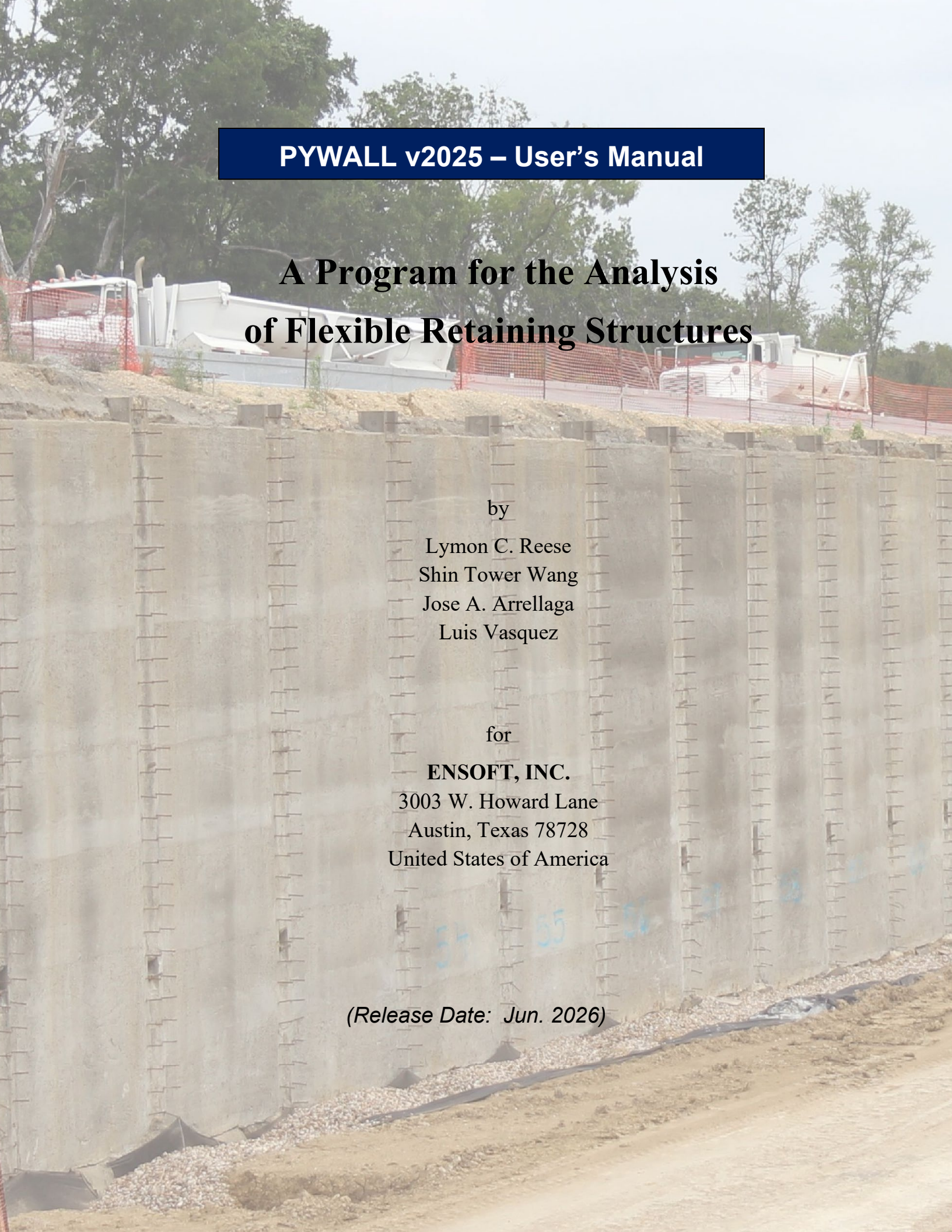


PYWALL v2025 – User's Manual

A Program for the Analysis of Flexible Retaining Structures

by

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

for

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

(Release Date: Jun. 2026)

COPYRIGHT

SOFTWARE LICENSE AGREEMENT & DISCLAIMER

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

Table of Contents

List of Figures

List of Tables

CHAPTER 1. Introduction

1.1 Behavior of Flexible Retaining Walls

There are many types of earth retaining structures such as gravity walls, sheet-pile walls, diaphragm walls, and reinforced-earth walls. Each type of wall has its specific advantages and disadvantages, depending on design and construction constraints.

Flexible retaining structures have received much attention since 1940 because of their widespread use in engineering construction. Flexible retaining walls, in contrast to rigid gravity walls, usually have a single row of piles, likely made of timber, reinforced concrete, or sheet steel, drilled or driven into the ground by hammers so that their lower ends are embedded in soil. Tschebotarioff presented the result of his famous large-scale model tests for retaining structures in 1948. One of the most important conclusions was that the distribution of earth pressure on sheet piles is highly influenced by the wall deformations.

The conventional method of design of flexible retaining walls is based on a limit-equilibrium theory that has a theoretical weakness for soil-structure-interaction problems. The behavior of retaining structures is largely a matter of soil conditions and the details of the structural system. Therefore, a rational method of design must include the nonlinear soil-resistance-displacement relationships, spacing, penetration depth, and structural properties of the wall element.

1.2 Available Methods of Design and Analysis

1.2.1 Limit-Equilibrium Analysis

The magnitude and the distribution of earth pressure, as mentioned before, is a function of the displacements of the soil and the related deflections of the structure. Because the problem involved is statically indeterminate, the real stress-deformation relations of soils have been ignored, and designs have been based on simple assumptions. Most of the so-called conventional methods employ the hydrostatic pressure-distribution theory in the limit-equilibrium analysis. Hydrostatic-pressure distribution is assumed for the active and passive pressures on the wall. The design is simplified because the effect of wall deformations on the magnitude or the distribution of the soil pressure is ignored. Unfortunately, the principles of soil-structure-interaction are violated, and an unrealistic pressure distribution can result in limit-equilibrium analyses. While a large number of flexible retaining structures have been designed as indicated above and have given good service, a more rational solution should be achieved.

1.2.2 Subgrade Reaction Method

The desire for improving the analysis of the development of soil resistance with wall deflection has led to the subgrade reaction method. The method solves the differential equation of the bending beam with the soil represented as linear springs that provide resisting forces which increase with lateral deflection. The method became popular and accepted by engineers after Terzaghi published a comprehensive document in 1955 discussing the application of this method and predictions of the subgrade modulus for a variety of cases. Both Rowe (1955) and Richart (1957) published methods of analysis for sheet-pile walls based on the elastic, subgrade-reaction theory.

Rowe took advantage of computers and developed solutions for three different cases of loading on a sheet-pile wall as:

Case 1: cantilever pile subjected to a line load at the top.

Case 2: cantilever pile subjected to triangular loading above the dredge line.

Case 3: cantilever pile subjected to rectangular loading above the dredge line.

The superposition of these results allows for solutions of sheet-pile walls with a variety of boundary conditions. An anchorage system, a surcharge behind a wall, or other combination of loading can also be treated.

Richart's method of analysis considers the sheet pile separately above and below the dredge line. Newmark's numerical procedure for beams, plus the assumption of zero deflection at the tie-rod level, is used to determine shear, moment, and pressure at the dredge line. After forces at the boundary are calculated, the theory of subgrade reaction is used to solve the beam analysis below the dredge line.

The subgrade reaction method discussed above is probably accurate for elastic materials, but these materials cannot generally represent real soils. However, it can be seen that this method might be much improved if the linear soil reaction could be replaced by a nonlinear soil reaction based on realistic test results. The resulting problem can be described as a beam on a nonlinear foundation. Ranhut (1966) and Haliburton (1968) both made attempts at the numerical analysis of flexible retaining walls where the soil response was a nonlinear function of wall deflection. The utility of such solutions is more suitable for the design of flexible retaining walls because the soil response is a function of wall deflection.

1.2.3 Finite-Element Method

It is well recognized that the determination of the actual earth pressure on a wall is extremely complicated, because it depends on the relative movement of the wall and the soil. The finite-element method has been proven to be a powerful tool for solving problems involving equilibrium and the compatibility of stresses and strains in a continuum. Most of the flexible retaining walls pose typical two-dimensional problems, and solutions to these problems have been obtained and well documented in many texts. The use of finite-element methods has allowed attempts at a complete solution of the retaining-wall problem to be made, including the computation of stresses and deformations in both the wall and the adjoining soil. Recent technological developments have opened up the possibility of obtaining improved solutions to the problem of interaction between a flexible structure and the nonlinear, inelastic soil medium. The problems that can now be handled by the finite-element analysis range from simple retaining structures to complex problems such as anchored and braced excavations (Clough and Tsui, 1971).

The significant advantage of finite-element analysis over other methods is the capability of studying the stresses and strains developed on the entire system, since this provides a better understanding of the behavior of the soil and the wall. However, the outcome of this method seems to depend strongly on the use of correct soil data. Limitations in using this method primarily derive from the inability to describe an appropriate constitutive law for soil and to determine the parameters needed for the constitutive models. Although the finite-element analysis is still not developed far enough to be used for design purposes, it has already at the present stage of development proved to represent a valuable tool in a study of the interaction between soil and structures.

As mentioned above, the interactions between the structure and the soil on retaining walls will be governed by the retained soil at the back, supporting soil in the front, as well as the structure itself. The response of the soil is a function of the soil-structure system and of the permissible deflections of this system. It is recognized that the limit-equilibrium analysis does not take into account the nonlinear mobilization of soil reaction with wall deflection in the analysis and so should not be recommended for

designs involving those factors. The finite-element method shows promise in handling the complicated stress-strain relationship for the retaining system. However, the constitutive law of soil has not been understood well enough for this method to be used in engineering practice with confidence. In addition, the finite-element method usually requires much larger modeling efforts than other options.

Recently, the method of analysis for beams on nonlinear foundations employs the soil response curves derived from full-scale experiments and has been accepted as a rational design method by many engineers. The method is commonly referred as the *p-y* method and has been successful in aiding the design of laterally-loaded piles. A flexible earth-retaining structure is similar to piles loaded by lateral forces. The *p-y* method for analyzing a beam on a nonlinear foundation is basically a modified form of the subgrade reaction method. PYWALL utilizes the nonlinear soil-structure-interaction concept from the modified subgrade reaction method for analysis of the behavior of flexible retaining walls.

1.3 Program Features and Change Log

1.3.1 PYWALL 1.0 (1999)

The initial version was able to consider the soil-structure-interaction by using a generalized beam-column model, and then analyze the behavior of a flexible retaining wall or soldier pile wall with or without tiebacks or bracing systems. The user could specify displacement or rotational restraint at any depth of the wall to simulate the bracing or tieback. Additionally, the user could also specify active earth pressure as the distributed load, or ask the program to generate the earth pressure. Passive soil resistance could be represented either by linear springs or nonlinear soil-resistance curves.

PYWALL could internally generate the nonlinear response of soil in the form of *p-y* curves for the passive soil resistance. If desired, the user could also input user-defined soil resistance curves externally from the program. The input parameters consisted of the geometry of the retaining wall, structural properties of the wall element, restrained conditions applied on the wall by the tiebacks or other bracing systems, soil resistance curves, soil unit weight, strength parameters, and the water table. From this, the program would calculate the earth pressure above the dredge line, the wall deflection, shear force, bending moment, and force on the strut.

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

1.3.2 PYWALL 2.0 (2002)

1. Double-precision algorithm was implemented in the procedures of nonlinear iterations for a better solution.
2. The new program introduces internal generation of group-reduction factors for timber-lagging walls and drilled shaft retaining walls based on the spacing between piles in a row. The user is also allowed to input the user-defined reduction factors externally from the program.
3. The program allows the user to enter a load factor to increase or decrease the magnitude of earth pressure that is internally generated by the program. The user may specify the water table differently for the backfill side and the excavation side.

4. Additional user-friendly features were included in the program, such as a short-list of recently-opened files, the option to view the wall geometry, soil layers, and restrains on the wall, and others.

1.3.3 PYWALL 3.0 (2007)

1. Soil-layer data structures and input screens were improved in Version 3. New data entry screens use only the soil parameters that are required for the selected soil type.
2. *p-y* criteria for weak rock was added onto the previously-existing six soil types.
3. The program allows the user to enter short-term and long-term parameters in one input file and provide the output and graphics for both cases for comparison.
4. PYWALL 3.0 also allows the user to study and compare the predicted deflection, bending moment, and shear at different stages of construction based on a simplified approach. However, the program is not able to trace the stress history of soil and structure at each construction stage. Instead, the program uses the earth pressure, restrained boundary conditions, and soil resistance curves (*p-y* curves) at each stage as the initial condition to evaluate the behavior of the system without considering any effect due to the stress history. However, the *p-y* resistance curves are generated based on the excavation depth, wall height, and water table at each new stage to reflect the change in soil resistance. This is not a rigorous approach but it provides the user with some general ideas about the possible pattern of deformation and stress in the system. It completely relies on the user's judgment to interpret the predicted results.

1.3.4 PYWALL (2012)

Maintenance services become more demanding and important for Program PYWALL, which has been used by many engineering companies worldwide. Program PYWALL starts to use the year number as the version sequence starting in 2012. The major improvements in PYWALL 2012 include:

1. For single level or multiple levels of anchors, the program generates trapezoidal shape of earth pressure distribution based on the recommended equations from the National Cooperative Highway Research Program (NCHRP) Report 611.
2. The user can input multiple surcharge loads and the program will add the earth pressure from the surcharge loads and plot the results with the earth-pressure distribution.
3. The program allows the user to enter nonlinear curves of bending moment-vs-flexural stiffness for the wall elements so it takes into account the nonlinearity of the flexural rigidity of the structural members.
4. The program allows the user to change the line width and font sizes in the generated graphical plots.
5. The program text book was updated and released in separate User's and Technical Manuals, both improved plus released electronically in protected PDF format.
6. Use of convenient units are introduced with entries such as lb/ft^3 for unit weight and lb/ft^2 for soil strength.

1.3.5 PYWALL (2013)

1. Introduces compatibility to the new Windows 8 operating system.
2. Improved input screens and overall program interface.

3. New installation routines and ability to perform 'silent' installations (with no user intervention) for network clients.
4. More stable on local area networks (applicable for network licenses only).
5. In large networks, this version introduces the ability to run in secondary software servers that may be automatically used in cases of failure of the primary software server.

1.3.6 PYWALL (2015)

Major improvements in PYWALL v2015 include the following:

1. An option is provided for the user to enter the data of multi-rows of tiebacks or ground anchors. With this new feature, the user can specify the cross-sectional area of the steel rod, the unbounded anchor length, the angle of the installed anchor from the horizontal plane, and the spacing between tiebacks (anchors) at each level.
2. The user may optionally enter an earth pressure coefficient for generating the lateral earth pressure that is equal to the vertical soil pressure multiplied by a user-specified coefficient. In models where this new feature is not needed the user can leave the optional earth pressure coefficient as zero and the program will use the active earth pressure coefficient that is calculated internally. The optional earth-pressure coefficient is helpful for users that want to apply equivalent fluid pressure as load in the wall.
3. Three-dimensional graphics was added to show the modeled wall configuration, soil profile, surcharge loading, restrain conditions and the deflected shape of the wall with the earth-pressure load. This is helpful to verify data entry and also for display in reports.
4. The user may now export graphical data (such as data of *p-y* curve, deflection, shear, bending moment, soil reaction, and earth pressure) to a formatted Microsoft Excel file for additional post-processing activities. This allows users to display in different units or for additional graphical effects in the charts.

1.3.7 PYWALL (2019)

Major improvements in PYWALL v2019 include the following:

1. Ability to enter the prestressing force of multi-rows of tiebacks or ground anchors.
2. New modeling of sloped surface at the excavation side away from the wall (ground sloping in backfill side has been enhanced for better computations of active loads).
3. Users can model the lateral earth pressure based on the wall movement above the excavation level, instead of using the triangular or trapezoidal earth pressure distribution based on the active state.
4. PYWALL v2019 introduces new models of wall sections with automatic computations of the nonlinear response of sections with round or rectangular reinforced concrete, circular/square or octagonal prestressed concrete, circular reinforced concrete with steel inserts, or AISC shapes (W, S, M, HP, HSS or Pipe).
5. The user may now apply the load factor and strength reduction factor for design and analysis based on LRFD procedures.
6. Improved the 3D presentation of the model to display more information of the model and to superimpose some of the results.

7. Added new plots on top of the 3D display so results can be displayed in graphical form and with maximum values (including resulting tieback forces).
8. Implemented a new solution using the "AASHTO Hinge Model" for the solution of retaining walls with multiple tiebacks using automatic computations of trapezoidal active loading distribution (based on the CALTRANS principle of zero moment at all tieback locations except the top one).
9. A new feature was added in PYWALL v2019 to define an equivalent trapezoidal pressure for uniform clay layers using Terzagui and Peck recommendations.
10. Additional surcharge loads can now define a vertical distance from load to wall (to account for truncated Boussinesq loading distributions on the retaining wall).
11. Soil layers can be displayed along the side of all applicable output graphics, as further guidance for the importance of each layer.

1.3.8 PYWALL (2022)

Several new features and enhancements are introduced with PYWALL v2022:

1. Added the option to define models with one or more levels of Struts/Rakers (Options > Apply Strut/Raker) supporting the retaining wall.
2. Introduced a new feature for models with continuous walls and tiebacks or struts/rakers to analyze wales within the tiebacks or struts/rakers (either continuous or simply supported). Analytical results (distributed loading, shear diagram and moment diagram) for wales are provided in the Computation > 3D View plus Show 3DView > Wales Results and in the Graphics menu.
3. Large expansion of database of sheetpile sections:
   3.1. Added the entire listing of Nucor sheetpiles (as of early 2022).
   3.2. Allows users to enter and build their own database of sheetpile sections (Options > User Defined Sheet Piling Sections) using new interface or in editable text files (which can be shared with other users).
4. Introduced a new feature so users can define walls that extend above the top of the retained soil layers in the backfill side.
5. Added a new feature to estimate internally the values of e_{50} and $k_{(py)}$ for applicable soil layers, if those entries are left as zero.
6. Added a new option to estimate trapezoidal loads for models with anchored walls (based on Peck and Terzaghi) (Options > Earth-Pressure Options > Generate Trap.-Distr. Pressure (Anchored Walls, Terzaghi and Peck).
7. All changes to plot/graphics (titles, legends, etc.) done by the user are now associated and saved with the PYWALL model.
8. Users are now able to enjoy the new/enhanced v2022 Technical Manual in standard PDF file format.

1.3.9 PYWALL (2025)

New features and enhancements in PYWALL v2025 are as follows:

1. Added new *p-y* model for Modified Stiff Clay w/o Free Water (similar to existing model in LPILE and GROUP).

2. Added a new feature to enter any user-specified distributed uniform lateral load that may be acting along the height of the wall.
3. Broadened the program so it can analyze models with only retained water (where the water in the retained side is above the retained soil).
4. Introduced new computational features and controls for evaluating the response of temporary braced and anchored excavations of retained clays using the FHWA method.
5. A new reporting tool produces a pre-formatted Microsoft Word file that contains input data of the model along with the user's selection of output tables and charts.
6. New display of wall/model geometry when opening existing models, with added functionality to double click on any element (soil, wall, tieback/strut, cross section, etc.) to display its associated data entry.
7. Users are provided with additional controls for the response of continuous walls (either secant or tangent walls). Soldier-pile walls that are spaced very closely may behave like continuous walls due to the arching effect of the retained soils. The new Options > Controls for Continuous Walls allow users to evaluate either condition.
8. Models with Anchored Walls (either FHWA or Terzaghi & Peck) are provided with additional control options (Options > Options for Anchored Walls) for the computation of equivalent trapezoidal active earth pressures.
9. Improved program functionality by allowing the entry of distances from top of wall, instead of nodal numbers, for the optional entries of concentrated loads on the wall, imposed displacement/rotation of the wall, specified lateral/rotational springs on the wall and/or user-specified *p-y* curves on the wall.
10. EnCPT Import. Ability to open data files with complete soil layering and properties defined by the separate EnCPT program. This separate software allows geotechnical consultants to offer to their clients new services for detailed interpretation of CPT tests into various layering and soil properties that can be used quickly and customized in PYWALL v2025 models.
11. Various computational improvements:
    - 11.1. Convergence for models with clay layers that may exhibit divergence behavior.
    - 11.2. Improved solution (moment, shear) for models with specified slope along wall.
    - 11.3. Smoother estimation of soil reaction along wall.
12. Introduction of Cloud Subscription Licensing option and management.

1.4 Organization of PYWALL Manuals

The documentation provided with the computer program PYWALL consists of three volumes: User's Manual, Technical Manual and Examples Manual. The User's Manual contains full documentation about the operation of the program on personal computers and covers installation, preparation of input data, program execution, view of computational results, and example applications.

The Technical Manual provides information about the theories of beam on elastic foundations and lateral load transfers in different types of soil/rock strata. The Technical Manual also includes other helpful technical references related to concepts utilized in program PYWALL. A separate Examples Manual has models for various sample application problems, some of which are compared to approximate hand solutions.

The distribution of all three manuals is made via standard Adobe© PDF file format (protected from external modifications). All three manuals are installed with the program and accessed via any one of the following three methods: i) from the Windows Start Menu, ii) from the Ensoft tile in Windows 8, or iii) from the Help menu within the PYWALL program.

1.5 Typographical Conventions

To orient the user to different program features, certain terms are set in typefaces that distinguish them from the body text. The following formatting conventions are used throughout the manuals:

- Commands that are typed directly on the keyboard: "Enter" ; "F1" ; "Esc"
- Commands with keys simultaneously pressed in the keyboard: "Alt+F" ; "Ctr+O"
- Menu items and other text displayed on the screen are in sans serif:
  From the File menu choose Open
- File names and names of directories and folder and icon names are in italics:
  Open the example file *example1.py5d*
- Internet and electronic mail addresses are underlined:
  Send email to support@ensoftinc.com

1.6 Contents of the PYWALL Package

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

- One USB Key (sometimes known as hardware key or dongle),
- 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).

1.7 Default Installation Directories

The directories for installation of program files and for example files are selected by the user during program installation. Default values are provided by the installer and confirmed or changed by the user.

1.7.1 Directory for Main Program Files

Program files are installed by default in the following directory:

Root Drive:\Program Files (x86)\Ensoft\Pywall2025

In most cases the *Root Drive* is the *c:* drive so the default is:

c:\Program Files (x86)\Ensoft\Pywall2025

Main program files are the following:

Pywall2025.exe ..... is the main Windows-program module (executable/application),

PYWALL2025_DLL.dll ..... is the program engine for direct computations.

User's, Technical and Example Manuals are also included in the main installation directory.

1.7.2 Directory for Example Files

Several modeling examples with input files (files with name/extension of the type *filename.py7d*) are installed on a separate folder that defaults to:

Root Drive:\Ensoft\Pywall2025-Examples

In most cases the *Root Drive* is the *c:* drive so the default is:

c:\Ensoft\Pywall2025-Examples

1.8 Hardware Requirements

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

- Any personal computer with an Intel® Pentium® or newer processor.
- A hard disk with at least 50 MB of free space.
- Microsoft Windows™ 2000, XP, VISTA, Windows 7, Windows 8, Windows 8.1, Win 10 or 11 in either 32-bit or 64-bit releases. A release of the Microsoft Windows™ system that is newer than those listed will need a new update of the software.
- Minimum of 640 KB of free RAM memory.
- Open (available) USB port (v2 or newer).
- Graphics card compatible to the Microsoft Windows™ operating system that is being used and also compatible with OpenGL.
- A mouse or similar pointing device.
- Optional: Any windows-compatible printer.

1.9 Technical Support

Although computer program PYWALL was designed to be distinguished by its ease of use and by its accompanying User's Manual, some users may still have questions. The technical staff at ENSOFT strongly supports all registered users with questions related to the installation or use of PYWALL, 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 includes free download of the latest version/update and free technical support for installation and usage issues as described below.

1.9.1 Preferred Methods of Software Support

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

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

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

- full software version/release/update (obtained from the Help > About dialog),
- a description of the user's problem or concern,

- attach a copy of the input-data file that is associated with the issue/concern (files with name/extension of the type *filename.py8d*), and
- 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).

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

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.

1.9.2 Upgrade Verification and Internet Site

Starting from PYWALL v2012 the software provides options for the user to check the most recent maintenance release through an internet connection by selecting Help > Check for Updates. This command starts the default internet browser and will display the user's maintenance expiration date, the user's software release number and the most recent release number that is available for downloading.

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

1.9.3 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 a program maintenance that has not expired can be found on the Ensoft website at https://www.ensoftinc.com/order_form

1.9.4 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 and Getting Started

2.1 Installation Procedures

The PYWALL software licenses are provided with an associated USB Key (hardware key or dongle) or as cloud subscriptions with yearly renewals (no USB Key). The USB Key consists of a device that is attached to an empty USB port (or USB hub) of the computer in use (or in the designated software server in the case of local network licenses).

Users with standard single-user licenses (USB Key) 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 (USB Key) can check the following link to a PDF with Network Installation Notes:

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

Users with cloud subscription licenses (no USB Key) can check the following link to a PDF with Installation Notes:

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

2.1.1 Installation of Single-User Version

This version of PYWALL is compatible with all recent versions of the Microsoft Windows® operating systems, including Windows 11, 10, 8.1, 8, 7, Vista, XP, 2000 in either 32 and 64-bit releases.

The following guidelines are provided for the installation process of PYWALL 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 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 Figure 2.1 should appear.
3. If the user installs from a downloaded file, then please run the downloaded file (double click) and go to instruction #4.
4. Click anywhere on the PYWALL 2025 icon and then click on the Install Standard button to start the installation of PYWALL.
5. The user should read the license agreement (partly shown in Figure 2.2). Users may also review the License Agreement online in the following link:
6. https://www.ensoftinc.com/doc/Ensoft%20License%20and%20Disclaimer.pdf
7. 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.

