associated geometry will be highlighted, and the corresponding boundary condition symbol will be displayed. Refer to the list of boundary condition symbols for additional details:

: 	‘roller’ on orthogonal edge
	‘pin’ on orthogonal edge
	‘pin’ on vertex
	concentrated force on vertex
	‘roller’ on inclined edge
	‘pin’ on inclined edge
	surface load on edge

	shear load on edge
	'roller' on beam
	pin on beam
	Embedded support on beam
	'rigid roller' on beam
	Concentrated load on beam
	Moment on beam
	Distributed load on beam

Clicking **"Show All"**, in the bottom left corner of the Boundary Condition Assignment dialog, will display all applied boundary conditions simultaneously. This image provides a convenient visual check to confirm correct assignment of a boundary condition (Figure 6-16).

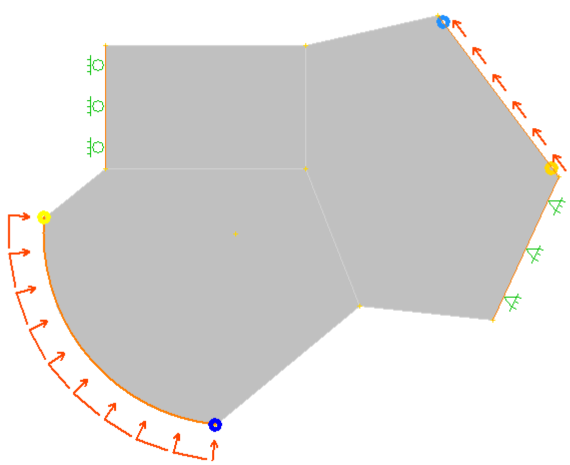


Figure 6-16: "Show all" B.C. for a visual check

After applying a *"body force/gravity"* to a selected region, the corresponding region will be shaded as shown in Figure 6-17.

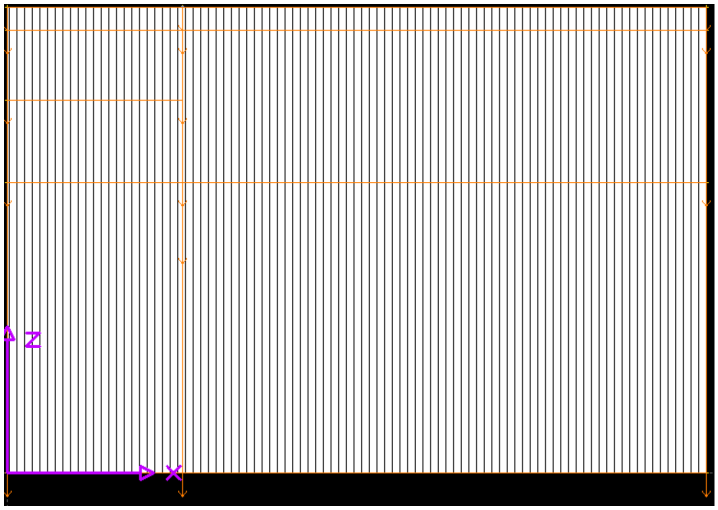


Figure 6-17: Example applied Body Force/Gravity B.C.

It is possible to apply the "body force/gravity" B.C. in incremental fashion if desired, by using multiple Stage and B.C definitions. However, the sum of the increments should be equal to 1.

CHAPTER 7. Set Management & Display Control

7.1 Introduction

Group-sets in EnFEM allow users to efficiently manage and operate on multiple geometry entities. These sets can include regions, edges, vertices, or structural members and are highly versatile in their applications. Common uses include displaying or hiding specific regions, assigning materials or structural sections, and facilitating post-processing tasks like plotting result curves along defined edge-sets.

In soil-structure interaction analyses, where materials like soil and concrete exhibit significantly different properties, group-sets provide a practical solution for managing visualization challenges. For example, stresses in concrete are often much higher than in soil, making it difficult to visually distinguish soil stress contours. By creating separate "selection-sets" for soil and concrete regions, users can isolate the soil region for independent contour plotting, ensuring clarity in the results.

Detailed examples and applications of group-sets, including the use of the “Set Tool” and “Display Tool” for managing “selection-sets,” are provided in the EnFEM Example Manual.

7.2 Set Tool

Operations to create a selection-set are only available in *“CAD-MODE”*. From the menu bar, click **"Tools->Set Management"** to open the Set Group Management window (Figure 7-1).

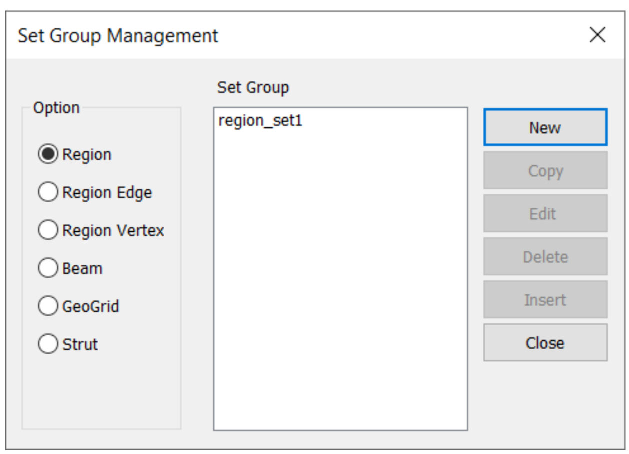


Figure 7-1: Define sets in EnFEM

According to the type of entity, sets can be grouped as region, region edge, region vertex, beam, geogrid or strut. Select the type of group, existing set group will be displayed in the middle window in Figure 7-1. Five operations are available to create or modify a set group:

1. “**New**”: define a new set group which has the selected type
2. “**Copy**”: duplicate a set group
3. “**Edit**”: modify an existing set group

4. “**Delete**”: delete an existing set group
5. “**Insert**”: insert a new set group in front of selected set group

These operations are described in detail in following sections.

7.2.1 Generate New Set

Procedures to define a new selection set are as follows:

1. Select an appropriate “Set Option”. Click “**New**” to pop up the set definition dialog (Figure 7-2)
2. Input a unique “Set Name”. The “Set Description” is optional.
3. Click **"Add Obj."** to select appropriate geometric entities. Depending on the “Set Option”, the related selection icon will be automatically activated. For example, the region filter will be activated for a region set (

).
4. If multiple entities need to be selected, hold SHIFT while LEFT-CLICKING to select or use a selection window. RIGHT CLICK and select “**End Selection**”.
5. Click **"Save & Exit"** to add new set data into model and exit to the set management dialog

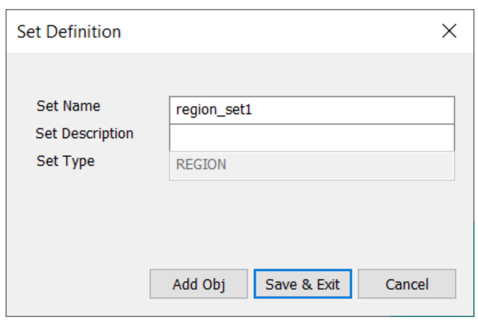


Figure 7-2: Define a new set group

7.2.2 Copy Set

Select an appropriate “Set Option”. Click the set group to be copied. Click “**Copy**” to duplicate a new set group as shown in Figure 7-3. The default name for the new duplication is the original set name plus “_copy#”, where # represents a number assigned by EnFEM.

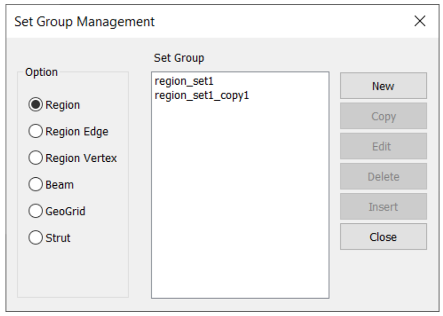


Figure 7-3: Copy set group

7.2.3 Edit Set

Procedures to update an existing selection set are as follows:

1. Select an appropriate set option. Select a set from the Set Group window. Click "**Edit**" to pop up the set definition dialog (Figure 7-4).
2. Change the *"*Set Name*"* and "Set Description*"*, if desired
3. Click **"Update Obj."**. Reselect geometric entities, if necessary. RIGHT-CLICK and select **"End Selection"** from the popup menu.
4. Click **"Save & Exit"** to save data to the model or click "Cancel" to withdraw modification.

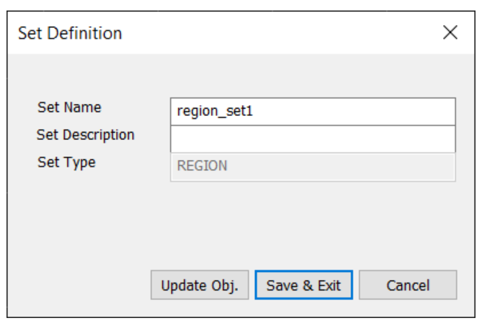


Figure 7-4: Edit set group

<u>NOTE:</u> If corresponding geometry objects associated with the selected are deleted, an error message box will pop up.

7.2.4 Delete Set

Select an appropriate set option. Select a set from the Set Group window. Click **"Delete "**. The selected group is removed from model data and its name is removed from the set group list window

7.2.5 Insert New Set

Select an appropriate set option. Select a set from the Set Group window. The new set will be inserted in front of this set. Click **"Insert"** to pop up the set group definition dialog. Input the new set name and select corresponding geometric entities. Click "**Save & Exit**" to exit to the set management window. The new set is generated and locates at the expected position.

7.3 Display Tool

This tool lets users control display of selected geometric entities conveniently. This feature is specifically helpful when users define soil-structure-interaction behavior. This tool is also essential in the post processing to plot contours on selected finite mesh or calculate integrated forces.

7.3.1 Use Display Tool to Control Hide/Display

From the menu bar, click **"Tools->Display Tool"** or click the  icon from the toolbar, to open the Display Control window as seen in Figure 7-5.

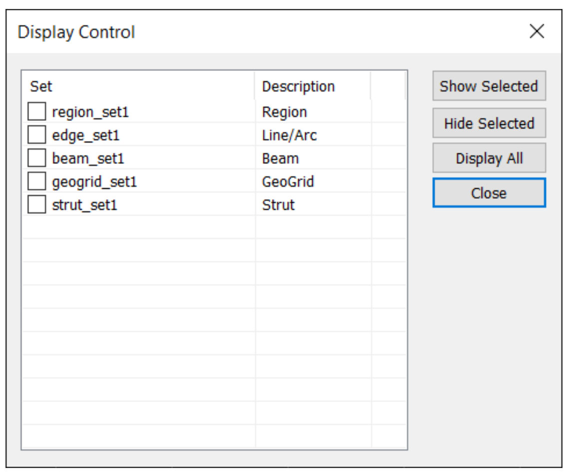


Figure 7-5: Display control by set group

Note that set group window has two columns: the left column shows all group names. Each set group has a check box before the name. The right column displays the set type. Only sets defined on independent entities will be listed here. Three commands are available in this display control window:

1. "**Show Selected**": only geometric entities belonging to selected sets are displayed. Other geometric entities will be hidden. If current window is the result window, finite elements belonging to these explicitly displayed geometric entities will be shown.
2. "**Hide Selected**": only geometric entities belonging to selected sets are hidden. Other geometric entities will be displayed. If the current window is the result window, finite elements belonging to these explicitly hidden geometric entities will be hidden.

3. "**Display All**": display all geometric entities. If the current window is the result window, all finite elements will be displayed.

Note: It is common that one geometric entity may belong to different set groups. EnFEM will merge repeated entities to find the maximum entity list marked as "selected". The command "**Show Selected**" or "**Hide Selected**" will be applied to all members in this list.

Display control by set group is available in both "CAD-MODE" and "RESULT-MODE". It is not available in "MESH-MODE".

7.3.2 Alternative Display Tool

EnFEM provides an alternative method to hide geometric entities. In "CAD-MODE", click the "Select All" icon, double click a region or a structure. Then select "**Hide Object**" from the popup menu. The selected geometry will be hidden. If users wish to recover display, click icon from the toolbar.

CHAPTER 8. Soil-Structure Interaction

8.1 Introduction

Contact is a crucial aspect of engineering analyses, particularly in geotechnical and mechanical engineering. Typical examples of interactions include soil-culvert systems, soil-retaining walls, and soil-pile systems. At their interfaces, these distinct materials must satisfy the condition of impenetrability, ensuring no overlap or penetration occurs. The inherent nonlinearity of contact behavior makes soil-structure interaction (SSI) problems challenging and computationally complex.

EnFEM provides two robust methods to address SSI problems:

- **Generic Contact Constraint Method:**
  This method applied to interaction between solid elements. Impenetrability condition is enforced at the interface using mathematical constraints. It is versatile and suitable for a broad range of SSI problems.

- **Zero-Thickness Interface Elements:**
  This approach models the interaction by using specialized interface elements with zero thickness. Structures are simulated as beam elements. These interface elements are designed to handle the distinct behaviors of the materials involved, such as soil and structural components, and are particularly useful for problems where interface behavior plays a dominant role.

The following sections will delve into the details of each method, highlighting their applications, strengths, and limitations.

8.2 Contact Constraint Method

This method is applicable to interaction between solid finite elements. Impenetrability on the interface is implemented by the penalty approach. Since the penalty value cannot be infinite, solid elements on the interface may have slight overlap (penetration). Contact pressure points in the normal direction of the interface. In the tangent direction, friction follows the classic Coulomb's friction law. In EnFEM, the contact interfaces can be frictionless or frictional. This method is versatile, which can be used to simulate sticking, finite sliding, and rebounding of contact objects.

The contact constraint method distinguishes the object of softer material as the ″slave body″, and the object of stiffer material, as the ″master body″. The surface involved in contact is called the "slave surface", if it is located on the slave body, or the "master surface", if it belongs to the master body. The implementation in EnFEM is based on the well-known ″Node-to-Segment″ algorithm. In other words, slave surfaces are represented by a series of *″*slave-nodes*″*, and master surfaces are represented by a series of *″*master-segments*″* after discretization. In EnFEM, linear finite elements, Q4 or T3, should be used to discretize the regions when the contact constraint method is used.

In 2D cases, contact interfaces are reduced to several lines or arcs, which are edges of regions. In the real world, potential bodies of contact may overlap initially or have gaps. These open gaps significantly increase the difficulty for contact search. Since gaps are not common in geotechnical engineering, EnFEM does not allow the existence of gaps along the interface surface initially. Master and slave surfaces should exactly overlap with each other.

After creating all region objects, users can define contact interaction group. Since contact objects overlap on the interface, it is necessary to define several regions set groups for convenient display control. Users can refer to chapter 7 to learn how to define object sets and how to control display.

To define contact group, in *"*CAD-MODE*"*, click **"Data->Interaction->Contact Constraint Method"** to open the Interaction Group Management window as shown in Figure 8-1.

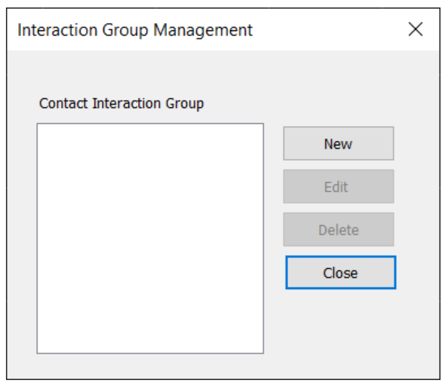
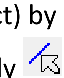


Figure 8-1: Dialog of Contact Interaction Group Management

Click **"New"** to create a new interaction group. Select an existing group from the "Contact *Interaction Group"* list and click **"Edit"** to modify it. Select an existing interaction group and click **"Delete"** to remove the unwanted group from the model.

8.2.1 New Contact Interaction Group

When creating a new interaction group, the *"*Interaction*"* window will open (Figure 8-2). Follow these steps to define the contact group:

1. Enter a group name for the interaction.
2. Select one "Interface Option" from the available choices.
3. Input the "Coefficient of Friction," typically ranging from 0.2 to 0.5 for soil-structure interfaces.
4. Define the master interface (on the stiffer object) by clicking "**Add Master Surf**." The line selection icon will be activated automatically . Use the appropriate geometry selection tools, and after making the selections, RIGHT-CLICK and choose "**End**

Selection." Use the "Display Tool" and zoom functions to simplify visibility and selection.

5. Define the slave interface by clicking "**Add Slave Surf**." The line selection icon is again activated . After selecting the relevant edges, RIGHT-CLICK and select "**End Selection**" from the popup menu.
6. Once all selections are made, click "**Save & Exit**" to close the "Interaction" window and save the newly created contact group

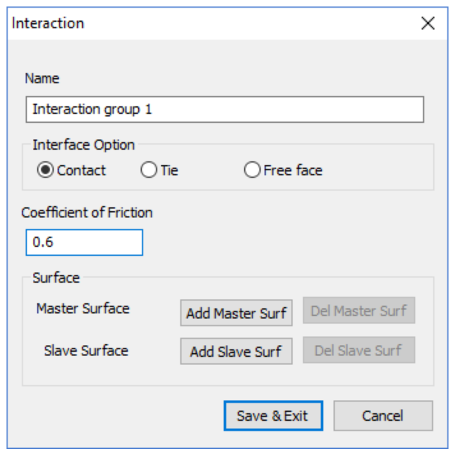


Figure 8-2: Create a new contact interaction group

The contact algorithm in EnFEM is based on the node-to-segment approach. This approach requires that the master body should be convex. Concave corners of a master body often bring convergence issues. Several examples of concave corners are shown in Figure 8-3.

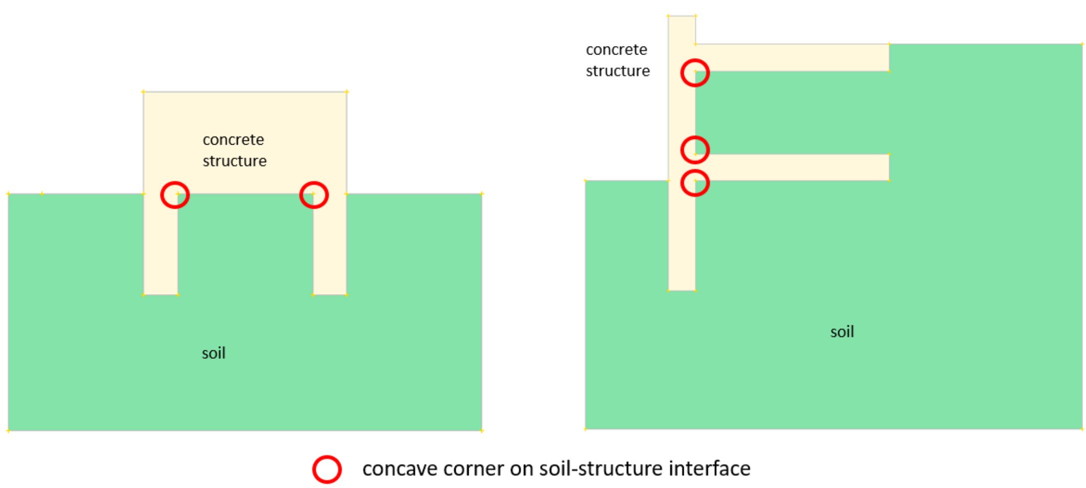


Figure 8-3: Examples of concave corners on soil-structure interface

There are three options to define the behavior of the interaction surfaces: "Contact", "Tie", or "Free Face".

8.2.1.1.1 Contact Option

The **"**Contact**"** option in EnFEM accurately simulates mechanical behaviors at an interface, including sliding, impenetrability, and separations. This condition is defined by two components: *Normal contact status* and *Tangent contact status*.

- **Normal Contact Status**
    - Represents whether the interface is in a state of *separation* (gap exists) or *in-contact* (no gap).
- **Tangent Contact Status**
    - Indicates whether the surfaces are *sticking* (no relative movement) or *sliding* (relative movement occurs).

These parameters ensure that the interface behavior is modeled realistically, accounting for the complexities of soil-structure interactions or other systems involving contact mechanics.

8.2.1.1.2 Tie Option

The "Tie" option is used to perfectly bond two bodies together along the common interface surface. This approach ensures that finite element nodes on the tied interface exhibit identical displacements in both the X and Z directions, effectively simulating a bonded condition. A "Tied" connection does not need iterations to search active "node-segment" pair. Therefore, the "Tie" option provides a fast solution if finite element nodes on the interface almost have no sliding. It can also be used temporarily to replace "Contact" option to help quickly troubleshoot convergence issues in the model. Final model can easily switch back to the "Contact" option later.

8.2.1.1.3 Free-Face Option

The "Free Face" option is used to create independent edges and is generally used in addition to the "Contact" option. If the "Free Face" option is used, adjacent regions on both sides of the free face will have their own independent meshes. Edges defined as a "Free Face" provide no deformation restraint to one another – allowing both separations and penetrations. This option is used generally to simplify contact interfaces and improve convergence when small edges appear in the model, such as the tip of a very long retaining wall or pile, or the tip of a long concrete soil nail. This issue is more noticeable if the geometry at these 'tip' edges is a sharp right angle rather than using a half-circle for a smoother transition. Using this option and setting the 'tip-edge' as the free face, may improve convergence performance, if a sharp right angle is used.

8.2.1.2 Coefficient of Friction

The coefficient of friction is used only if the "Contact" analysis option is selected. It is applicable to "Tie" or "Free Face" option.

The value of the coefficient of friction varies from 0 to 1. A coefficient of 0.0 is used for a perfectly smooth interface (frictionless). While a coefficient of 1.0 is the allowable maximum factor. In geotechnical engineering, the coefficient of friction is often in the range of 0.2 to 0.5.

8.2.1.3 Master and Slave Surfaces

As mentioned previously, the interaction algorithm in EnFEM employs is the "Node-to-Segment" method. The assignment of the "master" and "slave" surface are defined here, in the "Interaction" window. Click **"Add Master Surf"**, select the desired geometry (edges), and RIGHT-CLICK and select **"End Selection"** to complete the assignment – follow the same steps for the "slave" surfaces. "Master" surfaces are highlighted in blue (Figure 8-4), while "slave" surfaces are highlighted in green (Figure 8-5). If a "master" or "slave" surface has been previously or mistakenly defined, clicking **"Del. Master** (or Slave) **Surface"** will clear the assignment. Some recommendations for the assignment of the "Master-Slave" surfaces are as follows:

- "Contact" option:

If the interface is defined as "Contact" type, it is important to distinguish "master" bodies from "slave" bodies. Geometry bodies with the *stiffer* material should be the "master" body, while the "slave" body should be set on softer material. For example, if a concrete pile is interacting with soil, the pile should be defined as the "master" body, and the soil to be the "slave" one. Alternatively, reference may be used based on mesh refinement, whereby the "master" body (and corresponding surface) is defined as that with *less refinement* (larger element size), while the *"slave"* should be defined as the body with *more refinement* (smaller element size).