5. Select Single-User License in Figure 2.3 then click Next. For cloud installations see Section 2.1.2. For network installations see Section 2.1.3.
6. The user will be provided with an option to select a drive and directory for the installation of example files (see Figure 2.4). Default installation directory is the following:
8. (Root Drive):\Ensoft*Pywall2025*-Examples
7. The user will also be asked to select a drive and directory for the installation of PYWALL (see Figure 2.5). Default installation directory (varies according to the Windows release where it is installed) is one of the following:
9. *(Root Drive):\Program Files (x86)\Ensoft\Pywall2025*
10. *(Root Drive):\Program Files\Ensoft\Pywall2025*
11. If the desired directory does not exist, the installation program will automatically create a new directory in the chosen hard drive.

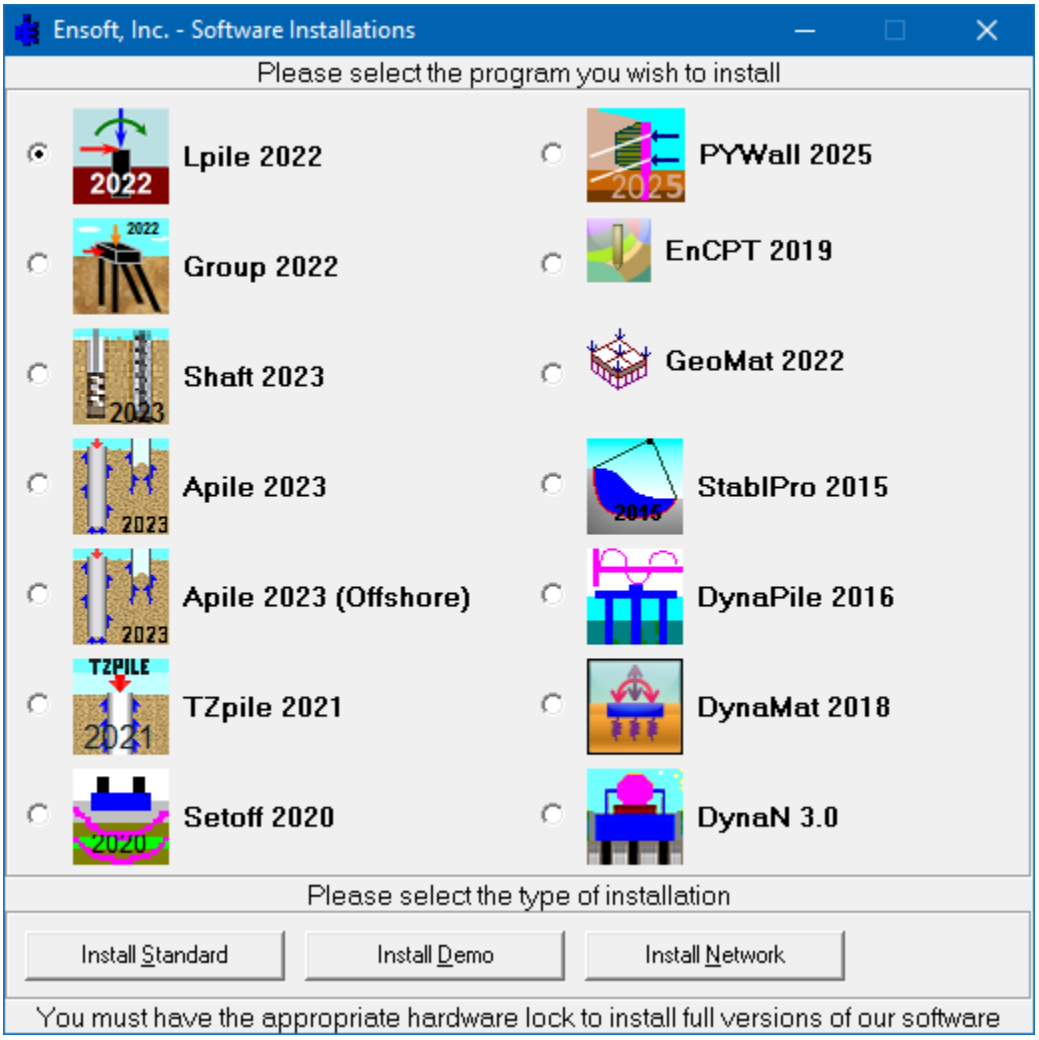


Figure 2.1 Main Installation Screen for ENSOFT Software (may change with time)

8. During the installation the user will be asked to set the file extension association for opening PYWALL v2025 input data files (see Figure 2.6). If the user agrees (leaves the default check mark)

then double clicking (or running) any input data file with extensions of the type *filename.py8d* will start the installed PYWALL v2025 software.

9. The user will be prompted to confirm the shortcut directory name that will be created in the Windows Start Menu (See Figure 2.7). The default is *Start Menu/Programs/Ensoft/Pywall2025*. Windows 11, 10 and 8 will automatically create an Ensoft tile with the same shortcuts.
10. After copying files, the installer may check if the user's Windows OS has the necessary Microsoft Visual C++ drivers. If those are not found the user will be prompted to install (Figure 2.8) and must agree to terms and select the Install button. If the drivers are present, the user will be prompted with Figure 2.9 where it can simply select the Close button.

After the installation is finished, it is usually not necessary to reboot Windows for the program to run. The user may run the program by selecting PYWall v2025 from the standard links installed in the Microsoft Windows® Start Menu: Start Menu > All Programs > Ensoft > PYWall2025

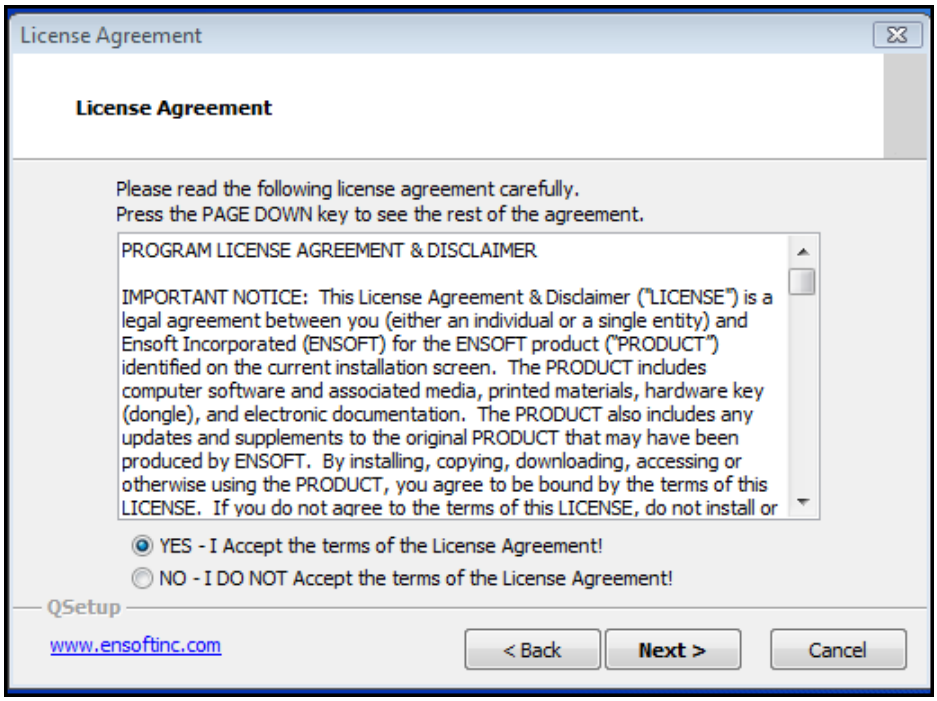


Figure 2.2 Installation Screen with License Agreement (may change with time)

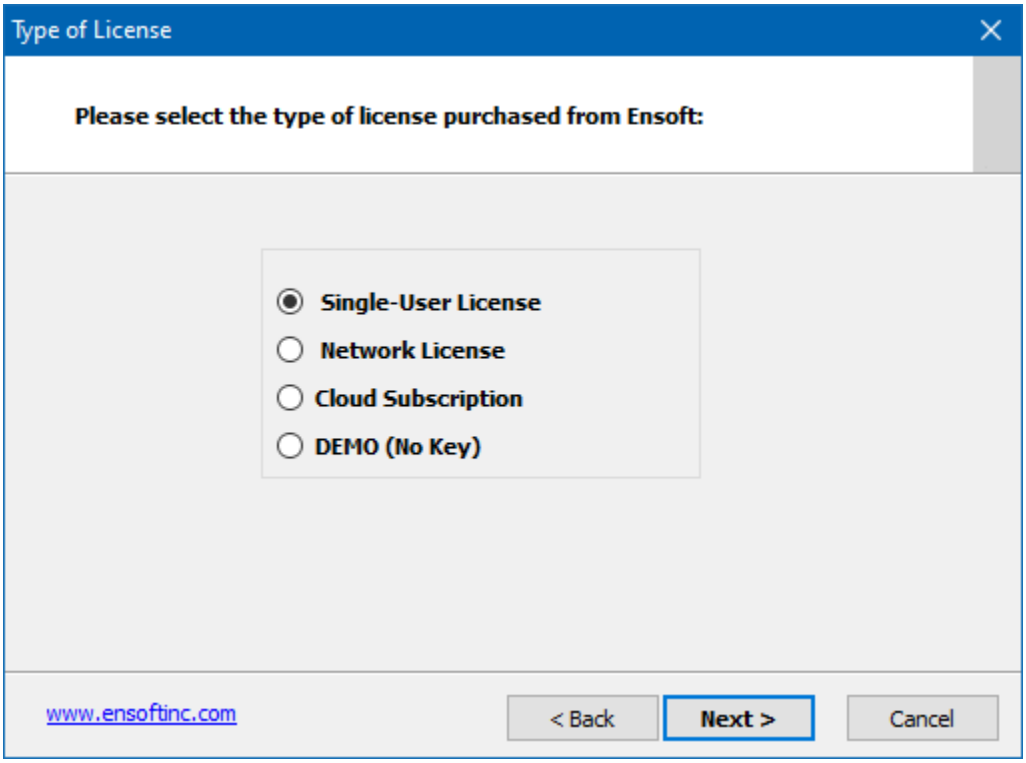


Figure 2.3 Selection of Single-User License (may change with time)

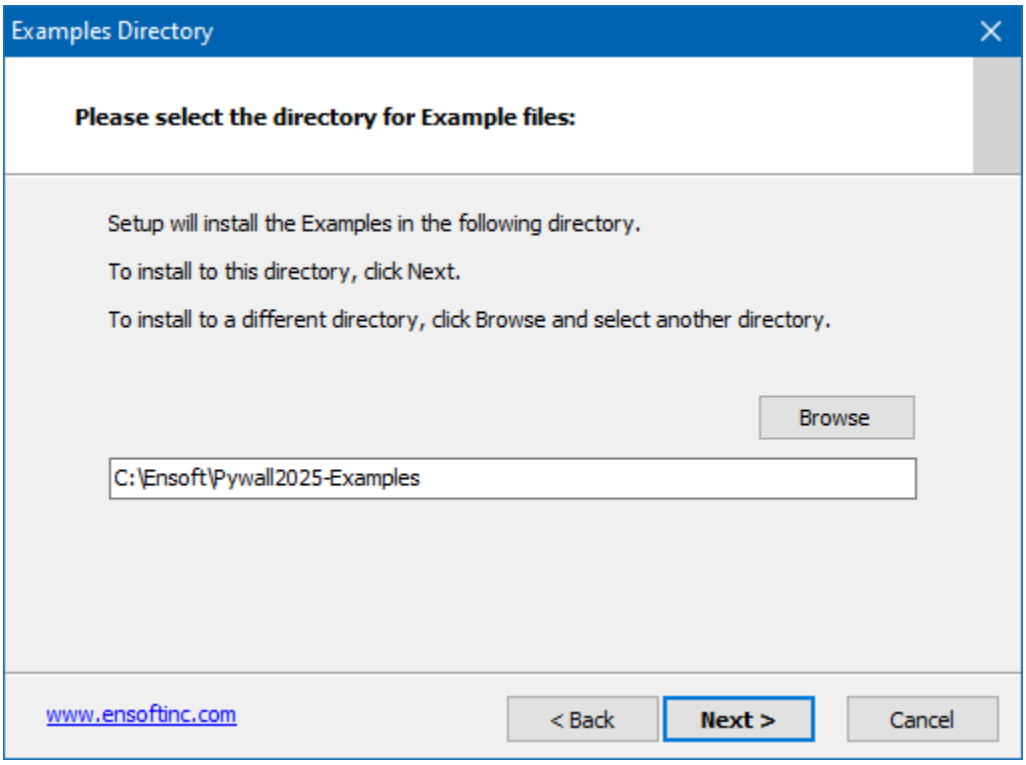


Figure 2.4 Default Installation Directory for Example Files (may change with time)

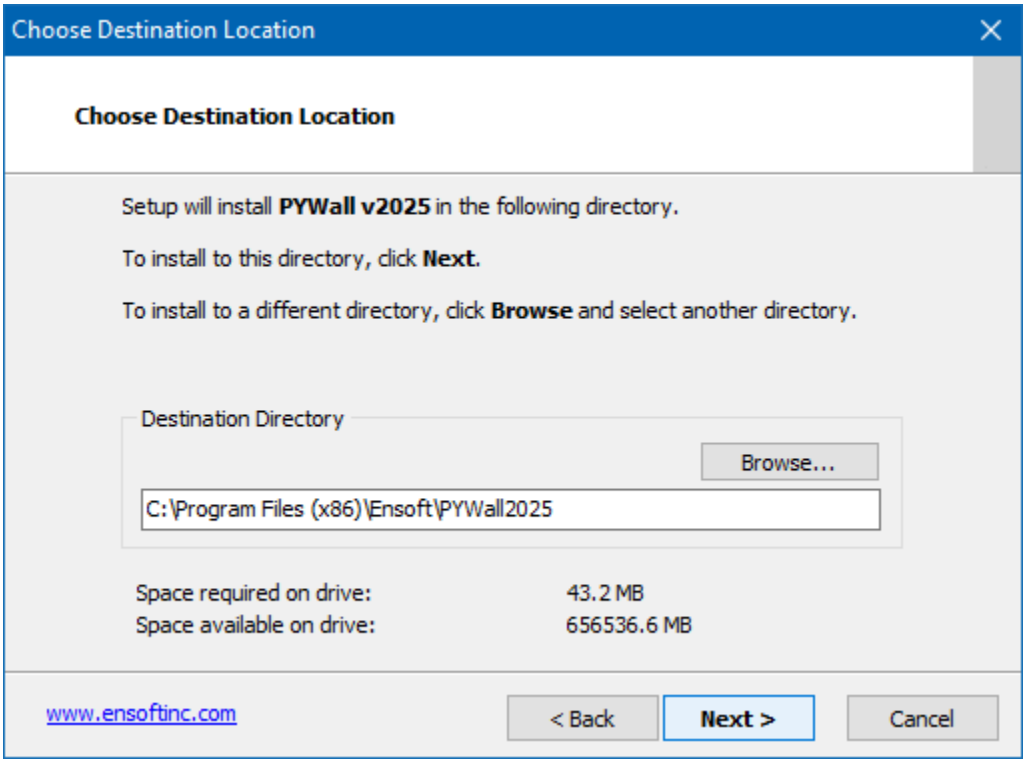


Figure 2.5 Default Installation Directory for Program Files (may change with time)

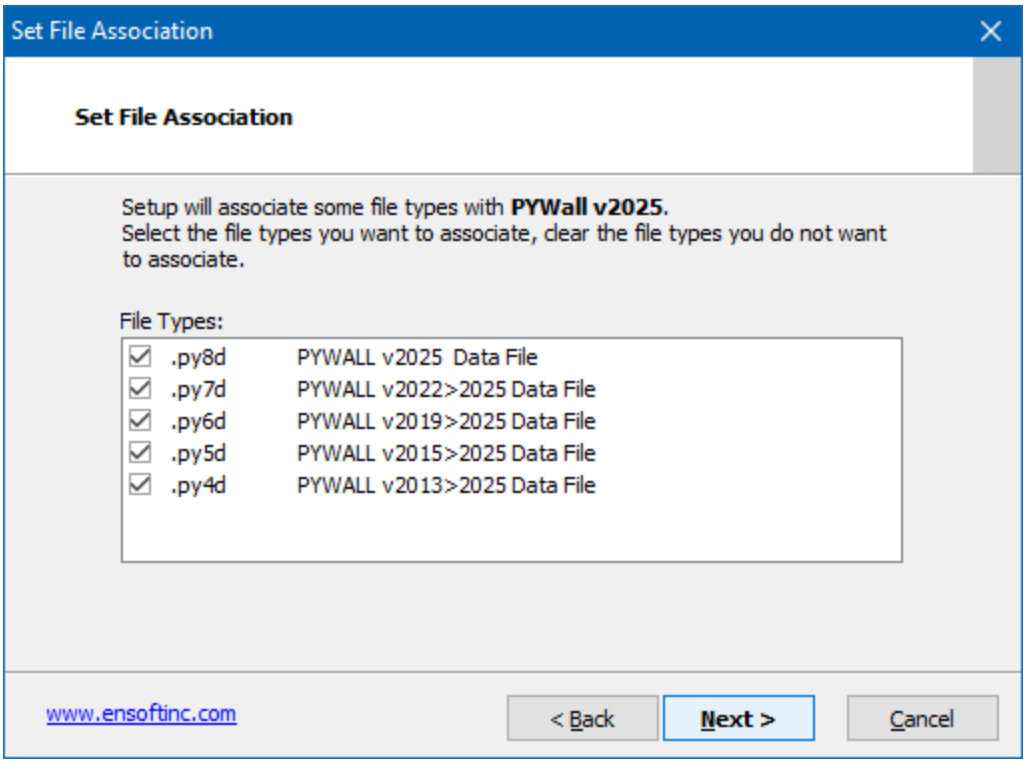


Figure 2.6 File Extension Association for PYWALL Data Files (may change with time)

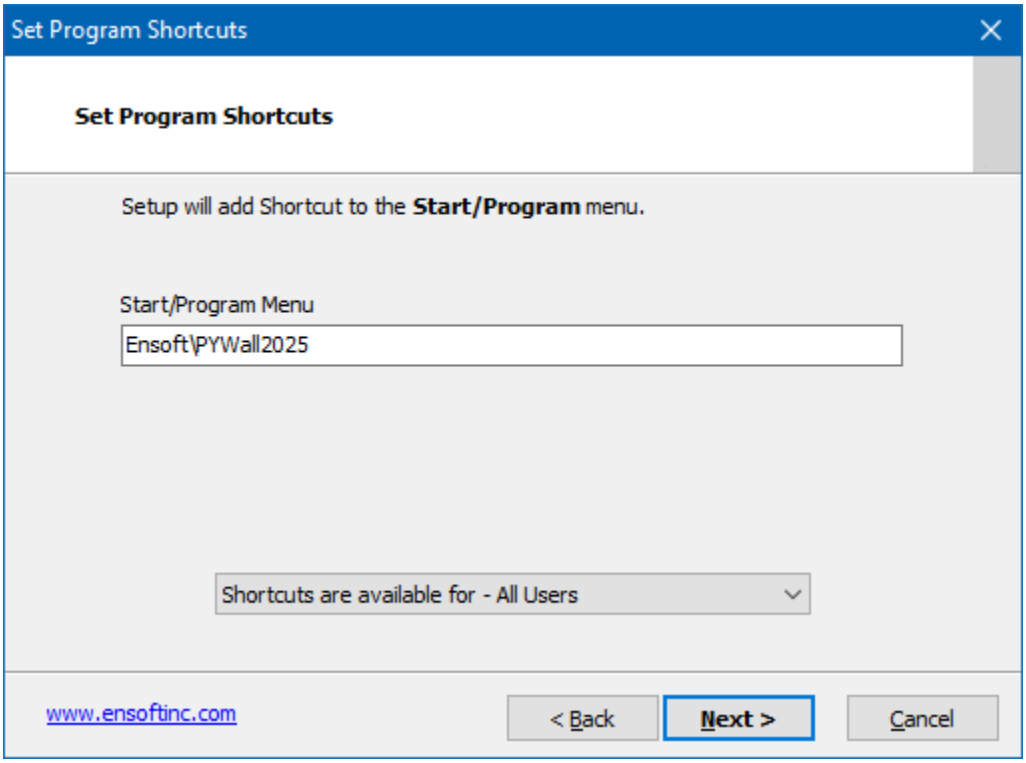


Figure 2.7 Default Shortcut Folder in Windows Start Menu (may change with time)

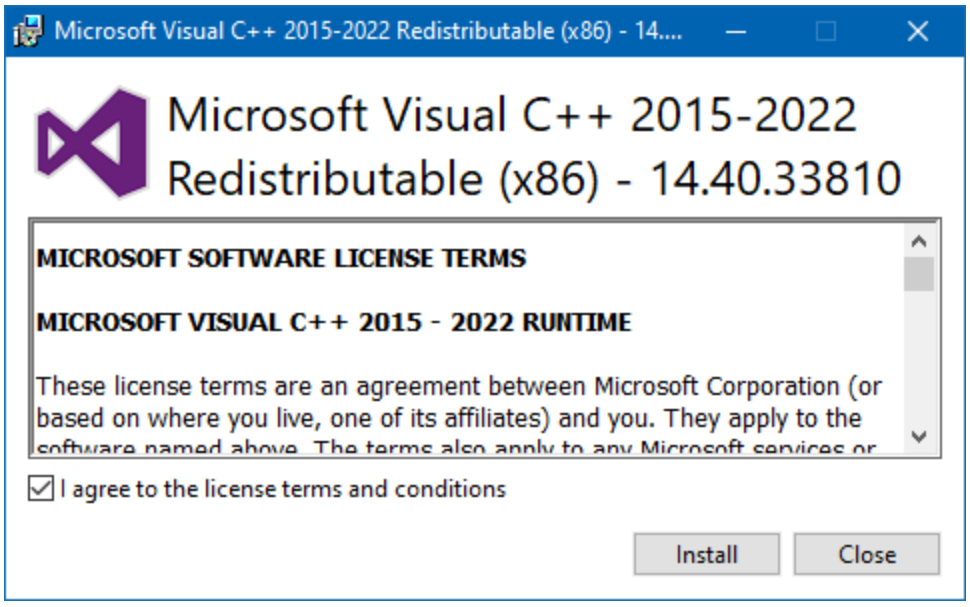


Figure 2.8 Consent for Installation of Microsoft Visual++ Drivers

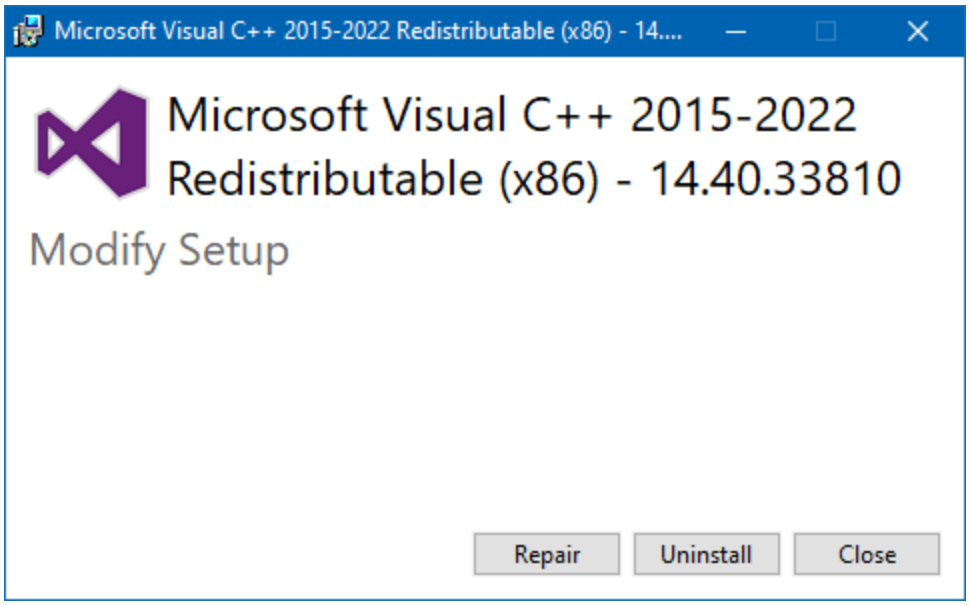


Figure 2.9 Found Installed Microsoft Visual++ Drivers

2.1.2 Installation of Cloud Subscription Version

The latest Ensoft Software updates are compatible with all recent versions of the Microsoft Windows® operating system, including 11, 10, 8.1, 8, 7, Vista, XP and 2000. The following guidelines are recommended for the installation of cloud licenses.

1. Download the purchased Ensoft Software from our website using the following link:
2. https://www.ensoftinc.com/downloads
3. Run (double click) the downloaded executable file and select Next to continue.
4. The user should read the license agreement (partly shown in Figure 2.2). Users may also review the License Agreement online in the following link:
5. 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.
3. Select Cloud Subscription in Figure 2.3 then click Next. For single-user installations see Section 2.1.1. For network installations see Section 2.1.3.
4. The user will be provided with an option to select a drive and directory for the installation of example files (see Figure 2.4). Default installation directory is the following:
7. (Root Drive):\Ensoft*Pywall2025*-Examples
5. The user will also be asked to select a drive and directory for the installation of PYWALL (see Figure 2.5). Default installation directory (varies according to the Windows release where it is installed) is one of the following:
8. *(Root Drive):\Program Files (x86)\Ensoft\Pywall2025*
9. *(Root Drive):\Program Files\Ensoft\Pywall2025*

10. If the desired directory does not exist, the installation program will automatically create a new directory in the chosen hard drive.

6. During installation the user will be asked to set the file extension association for opening PYWALL v2025 input data files (see Figure 2.6). If the user agrees (leaves the default check mark) then double clicking (or running) any input data file with extensions of the type *filename.py8d* will start the installed PYWALL v2025 software.

7. The user will be prompted to confirm the shortcut directory name that will be created in the Windows Start Menu (See Figure 2.7). The default is *Start Menu/Programs/Ensoft/Pywall2025*. Windows 11, 10 and 8 will automatically create an Ensoft tile with the same shortcuts.

8. After copying files, the installer may check if the user's Windows OS has the necessary Microsoft Visual C++ drivers. If those are not found the user will be prompted to install (Figure 2.8) and must agree to terms and select the Install button. If the drivers are present, the user will be prompted with Figure 2.9 where it can simply select the Close button.

After the installation is finished, it is usually not necessary to reboot Windows for the program to run. The user may run the program by selecting PYWall v2025 from the standard links installed in the Microsoft Windows® Start Menu: Start Menu > All Programs > Ensoft > PYWall2025

2.1.3 Introduction of Network Version

Special network licenses and USB hardware keys (network keys) are available for users that want to operate PYWALL on a Windows network. A network version is licensed for employees of one or more specific office sites that are within white listed subnets. Discounted rates apply for purchases of multiple network seats.

Network versions of PYWALL have special subroutines written for installations in "license manager" and for installations of "individual clients". The "license manager" is known as the computer that will be carrying the network key provided by ENSOFT, INC. The license manager is not necessarily the same as the existing network server but rather just a computer carrying the network key and managing the license. Any computer in the existing Windows network may be designated license manager for PYWALL 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 (or of the Ensoft Utilities) is installed on its hard drive. Software "clients" may be all other computers of the network that have the program installed as client. Client computers do not need any hardware key attached to their local system. The program installed in "client computers" will be allowed to run as long as the computer designated as "license manager" is accessible on the network with the proper operating system and with its network key secured in place.

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

2.1.4 Silent Installations of Single, Client or Cloud Licenses

For installation of single-user licenses, cloud subscription licenses and network licenses on local client computers (not on License Server) there is an option for command-based installations that are completely

silent (performed without other user input). Instructions for silent installations can be downloaded from the Ensoft web site using the following link:

https://www.ensoftinc.com/doc/Silent_Single-Client-Cloud_Installations.pdf

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

2.1.5 Backup of Original Software

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

2.1.6 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 PYWALL menu. Software users may freely download the latest program update from the PRODUCTS > Downloads link in the following site: *http://www.ensoftinc.com*

2.2 Getting Started

2.2.1 Starting the Program

In the very first start of the program, after a fresh installation, the user will be asked to personalize the installed copy of PYWALL (Figure 2.10). This dialog allows the user to enter any name to distinguish the software installation. The selected name will be included at the beginning of all output files that are generated by this particular installation of PYWALL. The selected name can be changed any time afterwards from the Help > User Information menu.

In any software start after the first, the program displays a blank screen containing the following top-menu choices: File, Data, Options, Computation, Graphics, Window, and Help along with a variety of speed buttons (Figure 2.11). If the user starts the software from an existing model (using Windows OS file extension association), the software opens with a background display of main graphical features of the working model.

As a standard Windows feature, pressing the "Alt" key displays the menu operations with underlined letters. Pressing the underlined letter after pressing "Alt" is the same as clicking the operation. For example, to open a New File, the user could press "Alt+F" followed by "N", or "Ctrl+N", or click File > New. Hovering the mouse cursor over a speed button displays a quick reference of its function.

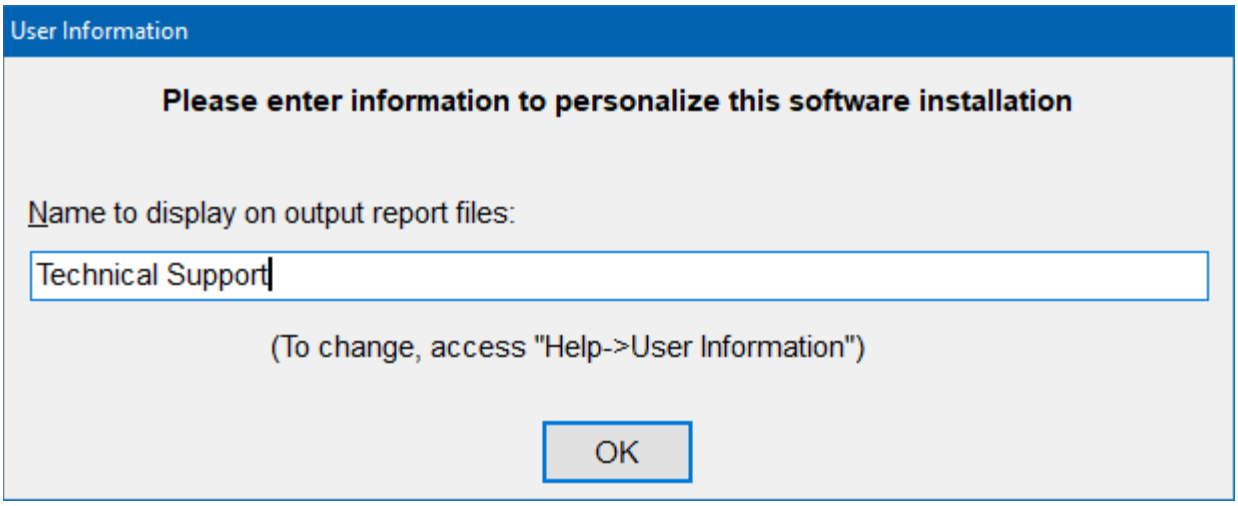


Figure 2.10 User Information on Initial Program Start (after a fresh installation).

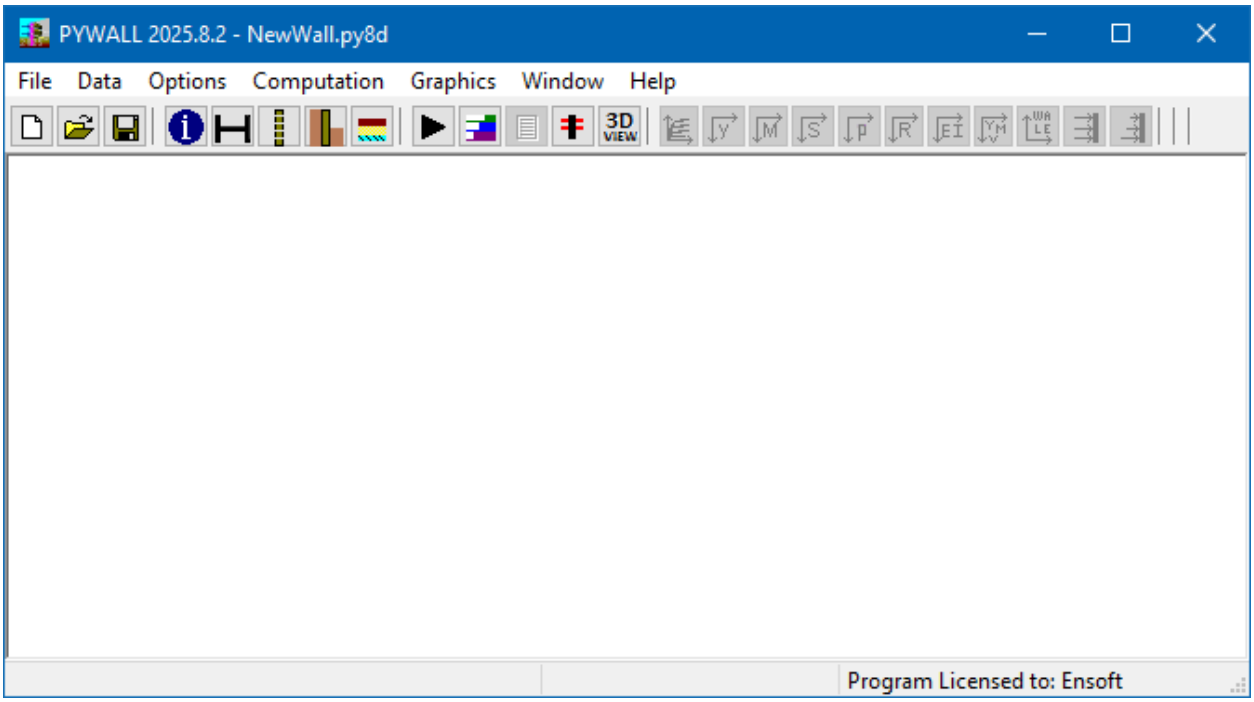


Figure 2.11 Blank Screen on Subsequent Program Starts.

2.2.2 File Management

The File menu option contains five standard entries, as shown in Figure 2.12; they are:

- New to create a new data file.
- Open… to open an existing data file.
- Save to save input data under the current file name.
- Save As… to save input data under a different file name.
- Exit to exit program PYWALL.

A history list of up to the last ten recently-opened data files is located between the Save As and the Exit options. This list varies on each user's computer and allows for easy access to recent data files.

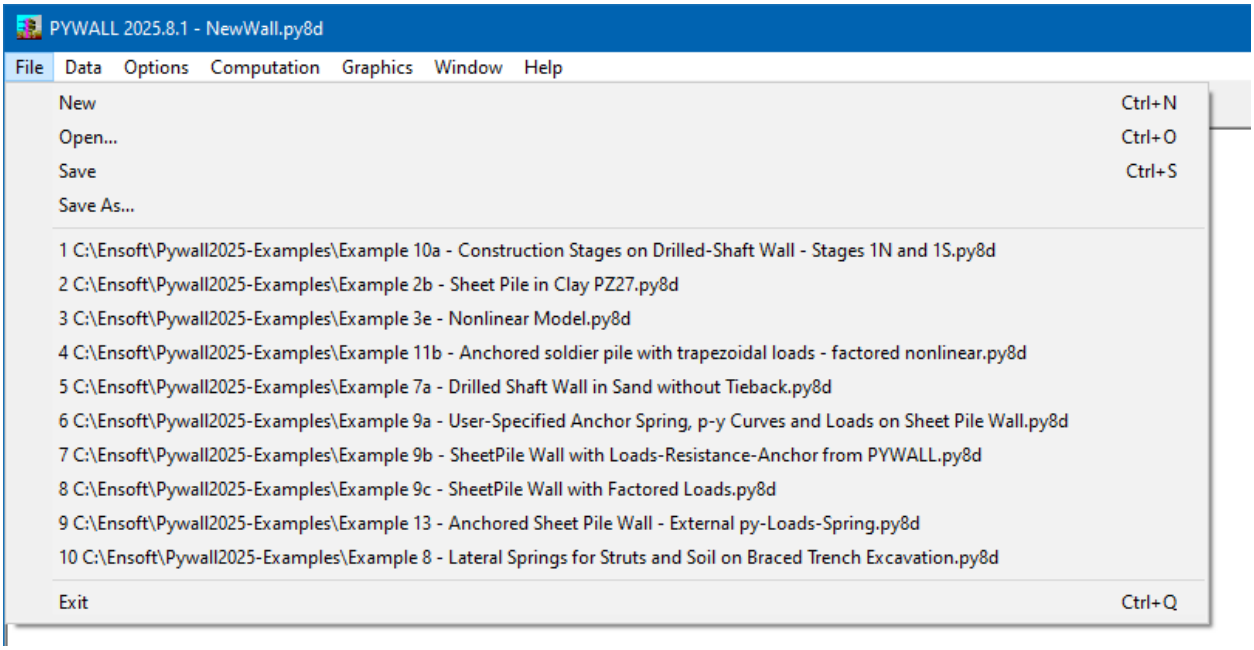


Figure 2.12 File Menu

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

File Name Extension	Usage Description	File Format	Example Files
*.py8d	Input-data file	Encrypted file	Example 1.py8d Example 2.py8d
*.py8o	Output-data file	Text file	Example 1.py8o Example 2.py8o
*.py8p	Plot-data file	Text file	Example 1.py8p Example 2.py8p

Table 2.1 Files created in PYWALL runs

2.2.3 Data Input of Application Problem

The Data menu contains several submenus, as shown in Figure 2.13. The choices are listed below, along with a general description of their use. In the default conditions (with no additional Options selected), only Title, Cross Sections, Wall Properties, Wall Configuration, and Soil Layer Data will be enabled (active). Please refer to the next section (Options menu) to learn how to activate other entries under the Data menu.

Title..... single line of text with a general description for the project.

Cross Sections..... structural properties of the wall elements.

Wall Properties ..... wall dimensions and assignment of the wall sections.

Wall Configuration..... geometry data of the wall, water-table depths and other parameters.

Soil Layer Data ..... type and depth of each soil/rock layer and to enter the associated soil/rock parameters. These definitions are used for the automatic generation of active earth pressure and soil resistance (*p*-*y*) curves.

Cyclic Model Data ..... data for models using cyclic response curves (may not produce any noticeable difference on some *p*-*y* curves). This entry is disabled by default but can be enabled if the user sets a check mark on Options > Use Cyclic Model for p-y Curves.

Distributed Active Earth Pressure….user-specified distributed earth pressure or loading that should be applied to the wall. This entry is disabled by default but can be enabled if the user sets a check mark on Options > Earth-Pressure Options > Earth Pressure Specified by User.

Additional Surcharge Loads .....additional surcharge loads on or near the wall. This submenu is disabled by default but can be enabled if the user sets a check mark on Options > Apply Additional Surcharge Loads.

Concentrated Loads on the Wall ....concentrated loads on any nodal point of the wall. This submenu is disabled by default but can be enabled if the user sets a check mark on the submenu Options > Apply Concentrated Loads.

Tieback (Ground Anchor) Data .....details for tiebacks in tabular entry. This submenu is disabled by default but can be enabled if the user sets a check mark on the submenu Options > Apply Tieback (Ground Anchor).

Strut/Raker Data..... details for struts and/or rakers in tabular entry. This submenu is disabled by default but can be enabled if the user sets a check mark on the submenu Options > Apply Strut/Raker).

Boundary Condition on Deflection or Slope restrained deflection or slope on any nodal point of the wall. This submenu is disabled by default but can be enabled if the user sets a check mark on the submenu Options > Apply Restrains on Deflection or Slope.

Lateral Springs and Rotational Restraints lateral springs and rotational restraints on any nodal points of the wall. This can be used to simulate a linear-elastic behavior of tiebacks or struts. This submenu is disabled by default but can be enabled if the user deletes the default check mark on the submenu Options > Apply Lateral Springs and Rotational Restraints.

Modification Factors for p-y Curves modification factors for soil resistance (*p*) and/or lateral movement of the pile (*y*) at user-specified depths. This submenu is disabled by default but can be enabled if the user sets a check mark on the submenu Options > Apply External p-y Modification Factors.

Sets of External p-y Curves..... user-specified relationships of soil resistance (*p*) and lateral movement of the pile (*y*) at user-specified depths. This can be used to specify lateral resistance of certain soil layers (when depth of external *p*-*y* curve is below excavation) or also for simulation of the nonlinear behavior of tiebacks or struts (when depth of external *p*-*y* curve is within the excavation). This submenu is disabled by default but can be enabled

if the user places a check mark on the submenu Options > Apply User-Specified p-y Curves (External).

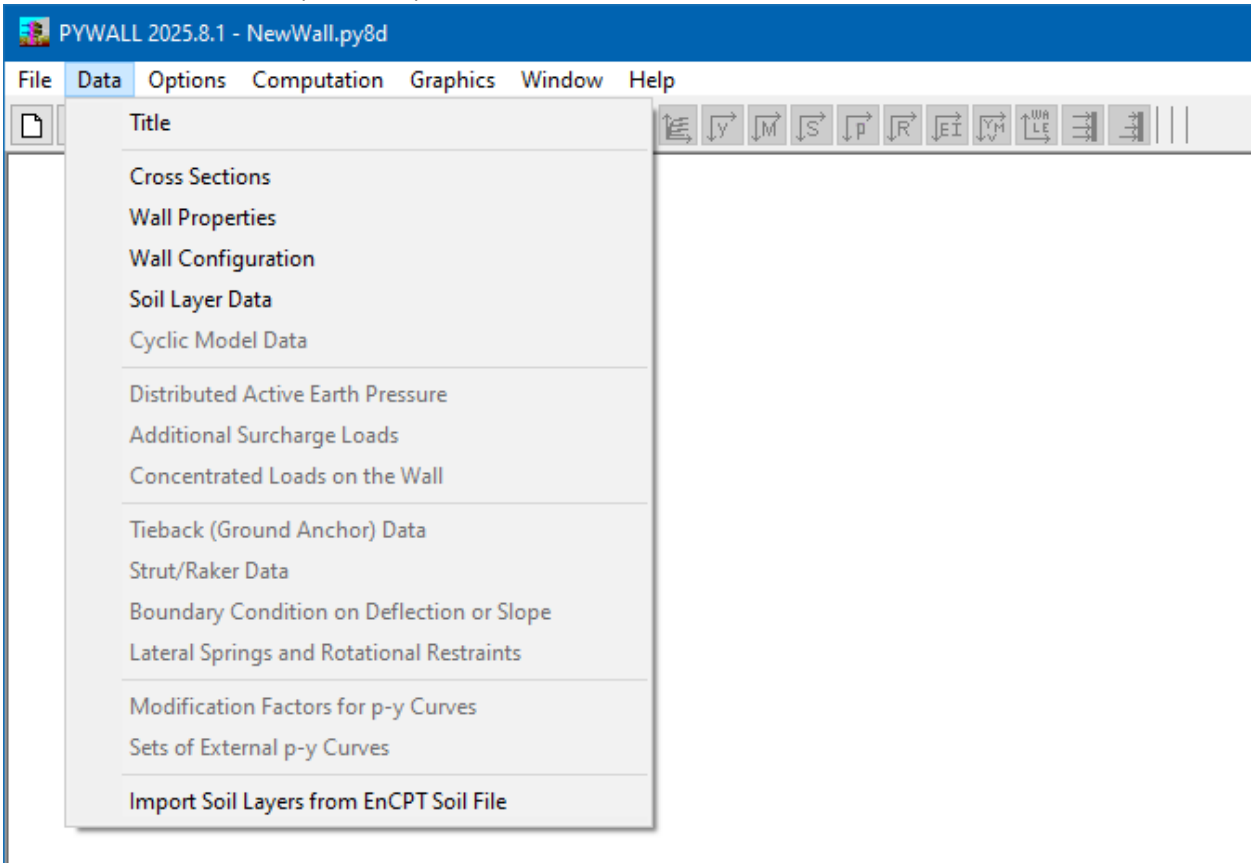


Figure 2.13 Data Menu

2.2.4 Options Menu

The Options menu provides access to several computation alternatives. Submenu choices, shown in Figure 2.14, are briefly described below.

Units .....this option allows the user to select either English Units (using pounds, feet, and inches) or SI Units (using kiloNewtons and meters) or any other consistent set of units of force and length. English units are enabled, as a default, for all new data files. The user may change the system of units as many times as desired and values that were previously inputted will be automatically converted by the program.

Earth-Pressure Options .....when selecting Earth Pressure Specified by User the user may enter any arbitrary type of active earth pressure or loading that is applied to the wall. A standard active earth pressure of triangular-shaped distribution is generated internally by the program when selecting Generate Triangular-Distribution Earth Pressure. The Generate Trapezoidal-Distribution Earth Pressure (Anchored Walls, FHWA), instructs the program to generate internally active earth pressures with a trapezoidal-shaped distribution. The Generate Traps-Distr. Earth Pressure (Anchored Walls, Terzaghi and Peck), instructs the program to generate internally active earth pressures with a trapezoidal-shaped distribution following the Peck and Terzaghi criteria (see Technical

Manual). The trapezoidal distributions are usually recommended by AASHO for walls that are anchored with tiebacks or supported by struts. The selection of Generate Earth Pressure Based on Soil Movements instructs PYWALL to generate earth pressures based on the wall movement as described in the PYWALL Technical Manual.

Generate p-y Curves .... this is selected by default and instructs the program to calculate internally the soil response (*p-y*) curves for the soil types and layers that are specified by the user following recommended criteria.

Use Cyclic Model for p-y Curves.....this is selected to instruct PYWALL for the internal generation of *p-y* curves that are modified for cyclic loads (not all curves are affected by the cyclic loading). The menu Data > Cyclic Model Data is enabled when a check mark is placed on this option.

Option for Continuous Walls.....this selection provides controls related to the response of continuous walls (either secant or tangent walls). This menu selection is enabled for all models, since soldier-pile walls that are spaced very closely may behave like continuous walls and the user may decide to evaluate both conditions.

Apply Concentrated Loads .....this is used if the user wants to specify any type of concentrated loads on any nodal point of the wall. The menu Data > Concentrated Loads on the Wall is enabled when a check mark is placed on this option.

Apply Additional Surcharge Loads .this is selected if the user wants to enter additional surcharge loads on or near by the wall. The menu Data/Additional Surcharge Loads is enabled when a check mark is placed on this option.

Apply Restraints on Deflection or Slope this is selected if the user wants to specify any restrained deflection or slope on any nodal point of the wall. The menu Data > Boundary Condition on Deflection or Slope is enabled when a check mark is placed on this option.

Apply Lateral Springs and Rotational Restraints this is selected if the user wants to enter lateral springs and/or rotational restraints on any nodal points of the wall. The menu Data > Lateral Springs and Rotational Restraints is enabled when a check mark is placed on this option.

Apply Tieback (Ground Anchor).....this is selected if the user wants to specify any type of tiebacks or ground anchors on any location of the wall. The menu Data > Tieback (Ground Anchor) Data is enabled when a checkmark is placed on this option.

Apply Strut/Raker ..... this is selected if the user wants to specify any type of strut or raker (compression elements) on any location of the wall. The menu Data > Strut/Raker Data is enabled when a checkmark is placed on this option.

Apply User-Specified p-y Curves (External) this is selected if the user wants to specify arbitrary soil response (*p-y*) curves externally from the program (in this situation, the external *p-y* curves must be calculated by the user using other methods and entered into the program). The menu Data > Sets of External p-y Curves is enabled when a check mark is placed on this option.

Apply External p-y Modification Factors this is selected if the user wants to enter modification factors for soil resistance (*p*) and/or lateral movement of the pile (*y*) at specified depths (instead of letting the program perform internal computations of these factors). The menu Data/Modification Factors for p-y Curves is enabled when a check mark is placed on this option.

Conduct Short-Term and Long-Term Analyses this is selected if the user wants to evaluate in one data file the effects of short-term and long-term soil properties and loading. The menu Data > Soil Layer Data/Short Term and Data > Soil Layer Data/Long Term are enabled when a check mark is placed on this option.

Conduct Staged-Construction Analysis… this is selected if the user wants to conduct analysis at different construction stages of the retaining wall. With this option the program will not trace the stress history of the soil and structure at each stage. Instead, the program will use the earth pressure, restrained boundary conditions, and soil resistance curves (*p*-*y* curves) at each stage as the initial conditions with which to evaluate the behavior of the system. The main menu Staged Construction appears when a check mark is placed on this option.

User-Defined Sheet Piling Sections… this is an option for users to open (and modify/expand) a file with their own custom table of user-defined sheetpile sections. The file '*SheetPiling_User_in.csv*' is installed and selected as default by the program and it can be further modified and expanded by the users.

Option of Text Editor….the user should input in this box the complete path and command line for the preferred text editor or word processor that will be used to examine and print the output and processor plain text files created by the program. As a default, the command line is used to operate the standard Microsoft Notepad text editor that is included with most Windows installations. However, there may be some instances when the output files are too large for Notepad and a different text editor or word processor should be used. In those cases, it is suggested to use Microsoft WordPad or similar.

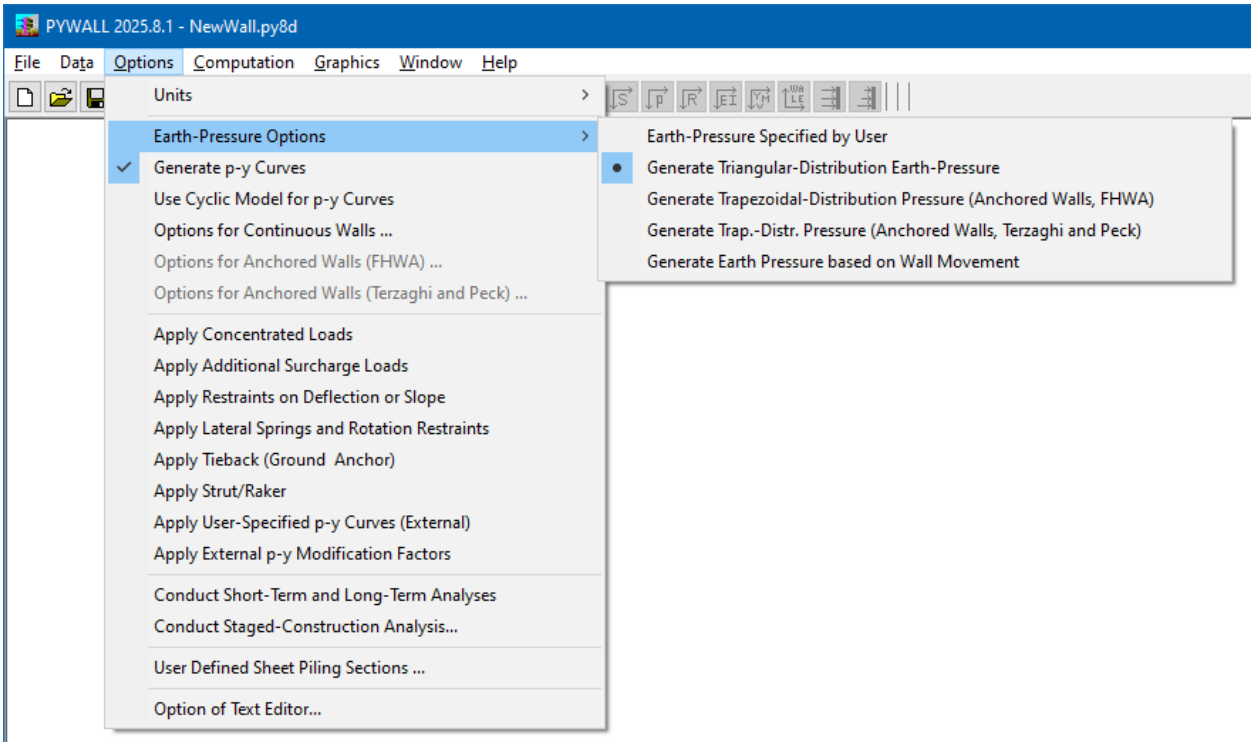


Figure 2.14 Options Menu

2.2.5 Computation Options

The Computation menu can be accessed to run the analytical computations after all data are entered and saved. After the computation is executed successfully this menu also allows for the reviews of plain-text input data, notes produced during computation, and output text data. It also provides access to the graphical observation of the modeled wall geometry and soil layers.