- "Tie" option:

The "Tie" option borrows the master/slave body concept. The "master" body in a "Tie" interface does not need to be the stiffer one.

- "Free Face" option:

The "Free Face" option does not distinguish the "master" body from the "slave" body. In general, either body each side of the adjacent edge, can be defined and the "master" or "slave".

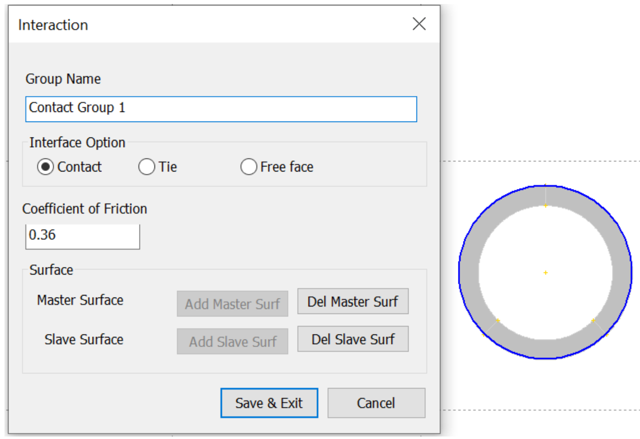


Figure 8-4: Define master interaction surfaces

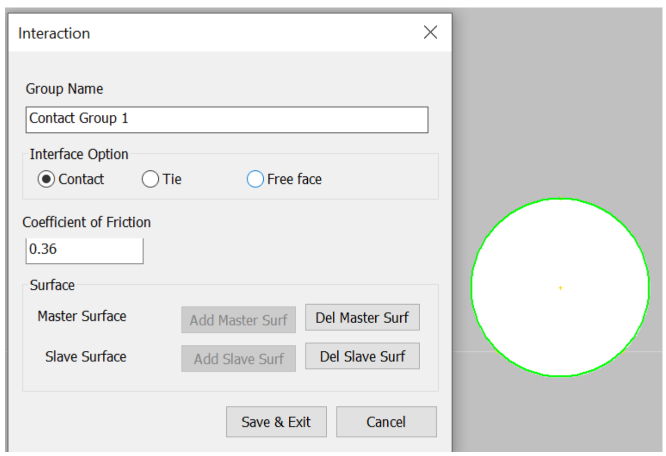


Figure 8-5: Define slave interaction surfaces

8.2.2 Edit Contact Interaction

This option allows the user to modify an existing interaction definition. Select the desired interaction definition from the Interaction Group Management window (Figure 8-1) and click **"Edit"** to open the "Interaction" window. Users can then modify the condition as desired, including changing the "interface option", "coefficient of friction", and reassignment of "master" and "slave" surfaces. After editing, click **"Save & Exit"** to save the changes and return

8.2.3 Delete Contact Interaction Group

This option allows the user to delete an existing interaction definition. Select the desired interaction definition from the Interaction Group Management window (Figure 8-1) and click **"Delete"** to remove the definition from the model database. Click **"OK"** to exit from the Interaction Group Management window when finished.

8.3 Zero-Thickness Interface

In geotechnical engineering, structures such as sheeting piles or culverts often have a high length-to-thickness ratio. Using solid finite elements for these slender structures can complicate mesh generation and make model development and analysis cumbersome. To address these challenges, EnFEM enables the simulation of slender structures as beams, offering a more convenient solution.

However, the generic computational contact algorithm, as described earlier, is not suitable for interactions involving beam-like elements. To overcome this limitation, zero-thickness interface elements are introduced in EnFEM as an alternative solution for soil-structure interaction (SSI) problems.

Key Features of Zero-Thickness Interface Elements:

- Adapted for Slender Structures: These elements are specifically designed for slender structures where solid finite elements are impractical.
- Efficient Interaction Simulation: The interface elements facilitate great modeling of soil-structure interactions without relying on the generic contact algorithm.
- Simplified Modeling: By treating the slender structure as a beam, the zero-thickness interface approach reduces complexity and enhances computational efficiency.
- Versatile Application: This method is well-suited for geotechnical systems involving sheeting piles, culverts, and other elongated structures where soil-structure interactions are critical.

The zero-thickness interface approach expands the flexibility and scope of EnFEM in handling diverse SSI problems, ensuring efficient analysis of slender structural systems

In “CAD-MODE”, from the menu bar, click **"Data->Interaction->Zero-thickness Interface"** to open the Interaction Group Management window as shown in Figure 8-6.

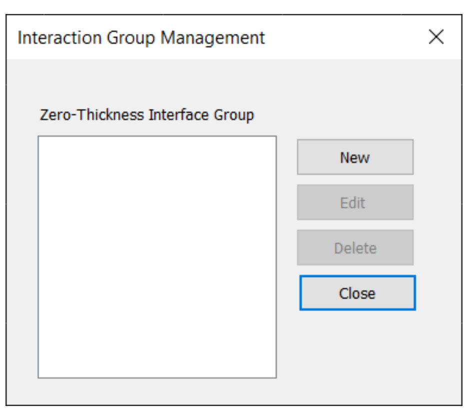


Figure 8-6: Dialog of Zero-thickness Interface Group Management

Click "**New**" to define a new zero-thickness interface group. Select an existing group and click "**Edit**" to modify it. Select an existing group and click "**Delete**" can remove this group from model database.

8.3.1 New zero-thickness Interface Group

The "Zero-thickness interface" window is used to create a new group (Figure 8-7). Procedures are listed here:

1. Key in the name of the interface group.
2. Select the desired "Interface Option." The default setting is "Zero Thickness".
3. Use the default values of "normal stiffness", "tangent stiffness" and "friction angle", or input user-defined values. If the normal and tangent stiffness are input as zero, program will use default values.
4. Click **"Add Beam."** to define the interaction geometry. The beam selection icon  will be automatically activated. Make the appropriate geometry selections, and then RIGHT-CLICK and select **"End Selection"** from the popup menu.
5. Click "**Create Interface**" to let program find appropriate interfaces applicable to be zero-thickness interface (Figure 8-8). Green dashed lines indicate successfully found soil-structure interfaces
6. Click **"Save &Exit"** to exit from the "Interaction" window.

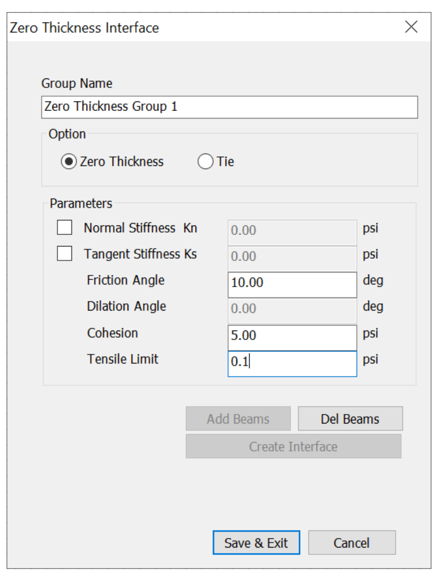


Figure 8-7: Define zero-thickness interface group

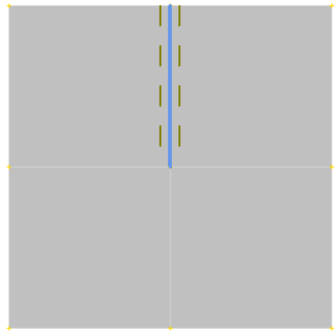


Figure 8-8: Example of zero-thickness interface group

8.3.1.1 Interface Options

There are two options to define the behavior of the soil-structure-interaction interfaces: *"*Zero Thickness*"* or *"*Tie*"*.

8.3.1.1.1 Zero Thickness Option

The zero-thickness interface element approach assumes a thin intermediate layer, with zero thickness, situated between the structural member and the soil part. This method is highly effective for simulating detailed interactions between soil and structures.

The intermediate layer connects the soil to the structure, with one side interacting with the soil element and the other with the beam element. Stresses, both normal and tangential, are calculated based on the relative movement between the two materials. The constitutive behavior of this layer follows an elasto-perfectly plastic Mohr-Coulomb algorithm. Tangential behavior begins with "sticking," transitioning to "sliding" when the tangential stress reaches the yielding threshold. Normal behavior is initially assumed to be "in contact," but when normal stress exceeds the cohesion of the material, the interface transitions to "separation."

The zero-thickness assumption is the default setting in EnFEM for this simulation approach. It effectively handles behaviors like "in contact," "sticking," and "small sliding," making it an efficient and accurate tool for modeling soil-structure interactions. By adopting the Mohr-Coulomb framework, it ensures realistic and physically consistent interface behavior, particularly for slender structures modeled as beams

8.3.1.1.2 Tie Option

This option is similar with that used in "contact constraint method". Structures are assumed to be perfectly bonded with soils. There are no relative displacements inside the interface layer.

8.3.1.2 Parameters of Interface Options

This zero-thickness interface utilizes the elasto-perfect plastic Mohr-Coulomb criterion. Material parameters of the zero-thickness interface are followings:

- Normal stiffness Kn
- Tangent stiffness Ks
- Friction angle

- Dilation angle
- Cohesion
- Tension limit

The value of Kn and Ks can be arbitrarily selected in theory. However, appropriate value can improve convergence performance and accuracy. A recommended range of Kn is 2 ~ 50 G, where G is the shear modulus of soil. In EnFEM, Ks is one-tenth of Kn by default. The friction angle can be determined from the laboratory test the Mohr-Coulomb model. The intermediate layer is assumed to have weaker strength than adjacent soil. The tension limit is the strength defined in the direction normal to the beams. Usually soils have poor resistance to the tension stress. Users can input a small number as the tension limit.

The dilation angle is always set to "0".

If the soil-structure interface involves multi soil layers, interface parameters can be averaged from these soil layers.

If Kn and Ks are input "0" or unchecked, EnFEM will estimate appropriate values automatically. Users can enlarge or reduce the default values by the scale factor as described in section 12.2.

8.3.1.3 Add Beams

Click "**Add Beams**" in Figure 8-7. The beam selection icon is automatically activated . If multi beams are needed, hold the SHIFT key and click to select these beams. After selection is completed, RIGHT-CLICK and select "**End Selection**" from the popup menu. The activated "**Add Beams**" button will be grayed and "**Del Beams**" will become activated which can be used to remove previously selected beams from current interface group definition.

If multi beams are selected in the same interface group, these beams should be "head-toe" connected without interruption. Non-continuous beams should be separated into different interface groups.

8.3.1.4 Create Interface

After adding beams and regions to the interface group, click the "**Create Interface**" button, as shown in Figure 8-7, to enable EnFEM to identify suitable soil-structure interaction (SSI) interfaces. Once the process is successfully completed, the button will turn gray, and green dashed lines will appear to indicate the identified interfaces between soil and beams, as illustrated in Figure 8-8. Users should verify the correctness of the identified interfaces to ensure accuracy.

Geometric compatibility between soil regions and beams must be maintained for proper interface definition. Beam segments can overlap with region edges, but specific conditions must be met.

Beam vertices should pair with corresponding region vertices to establish compatibility. Alternatively, a beam can pierce into a region, but compatibility at the vertices along the region edge must be preserved. This ensures that the mesh and interface behavior align with the physical and computational requirements of the model

8.3.2 Edit Zero-thickness Interface Group

Select an interface group (Figure 8-6) and click “**Edit**” to modify an existing interface group. Users can change the option, interface parameters, change beams or regions. After modification, click “**Save & Exit**” to save changes into model database.

8.3.3 Delete Zero-thickness Interface Group

Select an interface group (Figure 8-6) and click “**Delete**” to remove an existing interface group definition from model database. This operation has no “Undo” operation to recover from deleting.

CHAPTER 9. Activation/Deactivation of Geometric Entities

9.1 Introduction

Soil excavation is common in various geotechnical engineering projects, including the foundations of high-rise buildings, subway tunnels, slope stabilization, and buried culverts. To manage soil deformation near the excavation area, retaining walls are typically reinforced with struts or tie backs, which are installed in a sequential manner. To accurately model this construction process using Finite Element Analysis, it is necessary to partially remove solid finite elements to reflect the effects of excavation and to activate structural elements in a sequential fashion. EnFEM offers capabilities for activation and deactivation specifically for this type of excavation analysis.

9.2 Define New Deactivation Condition

9.2.1 Prerequisite model setup

Excavation analysis is generally characterized by nonlinearity, and to effectively manage this, the excavated area should be divided into several smaller sub-regions for incremental excavation. In EnFEM, the following steps are required to establish a deactivation condition:

1. Create Geometric Sub-Regions: Divide the excavated area into smaller layers.
2. Create Region Selection Sets: Set up selection sets for each sub-region. Note: It is advisable to name each set according to its order of excavation (e.g., “exca_set_layer_1,” etc..).
3. Define Excavation Stages: For each sub-region, establish a specific excavation stage-region. These excavation stages follow any "initial stress" stage.
    a. NOTE: It is recommended to name each stage according to its excavation order (i.e. *“exca_layer_1”*, etc.).
4. Finalize Other Model Parameters: Complete the definition of additional model parameters, including material assignments and boundary conditions

9.2.2 Deactivation of Regions

Once the prerequisite conditions are satisfied, navigate to "**Data -> Deactivate/Activate**" to open the Deactivation/Activation window, as illustrated in Figure 9-1. This dialog features four tabs, each dedicated to defining the deactivation or activation of regions, beams, struts, and geogrids. All stages defined in the model are automatically listed according to the sequence in the stage management dialog.

In the column to the right of each stage, you will find a list specifying which set will be deactivated or activated. By default, the set name appears as "None," indicating that no region set will be deactivated. For instance, in Figure 9-1 the initial excavation occurs during the “exca_1” stage. To assign a region set for this stage, click the cell adjacent to “exca_1,” then click the small arrow on the right side of the cell and select the region set “exca_1” from the dropdown list. This action establishes the deactivated region for stage “exca_1.” You can repeat this process to specify the region sets for stages “exca_2” and “exca_3,” as shown on the right side of Figure 9-1.

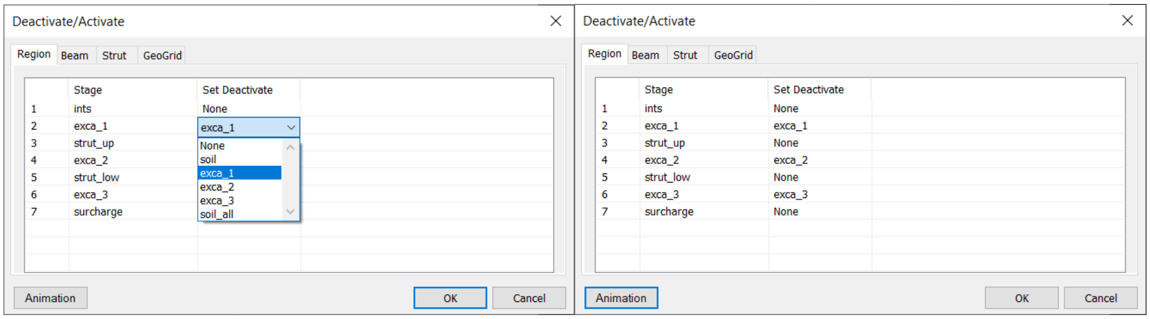


Figure 9-1: Region deactivation dialog – During (left) and After (right)

In the excavation simulation, the initial stage, named “ints”, represents the “initial soil stress” phase. As a result, there is no deactivated selection set associated with this stage. When a region set is deactivated, its weight is completely removed, and the finite elements from these deactivated regions do not contribute to the overall system stiffness. Additionally, deactivating a region also impacts related boundary conditions, including contact interfaces or zero-thickness interfaces, which will be automatically deactivated.

Users can preview the defined process by clicking the “**Animation**” button, which will cycle through the deactivation and activation steps once before stopping automatically. This can be used to visually confirm the deactivation settings. Once satisfied, click “**OK**” to complete the definitions and exit the “Deactivation/Activation Dialog” window.

For further details on this technique, refer to *Example 10 – Analysis of Excavation with Cantilever Retaining Wall*, found in the Example Manual.

9.3 New Activation of Structures

9.3.1 Define new activation

In the Deactivation/Activation window shown in Figure 9-1, navigate to the “Beam” tab to access the beam activation settings. The retaining wall set is labeled “wall” and will be activated during the “initial soil stress” stage. The corresponding zero-thickness interfaces associated with this wall will also be activated simultaneously. To proceed, click the cell following the initial stage “ints,” then click the small arrow on the right side of the cell and select the beam set “wall.” Since there is only one beam set to activate in this model, the activation process for the beam set is now complete (as illustrated on the left side of Figure 9-2).

In the example depicted in Figure 9-1, two struts are utilized to support the retaining wall: one strut is installed after the “exca_1” stage, and the other is added after the “exca_2” stage. To activate the struts, click the “Strut” tab to enter the strut activation window. Click the cell following the “strut_up” stage, then click the small arrow on the right side of the cell and select the strut set “strut_up.” Repeat this process to add the strut set “strut_low” in the “strut_low” stage. With these steps completed, the activation of the struts is finalized.

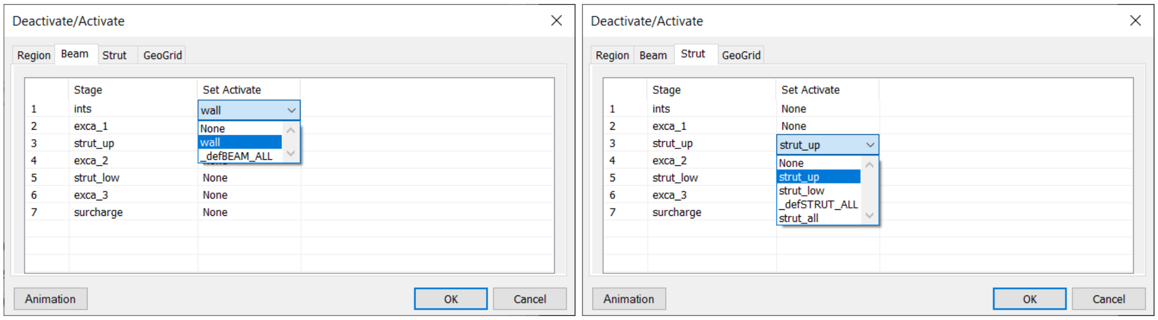


Figure 9-2: Activation of beams set (left) and activation of strut set (right)

Note: Usually, simulation EnFEM starts from the initial soil stress stage. If this model includes beams and soil-structure interactions, beams should be activated during the initial soil stress stage.

CHAPTER 10. Piezometric Line

10.1 Introduction

In geotechnical engineering, the presence of underground water is a common factor that significantly affects the mechanical behavior of soil materials. This impact must be considered during finite element analysis. EnFEM allows users to define a single piezometric line based on user-defined points. It is important to note that EnFEM treats this piezometric line as static. The water level remains constant throughout the analysis.

10.2 Define Piezometric line

In “CAD-MODE”, click menu “**Data->Piezometric Line**” to popup the input dialog as shown in Figure 10-1. Click “**Add Row**” to add one new data line at the end of the table. Click “**Insert Row**” to insert one data line at current cursor position. Click “**Delete Row**” to delete the data line at current cursor position.

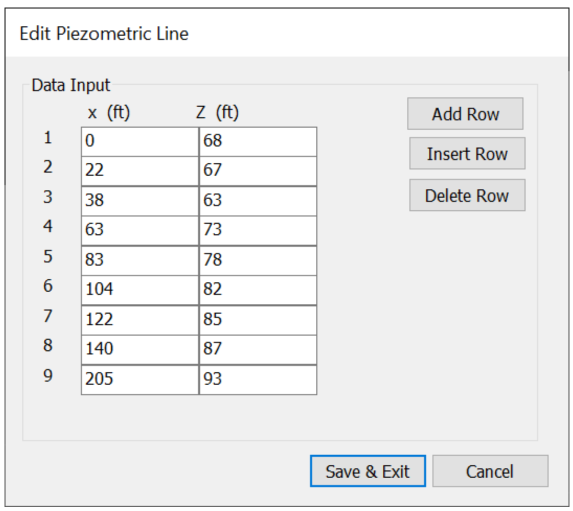


Figure 10-1: Input a piezometric line

The piezometric line will be shown in light green color in “CAD-MODE” (Figure 10-2) or in red color in “MESH-MODE”.

Users can toggle on/off the display of the piezometric line by clicking menu “**View->Show Piezometric Line**”.

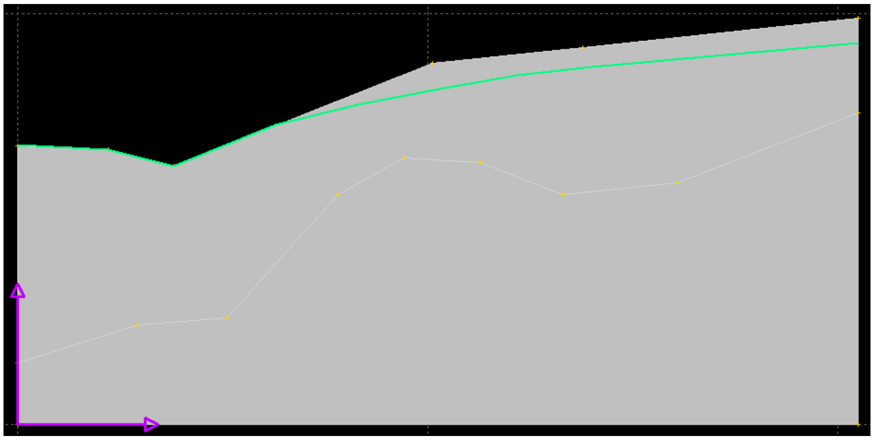


Figure 10-2: Display of a piezometric line in "CAD-MODE"

CHAPTER 11. Finite Elements & Mesh Generation

11.1 Introduction

The finite element mesh is crucial to the success of finite element analyses, as it significantly affects both computational efficiency and result accuracy. A mesh that is overly refined consists of numerous elements, leading to longer computation times and larger data files that can be challenging to handle. Conversely, a coarse mesh with an insufficient number of elements may produce quicker results, but those results may not be accurate. The key to creating a high-quality mesh lies in finding the right balance; it is generally advisable to develop the final mesh through several iterative trials and comparisons to assess how mesh quality impacts the results.