Run Analysis .. ..this is selected to run the analytical computations. This menu should be selected after all data have been entered and saved.

Edit Input Text .....calls the chosen Text Editor (selected under Options > Options of Text Editor…) to observe and/or edit the analytical input data in plain-text format. This selection becomes available after the input data has been saved to disk, or when opening an existing input-data file.

Edit Processor-Run Notes .....calls the chosen Text Editor (selected under Options > Options of Text Editor…) to observe, format, and/or print the notes provided during processing. This selection becomes available (or modified) after a run attempt has been made. The user is encouraged to read the processor-run notes if the program is not running successfully. The processor-run notes may provide the user with some additional information about possible causes of the problem.

Edit Output Text.....this selection is used to call the chosen Text Editor (selected under Options > Options of Text Editor…) to observe, format, and/or print the analytical-output data. This selection becomes available only after a successful run has been made. Certain output files

may be too large for the Microsoft Notepad editor, so other text editors would have to be used (Microsoft WordPad should be able to open most text files).

View Wall Geometry ..... this is to access a graphical representation of the elevation of the modeled wall and the location of the specified soil layers. Clicking the Print button immediately sends the graphic to the default Windows printer; without any other prompt.

3D View..... this selection provides a 3-dimensional graphical representation of the elevation of the modeled wall, soil profile, surcharge loading, restrain conditions, and the deflected shape of the wall.

Generate Report in MS-Word..... this selection allows users to export input and output information of the PYWALL model along with optional output charts into a Microsoft Word file that is pre-formatted for report presentations.

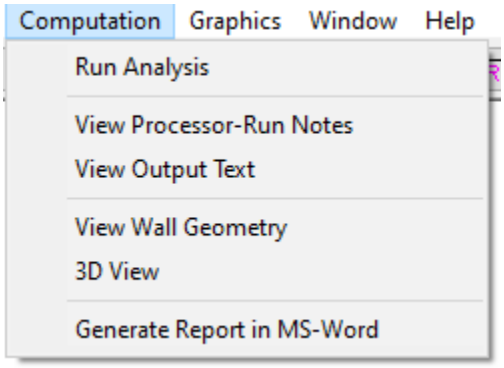


Figure 2.15 Computation Menu

2.2.6 Graphics Menu

The Graphics menu is used to observe plots of output data provided by a successful program run. Options for the observation of output curves under this menu are only enabled after a successful run has been made. Even after performing successful runs, some options may still be disabled since the amount of output data depends on specifications provided in the input file of each program run. Choices within this menu, shown in Figure 2.16, are briefly described below.

p-y Curves ..... this option provides a plot of generated soil resistance curves in each soil layer.

Deflection vs. Depth..... this option provides a plot of wall deflection versus depth.

Moment vs. Depth ..... this option provides a plot of wall bending moment versus depth.

Shear vs. Depth..... this option provides a plot of wall shear versus depth.

Reaction vs. Depth..... this option provides a plot of reaction from soil or tiebacks versus depth.

Deflection, Moment, and Shear ..... this option provides a plot of deflection, bending moment, and shear versus depth, all three on the same plot.

Wales Effect ... this option is only available for models of continuous walls with tiebacks or struts/rakers that had enabled the wales analyses. The graph provides a plot of equivalent distributed loads, shear and bending moment versus distance (from supports).

Earth Pressure Distribution.....this option provides a plot of distributed active earth pressure versus depth, as used internally by the program.

Earth Pressure Combined.....this option provides a plot of combined distributed active earth pressure (including those from surcharge loads) versus depth, as used internally by the program.

Export Plots to Excel.....this option provides options to export any selected plot data to formatted Microsoft Excel files.

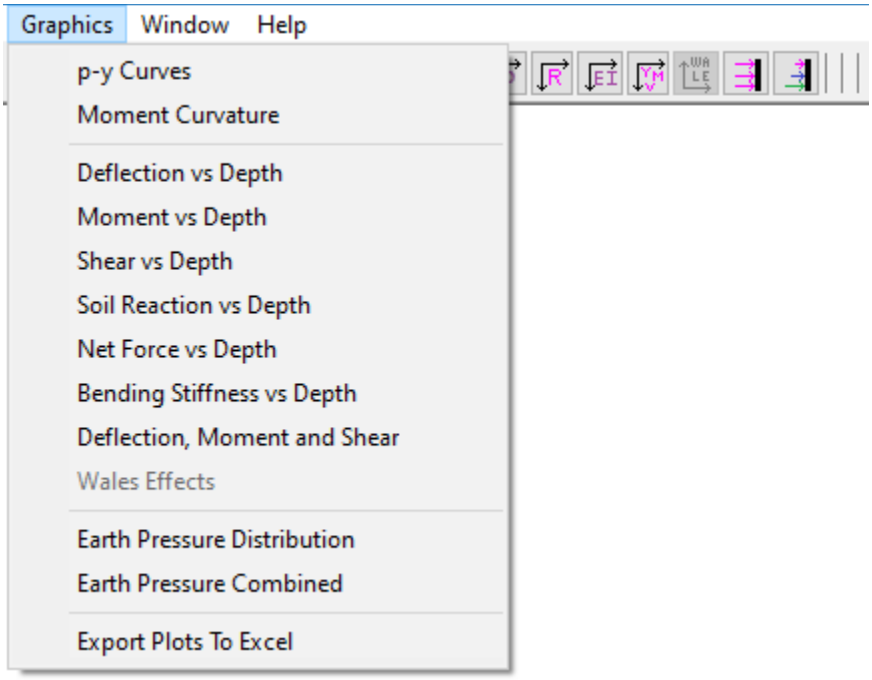


Figure 2.16 Graphics Menu

2.2.7 Window Menu

The Window menu provides two standard functions for organizing open- screen windows and/or minimized screen-window icons. Submenu options, shown in Figure 2.17, are briefly described below.

Cascade.....this option organizes all open-windowed menus so that all become visible with their tops cascading from the top left portion of the screen.

Arrange Icons. .....this option organizes the icons of all minimized windowed menus so that all become visible and aligned at the bottom portion of the screen.

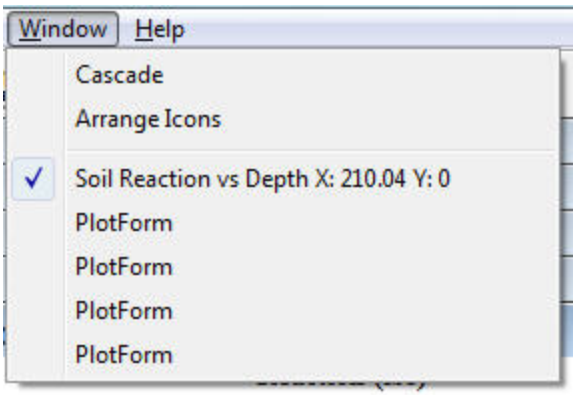


Figure 2.17 Window Menu

2.2.8 Help Menu

The Help menu provides access to the software manuals (PDF files) and to some helpful utilities that the user may need when running the program. The software manuals are helpful references on topics such as: using the program, entering data, information about variables used in the program and methods of analyses. Options in the Help menu, shown in Figure 2.18, are briefly described below.

User's Manual ..... this selection calls for any installed PDF reader (Adobe Acrobat, Acrobat Reader or similar) to open the User's Manual. This document is formatted as a protected PDF file. This manual describes all menu items to use the program, description of data entries and for output tables and graphs.

Examples Manual..... this selection calls for any installed PDF reader (Adobe Acrobat, Acrobat Reader or similar) to open the Examples Manual. This document is formatted as a protected PDF file. This manual describes several example problems that help to explain various program features.

Technical Manual ..... this selection calls for any installed PDF reader (Adobe Acrobat, Acrobat Reader or similar) to open the Technical Manual. This document is formatted as a protected PDF file. This manual contains background technical information and references regarding the theoretical methos used within the program.

User Information..... this is used to personalize the individual installation of APILE. The user-specified entry in this dialog will be included at the beginning of all output files that are generated by this particular installation of APILE.

About..... this provides a screen describing the program version, date, maintenance/subscription expiration date, license number and methods for accessing technical support.

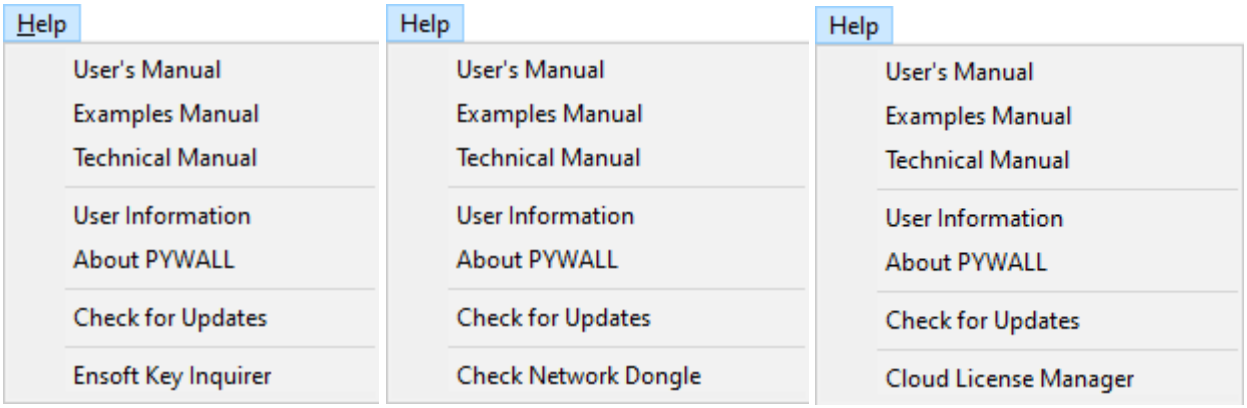
Check for Updates ..... starts the default Internet browser on a page that describes information about the user's license (release and maintenance expiration date) as well as the latest release that is available for downloading at the Ensoft site. Users may only run in full mode the maintenance updates that were released before the expiration date of the user's license.

Ensoft Key Inquirer..... this selection is only available for single-user licenses (not available in local network licenses). This utility allows the user to check information contained in the USB Key that is attached to the computer in use. With this utility the user can see the following

information: dongle serial number and expiration date, licensed program names and type of licenses, maintenance expiration date of each software license, licensed company name, allowed IP subnets. This utility is installed in the selected installation directory with the following filename: *ensoft_key_inquirer_v5.exe* (may change with time).

Check Network Dongle. this selection is only available in client computers of local network licenses (not available in single-user licenses). This calls the utility that is used to check the licensing and to troubleshoot communications with a license manager (or software server). This utility is very useful in case of problems in clients while trying to run the licensed software. This utility is installed in the selected installation directory with the following filename: *CheckNetworkDongle_ipv_511.exe* (may change with time).

Cloud License Manager this selection is only available on cloud subscription installations of the software product (it is not available for users with USB-based installations, such as single-user or network licenses). This menu provides access to information and management of cloud subscriptions licenses.



a. Single-User License b. Client on Local Network License c. Cloud Subscription License

Figure 2.18 Help Menu

CHAPTER 3. References for Data Input

3.1 File Menu

This menu contains options related to the management of input-data files and to exit the program. Input-data files created for PYWALL are provided with a standard file-name extension in the form of **filename*.py8d* (where **filename** represents any allowable file name). All input data files are standard text files and may be edited with any text editor or word-processing program.

3.1.1 File > New

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

3.1.2 File > Open

This is used to open a file that has been previously prepared and saved to disk. The File > Open window dialog, shown in Figure 3.1, is used to search for an existing input-data file. By default, the file is initially searched in the directory where PYWALL was installed. Standard windows-navigation procedures may be used to locate the name and directory of the desired project file. This menu option may be accessed with the Ctrl+O keyboard combination.

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

Opening partially-completed PYWALL input files or invalid data files may produce an information window reporting that an "invalid or incomplete" file is being opened (Figure 3.2). The user should click the OK button and all partial-input data that was previously prepared should become available.

The program allows users to read input-data files created for the previous PYWALL v2022, v2019, v2015, v2013, v3.0 or v2.0 versions by selecting the drop-down arrow at the bottom right corner of the File > Open dialog box. The program will automatically convert the opened older input files to the latest version of PYWALL when the user saves the opened file.

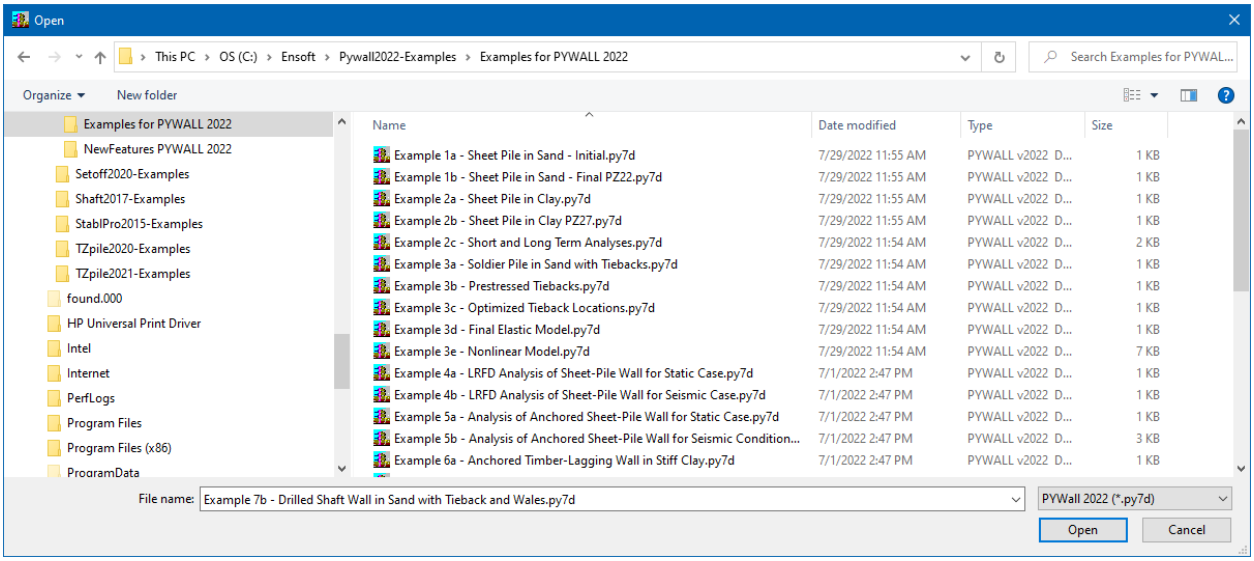


Figure 3.1 File > Open

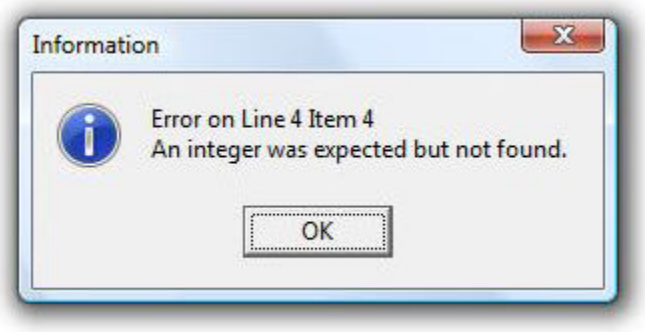


Figure 3.2 Message Window for Incomplete or Invalid Files

3.1.3 File > Save

This selection is used to save input data under the current file name. With this method of storing data to disk, any input data that was previously saved with the same file name is replaced with the current parameters. Input-data files are saved every time before proceeding with runs for analytical computation.

When saving a data file, the PYWALL program will automatically add an extension of the type **.py8d* to the name of the input data file. The program is not able to save data files that are compatible to older versions/releases of PYWALL. This menu option may also be accessed with the Ctrl+S keyboard combination.

3.1.4 File > Save As

This selection allows the user to save any opened or new input data file under a different file name or different directory. Any input data file saved under an existing file name will replace the contents of the existing file. When saving a data file the PYWALL program will automatically add an extension of the type **.py8d* to the name of the input data file. The program is not able to save data files that are compatible to older versions/releases of PYWALL.

3.1.5 File > Exit

This is selected to exit PYWALL. Any input-data file that was modified and not yet saved to disk will produce a confirmation window before exiting the program (see Figure 3.3).

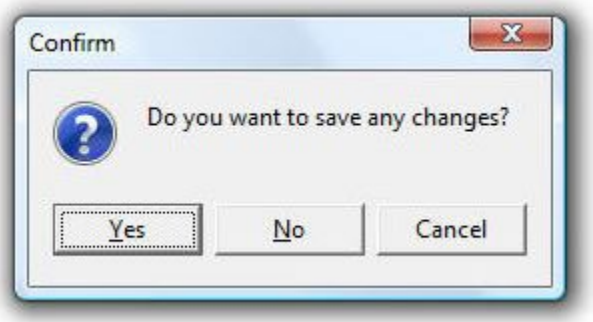


Figure 3.3 Message window advising that changes were not saved to disk

3.2 Data Menu

The input of specific parameters for an application is controlled under selections contained within this menu (sample shown in Figure 3.4). Not all of the entries under the Data menu may be enabled since many depend on other entries or selections made from the Options menu. It is recommended that the user choose each submenu and enter parameters in a consecutive manner starting from the top selection.

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

- clicking the OK button (if available), or
- clicking the X-box on the upper-right corner of the window, or
- double-clicking the PYWALL icon on the upper-left corner of the window, or
- clicking once on the PYWALL icon on the upper-left corner of the window and then choosing Close.

Open windows may optionally be left open on the screen. The selection of other menu options will then produce new windows on top of those that were left open.

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

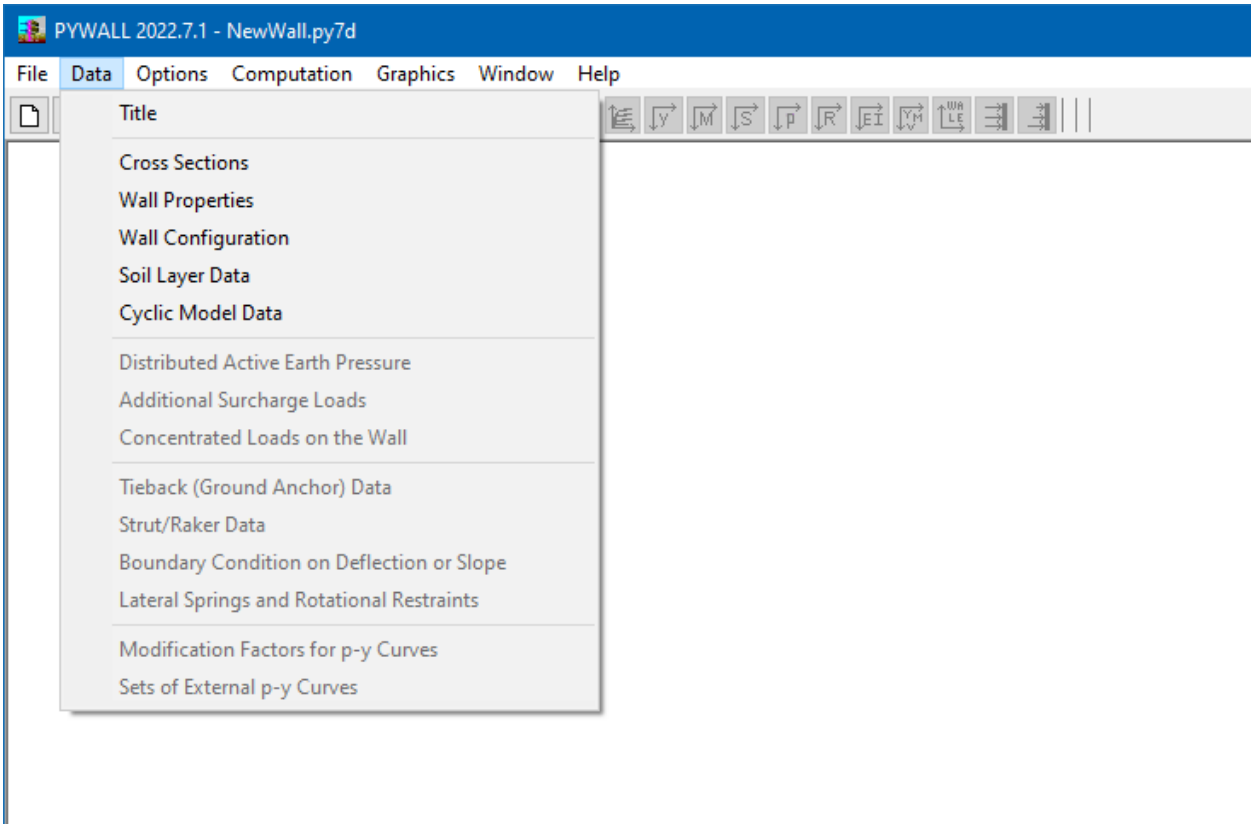


Figure 3.4 Sample Data Menu

3.2.1 Numeric Data Entries

Cells that require numeric data may accept entries of mathematical expressions in addition to simple numeric entries. Entering a mathematical expression works similarly to normal numeric data. The user types the expression that represents the data and presses the Return key to calculate the entered expression and to display the numeric result in the same cell.

Table 3.1 below shows the list of supported operations and constants. The order of operations follows the order in the list of Table 3.1. Note that implicit multiplication (i.e. 2(4+6)) is not supported (instead, use 2*(4+6) for this example).

The two constants that are currently supported are PI and e. Implicit multiplications using constants is not supported (use 2*e instead of 2e). Negation of the constants PI or e is not allowed. For instance, instead of entering -PI the user must enter - (PI).

Scientific notation (i.e. 1.65e8 or 1.65e-8) may be used to input very large or very small numbers. After an expression is calculated, very large or very small numbers will be displayed using scientific notation.

	OPERATORS
Symbol	**Description**
()	Parenthesis (may be nested)
^	Exponentiation
*****	Multiplication
/	Division
+	Addition
-	Subtraction
-	Negation (same as subtraction)
	CONSTANTS
Symbol	**Value**
PI (or pi)	3.1415927
e (or E)	2.7182818

Table 3.1 Supported Mathematical Operations and Constants

3.2.2 Data > Title

This activates the dialog box shown in Figure 3.5, where the user can enter a line of text containing a general description for the application problem. Any combination of characters may be entered in the text box in order to describe a particular application. The user input will be restrained automatically once the maximum length of text is reached. The Title entered by the user will be printed in the output text file of the model.

In models with the optional stage construction (Options > Conduct Staged-Construction Analysis), a unique Data > Title can be entered by the user for each Stage. The output file will provide a unique header for the results of each stage, for example: `Stage No : 1 TitleStage1` and/or `Stage No : 2 TitleStage2` where `TitleStage1` and `TitleStage2` are the unique Data > Title that were respectively defined by the user for Stage 1 and Stage 2.

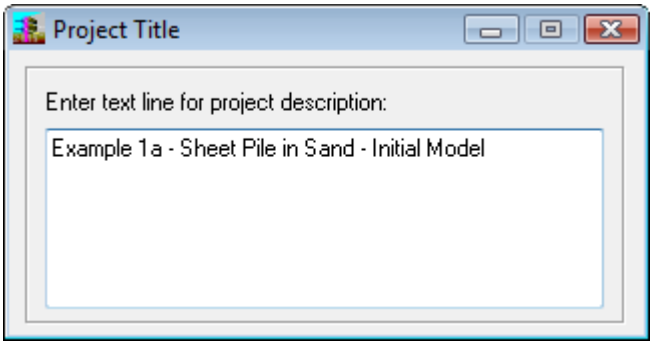


Figure 3.5 Sample Data > Title

3.2.3 Data > Cross Sections

The user has to define a "wall unit" with a specific flexural stiffness. For example, on soldier-pile walls the user normally analyzes a single pile as the "wall unit." In those cases, the pile may be a steel shape (HP pile, W section or a Circular Pile) or a concrete pile (drilled shaft or prestressed pile) with defined flexural properties for its direction of bending. On continuous diaphragm walls the "wall unit" is

usually a unit length of wall, so users enter the flexural stiffness for a 1-m or 1-ft length of wall. On sheet-pile walls the users may choose unit lengths (1-m or 1-ft) or may use the length of a sheet-pile section (eg, 24.8 in for a standard AZ36 sheet pile).

The Data > Cross Sections menu is used for the definition and input of structural properties for specific heights of the modeled wall element. A sample dialog box for Data > Cross Sections is shown in Figure 3.31 and the various data entry listed below.

More than one Cross Section Definition are necessary if the wall changes in flexural property along its height. For example, there are cases where a soldier-pile wall has HP piles embedded in concrete at the penetration below the excavation and single HP piles above the excavation. Two or more cross sections may also be needed on retaining walls with reinforced-concrete sections where the vertical section reinforcement changes at various heights. The user may enter up to 20 rows of cross sections (or wall heights) with different properties.

Section

This is a sequential number that is provided for each unit section of wall height/depth.

Section Name

User may enter any useful name to be associated with the wall section that is being defined.

Section Type

The defined wall section may be either Elastic or Nonlinear. It is usually recommended to start a model with linear elastic section properties so the PYWALL model has better mathematical convergences while not being limited to the flexural capacity of the section. However, the user should always make their final model using nonlinear material response.