In EnFEM, the typical workflow involves users selecting the mesh type, element shape, element type, integration rule, global element sizes, and any local refinements before generating the finite element mesh.

For added convenience, EnFEM also offers an automatic mesh generation feature. However, it is important to note that in “**Auto Mesh Generation**," the default element shape is restricted to triangles, and the mesh is generated with a uniform distribution.

11.1.1 Solid Finite Element

Throughout the history of Finite Element Analysis (FEA), numerous finite element types for solid objects have been developed to address a range of challenges in solid mechanics, fluid dynamics, acoustics, and electronics and magnetics. In solid mechanics specifically, the primary focus is often on analyzing stresses and deformations. Figure 11-1illustrates four commonly used solid finite elements in this domain. These elements are available in EnFEM.

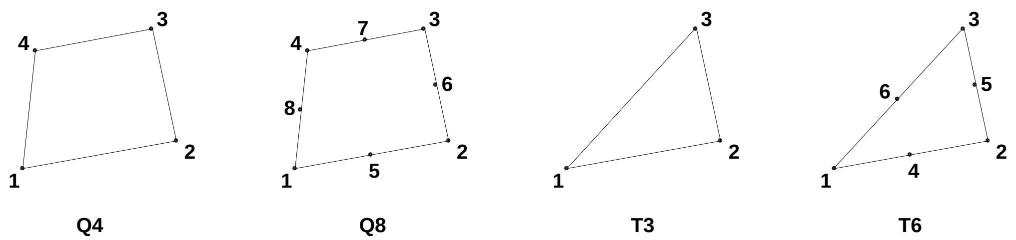


Figure 11-1: Types of finite elements used in EnFEM

The elements depicted in Figure 11-1 can be classified into two main categories: quadrilaterals and triangles. Within each category, there exists one linear element type and one quadratic element type. The quadratic elements feature additional nodes located at the midpoints of each edge, in addition to the nodes at the endpoints. The quadrilateral elements are represented by Q4 and Q8, while the triangular elements are represented by T3 and T6. The subsequent sections will provide detailed information on these various element types. For more technical details, please refer to the EnFEM Technical Manual.

11.1.1.1 Linear Quadrilateral Element - Q4

The Q4 is a linear quadrilateral element characterized by its four nodes and four straight edges, with nodes numbered in a counterclockwise manner. Figure 11-2 illustrates the Gaussian integration methods used in Q4 elements. Red crosses mark the Gaussian integration points. The left image depicts the standard Gaussian integration method, which utilizes four integration points, while the right image shows a reduced Gaussian integration method with a single integration point positioned at the element's center. Displacements within the element are interpolated linearly, resulting in field variables such as stresses or strains to be constant, based on the chosen method. Generally, the Q4 element performs well across various solid mechanics analysis scenarios. When bending behavior is dominant, such as in simulating a beam, this element may experience shear locking when applying the full integration rule. To mitigate shear locking effects, it is advisable to use multiple element layers. Another solution is to use reduced integration because it can significantly alleviate the impact of shear locking. However, if the Q4 mesh is structured (in a beam) and not confined by soil totally, computation may lose stability due to “zero-energy” motion mode.

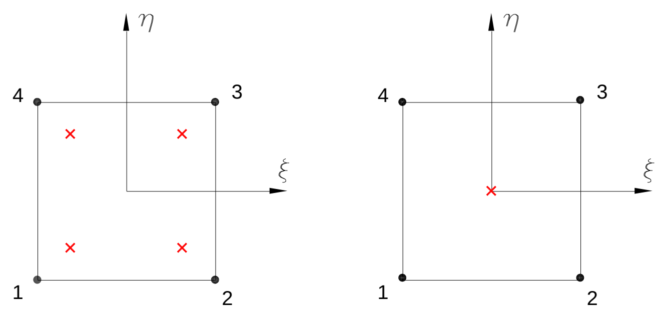


Figure 11-2: Gaussian integration rules for Q4 element

11.1.1.2 Quadratic Quadrilateral Element - Q8

The quadratic quadrilateral element, known as Q8, features eight nodes and four edges. Nodes are numbered in a counterclockwise manner, starting with the corner nodes and followed by the midpoints. Although initial model geometry can have straight or curved edges, the edges of Q8 elements are always treated as straight in EnFEM. Figure 11-3 illustrates the Gaussian integration methods applicable to the Q8 element. The left image depicts the full integration rule, utilizing nine integration points, while the right image shows the reduced integration rule, which employs four integration points. Inside the element, displacements are interpolated quadratically. Field variables, such as stresses and strains, are interpolated using quadratic functions for the full integration scheme and linear functions for the reduced integration scheme. Generally, the reduced integration rule tends to outperform the full integration rule in most solid mechanics applications, offering a balance of accuracy and efficiency while avoiding issues related to shear locking. In the context of EnFEM, Q8 elements are typically favored when contact interfaces are not part of the analysis.

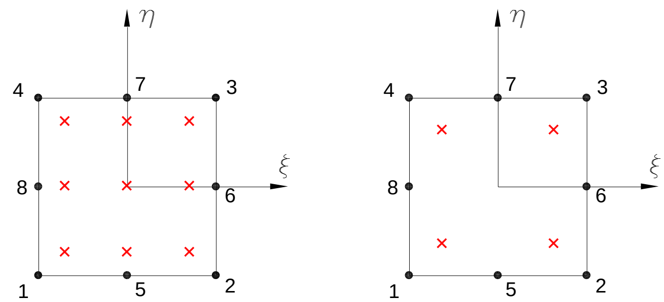


Figure 11-3: Gaussian integration rules for Q8 element

11.1.1.3 Linear Triangular Element - T3

The linear triangular element, referred to as T3, consists of three nodes and three straight edges, with nodes numbered in a counterclockwise fashion. The Gaussian integration method used for this element is depicted in Figure 11-4, where it employs a single integration point. Within the element's domain, both strains and stresses remain constant, leading to the designation of T3 as a "constant strain element". However, T3 elements are prone to shear locking, making them generally unsuitable for analyses dominated by flexural effects. To achieve reliable results in stress analysis when using T3 elements, it is necessary to employ a very fine mesh.

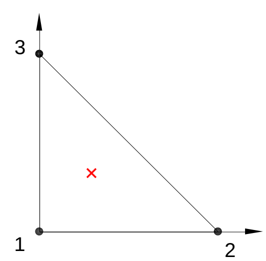


Figure 11-4: Gaussian integration rule for T3 element

11.1.1.4 Quadratic Triangular Element - T6

The quadratic triangular element, known as T6, features six nodes and three straight edges. Nodes are numbered in a counterclockwise direction, starting with the corner nodes and transitioning to the midpoint nodes. Initially, the model geometry can include both straight and curved edges; however, in EnFEM, the FE edges of T6 elements are always rendered as straight. Displacements within these elements are interpolated using a quadratic approach. The Gaussian integration method employed is illustrated in Figure 11-5, utilizing a three-point integration rule. T6 elements generally exhibit significantly improved performance compared to T3 elements, as the stresses and strains inside the element vary linearly.

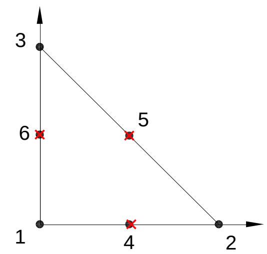


Figure 11-5: Gaussian integration rule for T6 element

11.1.2 Structure Element

To discretize structure entities like retaining walls, culverts or cables, three frame elements are provided in EnFEM: beam element, strut element and geogrid element.

11.1.2.1 Beam Element

A beam element has three nodes. Therefore, interpolation functions are quadratic. This element can simulate straight or curved beams. It has two integration points (marked as red cross in Figure 11-6). Each node has three displacements: translation in x direction, translation in z direction and rotation about y axis. Beam elements in EnFEM use a mixed integration rule to minimize the influence of shear locking. The benefit of using beam elements is that axial forces, shears and moments can be calculated directly from beam deformation. If solid elements are used to simulate a beam, internal forces have to be calculated by section integration inconveniently.

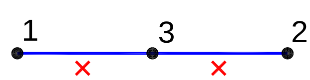


Figure 11-6: Three-node quadratic beam element in EnFEM

Beam elements in EnFEM are primarily used in soil-structure-interaction (SSI) analysis to simulate slender structural members like a sheeting pile or culvert. They can also be used independently to solve mechanical problems of frame analysis. When beams elements are used in SSI analyses, quadratic elements (Q8 or T6) should be used to keep compatibility with the beam elements.

11.1.2.2 Strut Element

The strut element in EnFEM is a two-node truss element. Each node has two freedoms: translation in x direction and translation in z direction. Only axial force is developed for this element. This element can be used to simulate a strut to support a retaining wall, or a tension cable of a tie back structure. It can be defined as tension only member or apply prestressed to it to simulate on-site tie back in more reality. Figure 11-7 illustrates the strut element in EnFEM.

Figure 11-7: A two-node linear strut element

11.1.2.3 Geogrid Element

The geogrid element in EnFEM is a three-node truss element. It is used to simulate the anchor component of a tie back, or synthetic reinforcements to strengthen soil. It is used frequently with beam elements in EnFEM to simulate behavior of a retaining wall. Therefore, has three nodes to be compatible with quadratic solid elements (Q8 or T6) for soil. A geogrid element is always assumed to be bonded with solid elements perfectly. There are no relative displacements developed between nodes of a geogrid element and nodes of a solid element. The available internal force of a geogrid element is the axial force which can be tension or compression. Figure 11-8 shows a typical geogrid element in EnFEM.

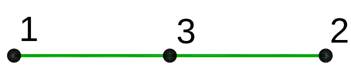


Figure 11-8: A three-node quadratic geogrid element

11.1.3 Structured/Unstructured Mesh for Solid Elements

After defining an element type, users must decide on the structure of the mesh. Both "structured" and "unstructured" meshes are available in EnFEM, and users will define either type for each region in the model. Both mesh types can be combined freely in a model with multiple regions, but only one mesh type may be defined per region. Figure 11-9 shows a sample of structured (at left) and unstructured (at right) meshes. The structured mesh is such that each internal node has a fixed number of neighboring nodes. The unstructured mesh is such that the number of neighboring nodes around an internal node is undetermined.

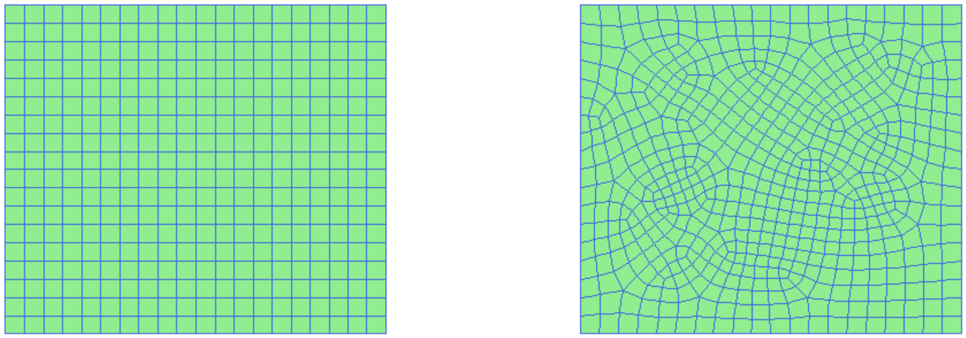


Figure 11-9: Structured and unstructured meshes

In general, finite elements in a structured mesh have less distortion and are considered to be of better quality. In Figure 11-9, two identical rectangles are assigned with the same global FE size. The number of elements in the structured mesh is clearly less than that of the unstructured mesh. A smaller number of elements is one benefit of using a structured mesh, as computation times typically will decrease with less elements. However, a structured mesh is applicable to regional objects which have four sides. Each side can have more than one line segment. Additional requirement is that the number of discretized elements on opposite sides of the region must be equal.

Retaining walls, piles, and/or culverts are frequently found in geotechnical engineering. These structures have slim shapes. Sometimes one side may have several line segments, but the opposite side does not. It is recommended to divide these structures into several small rectangles as shown in Figure 11-10. The retaining wall, made from polygon GFEKLMH (at left), has more than four sides due to the adjacent regions (not shown) representing soil layers. This polygon is divided into three "four-sided" sub-regions - rectangles GFEH, EKNH, KLMN (at right). The structured mesh can then be applied to each of these sub-regions. Additionally, in post processing, a structured mesh is required to plot results of integrated forces, like moments or shear forces.

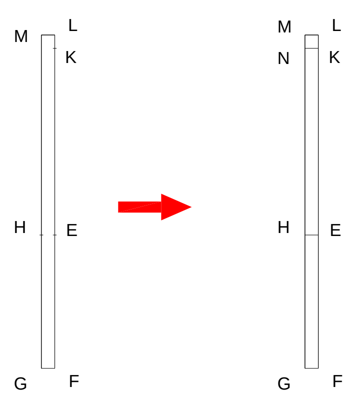


Figure 11-10: Sub-divide a multi-edge polygon into four-sided rectangles

11.1.4 General Mesh Generation Guidelines

To generate high quality meshes in EnFEM, users may follow recommendations here:

1. Use quadrilateral elements (Q4, Q8) as much as possible. If complex geometries are present, they can be divided into smaller regions with more regular shapes. This is helpful to improve mesh quality.
2. If contact analysis *is not present* in the model, it is recommended to use Q8 elements (reduced integration rule) exclusively
3. If contact analysis (contact constraint method) *is present*, it is recommended to use Q4 elements (full integration rule) exclusively. Adjust mesh refinement if necessary.
4. If soil-structural-interaction *is present* and structures are simulated by beam elements, solid regions should be discretized by quadratic elements (Q8 or T6) because beam elements are quadratic.
5. For a structural member in the model, like a retaining wall, a culvert, or a pile, divide the geometry into several "four-sided" subregions, if possible, and set the mesh type for this member as a structured mesh.
6. Refine the mesh at locations suffering from steep stress gradients (stress concentrations). Use coarser mesh at locations far away from the area of interest.

Mesh operations are only available in *"CAD-MODE"* and can be found in the **"pre-Mesh"** drop-down menu on the menu bar. Click  to switch to *"CAD-MODE"* if necessary. To generate a finite element mesh in EnFEM has these steps:

1. Click "**pre-Mesh->Advanced Mesh Control**" to define mesh controls. Select element shape, mesh type, element type, and integration rules
2. Click "**pre-Mesh->Global seed**" to define global seed control. This operation will assign a global element size if a uniform mesh is expected
3. Click "**pre-Mesh->Local Seed Control**" to define local seed controls (optional). Local seed controls can be used to adjust element size, refine bias direction, and refine bias patterns. If local seed control is applied to all region edges, global seed control can be ignored.
4. Click "**pre-Mesh->Show Edge Division**", "**pre-Mesh->Show Seeds**" and "**pre-Mesh->Show Mesh Assignment**" to verify mesh settings
5. Click "**pre-Mesh->Run Mesh Generation**" to generate mesh

11.2 Finite Element Mesh Control

11.2.1 Automatic Mesh Generation

EnFEM offers an automatic mesh generation feature. The element shape, mesh type, and element options are determined by EnFEM. To generate an auto mesh, simply click on the "**pre-Mesh -> Auto Mesh Generation**" menu in "CAD-MODE." The mesh will be created according to the following guidelines:

- If there are no defined contact interfaces between solid finite elements, quadratic triangle elements (T6) will be used, resulting in a uniformly distributed and unstructured mesh
- However, if contact interfaces between solid elements are present, linear quadrilateral elements Q4 with the full Gaussian integration rule will be employed.
- By default, the longest edge will be divided into 30 segments

11.2.2 Advanced Mesh Controls for Solid Elements

The advanced mesh control feature grants users' complete control over the meshing process. It allows for the specification of element shape, mesh type, element type, and integration rules. To access this feature, navigate to "CAD-MODE" in the menu bar and select "**pre-Mesh -> Advanced Mesh Control**." Choose one or several regions, then right-click and select "**End Selection**" from the context menu. This will launch the "Interaction" window (see Figure 11-11).

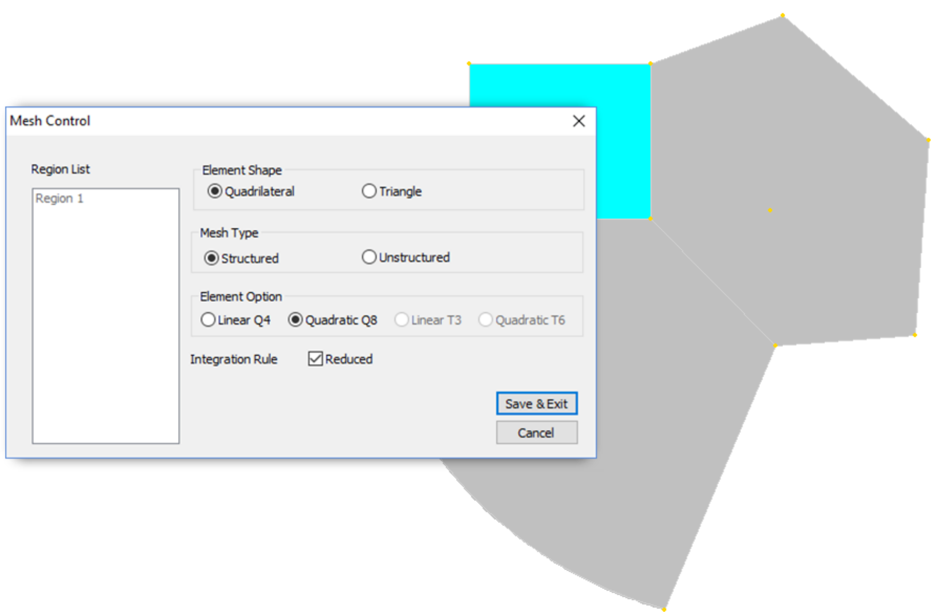


Figure 11-11: Mesh control dialog

In the left panel of this dialog, you will find a "Region List" that displays the selected regions. The region names are automatically generated by the program using the format "Region + number."

If the selected regions do not have predefined mesh properties, they will default to the "Quadrilateral" element with "Quadratic Q8" type. The default mesh type can either be "Structured" or "Unstructured," depending on the geometry's shape. If there are already assigned mesh properties on the selected regions, those mesh settings will be shown:

1. Choose the preferred options for element shape, mesh type, element options, and integration rule, keeping in mind the following guidelines: If the source region has fewer than four edges, only the "Unstructured" mesh type will be available.
2. If you select "Linear Q4" and contact interfaces are present in the model, it is advisable to adopt the full integration rule (ensuring that the "reduced" option is unchecked).
16. For the "Quadratic Q8" option, using the reduced integration rule is generally recommended.
17. Note that there is no integration option for the triangular element shape.

The mesh generator in EnFEM allows for the mixture of both structured and unstructured meshes within the same model. However, there are limitations to keep in mind during the mesh generation process:

1. Quadratic elements (Q8, T6) cannot be mixed with linear elements (Q4, T3)
2. To create a structured mesh, the region of interest must be a four-sided area with equal division numbers on opposite sides. Each side may have more than one segment.
   a. NOTE: There is no edge number limitation for an unstructured mesh.
18. EnFEM does not support a combination of quadrilateral and triangular elements.

11.2.3 Global Seed Control

Once the advanced mesh control settings have been established, additional steps are required to define the element size along the edges of each region. In EnFEM, this is achieved by adjusting the "seed spacing" on the perimeter of the regions. It's important to note that seed spacing cannot be regulated within the interior of a region. During the discretization process, a finite element (FE) node is placed at each seed position.

The " Global Seed Control " function allows you to apply a consistent element size across all regions in the model, ensuring a uniform mesh. To access the "Global Seed Control" dialog, navigate to the menu bar and select "**pre-Mesh -> Global Seed Control**" as illustrated in Figure 11-12.

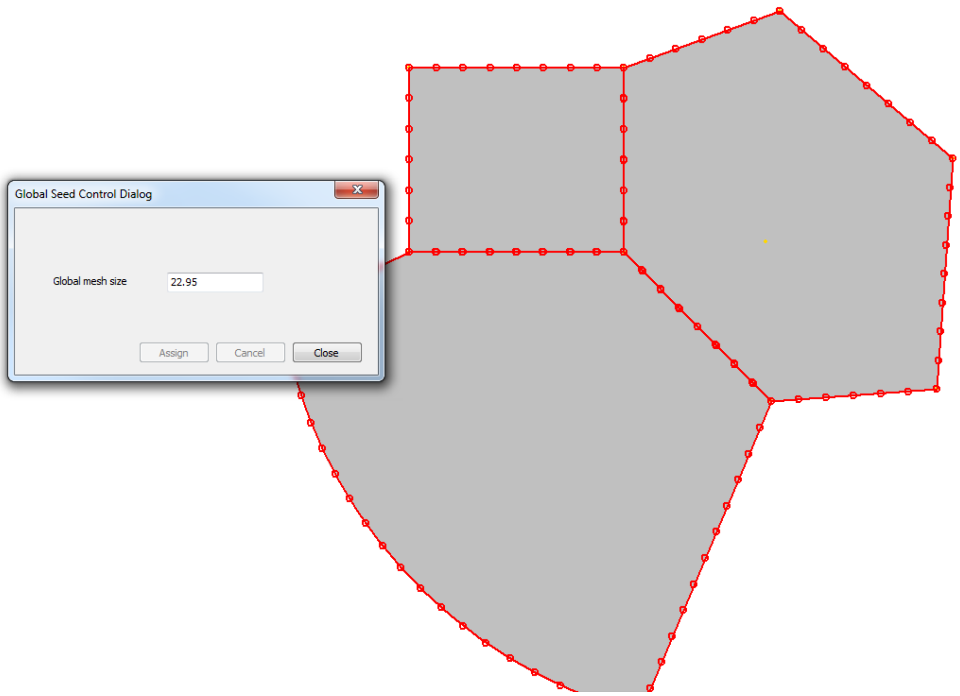


Figure 11-12: Global seed control

The default global mesh size is determined from the longest edge divided by 30.

Users can modify the default global mesh size by entering a specific value of their choice. After specifying the desired element size, click “**Assign**” to apply it to all regions. You can verify the distribution of red circles along each edge, which indicates the locations of the seeds (see Figure 11-12). To exit the dialog, click “**Close**”.

11.2.4 Local Seed Control for Solid Elements

Practical geotechnical challenges are often complex due to highly nonlinear mechanical behaviors, localized stress concentrations, and considerable differences in dimensions. Using a uniform mesh can lead to element sizes that are insufficient to capture the resolution needed in areas of stress concentration. On the other hand, employing an excessively fine uniform mesh can significantly extend computation time. To address this issue, local mesh control is beneficial. By implementing targeted local seed controls, a refined mesh can be generated that smoothly transitions from finely meshed regions to coarsely meshed areas, all while minimizing any increase in computation time.

To access local mesh control, navigate to the menu bar and select “**pre-Mesh -> Local Seed Control**.” Choose the desired edge(s), then RIGHT-CLICK and select "**End Selection**" from the popup menu to open the “Local Mesh Control” window (see Figure 11-13). Here, you will define options for “Seed Controls,” “Number of Divisions,” “Bias Direction,” and “Seed Distribution Style.” You can also view existing “Seed Information” if the selected edge has already been defined.

The input parameters available in this dialog will vary depending on your selections for "Seed Controls" and "Seed Distribution Style." For instance, when "Seed Control" is set to "By Number," the input for "Number of Divisions" will change based on your choice of "Seed Distribution Style." Similarly, this applies when "Seed Control" is set to "By Size" and the "Seed Distribution Style" option is modified.

Once you have made your selections and entered the necessary input, click "**Assign**" to apply these settings to the model. After completing the process, click "**Close**." You can repeat this procedure for any additional edges as needed. The following sections provide a more detailed discussion of these settings

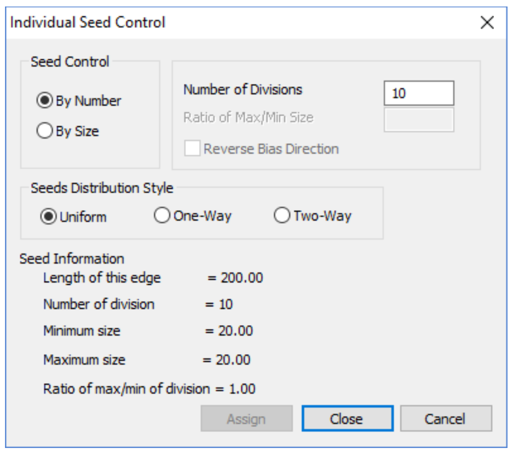


Figure 11-13: Individual seed control dialog

11.2.4.1 Seed Control and Sizing Options

There are two primary options for seed control: "By Number" and "By Size." Additionally, sizing controls include parameters such as "Number of Divisions," "Uniform Size," "Ratio of Max/Min," "Minimum Size," and "Maximum Size." It's important to note that the available sizing controls may differ depending on your choice of "Seed Control" and "Seed Distribution Style."

11.2.4.2 Seed Distribution Style

There are three options for seed distribution style: "Uniform," "One-Way," and "Two-Way"" (see Figure 11-14). The "Uniform" distribution style ensures equal spacing of seeds along the selected edges. The "One-Way" distribution style decreases seed spacing in a specified direction. The "Two-Way" distribution style adjusts seed intervals towards the center of the edge or towards both ends, based on the chosen "bias direction."

Note: When multiple edges are selected simultaneously, only the "Uniform" style will be available for local seed controls. You can use the keyboard shortcut "**CTRL+L**" to restart the "Local Seed Control" operation for further definitions.

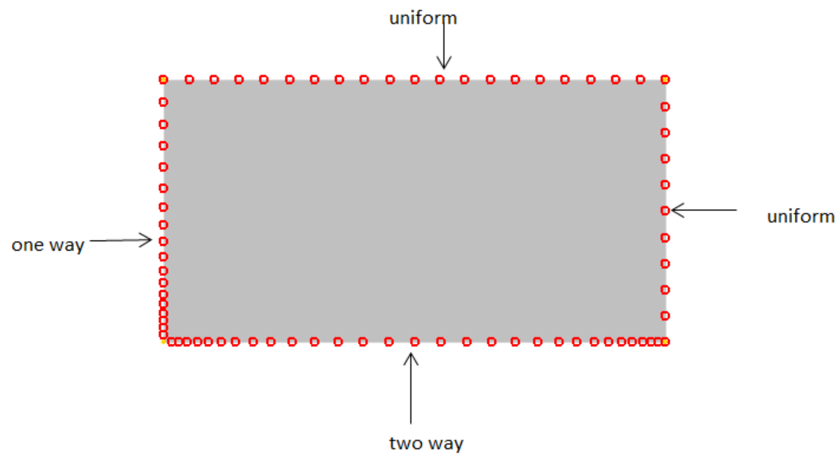


Figure 11-14: Seed distribution style in EnFEM

11.2.4.3 Bias

Both the “One-Way” and “Two-Way” distribution styles incorporate a directional bias, which means that seed spacing is minimized in a specified direction along the edge. The “One-Way” distribution style reduces seed spacing solely along the defined direction, while the “Two-Way” distribution style can decrease seed intervals toward the center of the edge or toward both ends, depending on the selected “bias direction.” When either of these styles is activated, red-highlighted arrows will appear on the selected edge to indicate the refining direction(s). If the arrows indicate the opposite direction of what is intended, the user has the option to check the box for “Reverse Bias Direction,” which will then adjust the arrow display to point in the opposite direction.

11.2.4.4 Example

A straightforward example illustrated in Figure 11-15 demonstrates the use of local mesh controls to refine the mesh around a specific vertex—Vertex B, in this instance (shown in the left image). The right image displays the final seed distributions, which are tightly refined around Vertex B and gradually transition to a less refined distribution outward. It is important to maintain consistent seed spacing around the vertex of interest to achieve high-quality meshes. When dealing with local stress concentrations, applying local refinement is crucial for minimizing the stress gradient.

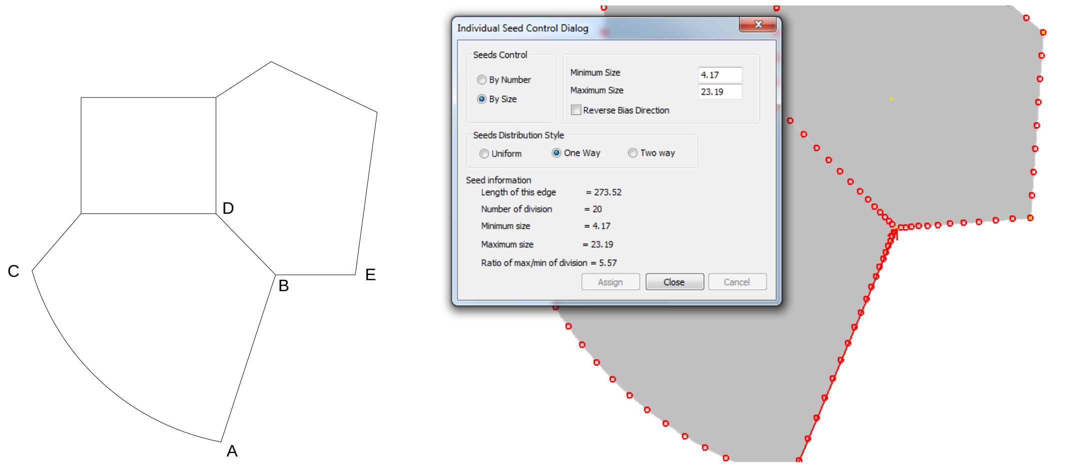


Figure 11-15: Local mesh refinement around Vertex B

11.2.4.5 Information of Seed Distribution

After assigning local seeds to an edge, or when selecting an edge that already has a defined seed, the calculated "Seed Information" will be displayed at the bottom of the “Individual Seed Control” dialog window (see Figure 11-13). Keep in mind that the calculated minimum and maximum element sizes may not perfectly match the values entered by the user.

11.2.5 Seed Control on Structure Elements

In EnFEM, the method for assigning seed control to a frame entity mirrors the process used for a region edge. Seed control is specifically required for beam and geogrid entities, while strut elements are automatically managed by EnFEM, eliminating the need for user involvement in strut discretization.

The global seed size setting applies to both beams and geogrids. To maintain compatibility between beams and region edges, it is recommended to select both region edges and overlapping beam edges simultaneously during the local seed control operation.

If a geogrid is located within a region, it can receive seed control through either global seed control or local seed control, depending on the modeling requirements. This flexibility ensures that the mesh remains consistent and accurate while adapting to the specific needs of the simulation

11.2.6 Other Mesh Tools

EnFEM provides some convenient tools to let users verify seed distributions and predict mesh quality before generating the mesh. To avoid any unexpected errors, it is recommended to review mesh definitions and properties using the following tools, before trying to generate the mesh.

11.2.6.1 Show Edge Division

To display division numbers on all edges, go to the menu bar and select "**pre-Mesh -> Show Edge Division**," as illustrated in Figure 11-16. You can toggle the edge divisions off by using the same command again. This feature is particularly beneficial when working with a structured mesh, as it ensures that opposing edges have matching division counts, which is essential for maintaining the structure of the mesh.

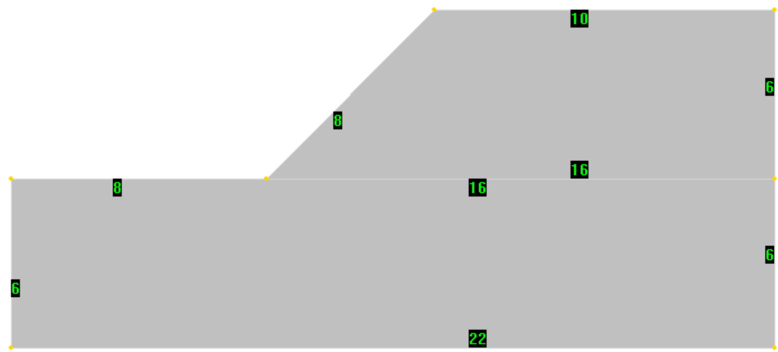


Figure 11-16: Division numbers displayed

Once the global seed control is established, the program calculates a division number for each edge. Typically, longer edges will have higher division numbers compared to shorter ones, as illustrated in Figure 11-17. However, this can create conflict with the requirements of a structured mesh. To resolve this issue, it is advisable to display the edge divisions and utilize local seed controls to ensure compliance with the mesh structure.

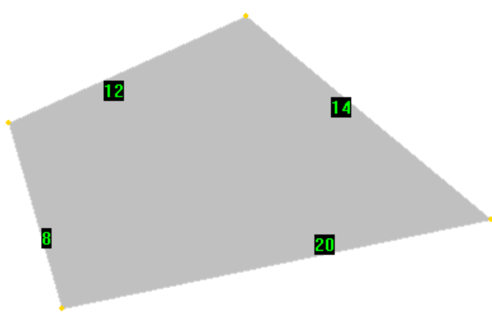


Figure 11-17: Incompatible division number for a structured mesh

11.2.6.2 Show Seeds

To display all specified seed distributions, go to the menu bar and select "**Mesh -> Show Seeds**," as shown in Figure 11-18. Using the same command again will turn off the seed visualization. This feature is helpful for verifying that the seed distributions are defined as intended.

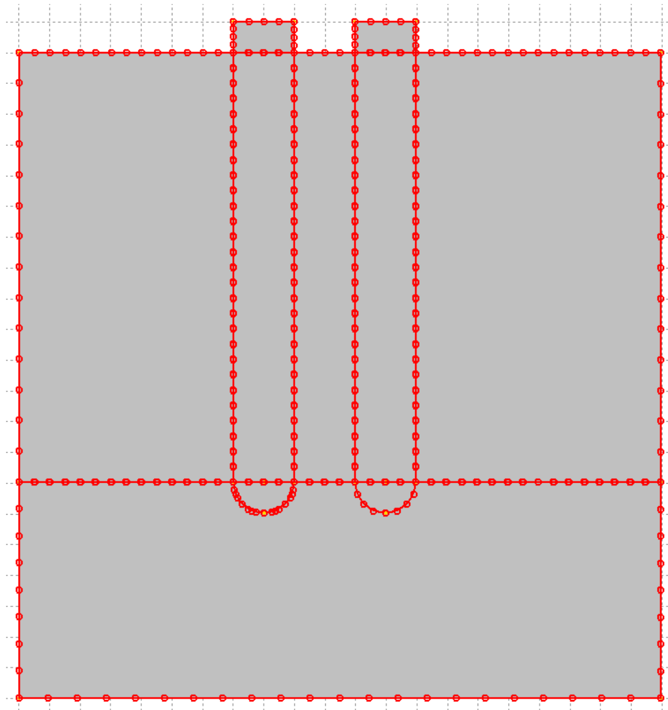


Figure 11-18: Display seed distribution

11.2.6.3 Show Mesh Assignment

From the menu bar, click **"pre-Mesh->Show Mesh Assignment"** to show all specified mesh property assignments. Following the same command, will turn off the assignment visual. This command shades regions in specific colors according to the assigned mesh properties, as seen in Figure 11-19. This provides another visual verification tool to confirm mesh properties are defined as expected, before the mesh is generated.

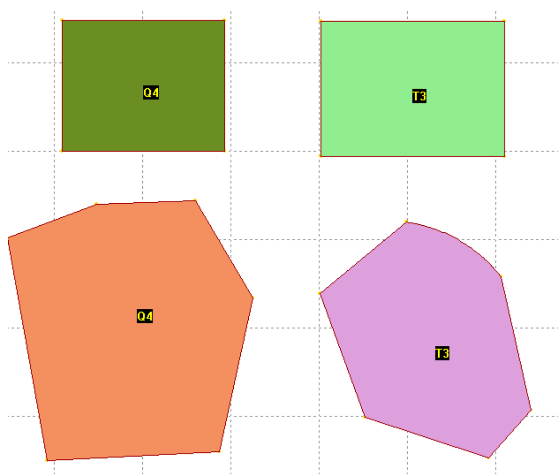


Figure 11-19: Display mesh assignment in color

Four colors are used to shade regions according to their mesh assignment:

- quadrilateral and structured
- triangle and structured
- quadrilateral and unstructured
- triangle and unstructured
- no mesh property defined

11.3 Special Cases

11.3.1 Mesh Control on Contact Surfaces

Contact surfaces necessitate careful consideration when establishing local seed control. The following general guidelines apply:

1. For curved interfaces, conforming mesh arrangements should be utilized Figure 11-21.
2. Conversely, for straight interfaces, it is strongly advised to use conforming meshes.
3. The element size on the slave surface must not exceed the element size on the master surface.

Figure 11-20 depicts a curved interface with unmatched meshes, for a retaining wall and soil interface. Since linear elements (Q4, T3) were used to discretize the interface, gaps are unavoidable in the initial undeformed configuration. Initial contact detection fails to find active contact pairs due to the existence of gaps, and thus the retaining wall will miss contact constraints. Because of this failure, some elements may suffer from rigid body motions and the computation may fail to converge no matter how small the load increment size.

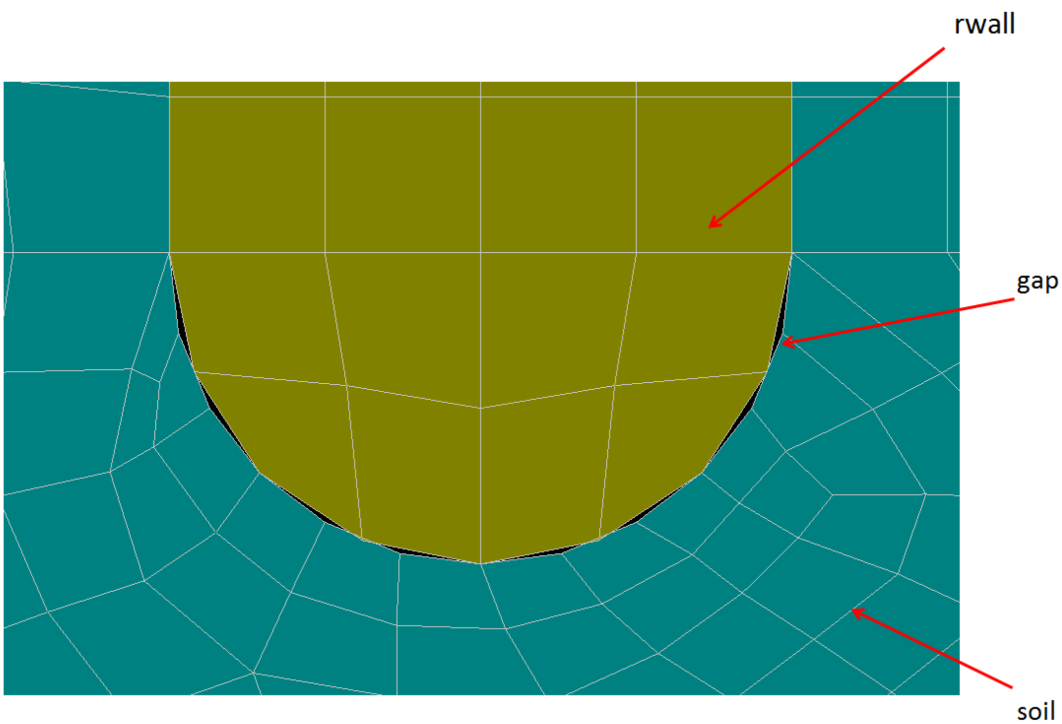


Figure 11-20: Non-matched mesh on curved interface

Figure 11-21 shows a matched mesh on the same retaining wall-soil interface. Initial contact detection will not fail with this interface.

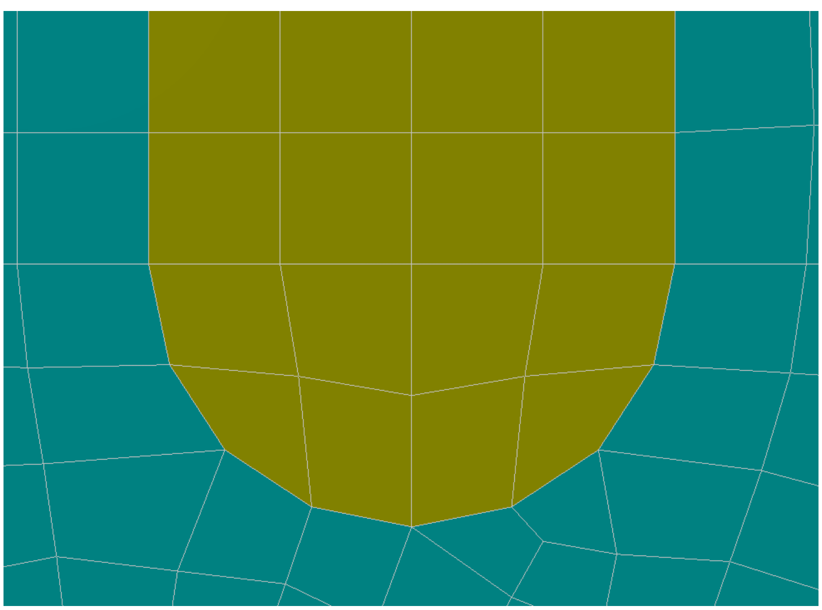


Figure 11-21: Matched mesh on curved interface

As discussed in previous sections, the interaction problem-solving algorithm in EnFEM utilizes a "Node-to-Segment" approach. In this context, the "node" refers to the slave body, while the "segment" pertains to the master body. The algorithm frequently checks for the penetration of the "slave node" into the "master segment" to maintain impenetrability between the contacting bodies. However, it is important to note that the algorithm does not check for the penetration of a "master node" against the "slave body." Consequently, if the element size of the "master body" is smaller than that of the "slave body," it may lead to the unrecognized penetration of the "master body" into the "slave body." Therefore, it is strongly recommended to implement matched meshes at the contact interfaces, regardless of whether these interfaces are curved or straight. The advantage of using matched initial meshes include:

1. easy to adjust mesh
2. easily satisfy mesh requirement for "master - slave" contact pair
3. combined with uniform seed distribution, contact stress has less oscillation

Figure 11-22 shows another conforming mesh on retaining wall-soil interfaces on the vertical edges. This meshing strategy is highly recommended in EnFEM.

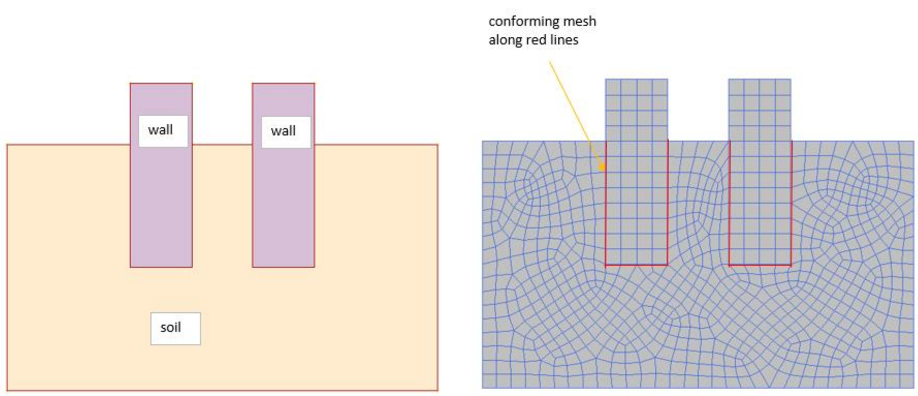


Figure 11-22: Conforming mesh on contact interfaces

11.4 Manual Mesh Generation

Once all parameters are defined for the mesh, click **"pre-Mesh->Run Mesh Generation"** from the menu bar to start the men generator. If this mesh is successfully generated, it will be displayed automatically, and the program will switch to *"MESH-MODE"* (Figure 11-23).