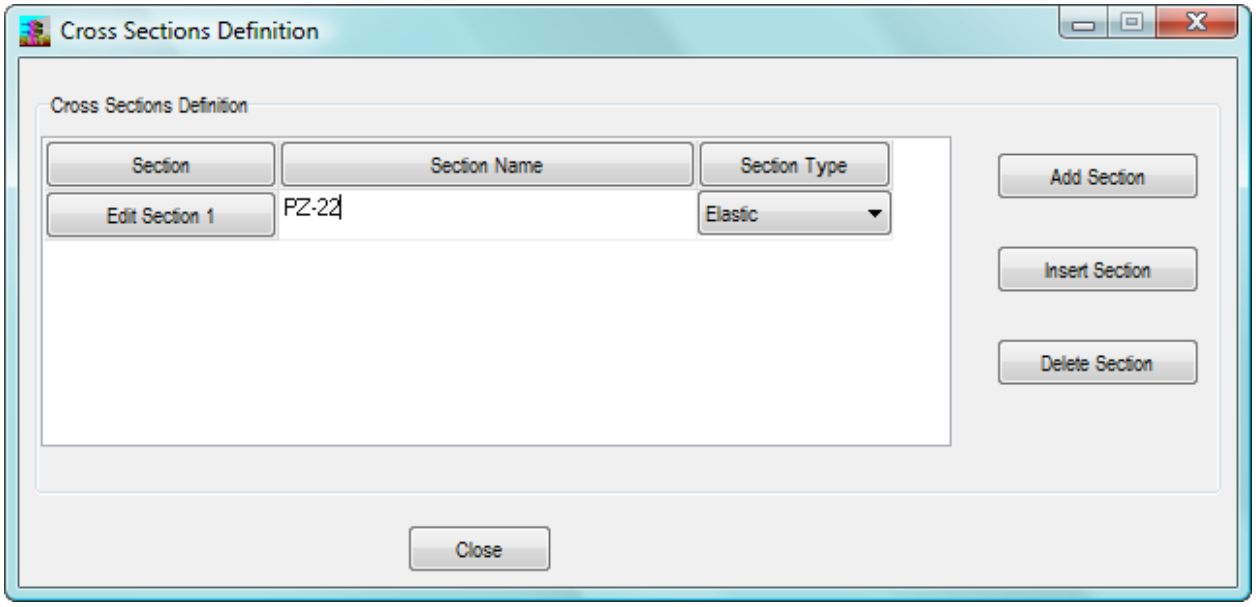


Figure 3.6 Sample Data > Cross Sections

3.2.4 Edit Section *x* for Elastic Types

The button Edit Section x (where x is any number from 1 to 20) in the Data > Cross Sections dialog box (while the column Section Type = Elastic) provides the user with a dialog screen that may be similar to Figure 3.7.

3.2.4.1 Select Section

Under the Select Section, the user can choose from 7 different types of sections for the retaining wall elements. The preset sections are listed below along with referenced sample screen captures. The definition of the various available entries is defined in the topic labeled Section Dimensions.

General Section. This is used to define a wall section using simple properties. This is the default section for any new model with elastic properties. In addition, this section is assigned automatically to all models created after importing data files from previous/older versions of PYWALL. For a reference on the screen interface of this section see Figure 3.7.

Circular Section. Used to define a solid or hollow circular section. For a reference on the screen interface of this section see Figure 3.8.

Rectangular Section. Used to define a solid or hollow rectangular section. For a reference on the screen interface of this section see Figure 3.9.

I Section. Used to define any section with the general I shape (rotating about its strong axis). For a reference on the screen interface of this section see Figure 3.10.

H Section. Used to define any section with the general H shape (or I shape rotated about its weak axis). For a reference on the screen interface of this section see Figure 3.11.

AISC Section. In this case users can select any W, M, S, HP, HSS or Pipe shape from the AISC manual rotating in strong or weak (rotated) axes. For a reference on the screen interface of this section see Figure 3.12.

Sheet Piling Section. This selection includes a variety of built-in (USS, Nucor) and user-defined sections for sheet piles. For a reference on the screen interface of this section see Figure 3.13.

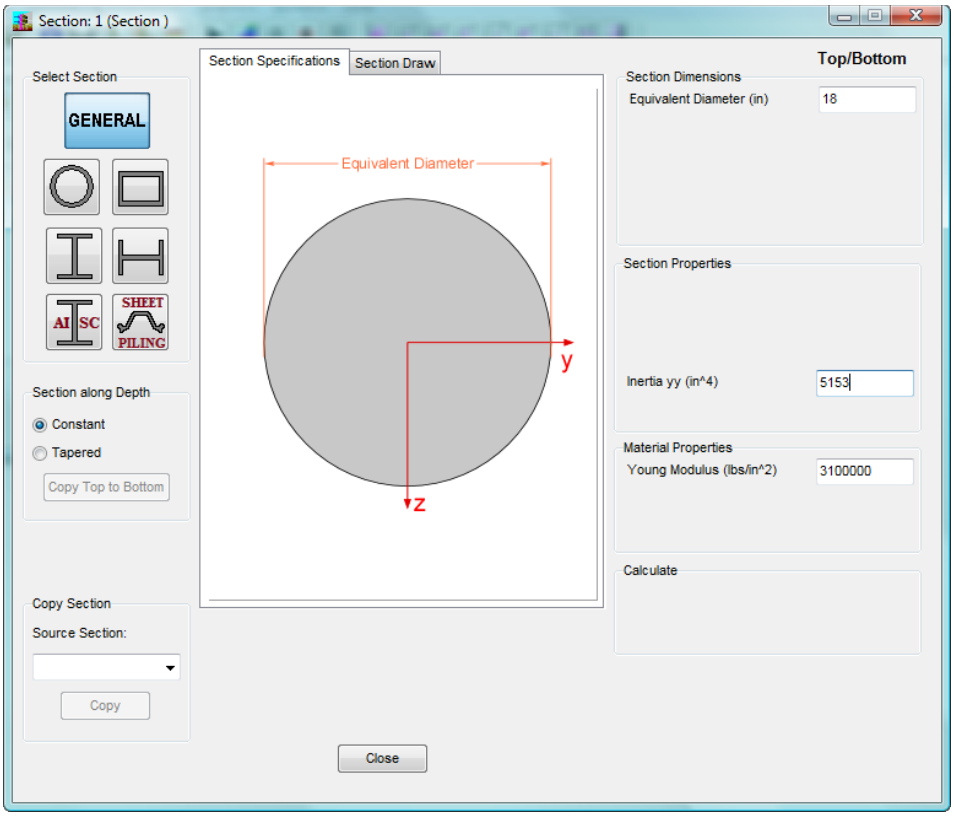


Figure 3.7 Sample General Cross Section with Linear Elastic Properties

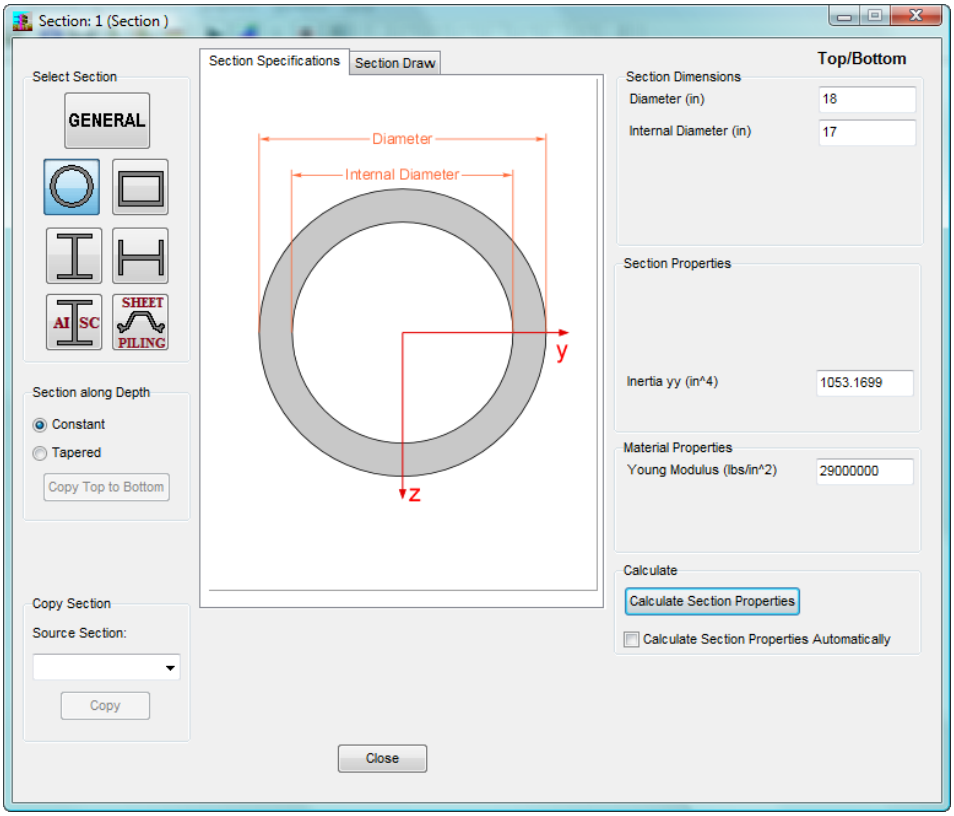


Figure 3.8 Sample Circular Cross Section with Linear Elastic Properties

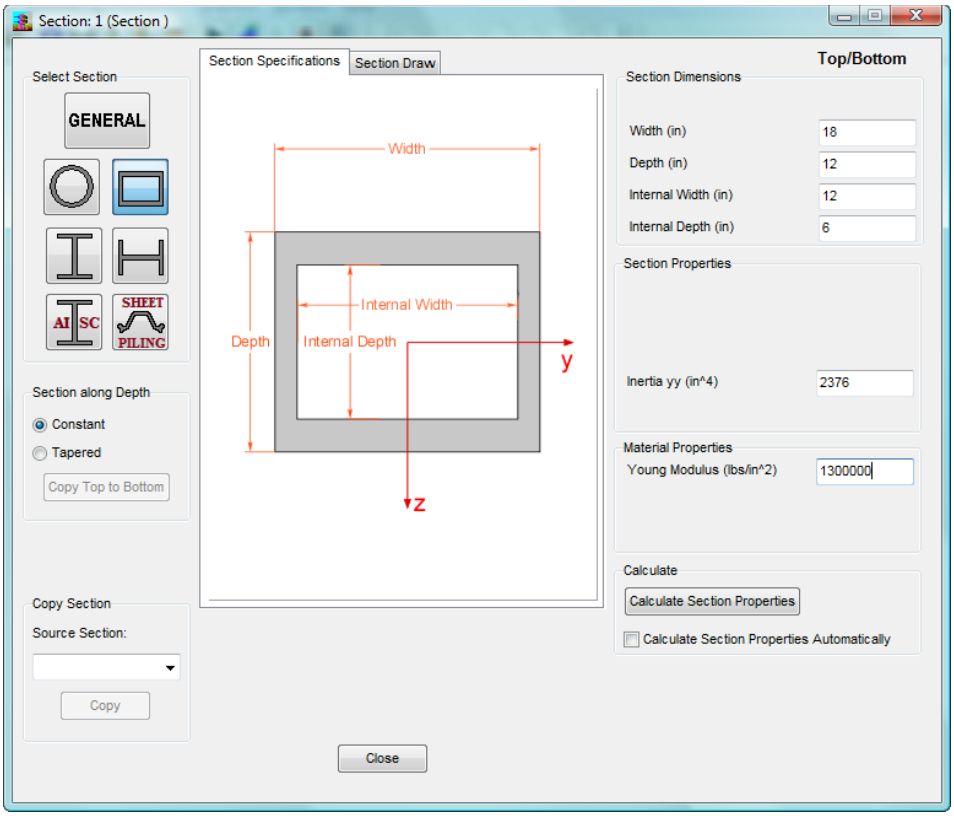


Figure 3.9 Sample Rectangular Cross Section with Linear Elastic Properties

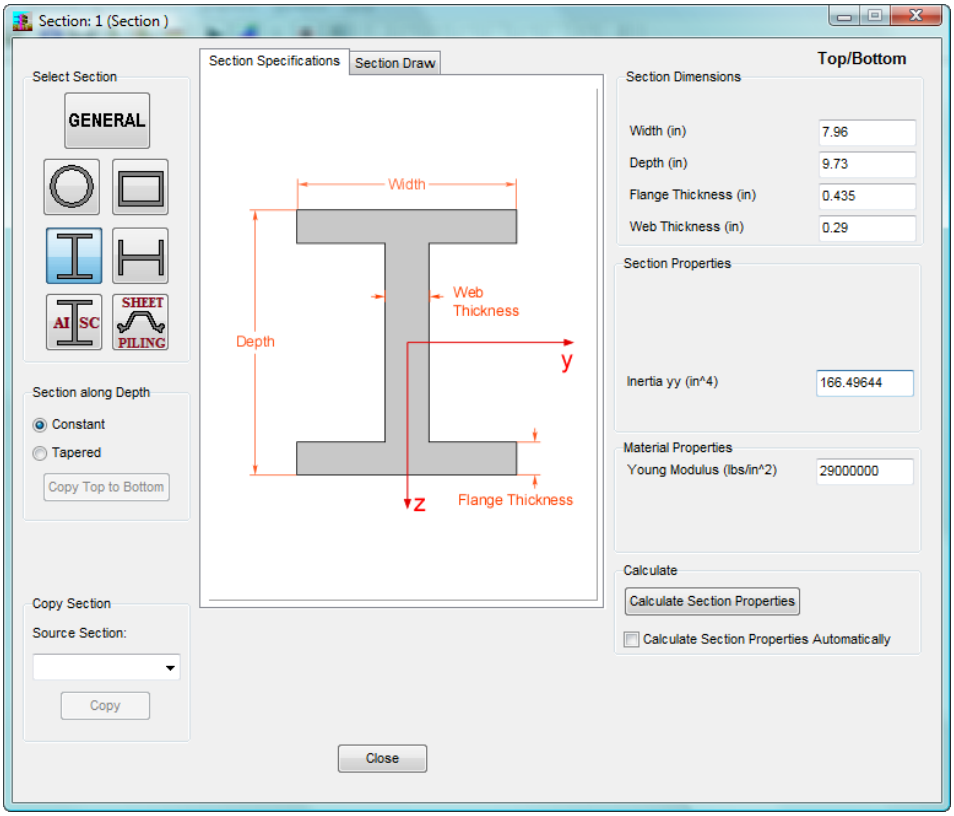


Figure 3.10 Sample I Cross Section with Linear Elastic Properties

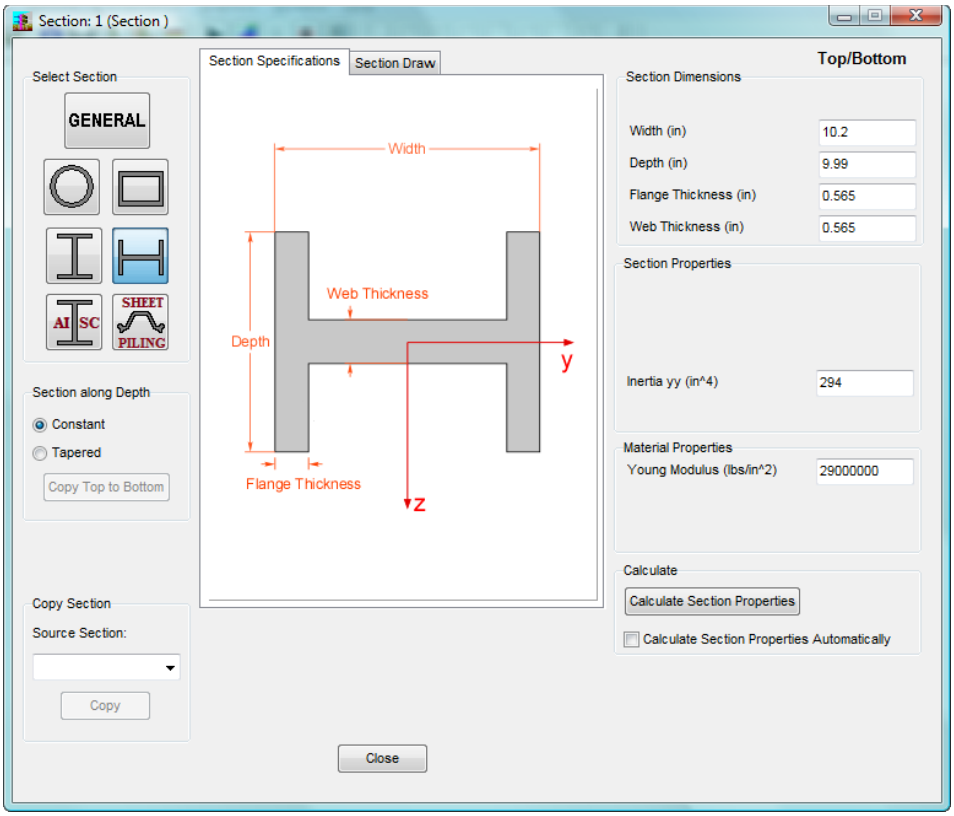


Figure 3.11 Sample H Cross Section with Linear Elastic Properties

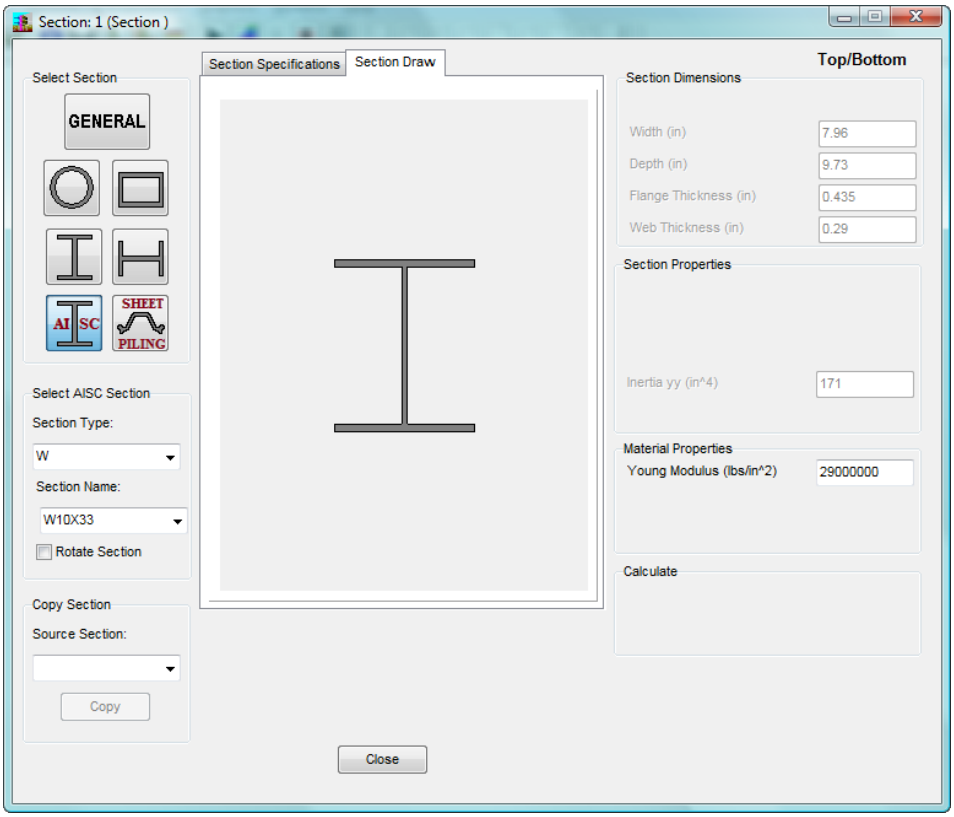


Figure 3.12 Sample AISC Cross Section with Linear Elastic Properties

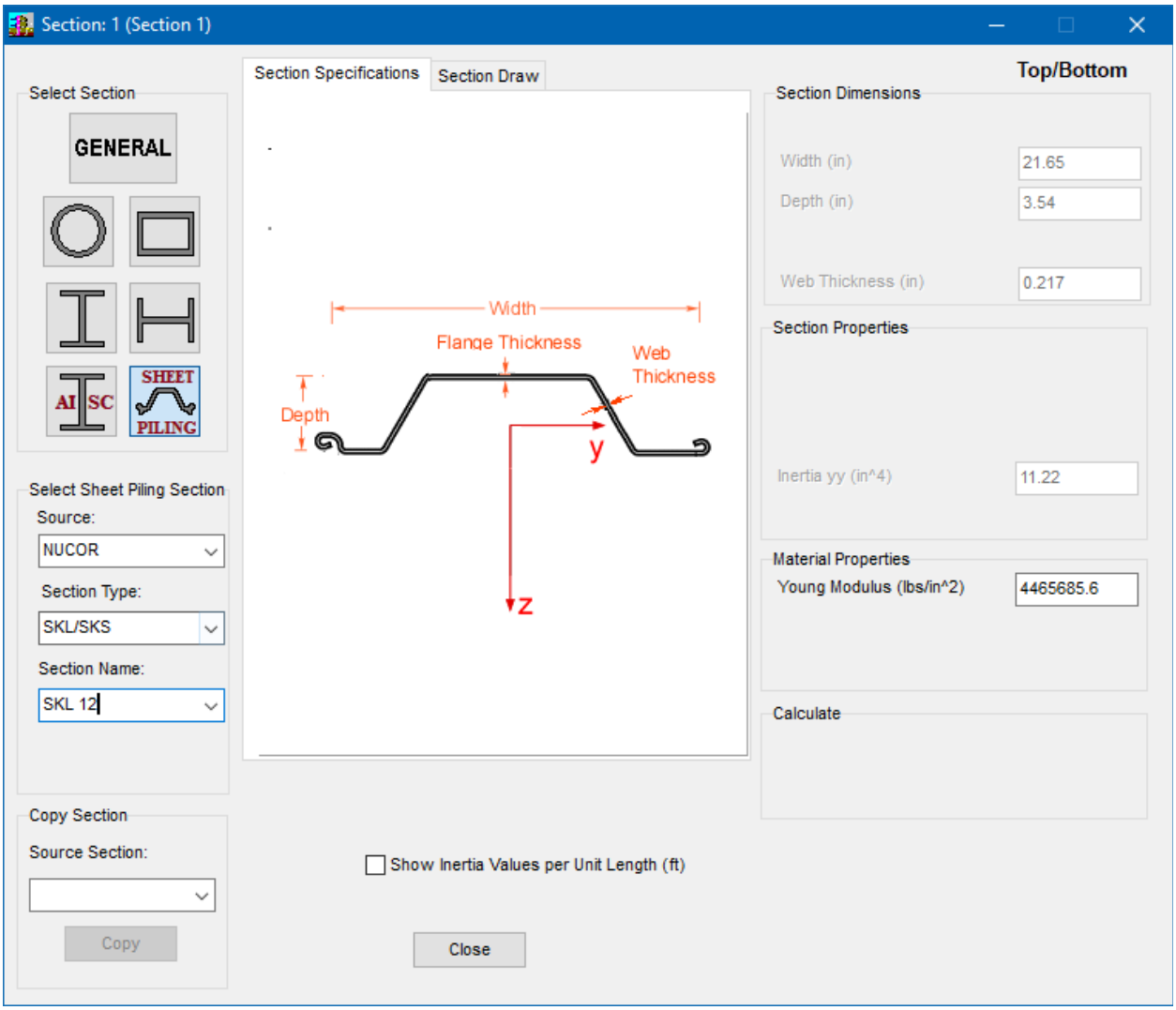


Figure 3.13 Sample Sheet Piling Cross Section with Linear Elastic Properties

3.2.4.2 Section along Depth

With the Section along Depth feature (which is not available for AISC or Sheet Piling),, the user can define a Constant section or a Tapered section along for the wall. For a constant section only one set of input parameters are required; however, for a tapered section properties at top and bottom of the section are required.

If a Tapered section is defined the Copy Top to Bottom button becomes available. This button can be used to copy all the parameters defined for the top to the bottom of the section. The user may edit the Bottom parameters separately and then click Calculate Section Properties to compute missing entries.

When a Tapered section is defined the program will use a linear interpolation for all the variables to estimate values between top and bottom of the section. A sample entry for a Tapered window screen using a general section is included in Figure 3.14.

3.2.4.3 Copy Section

The Copy Section is helpful to define sections that only have a few differences. This feature is available if at least two elastic wall sections are defined. While editing a new wall section, the user can click on the drop-down menu to select a previous section and then click on the Copy button to duplicate the data that was previously entered at the source.

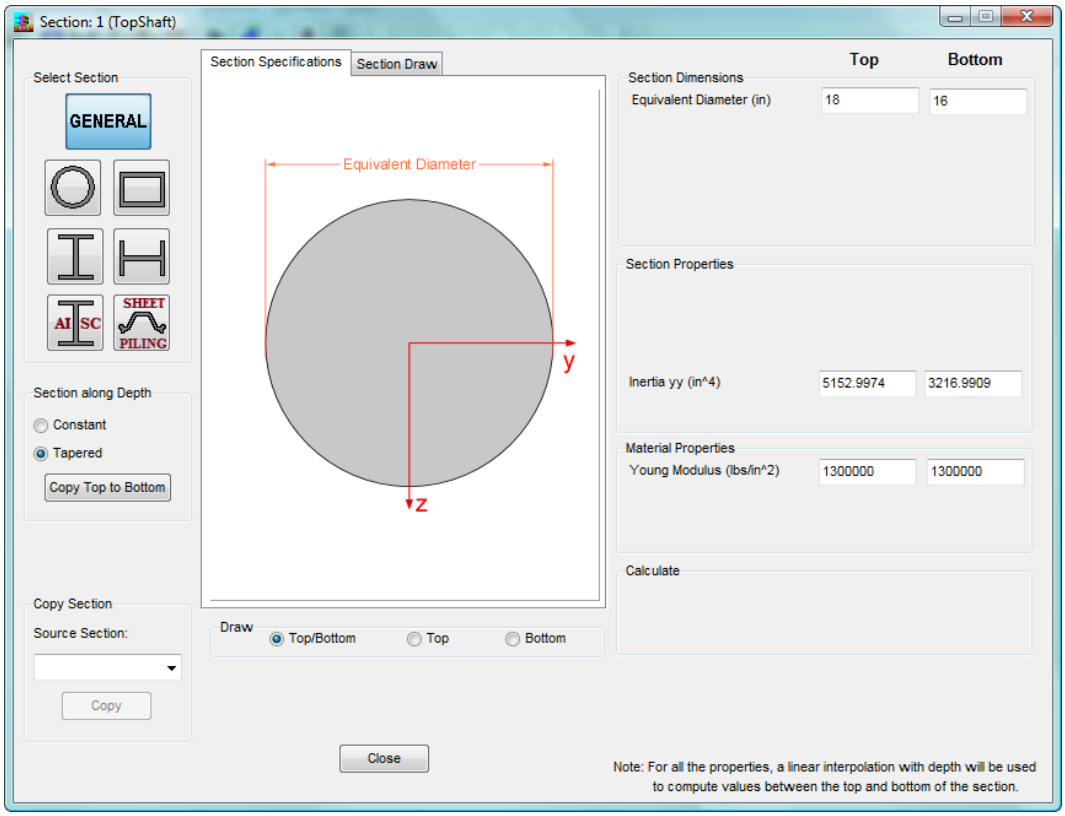


Figure 3.14 Sample Edit Cross Section Window for a Tapered Section

3.2.4.4 Section Specifications & Section Draw Tabs

The Section Specifications tab shows a reference picture of the cross section with appropriate dimensioning labels. When the user clicks on a dimensioning label, the corresponding entry field will be selected. This is an alternative method for entering section dimensions.