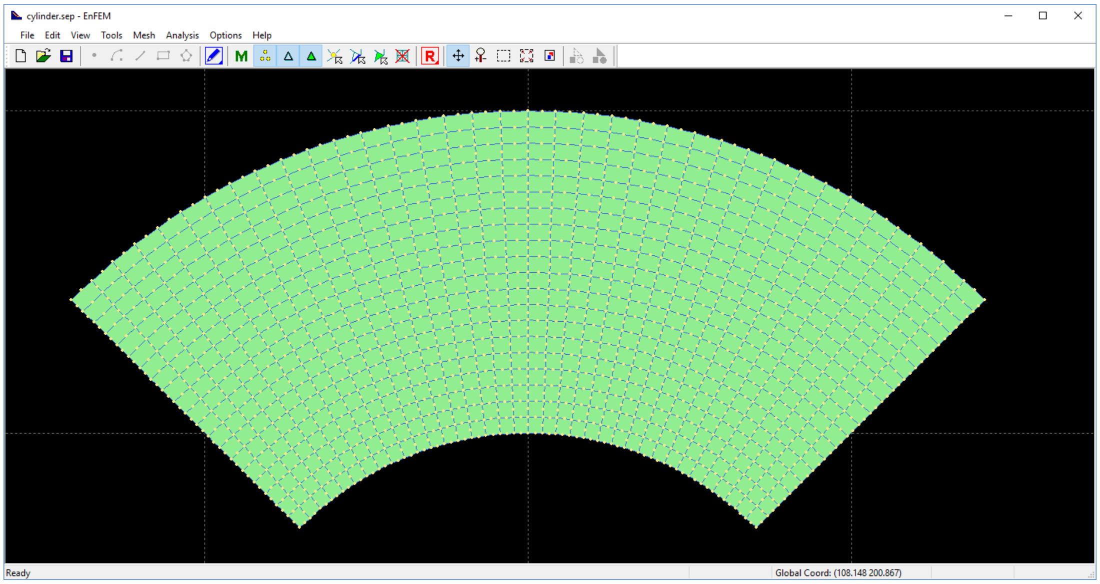


Figure 11-23: Mesh-mode interface (with nodal display on)

11.4.1 Unsuccessful Mesh Generation

If mesh generation fails, you may receive warning or error messages that provide details about the problem. Please make any necessary adjustments and attempt to regenerate the mesh. The following sections outline common issues that can occur during the mesh generation process.

11.4.1.1 Edge Division Numbers for Structured Mesh

If a region is assigned to be a structured mesh, and opposite sides of the region have different edge-divisions, an error message will appear like that seen in Figure 11-24.

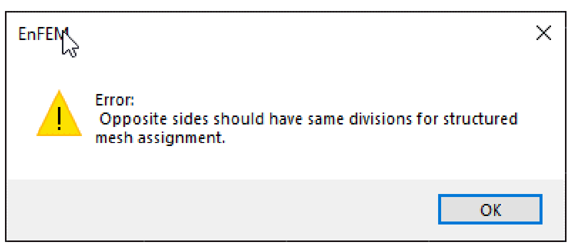


Figure 11-24: Edge-division error message for structured mesh

11.4.1.2 Overlapping Edges

Another common issue for new users is geometry created with invalid overlapping edges. Refer to Section 3.6 for reference on valid and invalid geometries, and best practices. If overlapping edges are present, this will generate a non-conforming mesh, which in turn may produce undesired analysis results. Before the mesh generation is complete, the program will check for these kinds of invalid edges and an error message will appear (like that seen in Figure 11-25) if present. EnFEM also will mark these invalid overlapped edges in red color for easy identification.

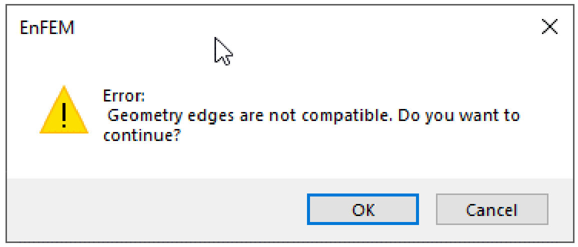


Figure 11-25: Invalid edge error message

11.5 Mesh-Mode

Once the mesh is successfully generated, EnFEM automatically switches from "CAD-MODE" to "MESH-MODE". "MESH-MODE" is a transition before running the analysis but still has some useful features for the user. From the menu bar, under the "Mesh" menu the following features are available.

1. **"Mesh->Delete Mesh"** to directly delete the generated mesh.
2. **"Mesh->Mesh Properties"** displays the current number of elements and nodes (Figure 11-26).
3. **"Mesh->Show Node Numbers"** displays node number assignments.
4. Select again to turn off display.
5. **"Mesh->Show Element Numbers"** displays element number assignments.
6. Click again to turn off the display.

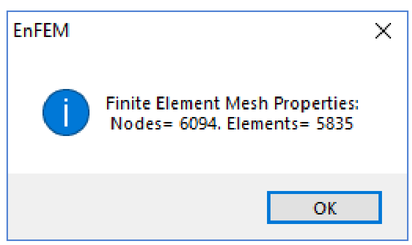


Figure 11-26: Mesh properties

In addition, in "MESH-MODE" (or in "CAD-MODE") the analysis settings can be defined and the analysis started, as noted in CHAPTER 12.

11.6 Probe an Element in Mesh-Mode

EnFEM provides a useful tool to check the element number or node name by clicking a solid element or a frame element in "MESH-MODE". Click icon  to switch to "MESH-MODE". Click menu "**Mesh->Probe Element**", select the type of element to be investigated (click icon , ,  or  to activate desired element type), click an element to view element number and node connection on screen (Figure 11-27). The yellow number on the left image is the element number of a solid element. The green number on the right image is the number of a beam element. Node numbers are displayed in the cyan color.

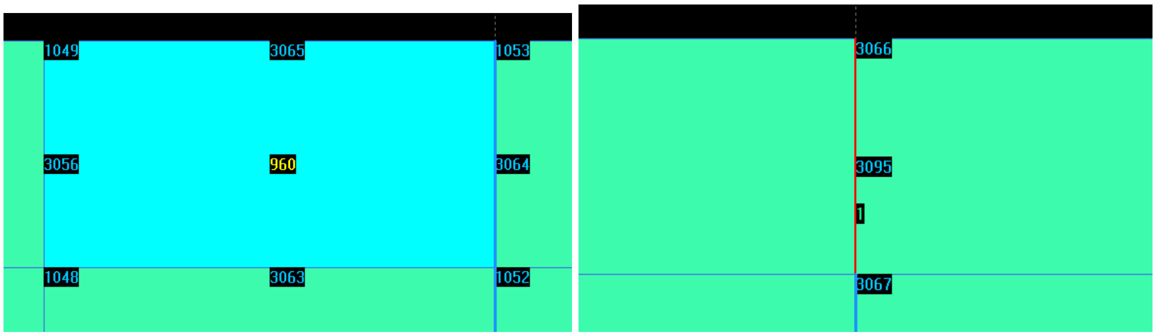


Figure 11-27: Probe a solid element (left) or a beam element(right)

This tool is specifically useful when users check element connections on the contact interface or the zero-thickness interface. Sides of finite elements overlap with each other on the interaction interface, it will be very hard to identity element connection just from command “Show Element Number” or “Show Node Number”.

CHAPTER 12. Analysis

12.1 Introduction

Nonlinear analysis through finite element methods in geotechnical engineering can be quite complex. There are instances when calculations fail to converge, leading to early termination, or when they take a significant amount of time to complete. EnFEM includes default solver settings that are suitable for a majority of applications; however, seasoned users may wish to modify these settings for more advanced analyses. To accommodate this, EnFEM allows users flexibility to adjust solver settings as needed, enhancing convergence rates.

12.2 Analysis Settings

From the menu bar, click **"Analysis->Analysis Setting"** to open the *"*Analysis Setting*"* window, as shown in Figure 12-1. Default settings are automatically entered in each field and may be changed as desired. The specific setting options are as described in the following sections.

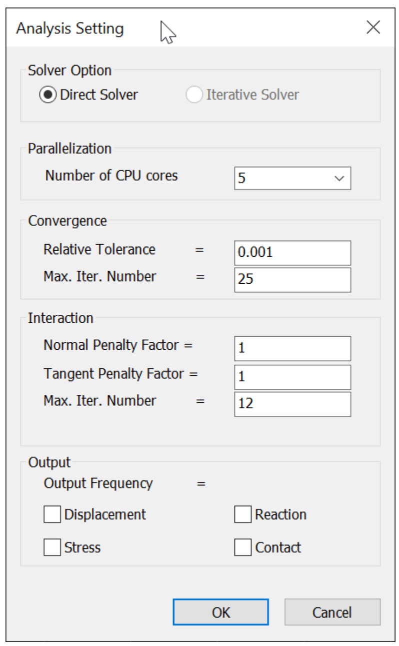


Figure 12-1: Analysis setting

12.2.1 Solver Options

EnFEM uses the direct solver for the solution of the system of equations. The solver is based on multithreading and parallelization techniques that usually reduce computational time in multi-core systems. EnFEM will detect and use available CPU cores automatically.

12.2.2 Parallelization

This option is used to set how many physical CPU cores are used in analysis. The default number is the maximum physical CPU cores minus-1.

12.2.3 Convergence Options

The convergence settings consist of two parameters: “relative tolerance” and “maximum iteration number.” The “relative tolerance” is defined as the ratio of the force residual to the total applied external loads. The force residual, obtained from each finite element, represents the difference between the internal nodal forces and the external loads. The default value of 1e-3 strikes a balance between accuracy and efficiency, making it suitable for most calculations. Users have the option to increase or decrease this value if they wish; however, selecting a smaller value may require additional iterations.

The “maximum iteration number” sets the upper limit on the number of iterations permitted for each load increment. If the actual number of iterations exceeds this limit, the program will indicate a convergence failure and attempt to resolve the issue by using a smaller load increment. The default limit of 25 iterations is generally effective for most scenarios.

12.2.4 Interaction Controls

The interaction settings for contact analysis include three parameters: “normal penalty factor,” “tangent penalty factor,” and “maximum iteration number.” EnFEM uses the penalty method for contact analysis, requiring an initial penalty value estimation before computations. Typically, the program's default penalty value is suitable for most applications. However, users can adjust the “normal” and “tangent” penalty values by applying a multiplication factor to the default value.

The “maximum iteration number” limits iterations to determine contact status. A new iteration begins if the contact status changes, continuing until the force residual falls below the tolerance and the contact status stabilizes. The default iteration limit for interaction checks is 12. For interaction interfaces involving many finite element nodes, where contact status changes extensively, the default setting may not converge. In such cases, the program reduces the increment size and retries with the smaller value.

Often, only the first few increments require additional iterations for contact status convergence. Increasing the maximum iteration number beyond 12, rather than letting the program reduce the increment size, can save CPU time.

12.2.5 Output Options

EnFEM uses a smart output control to save time and storage space of results. *“Output frequency”* controls how often to write calculated results to the file on the hard drive. This option is effective if a small increment size is being used, thus creating a large total number of increments. Results of the first and the last increments are always output.

Four output contents are available in EnFEM: "displacements", "boundary reactions", "element stresses" at Gaussian points, and "contact stresses". Users may "check" the content of interest desired for output.

12.3 Run Analysis

In "CAD-MODE" or "MESH-MODE", click **"Analysis->Run Analysis"** from the menu bar to open the "Run Analysis" window, as shown in Figure 12-2.

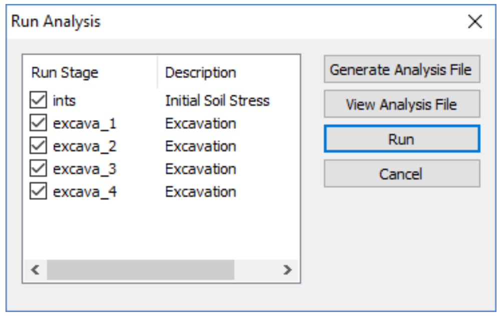


Figure 12-2: Job submission and options

All available stages of the analysis model will be displayed in this dialog window. Only stages with a check mark next to the name will be submitted to the solver, while those without the check mark will not be run.

12.3.1 Generate Analysis File

The button "Generate Analysis File" can be used to generate the solver input file (.dat) in ASCII format. This is a readable file primarily used for technical support purposes. Experienced users may use this file to track possible issues easily.

12.3.2 View Analysis File

The button "View Analysis File" will open this analysis file (.dat) automatically using the Windows Notepad program, for user review.

12.3.3 Run Analysis

Clicking "**Run**" button will submit the job to the nonlinear solver. A command window will pop up to display convergence information of each load increment. All information displayed in the command window, shall be saved to the "log-file" (.log). If the user wishes to terminate the analysis early, the keyboard shortcut **"CTRL+C"** may be used within the command window, or the command window can be closed directly by clicking the  button. When solver finishes the analysis, or terminates the analysis because of an error, the "log-file" (.log) will be opened automatically. It is recommended to review the history of convergence in this log-file. If any errors have occurred and EnFEM fails to generate the analysis input file, an error message will be produced, like that seen in - Figure 12-3. Related information can be found in the ".log file".

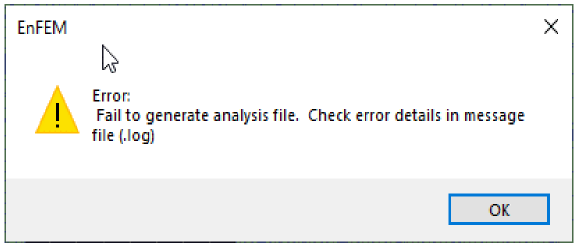


Figure 12-3: Analysis file error message

The output file (.out) and the log file (.log) are both available for users' review. They can be reached from the menu bar, by clicking **"Analysis->View Output File"**, or **"Analysis->View Message File"**, respectively.

12.3.3.1 Computation Result

In general, the computation will end in two ways – either a successful completion of the analysis, or an early termination.

12.3.3.1.1 Successful Computation

In this case, the analysis successfully ends without issue, or any error messages. Thus, users can go into post-processor directly to view the desired results.

12.3.3.1.2 Early Termination

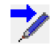
In this case, the analysis aborts due to errors. This case is common in nonlinear FE simulation. There are many reasons that may cause an abnormal abortion, such as: large load-increment size, local failure of material, non-convergence of a contact interface, or the model itself has some implicit errors. In this case, users need to carefully check the completed output, including the ".log file". If any partial results are available, judgment of the results based on engineering experience is necessary to find errors and make any needed corrections.

CHAPTER 13. Post-Processing of Results

13.1 Introduction

Visualization of results is generally known as post-processing in finite element analysis. Post-processing in EnFEM has four types of data visualization: "contour plots", "displacement vectors", "X-Y curve plots", and "cutline plots". These options are described in more detail in the following sections.

13.1.1 Result-Mode

Post-processing in EnFEM takes place in the "RESULT-MODE". With an analysis previously completed, users will click the  icon to switch into "RESULT-MODE". If results are available, the initial mesh will be plotted on screen (Figure 13-1). Users may click  or  to go back to "CAD-MODE" or "MESH-MODE", at any time.

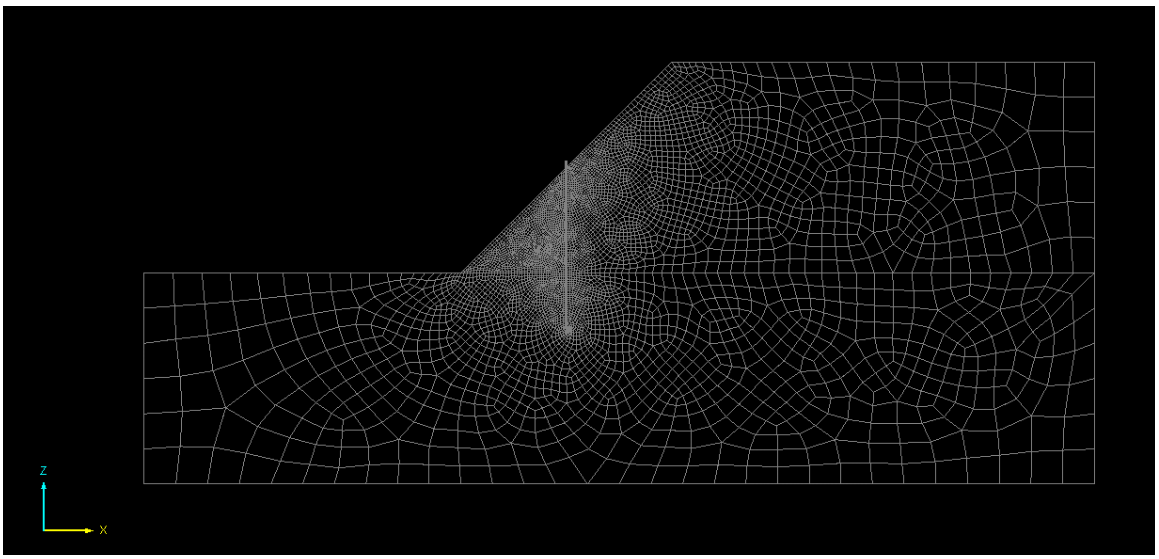


Figure 13-1: Initial "Result-mode" display

13.1.2 Stage Selection

-Before generating any results, users have the option to choose specific stages for output. To do this, navigate to the menu bar and click on "**Results -> Stage Selection**". This will open the "Stage Selection" window, as illustrated in Figure 132. In this dialog, stages that are marked as "checked" will be available for output operations, while those that are "unchecked" will be excluded. This feature is particularly useful for XY-Plots; for instance, if you want to plot only the loading stages and omit the initial stress stage. By default, all stages are selected for output. Additionally, columns following the stage name display details such as "start increment," "end increment," and "stage type".

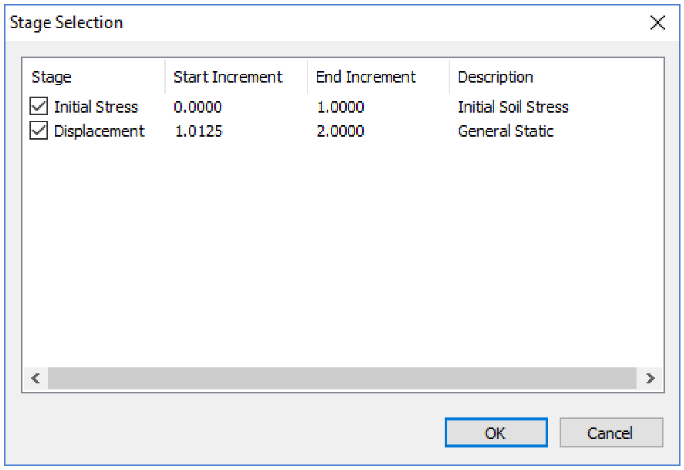


Figure 13-2: Select stages for output

13.2 Post-Processing Options

13.2.1 Contour Plot

The contour plot in EnFEM is based on Open-GL graphics technology which provides fast, accurate rendering of contour outputs on screen. Contour plots of field variables give users a visible color-based image to easily identify trends in displacement, local stress/strain concentrations, local failure of material, among others. From the toolbar, click the contour plot icon, , and the *"Contour Plot"* window will open, as shown in Figure 13-3.

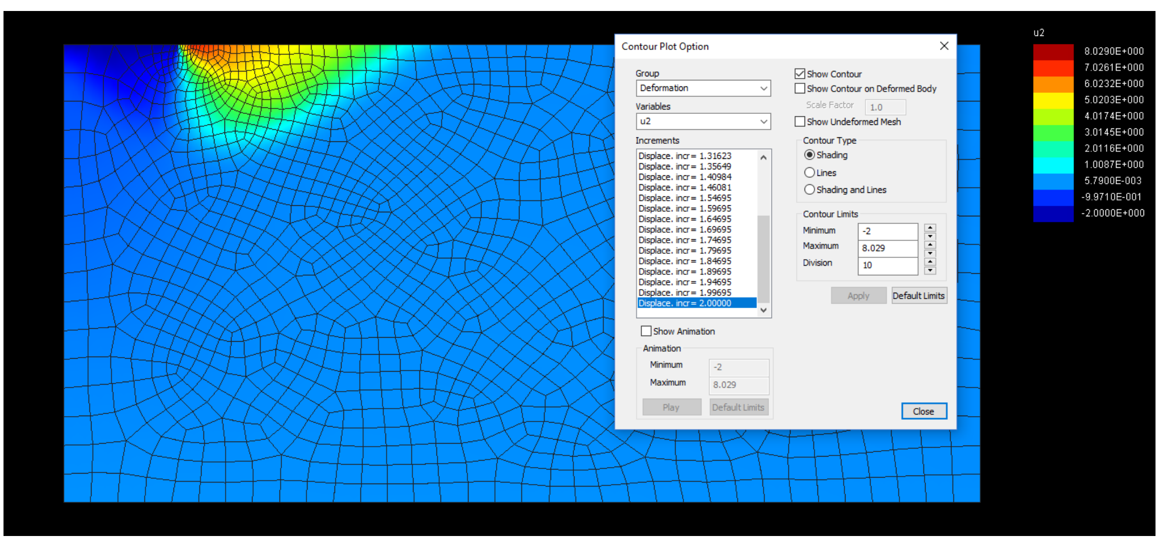


Figure 13-3: Dialog of contour plot

Variables available for display in the contour plot are divided into four groups: "Deformation", "Stress", "Strain", and "Soil variables". The contents of each group are listed as follows:

13.2.1.1 Deformation

1. *u1*: displacement component in X-direction (horizontal)
2. *u2*: displacement component in Z-direction (vertical)
3. *u_mag*: displacement magnitude

$$u_{mag} = \sqrt{u_{11}^2 + u_{22}^2}$$

(13.2-1)

13.2.1.2 Stress

1. σ_{11}, or S11: stress component in X-direction (horizontal)
2. σ_{22}, or S22: stress component in Z-direction (vertical)
3. σ_{12}, or S12: shear stress component
4. σ_{33}, or S33: stress component in Y-direction (perpendicular to screen)
5. mean stress *P*:

$$P = \frac{\sigma_{11} + \sigma_{22} + \sigma_{33}}{3}$$

(13.2-2)

6. equivalent shear stress *q*

$$q = \sqrt{3J_2} = \sqrt{\frac{(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6\sigma^2{}_{12}}{2}}$$

(13.2-3)

7. major principal stress, P_1, or PS1
8. minor principal stress, P_2, or PS2

13.2.1.3 Strain

1. ε_{11}, or E11: component of total strain in X-direction (horizontal)
2. ε_{22}, or E22: component of total strain in Z-direction (vertical)
3. ε_{12}, or E12: component of shear strain
4. ε_{33}, or E33: component of total strain in Y-direction (perpendicular to screen)
5. *EEQ*: equivalent magnitude of total strain

$$\varepsilon_{eq} = \frac{\sqrt{3}}{2}\sqrt{(\varepsilon_{11} - \varepsilon_{22})^2 + (\varepsilon_{22} - \varepsilon_{33})^2 + (\varepsilon_{33} - \varepsilon_{11})^2 + 3\varepsilon_{12}^2}$$

(13.2-4)

6. ε_{11}^{p}, or PE11: component of plastic strain in X-direction (horizontal)
7. ε_{22}^{p}, or PE22: component of plastic strain in Z-direction (vertical)

8. ε_{12}^{p}, or PE12: component of shear plastic strain
9. ε_{33}^{p}, or PE33: component of plastic strain in Y-direction (perpendicular to screen)
10. ε_{eq}^{p}, or PEEQ: equivalent magnitude of plastic strain

$$\varepsilon_{eq}^{p} = \frac{\sqrt{3}}{2}\sqrt{\left(\varepsilon_{11}^{p} - \varepsilon_{22}^{p}\right)^2 + \left(\varepsilon_{22}^{p} - \varepsilon_{33}^{p}\right)^2 + \left(\varepsilon_{33}^{p} - \varepsilon_{11}^{p}\right)^2 + 3\left(\varepsilon_{12}^{p}\right)^2}$$

(13.2-5)

13.2.1.4 Soil variables

1. Pore pressure
2. Consolidation stress, P_c
3. Void ratio:
4. Active yielding

Clicking on any item of "Variable" will automatically refresh the contour plot. The list under the label "Increments" shows the available increments for selected stages. Selecting any of the increments will also refresh the contour plot. There are several display options in the top-right part of the window. Options for displaying the contour, the deformed shape with scaling factor and the undeformed mesh, as well as various contour types and editing numerical limits are available to the user.