Selecting the Section Draw tab will produce a scale drawing of the actual dimensioned section. If a Tapered section is defined the user can choose to draw both sections (top and bottom), only the top or only the bottom section.

Notice that the bending of any section is about the Y axis so the Equivalent Diameter, Width, Diameter, entries (depending on the selected section type) are the ones activating the soil response.

The Section Specifications and Section Draw for a sample rectangular section are shown in Figure 3.15.

3.2.4.5 Section Dimensions

The parameters under Section Dimensions depend on the section type that is selected by the user. References for the various parameters are defined in the following topics.

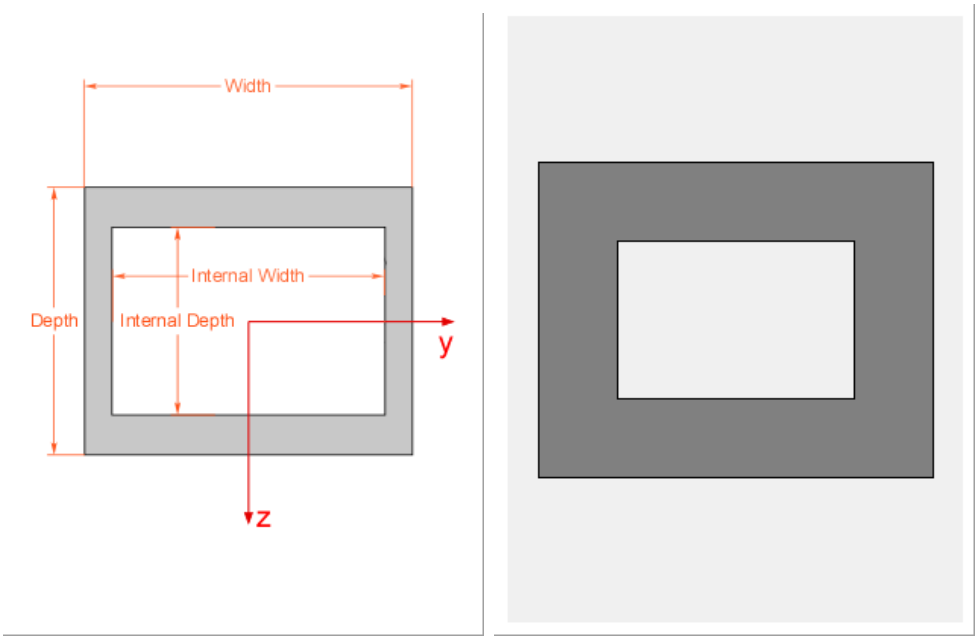


Figure 3.15 Section Specifications and Section Draw for a Sample Rectangular Section

General Section

Equivalent Diameter. The user should enter here the diameter of circular sections or an equivalent diameter for noncircular sections. This value is the one selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p-y* curve) for lateral analysis.

Circular Section

Diameter. This is the external diameter of the pile. This value is the one selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p-y* curve) for lateral analysis.

Internal Diameter. Internal diameter of the pile. Leave as zero for a solid pile.

Rectangular Section

Equivalent Diameter. The equivalent diameter for a rectangular pile can be computed, as an approximation, by finding the diameter of a circular section with the same area as the rectangular section. For more conservative estimates, the equivalent diameter may simply be equal to the Width of the rectangular section. This value is the one selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p-y* curve) for lateral analysis.

Width. Width of the cross section.

Depth. Depth of the cross section.

Internal Width. Internal width of the cross section. Leave as zero for a solid pile.

Internal Depth. Internal depth of the cross section. Leave as zero for a solid pile.

I or H Section

Equivalent Diameter. The equivalent diameter for an I or H section can be computed, as an approximation, by finding the diameter of a circular section with the same area as the rectangular section. For more conservative estimates, the equivalent diameter may simply be equal to the Width of the section. This value is the one selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p*-*y* curve) for lateral analysis.

Width. Width of the cross section.

Depth. Depth of the cross section.

Flange Thickness. Flange thickness of the cross section.

Web Thickness. Web thickness of the cross section.

AISC Section

The various parameters will be populated based on the section selected from the two drop-down menus. The Width of the section is the value selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p*-*y* curve) for lateral analysis.

Sheet Piling Section

The various parameters will be populated based on the section selected from the drop-down menu. The Width of the section is the value selected as Wp entry in the Data > Wall Properties when choosing the Get From Cross Sections. The Wp is used to estimate the contribution of soil resistance (*p*-*y* curve) for lateral analysis.

3.2.4.6 Section Properties

The only section property used in PYWALL computations is the moment of inertia of the section about the yy direction (Inertia yy). The user can input this property or let the computer estimate. For AISC and Sheet Piling sections the properties are populated after selecting the section and cannot be edited.

3.2.4.7 Material Properties

The user must enter the Young Modulus (Modulus of Elasticity) of the pile material. The PYWALL program uses this value to estimate the bending stiffness of the wall section for lateral analysis.

3.2.4.8 Calculate Section

The Calculate Section Properties button is used to calculate the section properties based on the section dimensions entered by the user.

A checkmark on Calculate Section Properties Automatically tells the program to automatically compute the section properties. Any change in the section dimensions will be reflected automatically in the section properties.

3.2.5 Edit Section *x* for Nonlinear Types

The button Edit Section x (where x is any number from 1 to 20) in the Data > Cross Sections dialog box (while the column Section Type = Elastic) provides the user with a dialog screen that may be similar to Figure 3.16.

3.2.5.1 Nonlinear Section Selection

At the top right of the windows screen in Figure 3.16 there are various preset cross sections that the user may choose for the wall element that is being defined with nonlinear properties. The preset sections are listed below along with sample screen captures. The definition of the various available entries are defined in the subsequent topics.

Rectangular Concrete Section. Used to define a solid rectangular section with or without reinforcement. For a reference on the screen interface of this section see Figure 3.17.

Round Concrete Section. Used to define a solid circular section with or without reinforcement. For a reference on the screen interface of this section see Figure 3.18.

Round Concrete Section with Casing. Used to define a circular section with or without: reinforcement, outside casing and/or structural steel insert. For a reference on the screen interface of this section see Figure 3.19.

Round Prestressed Section. Used to define a solid or hollow circular section with prestressing strands. For a reference on the screen interface of this section see Figure 3.20.

Square Prestressed Section. Used to define a solid or hollow square section with prestressing strands. For a reference on the screen interface of this section see Figure 3.21.

Octogonal Prestressed Section. Used to define a solid or hollow octagonal section with prestressing strands. For a reference on the screen interface of this section see Figure 3.22.

Steel Pipe Section. Used to define a hollow steel pipe section. For a reference on the screen interface of this section see Figure 3.23.

I or H Steel Section. Used to define a built-up steel section of I or H shapes. For a reference on the screen interface of this section see Figure 3.24.

AISC Steel Section. In this case users can select any W, M, S, HP, HSS or Pipe shape from the AISC manual (also with any rotated installation angle). For a reference on the screen interface of this section see Figure 3.25.

User-Defined Nonlinear Section. Users may also enter any values of moment vs curvature for the wall section that is being defined (when this is calculated outside the PYWALL program for a section that is not in the list or with different material properties). For a reference on the screen interface of this section see Figure 3.26. The Eq. Diameter entry is the equivalent diameter that should be used for the *p*-*y* curves corresponding to the defined nonlinear section.

Axial Thrust. In this entry the user can enter the thrust force (permanent axial force) at which the internal moment vs curvature shall will be computed (for any of the sections specified above). For a reference on the screen interface of this section see Figure 3.27.