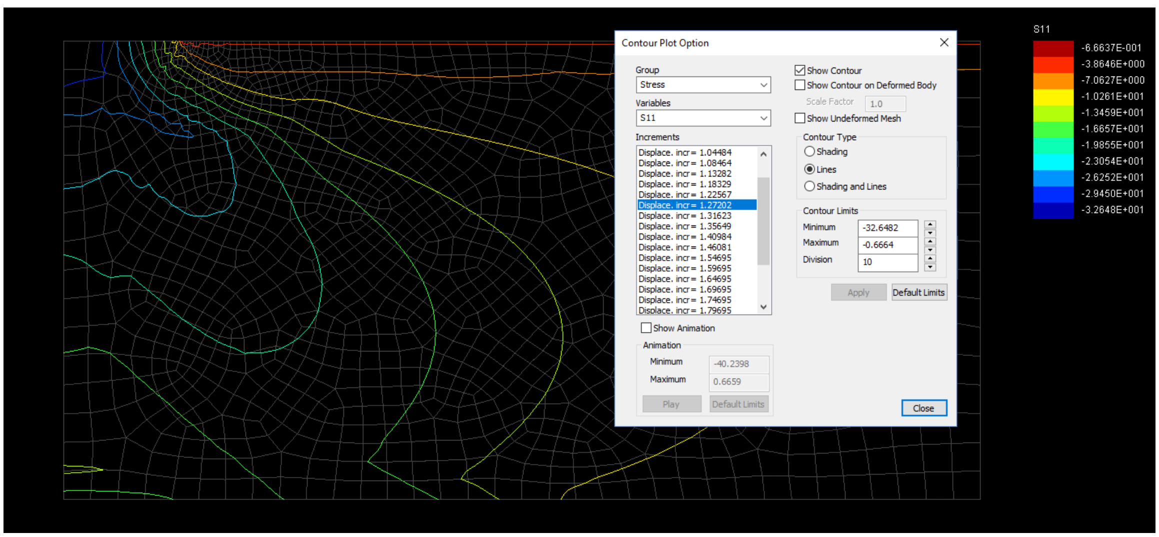


Figure13-4: Plot contour as line

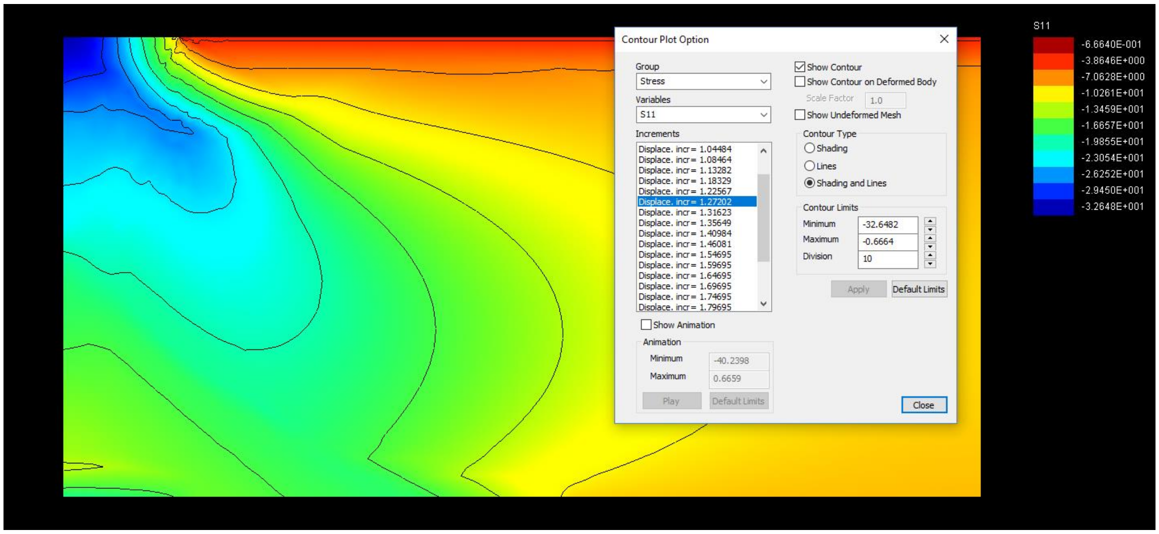


Figure13-5: Plot contour to show both shade and line

Three different options of "contour type" are available: "shading" (default), "lines", or both "shading and lines". Figure13-4 shows an example of line contours, whileFigure13-5is an example of the combined option.

Users can change the "contour limits" as desired by entering new minimum or maximum values, as well as the number of contour divisions. Click "**Apply**" to set the new limits and notice that the contour scaling reflects these changes. The button "Default Limits" is used to restore the original default contour limits, if necessary. Below the "Increment List", checking the box **"Show Animation"**, enables controls for an animated display of the contour results. Click the button Play, to play the contour animation. The animation will play once, from start to finish. If desired, contour limits for the animation may be changed using the **"Minimum/Maximum"** fields This can be repeated as necessary.

Click **"Close"** to exit the "Contour Plot" window. After exiting, the contour image will be cleared from the screen.

13.2.2 Incremental Contour Plot

The contour plot described in the previous section plots the total value of a variable, such as the total displacement magnitude at a specific load increment. Sometimes the change of a variable between two load increments may be helpful. For example, the change of plastic strain between of one load increment is useful to determine if a slope will fail soon.

In "RESULT-MODE", click menu "**Results->Increment Contour**", or click the icon . This increment contour plot in EnFEM looks like that in Figure 13-6. It has two increment history lists. Users can select any two different load increments to plot a contour.

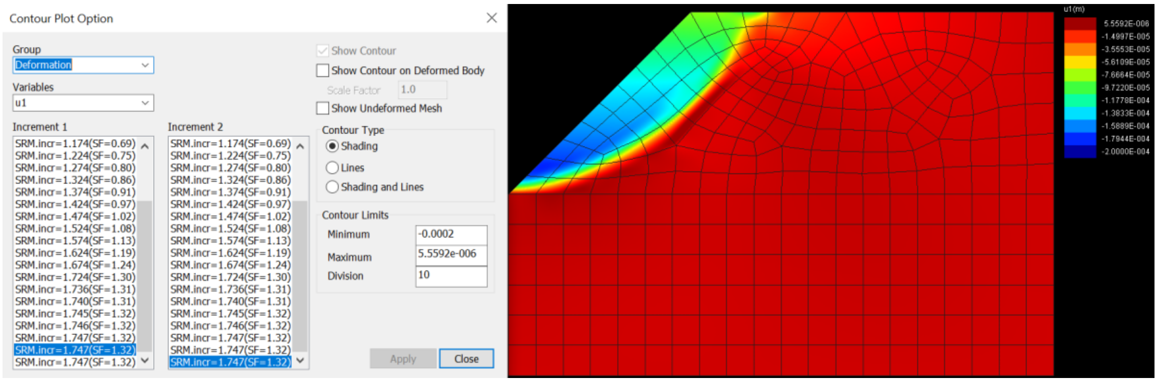


Figure 13-6: Incremental Displacement contour of the last load increment in example 3

13.2.3 Plot Displacement Vector

The displacement vector plot is a way to clearly illustrate displacement direction and magnitude of material parcels. From the toolbar, click the  icon to open the "Displacement Vector" window, as shown in Figure 13-7.This figure shows the displacement vectors on a foundation settlement example from *Example 2 - Analysis of shallow foundation subject to vertical settlement,* of the EnFEM Example Manual.

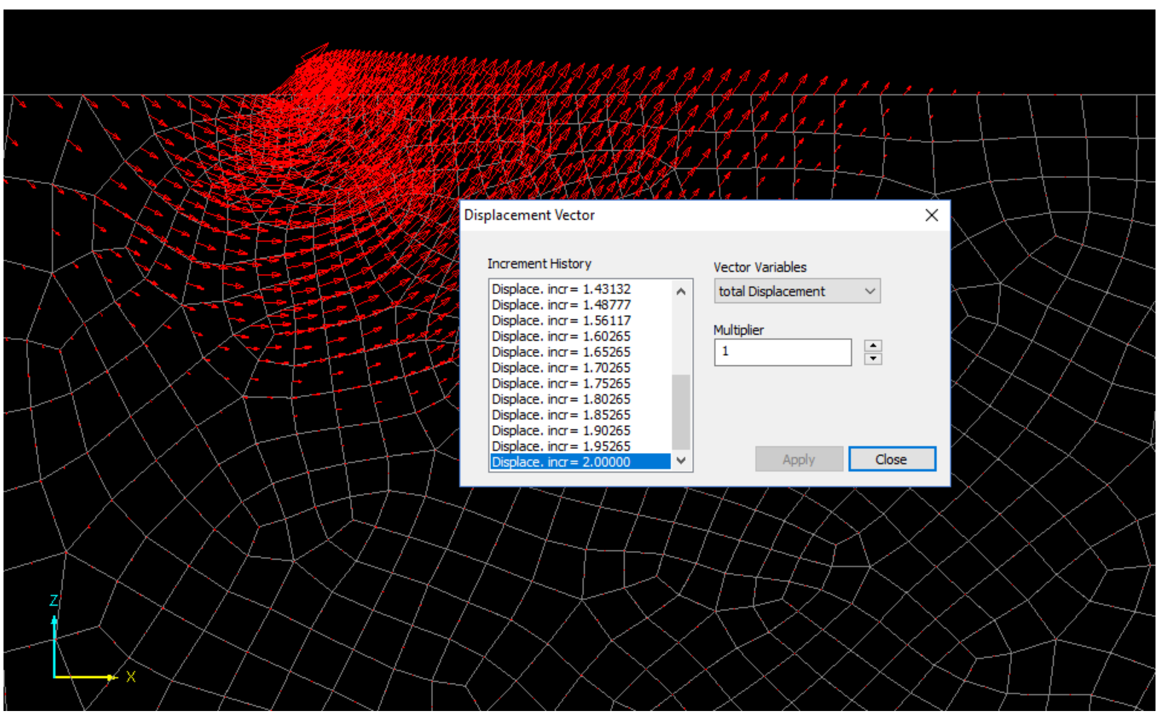


Figure 13-7: Displacement vector of soil due to settlement of foundation

User may select any available increment from the "Increment History" list, select the appropriate "Vector Variable" ("total displacement", is default), and select the "multiplier" value to increase or decrease of the size of the displacement vectors. Users can plot displacement vectors for components in the X-direction, Z-direction, or the combined magnitude and direction.

13.2.4 X-Y Plot

The X-Y plot is an important output option that illustrates the internal relationship between two parameters. The X-Y plot in EnFEM has two distinct categories:

1. Historic Result X-Y plot
    a. Plots results of a single node versus increment history.
    b. Only "Primary Variables" (deformation, stress, strain, etc.) or "Reaction Forces" are available "Categories" for plotting.
2. Field Result X-Y Plot
    a. Plots results in selected increments only.
    b. For plotting results along a defend node-path.
    c. All result "Categories" are available for plotting (including interaction interfaces).

The following sections detail options for X-Y Plots. Additional step-by-step detail using practical examples can be found in the EnFEM Example Manual.

13.2.4.1 Historic Result X-Y Plot

Historic X-Y Plot is used to plot results of a single specified node versus the increment history. Available "Categories" for historic X-Y Plots are "Primary Variables" (deformation, stress, strain, etc.) and "Reaction Forces".

Click the  icon on the toolbar to open the "X-Y plot" window seen in Figure 13-8. This figure shows an example plot of displacement in the X-direction versus the increment history of the selected stages (in this case the initial stress stage is turned off in the result).

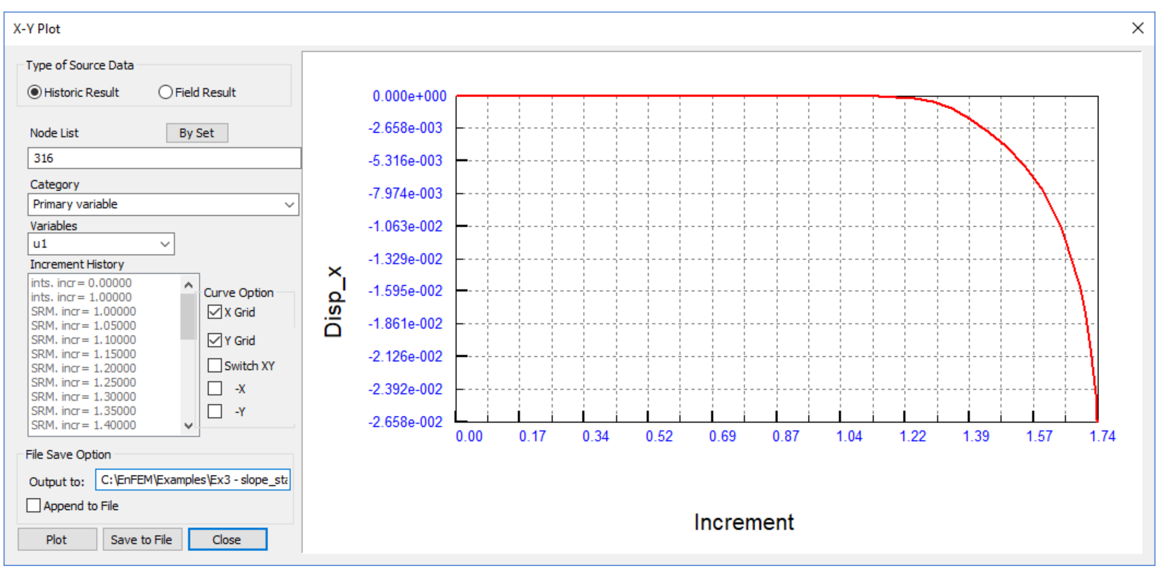


Figure 13-8: X-Y Plot for Historic Result

With the *"X-Y plot"* window open, the following procedures should be followed:

1. Select **"Historic Result"** as the data source.
2. Input the desired node number into the *"Node List"* entry.
   a. <u>NOTE:</u> Users many need to turn on the node numbers (**"Mesh->Show Node Numbers"**) and use zoom operations to determine the exact node of interest.

19. Select the desired option from the **"Category"** drop-down list.
    a. <u>NOTE:</u> Only *"Primary Variables"* (deformation, stress, strain, etc.) or *"Reaction Forces"* are available for *"Historic Plots"*.
20. Select the desired option from the **"Variables"** drop-down list.
21. Note that the content of increment history becomes grey because the historic option is selected.
22. Click **"Plot"** to plot the desired curve.

13.2.4.2 Field Result X-Y Plot

The Field Result X-Y Plot is used to plot results of specified variables along a path defined by a node list. Data in this plot are extracted from a selected load increment, and all result *"Categories"* are available for plotting (including interaction interfaces).

Click the  icon on the toolbar to open the *"X-Y plot"* window as seen in Figure 13-9. This figure shows an example plot of stresses along a selected node path at the selected load increment. Notice the node path that is entered in the *"Node List"* rather than just a single node. When a node path is used, the X-axis of the plot is the real distance of each node, in relation to the first node in the list.

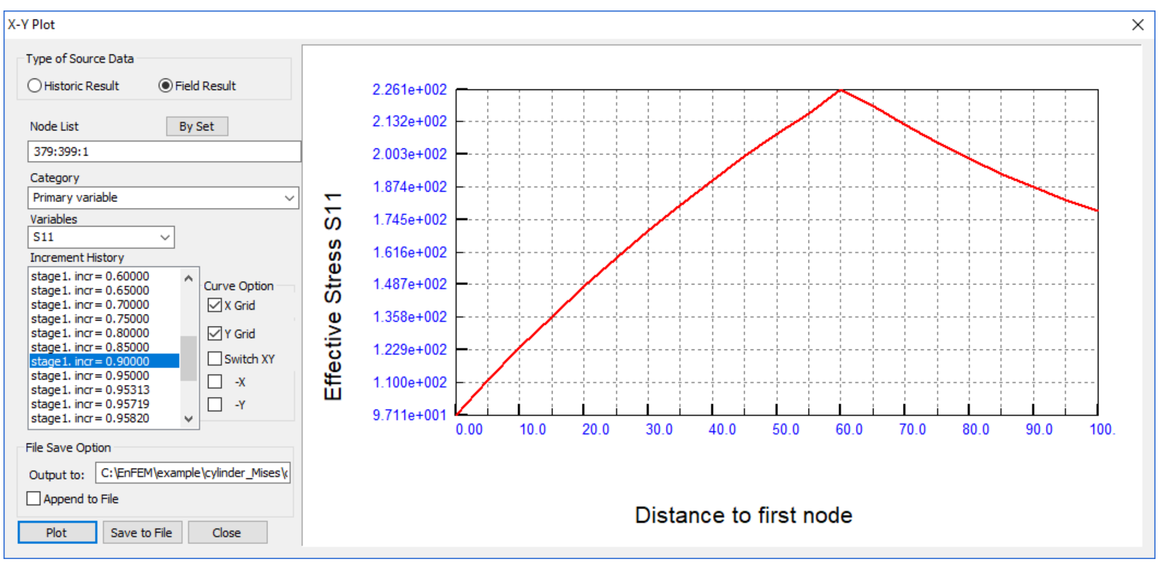


Figure 13-9: X-Y Plot for Field Result

With the *"X-Y plot dialog"* open, the following procedures should be followed:

1. Select **“Field Result”** as the data source.
2. Input the desired node-path into the *“*Node List*”* entry.
   a. NOTE: Users many need to turn on the node numbers (**“Mesh->Show Node Numbers”**) and use zoom operations to determine the exact node of interest.
   b. NOTE: See the following sections for options of node-path entry.

23. Select the desired option from the **“Category”** drop-down list.
    a. NOTE: All result *“*Categories*”* are available for plotting (including interaction interfaces).
24. Select the desired option from the **“Variables”** drop-down list.
25. Select the desired load increment.
26. Click **"Plot"** to plot the desired curve.

The following sections detail node-path entry options and formats.

13.2.4.2.1 Manual Entry of Node-Path

When specifying a Field Result X-Y Plot, a desired series of nodes must be input. The first option is to input this series manually, for which there are several options detailed below.

13.2.4.2.1.1 Individual Entry

The first option is a simple input of nodes, each separated by a comma - *“1,3,5,6,10…X”*. The user may choose to use this option if the nodes along the desired path are not in any fixed numerical order. This method can be used in combination with the *“fixed interval”* input described below, by simply using a comma for separation.

13.2.4.2.1.2 Fixed Interval

The second option provides more structured input, if the series of nodes has a fixed interval. In this case, the input can be written as:

`13:45:2` or, `117:112:-1`

The first number is in reference to the *“*start*”* node, while the second is the *“*end*”* node. The third number is in reference to the *“*interval*”*. Therefore, the first input will extract a node series from 13 to 45, using every other node - *“`13,15,17,19, ..., 45`”*. Whereas the second input is equal to a node series of *“`117,116,115,114,…, 112`”*. This method can be used in combination with the *“random entry”* input as described above, by simply using a comma for separation.

13.2.4.2.2 Node-Path Entry Using Predefined ‘Edge-Sets’

The alternative option to manual entry for specifying a node-path is using a predefined ‘edge-set’. This is the most convenient way to define the path, but does require that an *“edge selection-set”* (see CHAPTER 7) be previously defined, for the desired path.

With the *“*X-Y plot*”* window open, the following procedures should be followed:

1. Select the **“Field Result”** option

2. Click the button **"By Set"**, above the *"*Node List*"* entry, to open the *"*Set Selection*"* window, as seen in Figure 13-10.

   a. All of the previously defined selection-sets of the model are displayed in this window, detailed by their *"*Set-Name*"* and *"*Set-Type*"* (description).

27. Select one edge-set and click **"OK"**.

   a. NOTE: The "Node List*"* entry will now display *"*User Defined Set*"*.

28. Select the desired *"*Category*"*, *"*Variable*"*, and *"*Load increment*"*, and click **"Plot"** when finished.

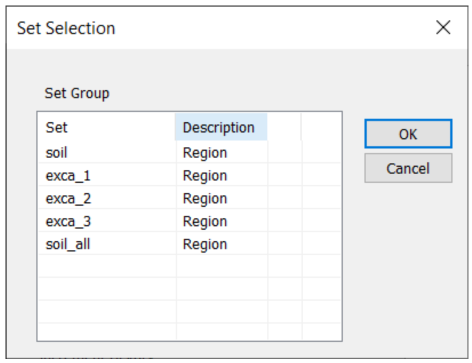


Figure 13-10: Select an 'edge-set' for Field Result X-Y Plot

13.2.4.2.3 Plotting Integrated Variables (moment, shear, or thrust)

The category of *"*Integrated Forces*"* for Field Result X-Y Plots is specifically designed to show integrated force output for structural elements in geotechnical engineering. For example, an engineer may be interested in moments and shears of a wall structure in retaining wall-soil interaction analysis or thrust forces inside a culvert while evaluating the design of the structure. Creating an integrated force plot comes with some additional steps that supplement the normal process for producing a Field Result X-Y Plot. They are as follows:

1. Integration forces can only be plotted from the single object of interest – typically a structural member. This structural member may be represented by more than one region. Use the *"*Set Tool*"* and *"*Display Tool*"* to define a selection-set and display *only* the member of interest - such as a culvert or wall region.

   a. NOTE: If there is more than one structural member that requires integrated force output, only display one structural member at a time.