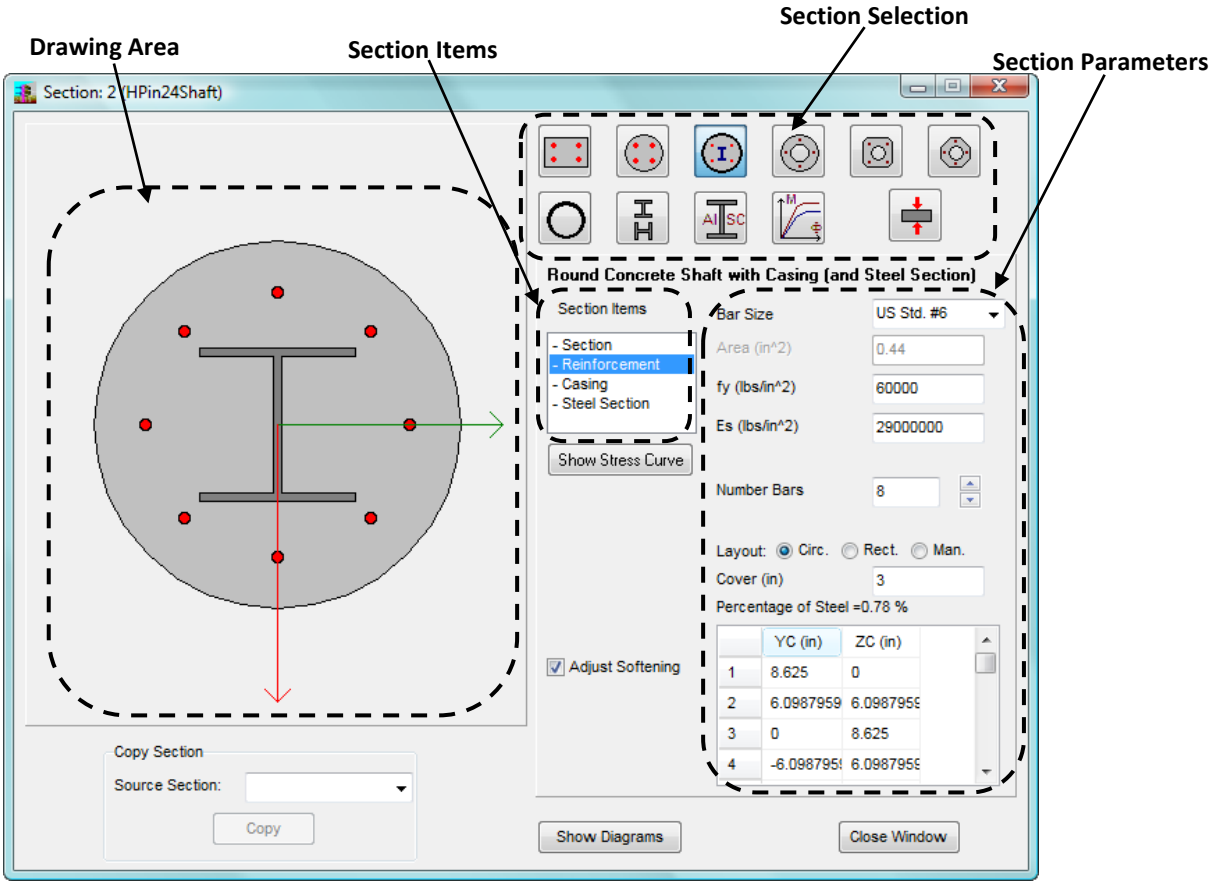


Figure 3.16 Sample Cross Section with Nonlinear Properties

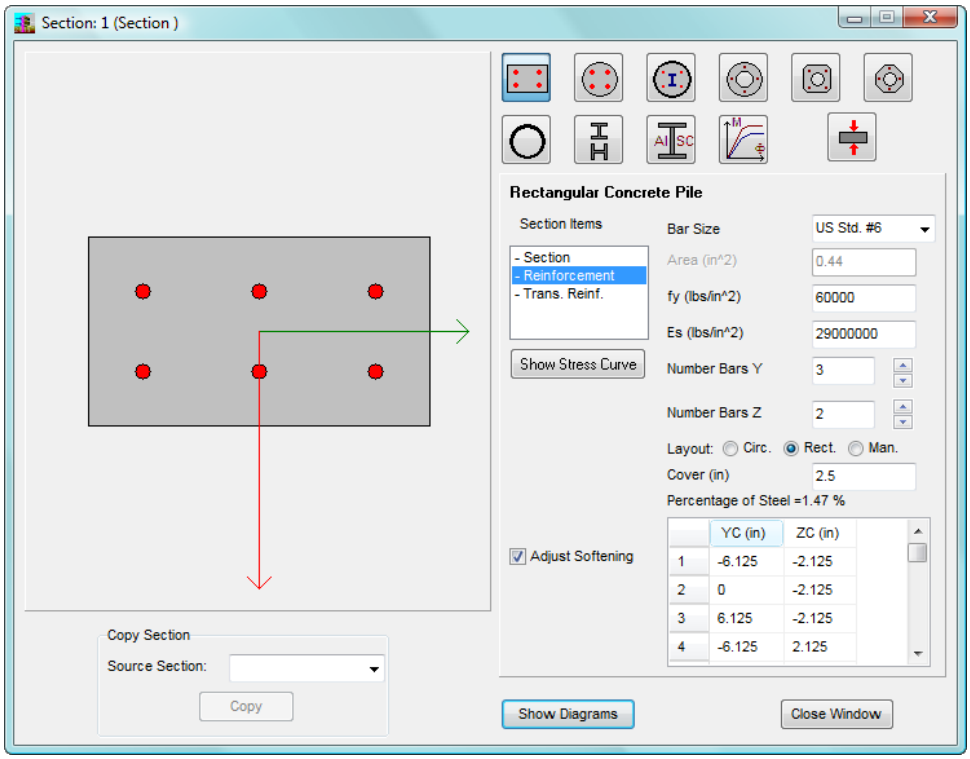


Figure 3.17 Sample Rectangular Concrete Cross Section with Nonlinear Properties

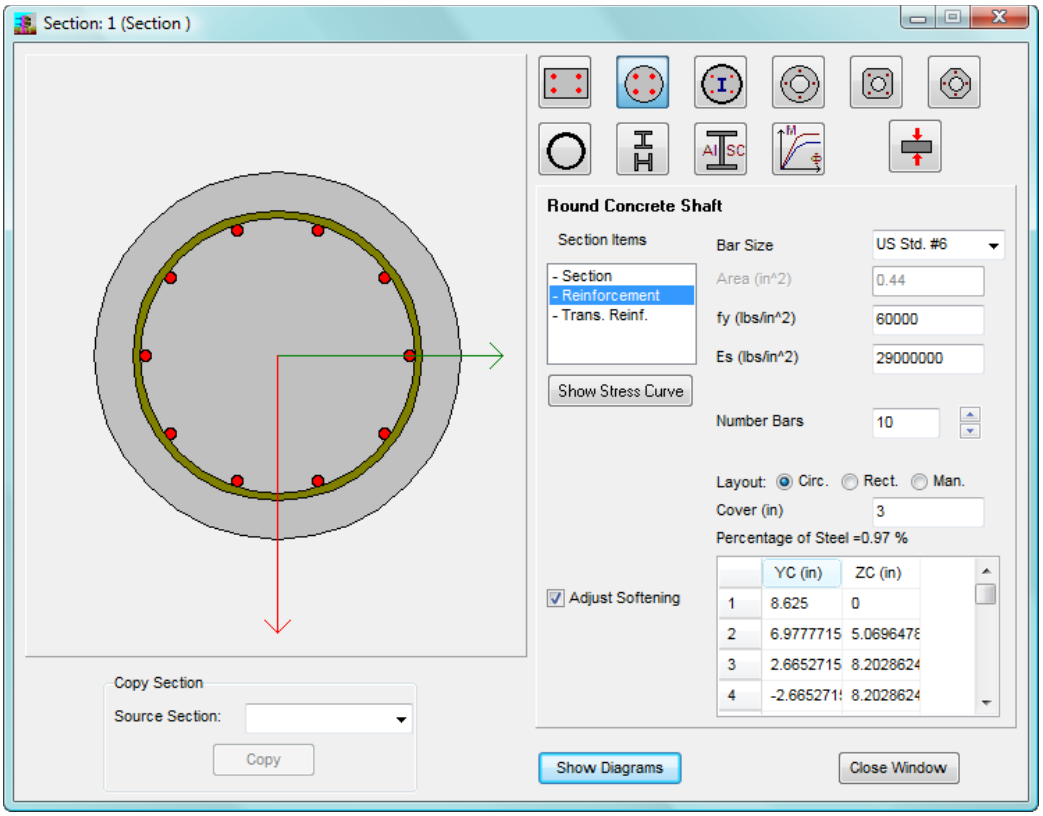


Figure 3.18 Sample Round Concrete Cross Section with Nonlinear Properties

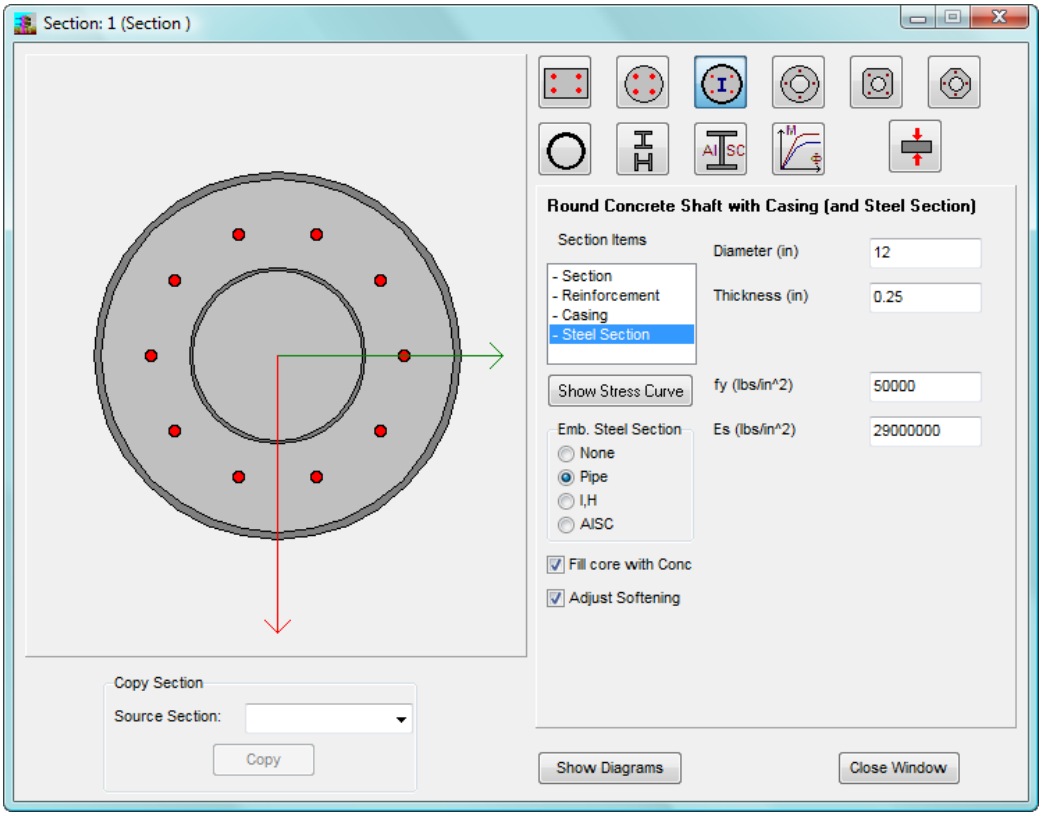


Figure 3.19 Sample Round Concrete Section with Casing with Nonlinear Properties

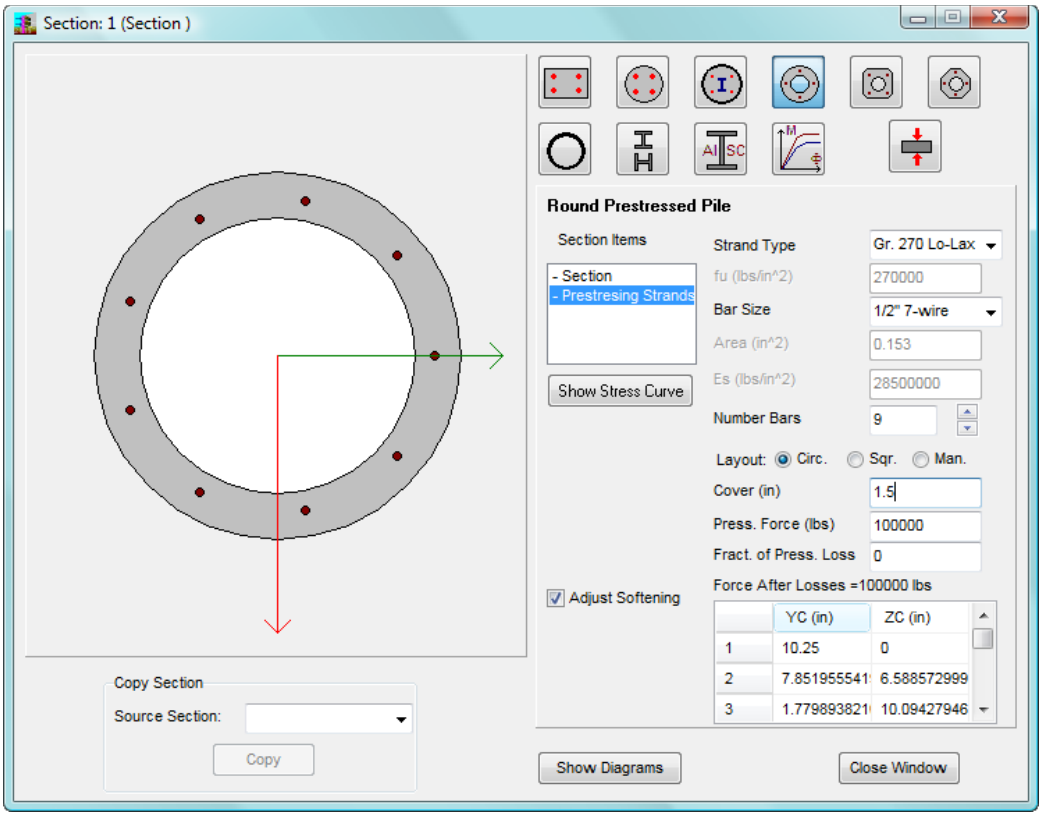


Figure 3.20 Sample Round Prestressed Section with Nonlinear Properties

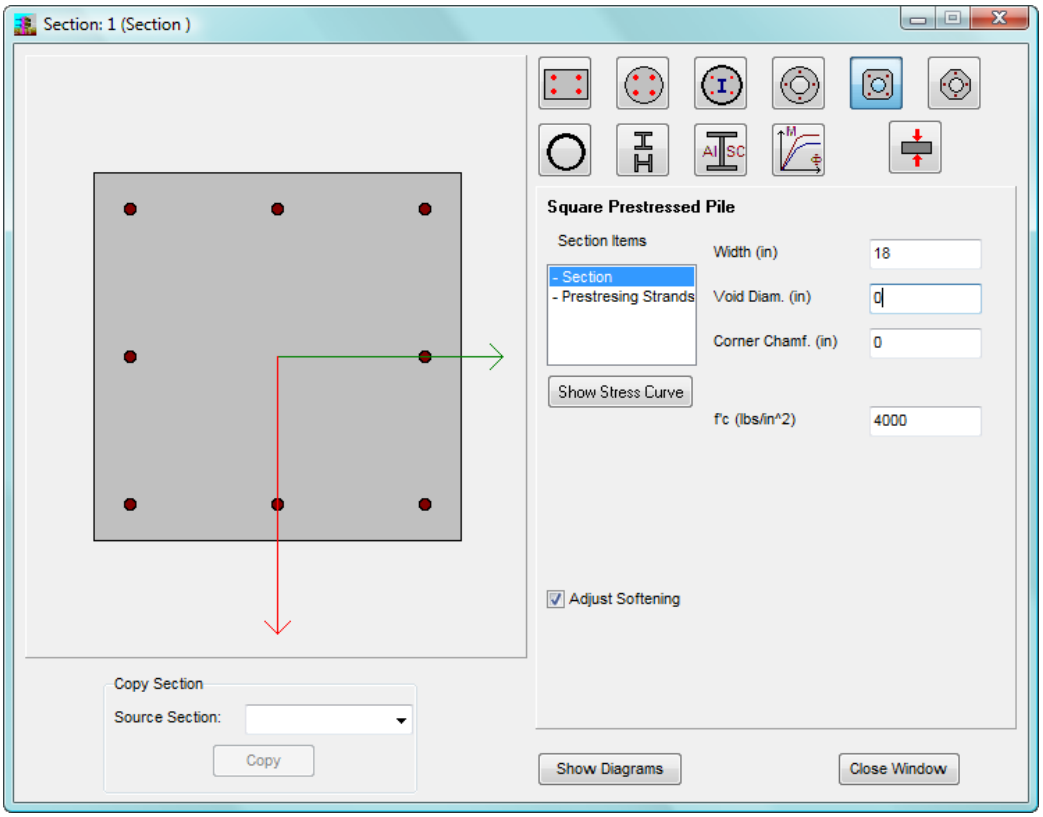


Figure 3.21 Sample Square Prestressed Section with Nonlinear Properties

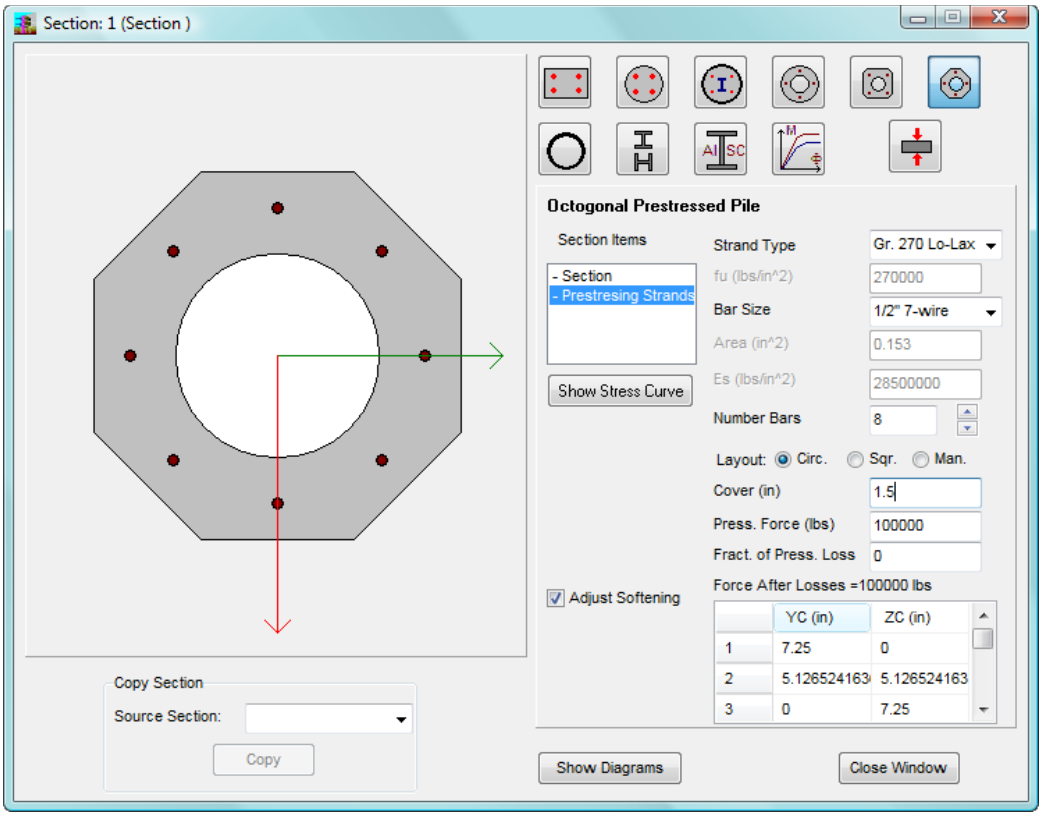


Figure 3.22 Sample Octogonal Prestressed Section with Nonlinear Properties

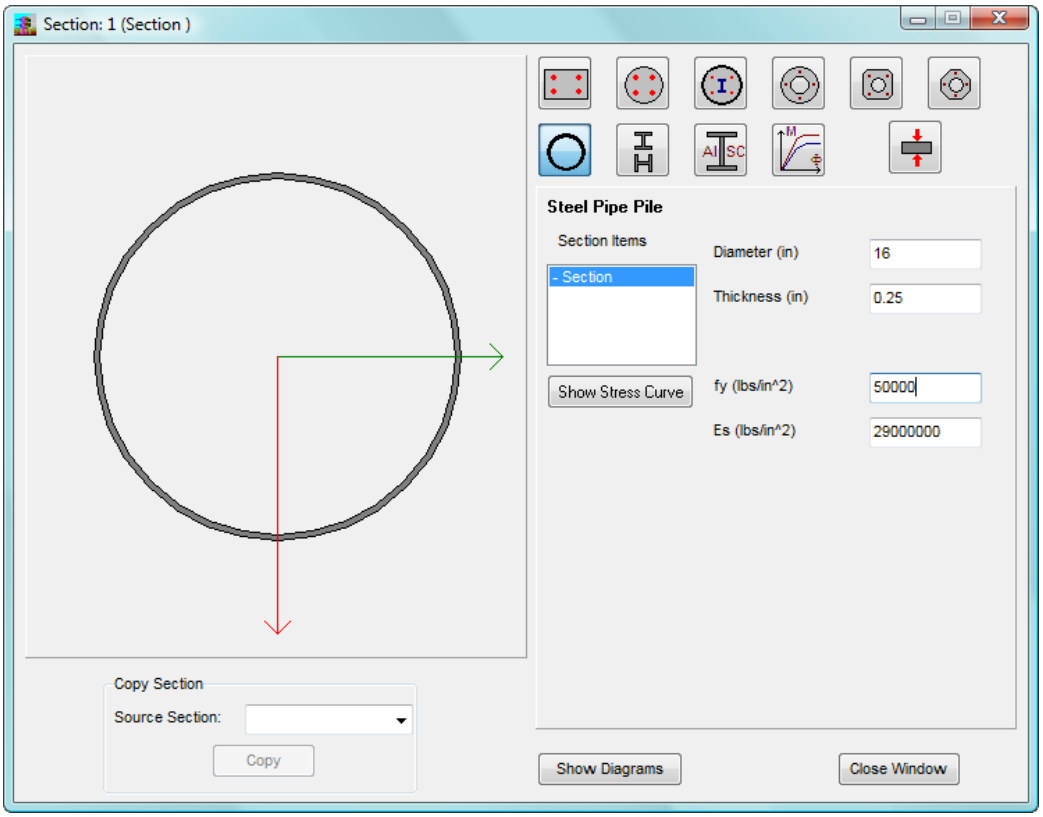


Figure 3.23 Sample Steel Pipe Section with Nonlinear Properties

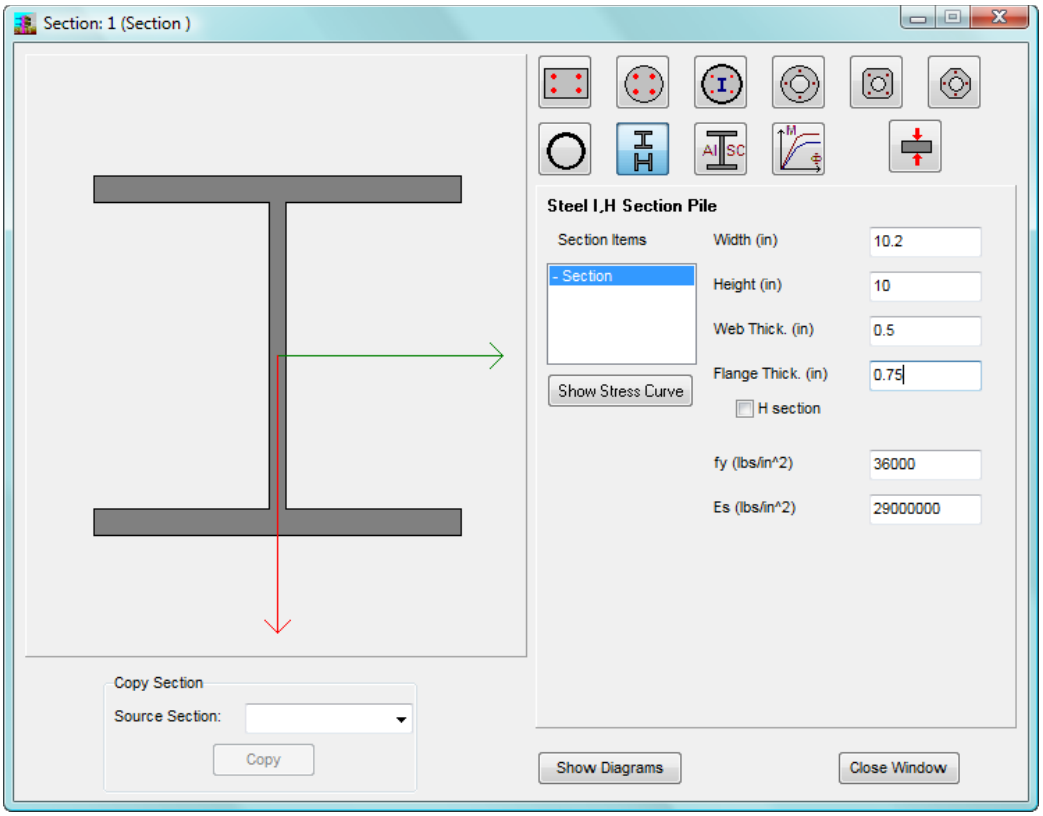


Figure 3.24 Sample I or H Section with Nonlinear Properties

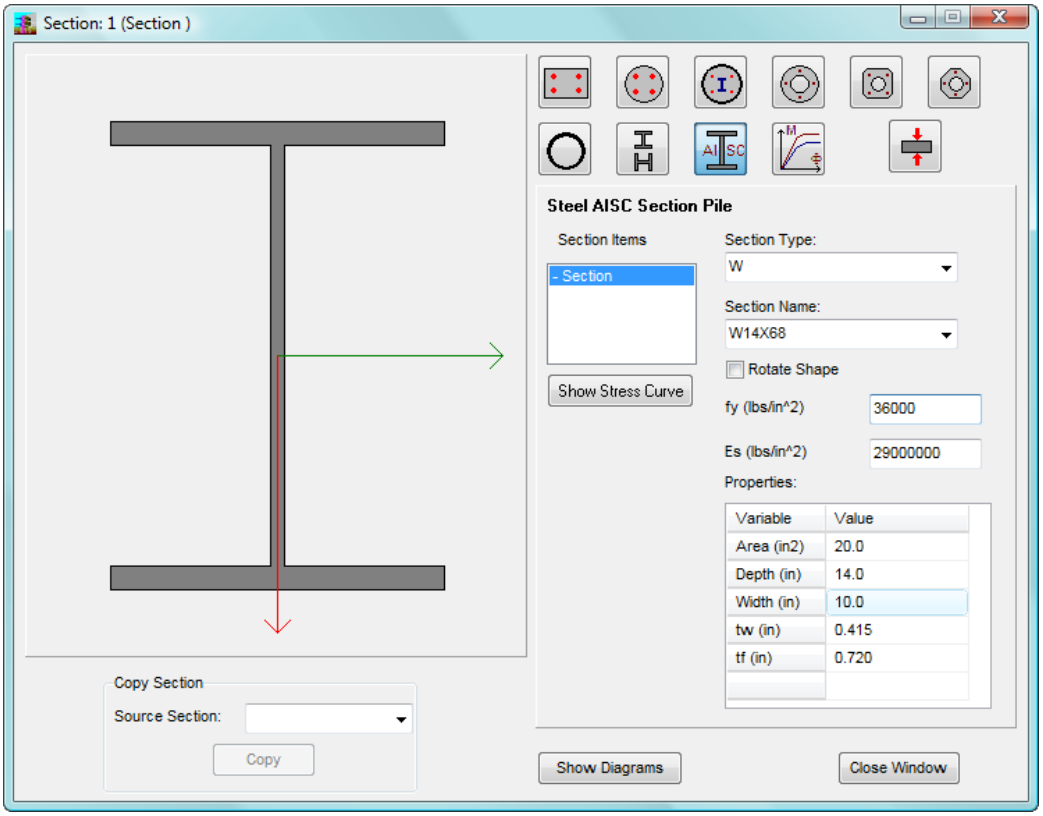


Figure 3.25 Sample AISC Steel Section with Nonlinear Properties

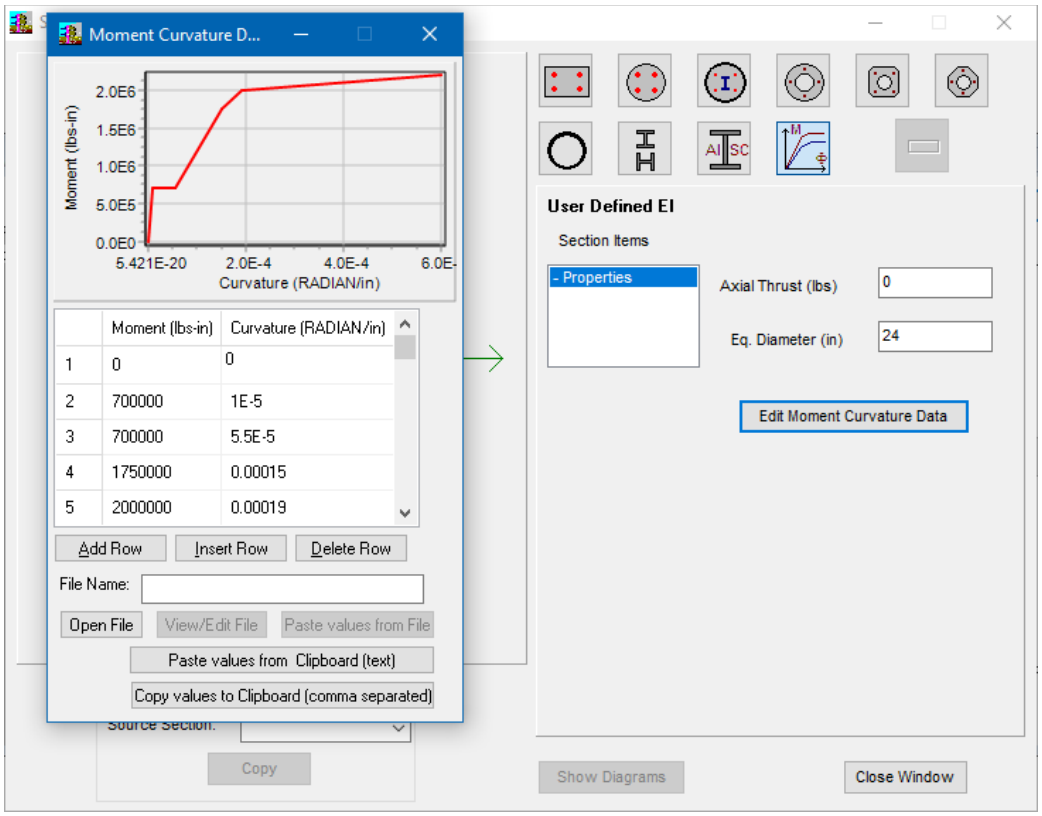


Figure 3.26 Sample Section with User-Defined Nonlinear Properties

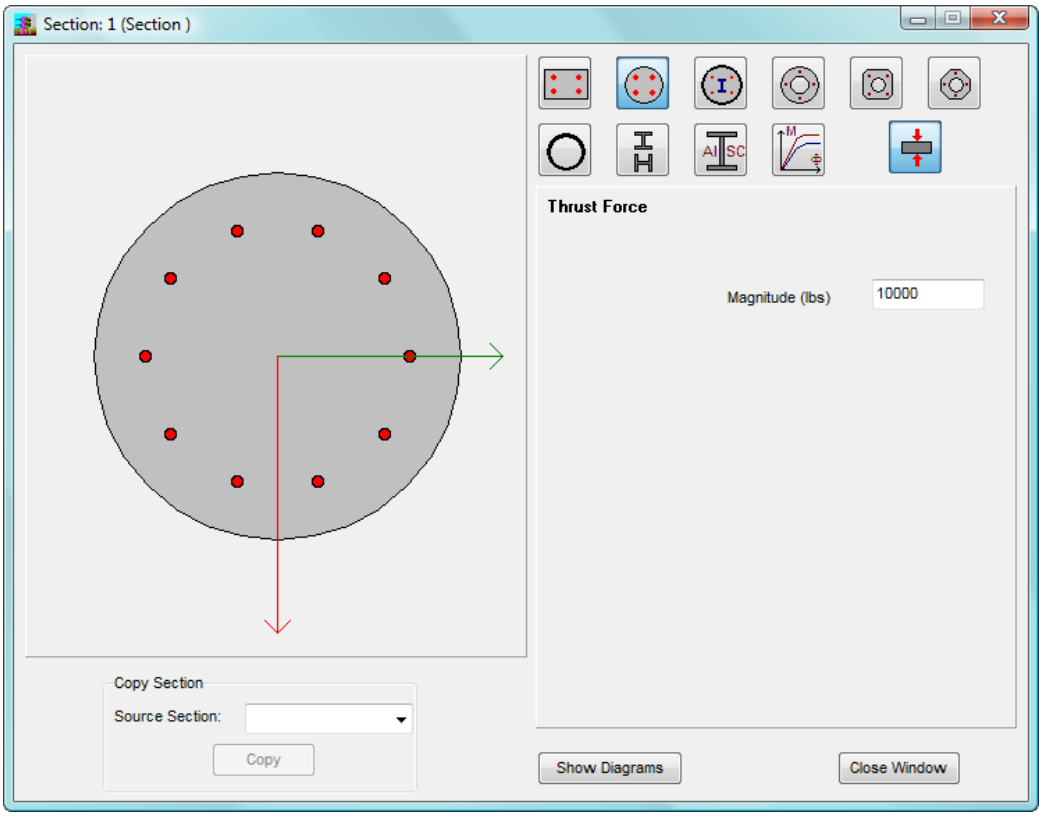


Figure 3.27 Sample entry of Axial Thrust Force

3.2.5.2 Section Items

Each nonlinear section defined in the previous topic has an association to different options under the Section Items. The parameters to the right of the Section Items will adjust depending on the item that is selected by the user. The following Section Items are available:

Section. Available to all nonlinear sections except for User Defined EI. Input parameters are the section dimensions and concrete compressive strength for concrete sections or yield strength and Young modulus of elasticity for steel sections.

Reinforcement. Available for all reinforced concrete nonlinear sections. Input parameters are the reinforcement properties and reinforcement layout.

Trans. Reinf. Available for both rectangular and round reinforced concrete sections. Input parameters are the transverse reinforcement to estimate the confinement following Mander equations.

Prestressing Strands. Available for all prestressed concrete sections. Input parameters are the strands properties and strands layout.

Casing. Available only for the round concrete shaft with casing section. Input parameters are the casing dimensions and yield strength and Young modulus of elasticity for the casing.

Steel Section. Available only for the round concrete shaft with casing section. While selecting this option the Emb. Steel Section will appear. The required inputs will depend on the embedded section that is selected by the user.

Properties. Available only to the User Defined EI. Allows the user to enter the Eq. Diameter (the equivalent diameter that should be used for the *p-y* curves corresponding to the defined nonlinear section), Axial Thrust (the axial load for the section), and Moment vs Curvature Data associated with the section.

3.2.5.3 Embedded Steel Section

This is available only for the round concrete shaft with casing section when the Steel Section is also selected under Section Items. The parameters to the right of the Section Items will adjust depending on the Emb. Steel Section that is selected by the user. The following items are available:

None. For no embedded steel section.

Pipe. To insert a steel pipe of defined dimensions and properties.

I, H. To insert a built-up I or H type steel section of defined dimensions and properties.

AISC. To select any W, M, S, HP, HSS or Pipe shape from the AISC manual.

3.2.5.4 Section Parameters

The input parameters depend on the choices of Nonlinear Section Selection (Section 3.2.5.1) and on the Section Item (Section 3.2.5.2) that were selected by the user. All the possible input parameters are listed below.

Width. Width of the cross section.

Height. Height of the cross section.

f'c. Concrete compressive strength.

Void Diam. Internal diameter of the section for prestressed sections. Leave as zero for a solid section.

Corner Chamf. Length of the corner chamfer. Only used for Square Prestressed Section.

Diameter. External diameter of the section. In addition to estimate the bending stiffness of the section, the program uses this value to estimate the contribution of soil resistance (*p-y* curve) for the lateral analysis.

Thickness. Thickness of the steel pipe section.

fy. Yield strength of the steel section/pipe.

Es. Young's Modulus of Elasticity of the steel section/pipe.

Web Thick. Thickness of the web for I or H steel section.

Flange Thick. Thickness of the flange for I or H steel section.

H Section. Check this option to assign an H section (ie., rotated I section).

Section Type. To select the AISC section type. Users can select any W, M, S, HP, HSS or Pipe shape from the AISC manual (also with any rotated installation angle).

Section Name. To select the AISC section size. After selecting the desired Section Name, the Properties table will be populated automatically (these entries cannot be edited by the user).

3.2.5.5 Reinforcement Parameters

These input parameters are used to define the size and layout of the longitudinal reinforcement on the wall section.

Bar Size. Select the size of longitudinal reinforcement from the drop-down list or choose User-Defined to enter the user-defined area of a non-standard rebar.

Area. Enter the cross-sectional area of a single rebar for User-Defined bar size, otherwise the program automatically estimates based on a selected bar size.

fy. Yield strength of the reinforcing bar.

Es. Young's Modulus of Elasticity of the reinforcing bar.

Number Bars. This is used to assign the total number of reinforcement bars in the entire cross section of the wall element. This entry is only used for Layout: Circ. (circular).

Number Bars Y. Assign the total number of reinforcement bars in the Y direction (following horizontal green axis). This entry is only used for Layout: Rect. (rectangular).

Number Bars Z. Assign the total number of reinforcement bars in the Z direction (following vertical red axis). This entry is only used for Layout: Rect. (rectangular).

Circ. By checking this option the program will automatically compute the rebar layout/coordinates based on a circular layout.

Rect. By checking this option the program will automatically compute the rebar layout/coordinates based on a rectangular layout.

Man. By checking this option the user should enter the rebar coordinates manually by selecting the Edit Bar Positions button.

Cover. This is the clear cover or distance from edge of section to outside edge of rebar. This entry is only used for Layout: Circ. (circular) or Layout: Rect. (rectangular). Not used for Layout: Man. (manual).

YC and ZC These entries are for the Y and Z coordinates of the individual reinforcing bar. The program will automatically compute these values if circular or rectangular layout are selected. Otherwise, user should enter these values.

3.2.5.6 Trans. Reinf. Parameters

These input parameters are used to define the size and layout of the transverse reinforcement on the wall section.

Confined Section. Check on this option to let the program estimate the confinement due to the transverse reinforcement.

Spiral. Available for circular sections. Select this option for placement of transverse reinforcement with spiraled lengths of bars.

Hoop. Available for circular sections. Select this option if the transverse reinforcement is made up of individual hoop bars.

Bar Size. Select the size of the transverse reinforcement from the drop-down list or choose User-Defined to enter the user-defined area of a non-standard rebar.

As. Enter the cross-sectional area of a single rebar for User-Defined bar size, otherwise the program automatically estimates based on a selected bar size.

Number Bars. Number of transverse reinforcement bars on each placed location along the spacing. This entry allows the use of bundled hoop bars that may be used in some seismic areas.

Set Legs from Long. Reinf. This check-mark box is only available for rectangular sections. Select this option to assign automatically the transverse bars based on the already provided longitudinal reinforcement.

Number Legs Y. Only available for rectangular sections. This is to set the number of transverse rebar legs along the Y direction.

Number Legs Z. Only available for rectangular sections. This is to set the number of transverse rebar legs along the Z direction.

Spacing. Distance between transverse reinforcement bars along the wall element, from center to center.

fys. Yield strength of the transverse reinforcing bar.

Use Strain Hardening Model. Check on this option to let the program use the strain hardening model for longitudinal reinforcement.

fu/fy. Enter the ratio of ultimate stress (fu) over yield stress (fy). This is only necessary for a strain hardening model.

esh. Enter the strain at which the hardening starts. This is only necessary for a strain hardening model.

esu. Enter the ultimate strain. This is only necessary for a strain hardening model.

3.2.5.7 Prestressing Strands Parameters

These input parameters are used to define the size and layout of the prestressing strands on the wall section.

Strand Type. Select the type of prestressing strand from the drop-down list or choose User-Defined to enter the user-defined strength of a non-standard strand type.

fu. Enter the ultimate strength of the strand for an User-Defined strand, otherwise the program estimates this entry based on a selected Strand Type.

Bar Size. Select the size of the prestressing strand from the drop-down list or choose User-Defined to enter the user-defined properties of a non-standard strand size.

Area. Enter the cross-sectional area of an individual prestressing strand for an User-Defined strand, otherwise the program estimates this entry based on the selected Bar Size.

Number Bars. This is used to assign the total number of prestressing strands in the entire cross section of the wall element. This entry is only used for Layout: Circ. (circular).

N. Bars Side. This is used to assign the total number of prestressing strands in each side of a rectangular cross section of the wall element. This entry is only used for Layout: Sqr. (square).

Circ. By checking this option the program will automatically compute the prestressing strand layout/coordinates based on a circular layout.

Sqr. By checking this option the program will automatically compute the prestressing strand layout/coordinates based on a square layout.

Man. By checking this option the user should enter the prestressing strand coordinates manually by selecting the Edit Strand Positions button.

Cover. This is the clear cover or distance from edge of section to outside edge of the strand. This entry is only used for Layout: Circ. (circular) or Layout: Sqr. (rectangular). Not used for Layout: Man. (manual).

Press. Force. Enter the initial force in the section before losses.

Fract. of Press. Loss. Enter the percentage of losses for the initial prestressing force.

Force After Losses = This is the calculated prestressing force after Press. Force and Fract. of Press. Loss entries above.

YC and ZC These entries are for the Y and Z coordinates of the individual prestressing strand. The program will automatically compute these values if circular or square layout are selected. Otherwise, user should enter these values.

3.2.5.8 Casing Parameters

These input parameters are used to define the size and properties of a permanent outside casing used for a Round Concrete Section with Casing nonlinear section.

Diameter. This is the external diameter of the permanent steel casing. In addition to estimate the bending stiffness of the section, the program uses this value to estimate the contribution of soil resistance (*p-y* curve) for lateral analysis.

Thickness. Thickness of the permanent steel casing.

fy. Yield strength of the permanent steel casing.

Es. Young's Modulus of Elasticity of the permanent steel casing.

3.2.5.9 Steel Section Parameters

These input parameters are used to define the size and properties of an embedded structural insert used for a Round Concrete Section with Casing nonlinear section. Options for embedded structural insert were defined in Section 3.2.5.3.

Diameter. This is the external diameter of the Pipe section.

Thickness. Thickness of the Pipe section.

fy. Yield strength of the Pipe section.

Es. Young's Modulus of Elasticity of the Pipe section.

Width. Width of the I,H section.

Height. Height of the I,H section.

Web Thick. Thickness of the web for I or H steel section.

Flange Thick. Thickness of the flange for I or H steel section.