2. When inputting the node-path of interest, using nodes along the centerline of the structural object is required. Therefore, a structured mesh should be used on such structural objects, while specifying the edge division such that nodes are present along the centerline. In this case, the node-path along the centerline is then manually entered. This may be tedious, and difficult depending on the mesh refinement. Taking some extra steps while creating the geometry of the structural object, by using sub-regions - in which a geometric edge is defined along the centerline - will allow the user to create a selection-set for these desired nodes and simplify plotting procedures in post-processing.

a. NOTE: Using nodes on an external boundary of the structural object is not recommended

Figure 13-11 shows the moment distribution along a culvert at the last load increment, using a manually entered node-set. Additional step-by-step detail using practical examples can be found in the EnFEM Example Manual.

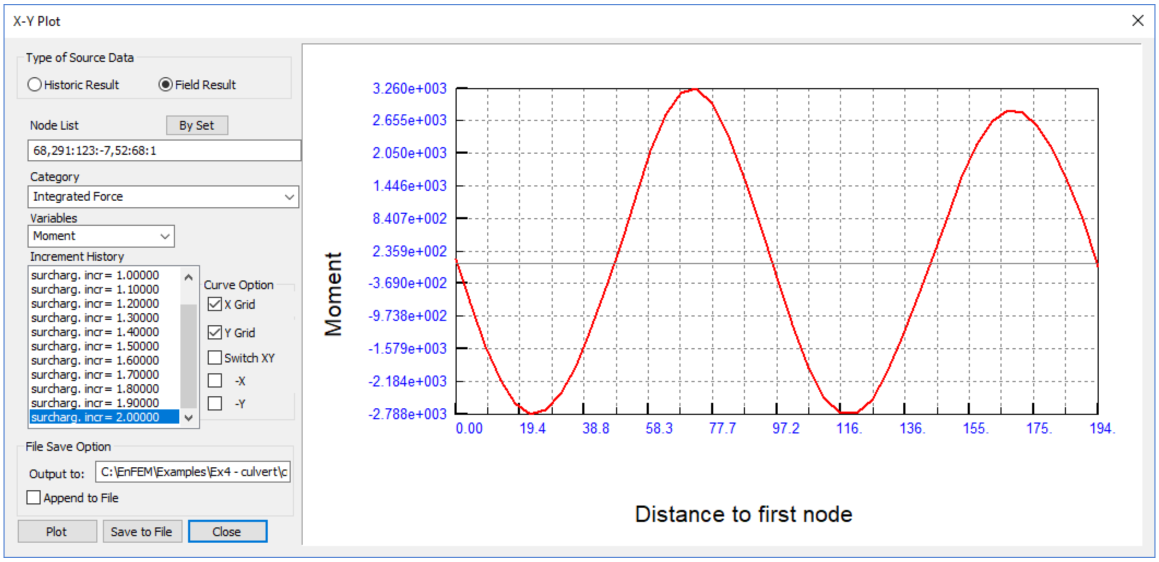


Figure 13-11: Moment of culvert along given node path

13.2.4.2.4 Plotting Interaction Variables

Plotting interaction variables is another useful feature in EnFEM that gives valuable information to the user in soil-structure interaction problems. For example, plotting pressure on the retaining wall interface, or plotting the contact status along the interface to determine if gaps are present. Creating an interaction variable plot comes with some additional steps that supplement the normal process for producing a Field Result X-Y Plot. They are as follows:

1. Use the *"Set Tool"* to define edge selection-sets for each of the desired interfaces. For example, one edge-set for the active side of a retaining wall, and a second for the passive side.
    a. NOTE: These edges should be continuous, and only nodes on the *'slave interaction-surface'* are available to plot contact stresses in EnFEM, so make sure to define edge-sets appropriately.

Figure 13-12 shows an example of normal pressure VS depth along an interaction interface. The start node is determined by EnFEM implicitly. If users want to check where the XY plot starts, users may compare the node list to the nodal location in the model, from an externally saved file (process detailed in following sections).

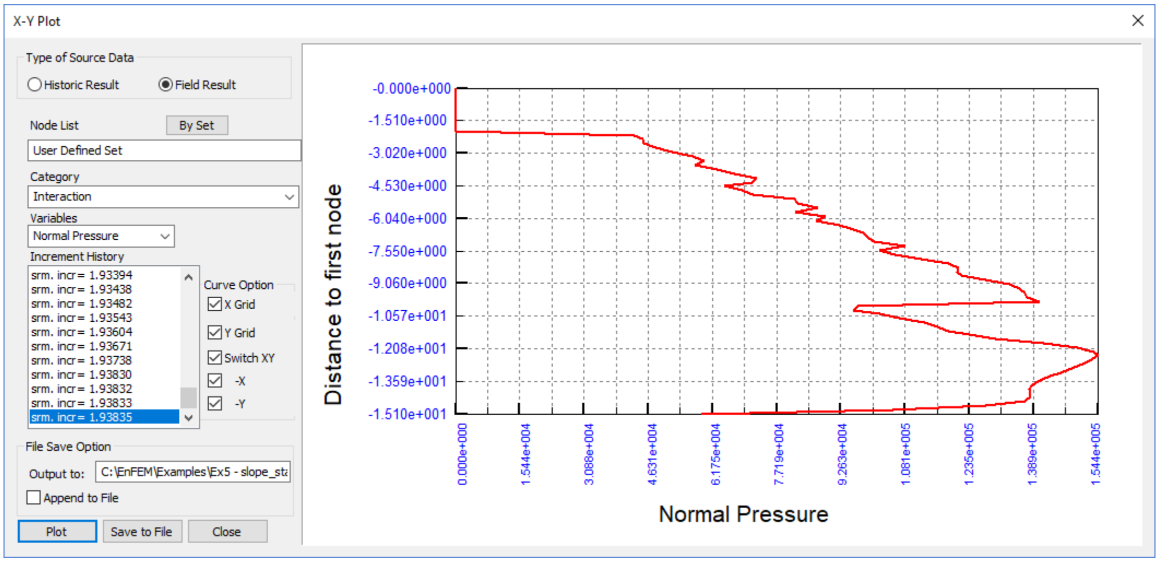


Figure 13-12: Normal interaction pressure versus depth

13.2.4.3 Other Plot Options

Additional options for X-Y Plots are available to add flexibility.

13.2.4.3.1 Curve Options

A number of options exist to manipulate the plot, at right of the *"*Increment History*"*, under the name *"*Curve Options*".* Activating these options will enact various changes in the visual representation or direction of the plot.

13.2.4.3.2 'Save to File' Options

A second option available is to export the X-Y Plot data to an external file, in order to save for further processing.

The default save directory is the same as the model file, and the output file name is that of, *"current analysis filename_xy.dat".* Under the *"*File Save Option*"* users may change this to point to a specific file directory or specify another filename as desired. Clicking **"Save to File"** will export the file, and a message window will appear to confirm the export was completed successfully.

If the option **"Append to File"** is checked, new data exported will be appended to the end of the file, provided the path and filename are the same as already exists.

Additional step-by-step detail using practical examples can be found in the EnFEM Example Manual.

13.2.4.4 Available Categories and Variables for Plotting

In addition to those output variables available for Contour Plots, the following are also available for X-Y Plots.

13.2.4.4.1 Integrated Force Category

1. Moment: Integration of normal stresses to calculate the moment on defined sections along a node-path. It has the unit: "Force*length"
2. Shear: Integration of stresses tangent to sections. It has the unit: "Force"
3. Thrust: Integration of stresses normal to sections. It has the unit: "Force"

> NOTE: These "sections" are perpendicular to the node path and located at each nodal position in the path. For example, at each node along the node-path, an integration is performed for the nodes at the specific cross-section of the object of interest. Only one object of interest, subject to the X-Y Plot, should be displayed. Additionally, a structured mesh is recommended for producing this type of result.

13.2.4.4.2 Interaction Category

1. Normal Pressure: Calculated stress components which are in the normal direction to the contact interface. It has the unit: "Force/sq. unit length".
2. Tangent Shear: Calculated tangent (frictional) stress components which are in the tangent direction to the contact interface. It has the unit: "Force/sq. unit length".
3. Normal Status: Numerical values are plotted corresponding to the status of the contact condition: 1=interface is open (gap), 2=interface is closed (no gap)
4. Tangent Status: Numerical values are plotted corresponding to the status of the contact condition: 0=not in contact, 1=in contact and "sticking condition", 2=in contact and "sliding condition".

13.2.4.4.3 Reaction Force Category

1. X Reaction: Calculated reaction forces in the direction of the X-axis. It summarizes all forces from the provided node-path. It has the unit: "Force".
2. Z Reaction: Calculated reaction forces in the direction of the Z-axis. It has the unit: "Force".

13.2.5 Cutline Plot

If the mesh is unstructured, it is almost impossible to find a straight-line node-path set inside the FE mesh, to produce a typical X-Y Plot. In this case, the "Cutline Plot" option is a convenient and useful tool. From the menu bar, click **"Result->Cutline"**, or use the toolbar icon, , to open the "X-Y Plot on Cutline" window as shown in Figure 13-13. This figure shows an example plot of displacement in the Z-direction at the selected load increment.

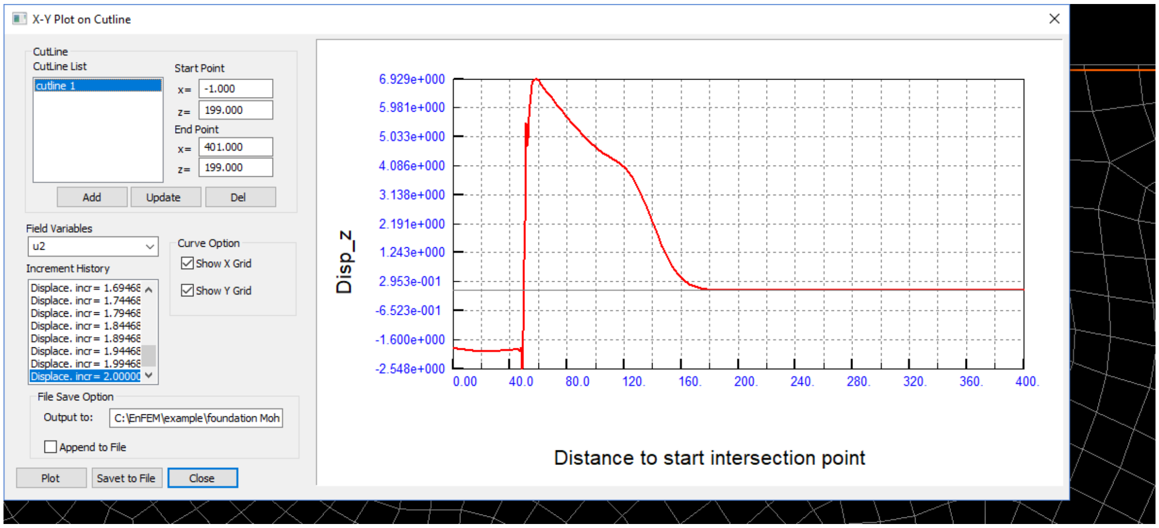


Figure 13-13: X-Y Cutline Plot

With the *"X-Y Plot on Cutline window"* open, the following procedures should be followed:

2. Input the X and Z-coordinates of two ends of the desired cutline.
2. Click **"Add"** to add this new cutline into the *"Cutline List"* window.
   a. A red straight line is then displayed on screen to show the cutline location in reference to the model.
29. Select the field variable, and load increment of interest
   a. NOTE: Only *"Primary Variables"* (deformation, stress, strain, etc.) are available for Cutline Plots.
30. Click **"Plot"**.

Note that the X-axis refers to the distance along the cutline. The *"start point"* of the cutline refers to zero on the X-axis, and *"end point"* will refer to the maximum value (distance) of the axis.

If a cutline line passes through an internal void (hole), the plot connects the two points on the inner faces of the void by a straight line.

Similar to typical X-Y Plots, the option is available to export the Cutline Plot data to an external file, in order to save for further processing. The default save directory is the same as the model file, and the output file name is that of, *"current analysis filename_cut.dat"*. Under the *"File Save Option"* users may change this to point to a specific file directory and specify another filename as desired. Clicking **"Save to File"** will export the file, and a message window will appear to confirm the export was completed successfully.

If the option **"Append to File"** is checked, new data exported will be appended to the end of the file, provided the path and filename are the same as already exists.

13.2.6 Curve Plot of Structural Element

This function is to plot deformation (translation, rotation) and internal forces (axial force, shear, moment) of selected structural elements. The X-Y curve plot function in section 13.2.4 plots results verse increments or nodal position in X-Y type. The curve plot in this section will plot an actual view of structural elements and result curve at each node. If users define structural entity sets in advance, input of node list is totally not necessary.

Click icon to switch to “RESULT-MODE”. Click menu “**Results->Curve Plot for Structural Elements**” or icon on the toolbar to open the curve plot dialog window as shown in Figure 13-14. The following procedures should be followed:

1. select an appropriate type of structural element (Beam, strut or geogrid)
2. Click button “**Select By Set**”. Select one set of structural entities from the popup dialog. Click “**OK**” to exit to the curve plot window again.
3. Select one appropriate category (deformation, internal forces or interface)
4. Select one increment from the drop list of increment history
5. Click “**Plot**” to show the curve on the right of the dialog.

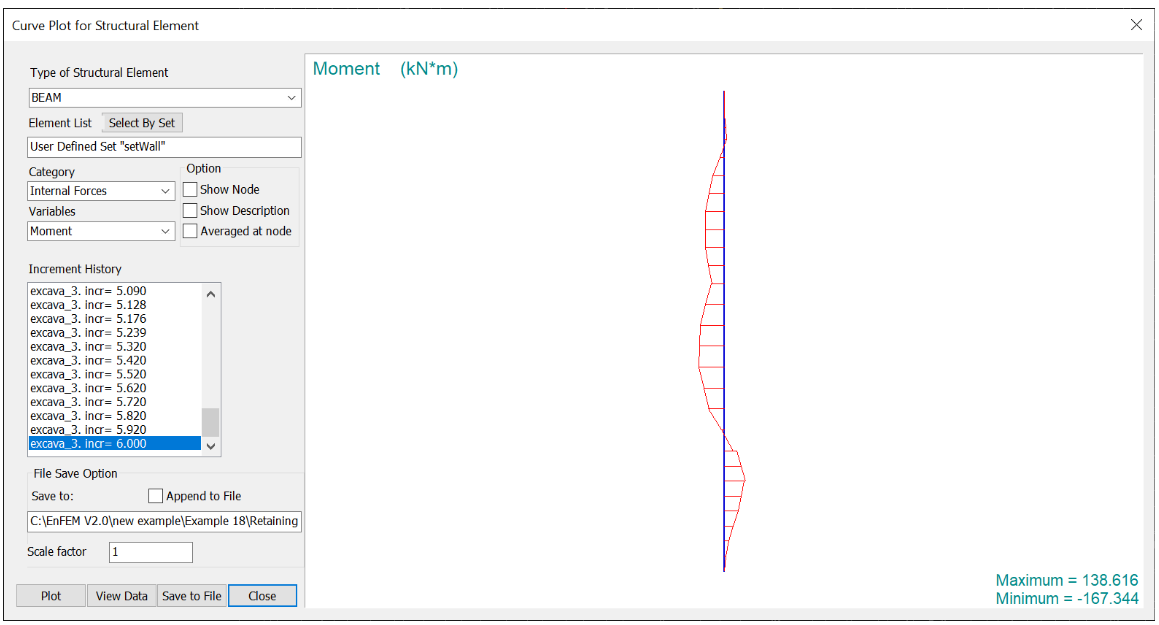


Figure 13-14: Curve plot for structural elements

Figure 13-14 plot the moment curve along a retaining wall at selected load increment.

13.2.6.1 Type of structural element

Three types are available to select: “BEAM”, “STRUT” or “GEOGRID”. This selection determines the contents of the other drop list: element set list, items of category and lists of variables will follow the same type.

13.2.6.2 Element list

Users should define the list of structural elements for curve plot. This list can be input by users directly using the same rule as in section 13.2.4.2.1.1. An alternative convenient way is to select from geometric entity sets which should be defined in advance. Click “**Select By Set**” to pop up the “set selection dialog”. Note that only geometric entity sets with matching type will be listed in “set selection dialog”. Here, users should note a restriction to plot curve by sets. Finite elements of every geometric entity set will be generated by EnFEM automatically. However, not all geometric entity sets are applicable to plot a curve. To plot curve by sets, geometric entities in one set should be head-toe connected without interruption or closed. If a geometric entity set contains non-continuous structural entities, this set cannot be used to plot a curve.

13.2.6.3 Category

Depends on which type is selected by users, following items are available:

- Type of “BEAM”: deformation, internal forces, interface
- Type of “STRUT”: deformation, internal forces,
- Type of “GEOGRID”: deformation, internal forces

13.2.6.4 Variables

The list of available variables depends on the selected type.

For “BEAM” type, following variables are available:

- Deformation: translation in x direction, translation in z direction, rotation
- Internal Forces: axial force, shear, moment
- Interface: normal stress, tangent stress, normal status, tangent status. In general, a beam has two zero-thickness interfaces. Correspondingly, variables for “Interface” category are distinguished by suffix “R” (right side) or “L” (left side). Figure 13-15 displays an example of normal stress of a retaining wall. The red arrow on the beam represents the beam direction. The red circle indicates the start node of this plot. The green dashed line adjacent to the wall represents an interesting side (left or right).

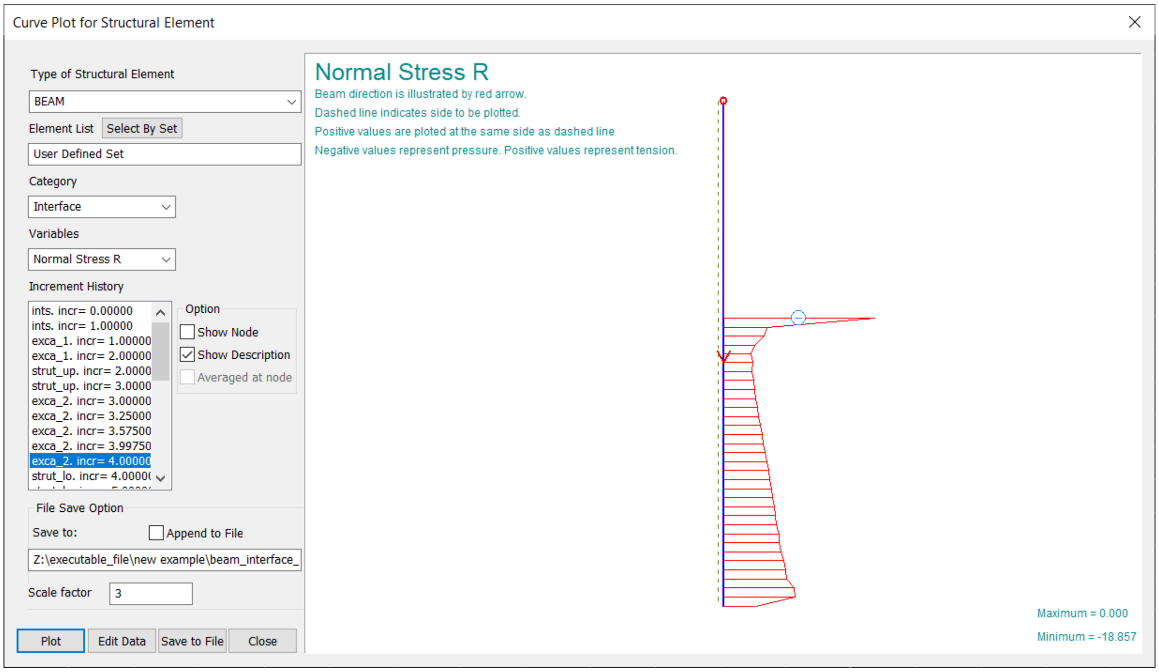


Figure 13-15: Example of normal stress on a retaining wall

For “STRUT” or “GEOGRID” type, following variables are available:

- Deformation: translation in x direction, translation in z direction
- Internal Forces: axial force

13.2.6.5 Increment history

All successfully completed increments are listed in the increment history window. A load increment should be determined to plot a curve.

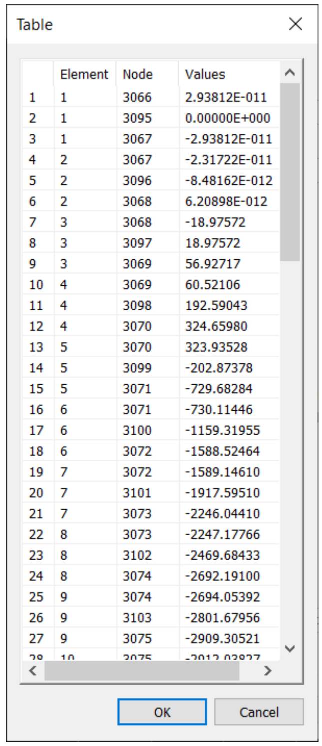
13.2.6.6 Option of curve plot for structural elements

Several options for curve plot are available:

- Show Node: the default curve plot does not plot node mark and node number. If this option is selected, node marks are plotted. Selected node numbers are also plotted adjacent to nodal position.
- Show Description: Title of each curve and detailed information such as sign convention are plotted
- Averaged at Node: Curve plot is based on values at nodal position. However, internal forces of beams are computed at integration points and extrapolated to nodal positions. If this option is selected, nodal values will be averaged from adjacent elements. It is not recommended to average nodal values for shear forces.

13.2.6.7 Edit Data

Click "**Edit Data**" to pop up a data window as shown in Figure 13-16. Results in this table is *element-based*. Click "**OK**" to exit from this data table window.



	Element	Node	Values
1	1	3066	2.93812E-011
2	1	3095	0.00000E+000
3	1	3067	-2.93812E-011
4	2	3067	-2.31722E-011
5	2	3096	-8.48162E-012
6	2	3068	6.20898E-012
7	3	3068	-18.97572
8	3	3097	18.97572
9	3	3069	56.92717
10	4	3069	60.52106
11	4	3098	192.59043
12	4	3070	324.65980
13	5	3070	323.93528
14	5	3099	-202.87378
15	5	3071	-729.68284
16	6	3071	-730.11446
17	6	3100	-1159.31955
18	6	3072	-1588.52464
19	7	3072	-1589.14610
20	7	3101	-1917.59510
21	7	3073	-2246.04410
22	8	3073	-2247.17766
23	8	3102	-2469.68433
24	8	3074	-2692.19100
25	9	3074	-2694.05392
26	9	3103	-2801.67956
27	9	3075	-2909.30521

Figure 13-16: Data table of plotted curve

13.2.6.8 File Save

Results of this curve plot can be saved to a file on hard disk. Figure 13-17 shows contents of a saved data file. Output is based on nodes. If "**Averaged at node**" is selected, output will calculate the average value at each node and merge repeated nodal output.

```
Curve Plot for Structural Element Set

   Type      = BEAM
   Category  = Internal Forces
   Variable  = Moment
   Increment = exca_2. incr= 4.00000

Output of Node List in Set
   User Defined Set
    3066     3095     3067     3096     3068     3097     3069     3098     3070     3099
    3071     3100     3072     3101     3073     3102     3074     3103     3075     3104
    3076     3105     3077     3106     3078     3107     3079     3108     3080     3109
    3081     3110     3082     3111     3083     3112     3084     3113     3085     3114
    3086     3115     3087     3116     3088     3117     3089     3118     3090     3119
    3091     3120     3092     3121     3093     3122     3094

     Node      Dist.     Moment
---------------------------------------------------------------------
     3066          0     2.93812e-011
     3095       6.25                0
     3067       12.5     -2.93812e-011
     3096      18.75     -8.48162e-012
     3068         25     6.20898e-012
     3097      31.25       18.9757
     3069       37.5       56.9272
     3098      43.75        192.59
     3070         50        324.66
     3099      56.25      -202.874
     3071       62.5      -729.683
     3100      68.75      -1159.32
     3072         75      -1588.52
```

Figure 13-17: Curve data saved to a file

13.2.6.9 Scale Factor

The scale factor is used to enlarge or shrink the curve altitude. The default value is "1". Users can change this value and click "**Plot**" to refresh the curve.

13.2.6.10 Sign Convention

The sign convention of curve plot for structural elements follows these rules:

- Positive displacements are in the global X or Z direction
- Compressive forces are negative. Tensile forces are positive
- Moments are plotted on the tension side
- Counterclockwise rotation of cross section is positive

Additional rules exist for plot of zero-thickness interface:

- Interested side (left or right) is marked as green dashed line
- Positive values are plotted on the same side as the green dashed line. Negative values are plotted on the opposite side of the green dashed line

CHAPTER 14. Import Models

14.1 Import from UTEXAS

UTEXAS is general-purpose software for limit equilibrium slope stability computations. EnFEM can import the model from UTEXAS for further finite element analysis.

14.1.1 Import from UTEXAS

In "CAD-MODE", click the menu "**File->Import from EnSlope/Utexas4**" to pop up the importing window as shown in Figure 14-1. Click the button "**Import from UTEXAS4**". In the file explore window, select the UTEXAS input file which has the suffix ".preinput-utexas". Click "**Open**" to complete the importing operation.

Profile lines are imported to build all soil regions. Materials are imported and converted by EnFEM automatically. Side "roller" type displacement boundary and bottom "pin" type displacement boundary are created and assigned to appropriate region edges. The gravity boundary is also created and assigned to all soil regions.

Piezometric line is imported into EnFEM model. Finite elements are located above the underground water table, regular unit weight is used when gravity load is applied. Finite elements locate below the underground water table will use the floating unit weight where the unit weight if water has been subtracted.

14.1.2 Requirements for profile line definition

There are some requirements for profile lines when users try to import a model from UTEXAS:

1. One profile line should be defined from left to right horizontally
2. Profile lines are recommended to be defined from bottom to upside sequentially
3. Two adjacent profile lines cannot have overlapped segments
4. The start/end vertex of any profile line should be the case from the left boundary, the right boundary, or intersection point of previous profile line. The left/right boundary is the farthest side of the model range.

14.1.3 Limitations

There are some limitations in the importing operation:

1. Only one stage can be imported from UTEXAS. The second stage is ignored
2. Materials of "Very Strong" in UTEXAS are converted to linear elastic material. Materials of "Conventional" in UTEXAS are converted to "Mohr-Coulomb" materials. Materials in UTEXAS defined as "Linear increase below profile line", "Anisotropic strengths", "Nonlinear envelope" and "Interpolate Strengths" are replaced by "Mohr-Coulomb" material, but no material parameters are filled. Users need to complete these soil parameters
3. Young's modulus and Poisson's ratio are estimated by EnFEM if possible. Usually, users need to update these parameters according to lab test results.
4. The minimum cohesion is set to 72psf(0.5psi)

14.2 Import from EnSlope

EnSlope is another software for slope stability analysis. EnFEM can import geometric data and materials from an existing EnSlope model. Boundary conditions, soil-structure interactions and stages are added by EnFEM automatically.

14.2.1 Import from EnSlope

In “CAD-MODE”, click “**File->Import from EnSlope/Utexas4**” to pop up the dialog for model importing as shown in Figure 14-1. Click “**Import from EnSlope**”, select the model file *.slpd from the file explore window, then click “**Open**” to complete the importing operation. If the model is successfully imported, a ENFEM model will be displayed on the screen as in “CAD MODE”.

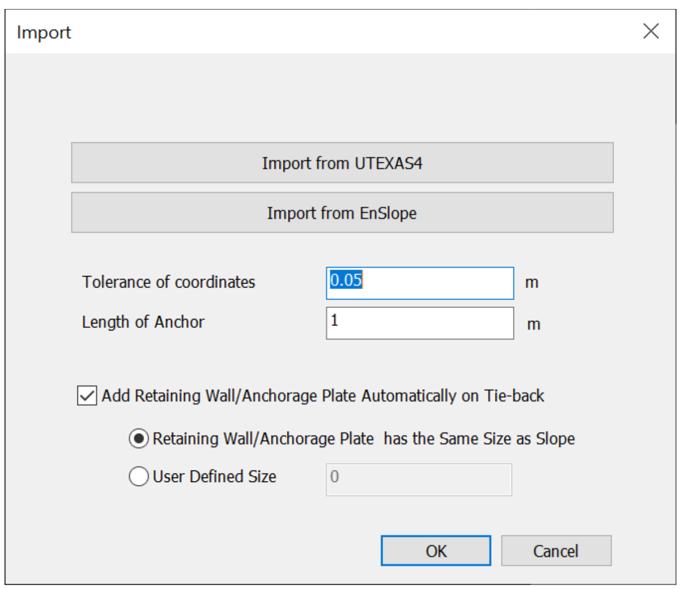


Figure 14-1: Import model from EnSlope/UTEXA4

The rules to import an Enslope model are described here:

1. Points have coordinate difference smaller than the tolerance will be merged into one point
2. Enslope does not define the anchor length of a tie-back. EnFEM will add a geogrid as the anchor whose length is 1m or 3 ft by default
3. EnSlope does not define the structure to hold the tie-back on the slope side. EnFEM will add beams automatically as the left anchorage of a tie-back. The soil-structure interaction follows the “tie” type. If the option “Add Retaining Wall/Anchorage Plate

Automatically" is unchecked, users take the responsibility to add beams as the anchorage on the slope side.

4. Young's modulus and Poisson ratio are not defined in EnSlope model. EnFEM estimates these parameters according to the friction angle/cohesion.
5. If the Enslope model has more than one piezometric line, only the first one will be imported
6. Axial forces of tie-backs defined in the EnSlope model will be imported as pre-stress forces
7. After adding a geogrid on the right side of tie-back, the right end of the geogrid may locate outside or too close to the right boundary. In this case, EnFEM will extend the right soil boundary automatically
8. If a material in EnSlope has cohesion = 0 and friction angle = 0, this material will be imported as linear elastic material. Young's modulus and Poisson ratio are estimated by EnFEM using the default values.
9. If a material has friction angle ≥20° and cohesion = 0, EnFEM will set the cohesion as 3.5kPa or 72 psf automatically

14.2.2 Estimation of Young's modulus and Soil Strength

14.2.2.1 Estimation of soil parameters

Young's modules of soils are estimated from the friction angle and cohesion: (unit: psf)

Cohesion c (psf)	fric. angle<10°	30° >fric. angle>=10°	30° <=fric. angle<48°	
		2e5	2e5 ~ 2e6	
>3000	250*cohesion			
3000>= c >1000	500*cohesion			
1000 >=c >600	1000*cohesion			
<600	1500*cohesion			

14.2.2.2 Young's modulus of Geogrid

Anchor of a tie-back:

Young's modulus E=5.2e8 psf (2.5e7 kPa). Area=0.76 ft^2 (0.07 m^2).

Reinforcement of soil:

Young's modulus E=12e8 psf (6e7 kPa). Area=4.3e-4 ft^2 (4.0e-5 m^2).

14.2.2.3 Young's modulus of strut

Young's modulus=4.2e9 psf (2e8 kPa). Area=0.007 ft^2 (0.00065 m^2).

14.2.3 EnSlope Example Import settings

The following parameters are set/changed when examples are imported from EnSlope.

ENSLOPE EX1: ENFEM imports materials with zero strength as concretes, so results from ENFEM are not comparable to those from EnSlope. Reduce the Young's Modulus of the top layer to 1 psf (Data > Material properties > Material_2) to simulate a very soft material. Reduce the global mesh seeds to a size of 3 to better define the failure surface.

ENSLOPE EX1 Surcharge: Reduce the global mesh seeds to a size of 3 to better define the failure surface.

ENSLOPE EX2: Set geogrid length as 0.1m when import from EnSlope. This relatively short geogrid (anchor) will provide about 650 kN of tension on the tieback. User can fine tune the length of the geogrid for better comparison to results from EnSlope if needed.

ENSLOPE EX3: Set geogrid length as 3 ft when import from EnSlope. Deselect prestress option in tieback properties (Data > Structure section > truss). Add seismic forces in the gravity force setting of boundary conditions (Data > Boundary Conditions > Gravity).

ENSLOPE EX4: Increased the size of the local seed control (Data > Pre-mesh menu > local seed control) at the far edges to speed up the calculation.

ENSLOPE EX5: Set geogrid length as 3 m when import from EnSlope.

ENSLOPE EX6 Overall: Change the SRM range from 0.2 to 2.0. Increased the size of the local seed control (Data > Pre-mesh menu > local seed control) at the far edges to speed up the calculation. Result is comparable to "ENSLOPE EX6 upper slope.slpd".

ENSLOPE EX6 Overall_TopRein: This example added tiebacks in the upper portion of the slope, so the result is comparable to "ENSLOPE EX6 weak layer Janbu.slpd". Set geogrid length as 1 m when import from EnSlope. Deselect prestress option in tieback (truss) properties. Change the wall section area to 1 m^2 with a spacing of 1m (Data > Structure section > Wall).

ENSLOPE EX6 Overall_TopMidRein: This example added tiebacks in the upper portion of the slope, so the result is comparable to "ENSLOPE EX6 Overall.slpd". Set geogrid length as 1 m when import from EnSlope. Deselect prestress option in tieback (truss) properties. Change the wall section area to 1 m^2 with a spacing of 1m (Data > Structure section > Wall).

ENSLOPE EX7 Pile: Assign Region 5 to Material_1 (Data > Material Assignment > Region 5 > Select Material_1 for Material Group). Change the pile spacing to 5 ft (Data > Structure section > Pile).

14.3 Import from PYWall

PYWall is the tool to analyze the behavior of retaining structures which depends on realistic soil conditions and nonlinear soil-structure interactions. Based on finite element approach, EnFEM provides users more information about behavior of the retaining wall. After importing the PYWall model, geometry, materials, construction stages and boundary conditions can be created automatically by EnFEM.

14.3.1 Import from PYWall

In "CAD-MODE", click the menu "**File->Import from PYWall**" to pop up the dialog for model importing as shown in Figure 14-2. Users can change the default soil width on the back/excavation sides or soil depth as long as the anchor length. Usually, excavation constructions are divided into several stages, such as excavation of soil or installation of tiebacks/struts. The EnFEM model assumes that the analysis of a retaining wall is multi-stage based by default. If the user unchecks the option "Use Multi Stages", the imported model will have only one construction stage without geostatic stage. All structures (tie-backs, struts) will be activated in this stage, and no simulation of soil excavation. If the PYWall model does not have tie-backs or struts, the option "Number of excavations" should be selected. Otherwise, the option "Number of excavations is determined by structures" should be selected if the PYWall model has tie-backs/struts as lateral support of the retaining wall.

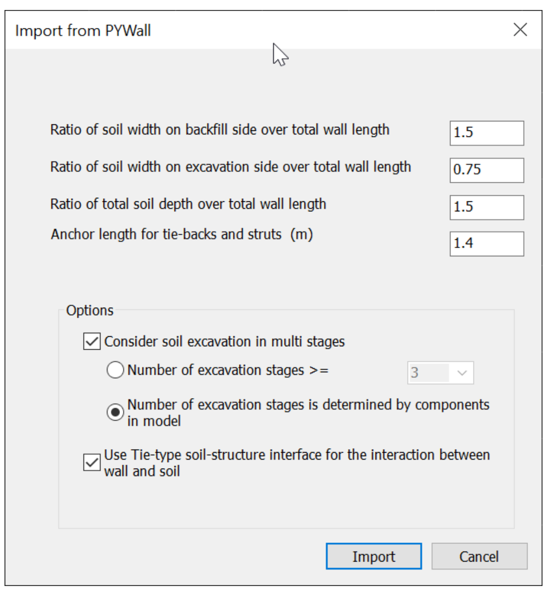


Figure 14-2: Import model from PYWall

The default soil-structure interface is set to "tie" type. Uses may change it to regular zero-thickness interface after importing.

14.3.2 Limitations

Some features in the PYWall model cannot be imported:

1. The material of the retaining wall is always elastic in EnFEM. Non-linear wall section is not supported
2. Inclined ground on the back/excavation side cannot be imported
3. Assumption of zero-bending model at the wall connected with a tie-back is not supported. Wales are not supported in EnFEM
4. Multi-stages defined in the PYWall model cannot be imported totally. Only the last construction stage will be imported. Additional wall deflection constrains during excavation construction will be applied immediately after the first excavation. The excavation soil layers created by ENFEM may be different from the original PYWall model.
5. Point surcharge loads or buried surcharge loads are not supported. Line surcharge loads will be replaced by point loads after importing.

14.3.3 Estimation of Young's modulus and Soil Strength

Estimation of Young's modulus follows the same rule as importing from UTEXAS4/EnSlope.

CHAPTER 15. Miscellaneous Tools & Options

15.1 “CAD-MODE” Option Menu

15.1.1 Grid Space

This tool is used to change the spacing of the background gird-lines in *“CAD-MODE”*. From the menu bar, click **“Options->Grid Size”** to open the *“Grid Size”* window as seen in Figure 15-1.

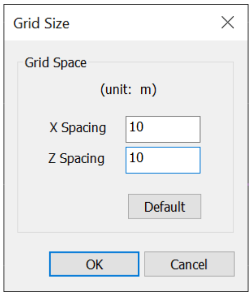


Figure 15-1: Grid size dialog

Input new values to change the spacing of background grid lines in the X and Z directions. Click **“OK”** to exit. Clicking **“Default”** will reset to the default grid size. The default grid size is 100x100. This can be a useful tool for model creation, as users can *“Snap”* to the grid when creating various geometric entities.

15.1.2 Drawing Space

This tool is used to change the default distance of the model extents on screen. From the menu bar, click **“Options->Drawing Space”** to open the *“Drawing Space Extents”* window as seen in Figure 15-2. Input new extents and click **“OK”** to exit. Clicking **“Default”** will reset to the default space extents.

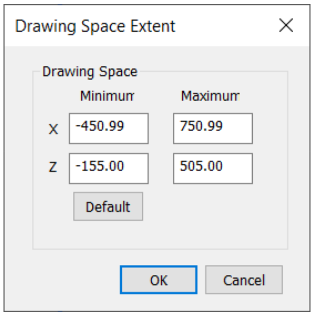


Figure 15-2: Drawing space extent dialog

15.1.3 Background Color

The background color for the *“CAD-MODE”* and *“MESH-MODE”* display windows is set to black by default. Users can change this setting to improve visualization. From the menu bar, click **"Options->Background Color"** to open the *“Background Color”* window, as seen in Figure 15-3. Use the drop-down menu to pick the desired color and click **"OK"**. This information is saved into the model database, so that when the model is closed and re-opened, this setting will be preserved. Clicking **"Use Default"** will reset these options.

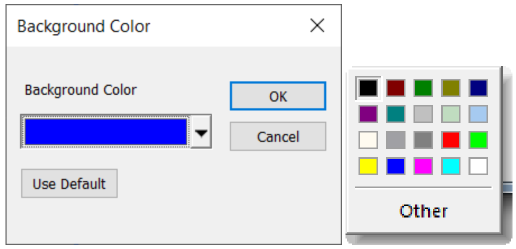


Figure 15-3:: Change background color

15.1.4 Toolbar options

15.1.5 Show CAD toolbar

By default, CAD tool bar is always shown on. If this option is not checked on, the CAD tool bar will not be displayed. If the user turns off the CAD bar accidently, recheck this option can recover display of the CAD tool bar.

15.1.6 Show CAD Edit toolbar

Toggle on/off edit tool bar of "CAD MODE".

15.1.7 Show CAD selection toolbar

Toggle on/off selection tool bar of "CAD MODE".

15.1.8 Show CAD option toolbar

Toggle on/off option tool bar of "CAD MODE".

15.2 Zoom Operations

The following zoom operations are available in EnFEM:

15.2.1 Pan View

Pan view in EnFEM is enabled by clicking and holding the middle mouse key. The view will be updated when the mouse moves.

15.2.2 Zoom In/Out

Zoom in/out is effective when users rotate the mouse wheel.

15.2.3 Zoom to Window

Click to activate the  icon, move the mouse cursor to an appropriate location on screen. LEFT-CLICK and hold and drag the mouse creating a rectangular zoom-window. Release the mouse button when the desired geometry is within the rectangle. The view inside the zoom window will be enlarged to fill the whole EnFEM window.

15.2.4 Zoom to Extents

Click the icon to zoom to the model extents. This operation is used to show the whole model and scale as large as possible.

15.3 "Result-Mode" Options Menu

15.3.1 Font and Color

In "RESULT-MODE", users can change the background color, and the font color, size, and type. From the menu bar, click **"Options->Font and Color"** to open the "Font and Color" window as shown in Figure 15-4.

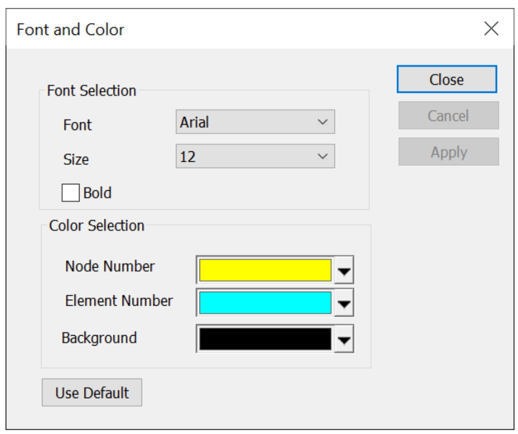


Figure 15-4: Change color and font for result window

Make the appropriate selections for the various options available and click **"Apply"** to activate these changes. Click **"Close"** to exit. Clicking **"Use Default"** will reset these options.

List of Technical References

Nonlinear FE analysis involves knowledge of the finite element method, engineering judgment, and some experience with numerical simulation. Several reference books are listed here as reading materials:

1. C.S. Desai and John T. Christian, *Numerical methods in geotechnical engineering*, McGrawHill,1977
2. David M. Potts and LidijaZdravković, *Finite element analysis in geotechnical engineer: theory*, Thomas Telford,1999
3. EA de Souza Neto, D perić and DRJ Owen, *Computational methods for plasticity: theory and applications*, John Wiley and sons, 2008
4. R.D. Cook, D.S. Malkus, M.E. Plesha and R.J. Witt, *Concepts and applications of finite element analysis* 4th version, John Willey & sons, 2002

License Agreement & Disclaimer

PROGRAM LICENSE AGREEMENT & DISCLAIMER

IMPORTANT NOTICE: The software you are about to install or previously have installed from Ensoft Incorporated (“ENSOFT”) is licensed only on the condition that you agree to the terms and conditions set forth below. Please read the terms of this License Agreement & Disclaimer ("LICENSE") carefully.

If you agree to be bound by the terms of this LICENSE, please check mark the box labeled “YES – I Accept the terms of this License Agreement!” during software installation so it can be installed on your computer.

If you do not agree to the terms of this LICENSE, please check mark the box labeled “NO – I DO NOT Accept the terms of this License Agreement!” which will terminate the software installation.

If the software is already installed and licensed, you have already agreed to the terms and conditions of this license, or someone has done so on your behalf.

This LICENSE is a legal agreement between you (“USER”) – either an individual or a single entity– and ENSOFT for the ENSOFT software product that you are about to install or previously have installed (“PRODUCT”). The PRODUCT collectively includes computer software and associated media, printed materials, hardware key (dongle), and electronic documentation. The PRODUCT also includes any updates and supplements to the original PRODUCT that may have been produced by ENSOFT. By installing, copying, downloading, accessing or otherwise using the PRODUCT, the USER agrees to be bound by the terms of this LICENSE. If you (the USER) do not agree to the terms of this LICENSE, do not install or use the PRODUCT.

SOFTWARE LICENSE & DISCLAIMER

The PRODUCT is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The PRODUCT is licensed, not sold.

1. GRANT OF LICENSE

This LICENSE grants you the following rights:

1.1 INSTALLATION AND USE

1.1.1 Single-User Licenses

This software is licensed only to the USER (company or individual) whose name is registered with ENSOFT and for one specific physical site (“SITE”) registered with ENSOFT. You may install and use the PRODUCT on any number of computers at the licensed SITE. However, the PRODUCT is fully operational only in the SITE computer that carries the appropriate PRODUCT’s hardware key (“KEY”) that is used as software protection device.

1.1.2 Network Licenses

This software is licensed only to the USER (company or individual) whose name is registered with ENSOFT and for the specific physical site(s) (“SITE(S)”) and IP range registered with ENSOFT. Each network seat that is purchased is provided with access for SITE(S) clients located within up to 2 subnets with unique third octets (“CLIENT SUBNETS”). The minimum two-seat network license can be associated to a maximum of 4 CLIENT SUBNETS (within up to 4 unique third octets). Purchase of additional network seats can extend the allowable range of CLIENT SUBNETS.

The USER may install and use the PRODUCT on any computer in the licensed SITE(S) that is within the allowable CLIENT SUBNETS registered with ENSOFT. Any one computer in the network can be designated as license server for the PRODUCT by carrying the appropriate PRODUCT’s hardware key (“KEY”) that is used as software protection device.

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 shallprovide 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, eg., 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.