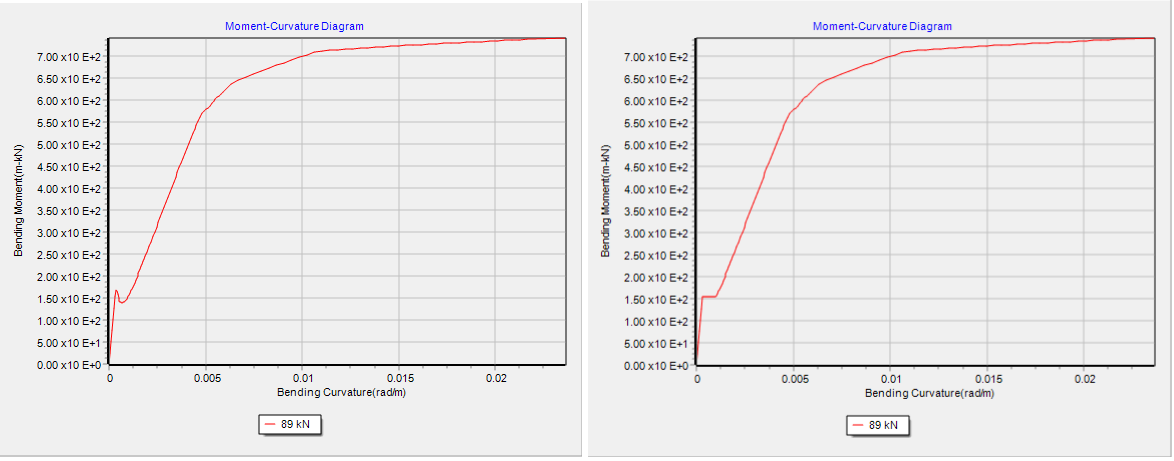
H Section. Check this option to assign an H section (ie., rotated I section).

Section Type. To select the AISC section type. Users can select any W, M, S, HP, HSS or Pipe shape from the AISC manual (also with any rotated installation angle).

Section Name. To select the AISC section size. After selecting the desired Section Name, the Properties table will be populated automatically (these entries cannot be edited by the user).

3.2.5.10 Adjust Softening Parameter

This option is selected by default and instructs the program to adjust the computed nonlinear flexural response of the wall section so as to eliminate abrupt nonlinearities that may occur from the mathematical approximations. The computed moment vs curvature of a concrete section may report a sudden reduction in bending moment mainly due to the initial cracking in the tension side of the wall section. See Figure 3.28 for a comparison of the effect of this parameter in a sample computation of moment vs curvature.



a) Without Adjust Softening b) With Adjust Softening

Figure 3.28 Adjust Softening Parameter

3.2.5.11 Show Stress Curve

The Show Stress Curve button opens a window screen displaying the stress curve used for the material which properties are being defined. It could show the stress curve for concrete, steel reinforcement or prestressing strand. See Figure 3.29 for an example displaying a concrete curve where the user also selected the option of Confined Section.

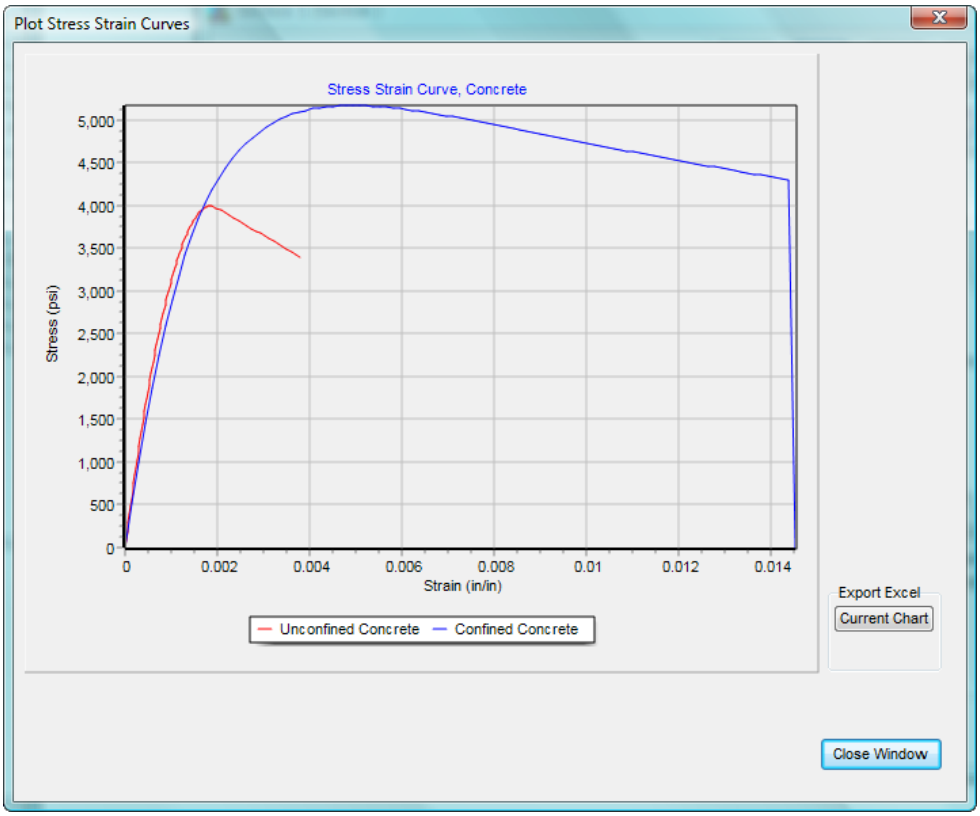


Figure 3.29 Sample Show Stress Curve for Concrete

The Export Excel Current Chart button can be used to export the observed plots into a new pre-formatted excel spreadsheet with tabs containing the numerical entries and other tabs containing the resulting graphs.

3.2.5.12 Show Diagrams

The Show Diagrams button opens a window screen that is used to the various nonlinear flexural response diagrams that were computed by the program for the current wall section parameters. Available diagrams are: Bending Moment-vs-Curvature, Bending Stiffness-vs-Curvature, Bending Stiffness-vs-Bending Moment and Axial Force-vs-Bending Moment Interaction. See Figure 3.40 for an example displaying a Bending Moment-vs-Curvature for a reinforced concrete wall section.

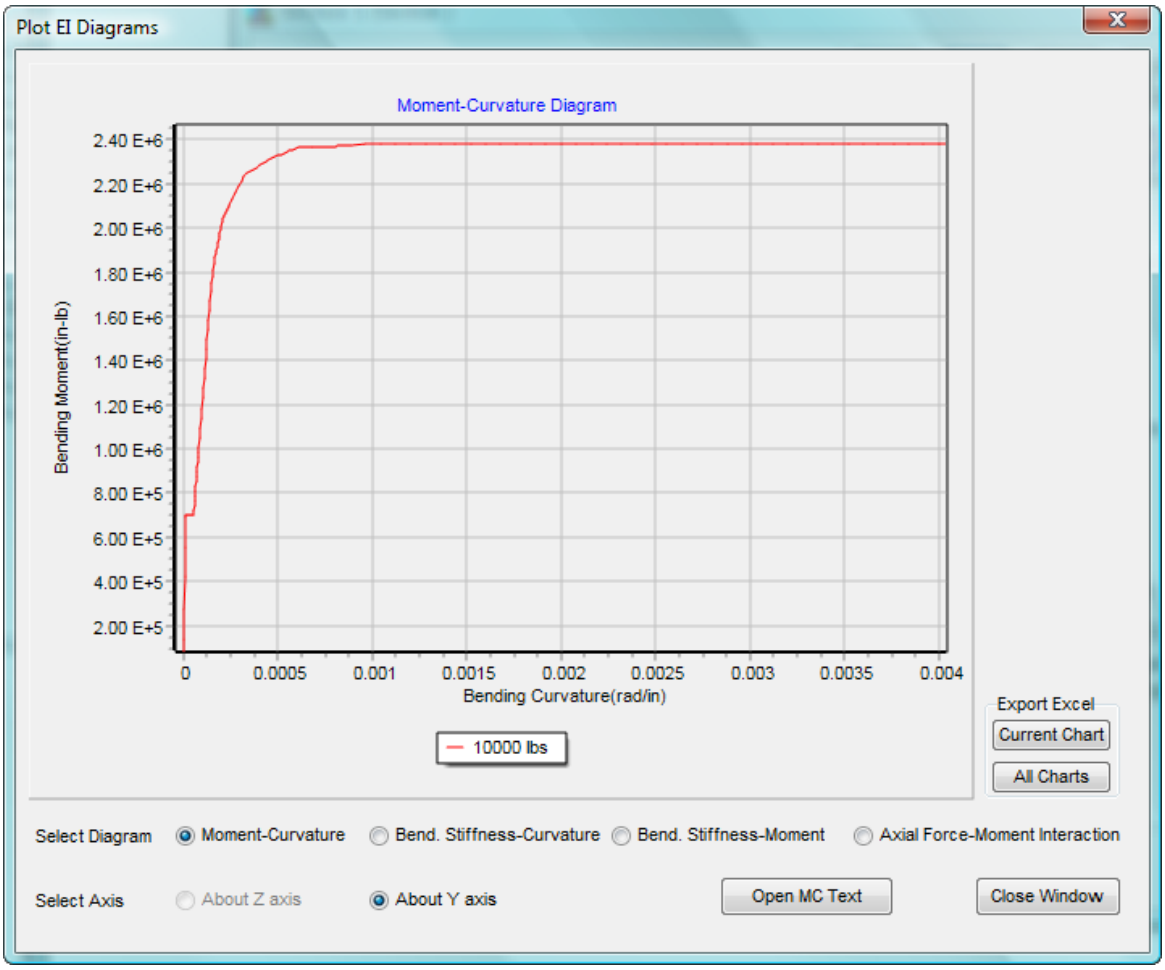


Figure 3.30 Sample Show Diagram for Reinforced Concrete

The Export Excel Current Chart or All Charts buttons can be used to export the observed plot or all plots into a new pre-formatted excel spreadsheet with tabs containing the numerical entries and other tabs containing the resulting graphs.

3.2.6 Wall Properties

This dialog screen allows the user to assign the previously-defined sections under Data > Wall Sections to specific section heights of the retaining wall. The menu also contains various parameters used for the numerical discretization of the wall. A sample Wall Properties dialog box is shown in Figure 3.31. and descriptions for the various entries are listed in subsequent topics.

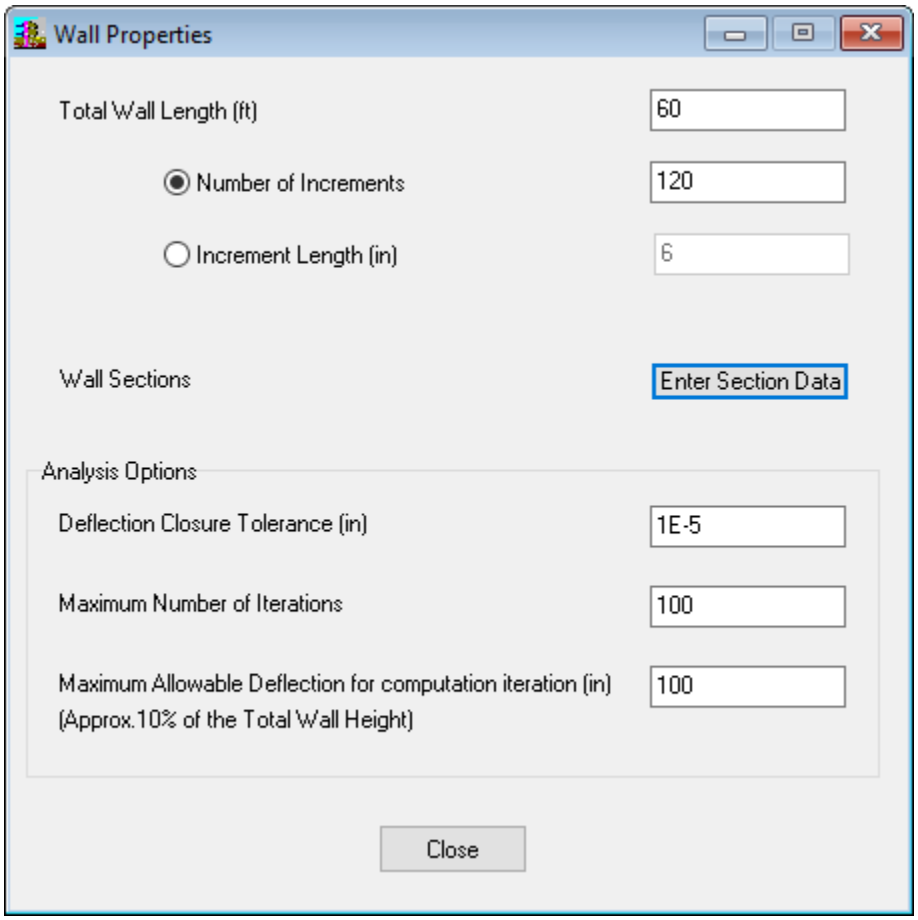


Figure 3.31 Data > Wall Properties

3.2.6.1 Total Wall Length

This entry (always positive) represents the total length (in vertical direction) of the wall, which is the free height of the wall plus the embedded length of wall below the excavation.

3.2.6.2 Number of Increments

The entry for Number of Increments allows the user to specify into how many vertical segments the previously entered Total Wall Height will be divided. In theory, the higher the number of increments the more precise the solution and the longer the computation time. The maximum number of increments that a wall height may be divided into is 300.

While choosing the appropriate number of increments, notice that if the number of increments is too great, the differences between the finite increments would become very small and mathematical convergences may not be reached between extremely small differences. Conversely, if the number of

increments is too large, the wall may be improperly modeled. Using increments from 50 to 150 has been found to produce results with acceptable accuracy in most models. However, for some problems, say for short penetration below the excavation line, the user may wish to investigate the impact in the results with lower number of increments.

Users may enter either the Number of Increments or the Increment Length. The division of Total Wall Height by the Number of Increments defines the Increment Length parameter which is calculated automatically by the program when user selects to enter Number of Increments.

3.2.6.3 Increment Length

This entry specifies the length (in vertical direction) of each finite increment in which the Total Wall Height shall be divided. Users may enter either the Number of Increments or the Increment Length. The division of Total Wall Height by the Increment Length defines the Number of Increments parameter which is calculated automatically by the program when user select to enter Increment Length. Users must enter an Integer Length that is divisible by the Total Wall Height or an error is produced.

When using models in English Units, it is usually beneficial to define integer numbers as Increment Length. For example, on a 25-ft total wall height (free height plus embedded length) the model may use 6-in increments (resulting in Number of Increments = 50). On models with SI Units the Increment Length may be equal to decimal numbers (integers when read in cm or in mm). For example, on a 10-m total wall height (free height plus embedded length) the model may use 0.2-m (20 cm) increments (resulting in Number of Increments = 50). It is usually recommended to use Increment Length between 3 and 10 inches (0.08 to 0.25 m).

3.2.6.4 Wall Sections - Enter Section Data

The Enter Section Data button allows users to assign any of the previously-defined sections under Data > Cross Sections (Section 3.2.3) to specific section heights/depths of the retaining wall. The user may enter up to 20 rows of data with different assignations. A sample Wall Sections dialog box is shown in Figure 3.32. and a general description for the various entries is listed below.

Sections

This is a sequential number that is provided for each unit section of wall height/depth.

Top

This numeric entry is the coordinate for the beginning of the row/section that is being assigned. The origin of vertical coordinates is zero at the top of the retaining wall and increases downwards.

Bottom

This numeric entry is the coordinate for the end of the row/section that is being assigned. The origin of vertical coordinates is zero at the top of the retaining wall and increases downwards. The user must make sure that the last Bottom entry is equal to the Total Wall Length (Section 3.2.6.1, total height of wall including free plus embedded heights).

Cross Section Index

This entry is for the integer number that represents one of the previously-defined sections under Data > Cross Sections (Section 3.2.3). The user is thus able to assign in this row the chosen Cross Section to the specific height/depth of retaining wall that is being defined.

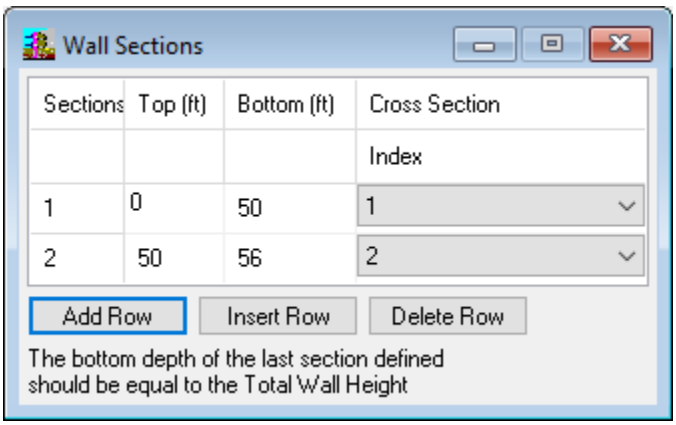


Figure 3.32 Data > Wall Properties > Enter Section Data

3.2.6.5 Deflection Closure Tolerance

This entry is for the tolerance (in units of Length) that is specified for mathematical convergence. When the maximum change in deflection at any node for successive iterations is less than this specified tolerance then iteration stops.

There is no specific guideline regarding acceptable ranges of this input parameter but usually the default of 0.00001 inches covers a great range of numerical models. However, in some cases with large wall movements the user may need to increase this input value to 0.001 inch or even larger values to see if a failed model can reach mathematical convergence.

3.2.6.6 Maximum Number of Iterations

This entry is always positive and represents the maximum number of iterations that the program shall perform to achieve mathematical convergences. The default of 100 is usually sufficient for the majority of the models. Maximum is 500 iterations.

3.2.6.7 Maximum Allowable Deflection

This entry is always positive and represents a value of deflection that is considered excessive for computation iterations between finite increments of the wall. The user should enter a large value (such as 10 times of the wall diameter/width) to ensure the program completes the nonlinear iterations without an early termination. This is not a value that can be used to control the resulting maximum lateral deflection of the modeled wall.

3.2.7 Wall Configuration

This dialog box allows the user to enter the geometry data of the retaining wall along with water table depths and any uniform loads on top of the retained surface. This submenu is enabled when the user selects to have the program generate active earth pressure and soil resistance curves internally. A sample Wall Configuration dialog box is shown in Figure 3.31. and descriptions for the various entries are listed in subsequent topics.

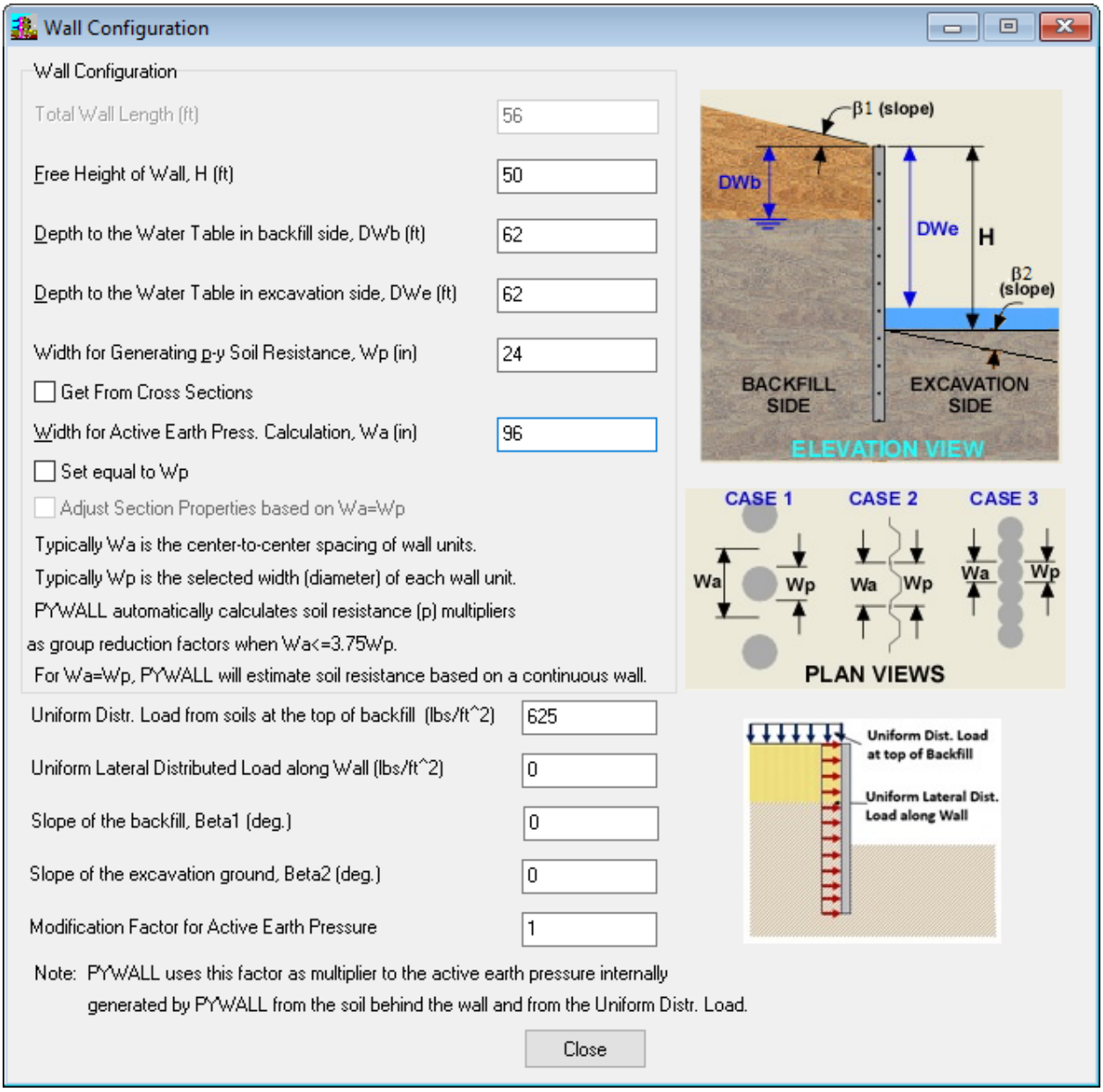


Figure 3.33 Data > Wall Configuration

3.2.7.1 *Total Wall Height*

This is a reference to the Total Wall Length that was previously defined by the user under in Section 3.2.6.1 under Data > Wall Properties (total height of wall including free plus embedded heights).

3.2.7.2 *Free Height of Wall, H*

Enter here the length/height of wall that is above the dredge line. This entry must be lower than the Total Wall Height.

3.2.7.3 Depth to the Water Table in Backfill Side, DWb

The user needs to specify the distance to the water table in the backfill side, measured from the top of the wall.

The program uses this value internally for two purposes:

i) calculation of any contributions to internal computations of active earth pressure (unless the user selects Options > Earth-Pressure Options > Earth Pressure Specified by User), and

ii) depth for calculation of submerged unit weight of soils which affects certain internal *p-y* curves (unless the user removes the default check mark under Options > Generate Internal p-y Curves and places a check mark under Options > Apply User-Specified p-y Curves (External)).

Any contributions to internal computations of active earth pressures from water differential height will be displayed in the output graphics with the label from Water under Graphics > Earth Pressure Combined.

3.2.7.4 Depth to the Water Table in Excavation Side, DWe

The user needs to specify the distance to the water table in the excavation side, measured from the top of the wall.

The program uses this value internally for two purposes:

i) calculation of any contributions to internal computations of active earth pressure (unless the user selects Options > Earth-Pressure Options > Earth Pressure Specified by User), and

ii) depth for calculation of submerged unit weight of soils which affects certain internal *p-y* curves (unless the user removes the default check mark under Options > Generate Internal p-y Curves and places a check mark under Options > Apply User-Specified p-y Curves (External)).

Any contributions to internal computations of active earth pressures from water differential height will be displayed in the output graphics with the label from Water under Graphics > Earth Pressure Combined.

3.2.7.5 Width for Generating p-y Soil Resistance, Wp

The user needs to specify the unit width of the wall used for computations of soil resistances below the dredge/excavation line. For example, if a soldier-pile wall has a center to center spacing of 5 ft (1.5 m) between piles and the diameter is 2 ft (0.61 m) for each pile, the width for passive soil resistance (Wp) computations is only 2 ft (0.61 m). On continuous diaphragm walls and sheet-pile walls the Wp entry is usually an unit length of wall (1-m or 1-ft) and equal to the Wa entry. On sheet-pile walls some users may choose to use the length of a sheet-pile section (eg, 18 in for a standard PZ27 sheet pile).

A check mark under Get From Cross Sections instructs the program to use the same width/diameter that was defined under Data > Cross Sections. The program averages values when more than one cross section was defined.

3.2.7.6 Width for Active Earth Press. Calculation, Wa

The user needs to specify the unit width of the wall that is used for internal computation of earth pressure above the dredge line. The user should consider the whole tributary width for each wall support unit. For example, if a soldier-pile wall has a center to center spacing of 5 ft (1.5 m) between piles, the width for active-earth-pressure computation (Wa) is 5 ft (1.5 m) even though the diameter of each pile is

smaller. On continuous diaphragm walls and sheet-pile walls the Wa entry is usually an unit length of wall (1-m or 1-ft) and equal to the Wp entry. On sheet-pile walls some users may choose to use the length of a sheet-pile section (eg, 21 in for a standard PZ32 sheet pile).

A check mark under Equal to Wp instructs the program to use the same width/diameter that was defined for Wp. Notice that the program automatically computes soil resistance (p) multipliers for "group" effects (for closely-spaced piles) when Wa<=3 Wp

3.2.7.7 Adjust Section Properties based on Wa=Wp

When the user enters a value for Wp and also selects (places a check mark) Equal to Wp the program will automatically place a check mark on Adjust Section Properties based on Wa=Wp. This is necessary to instruct the program to use the flexural stiffness that corresponds to the Wp value that was entered by the user. The adjustment is made proportionally, based on the ratio of Wp and the Diameter (or Section Width) that were entered by the user (under Data > Cross Sections explained in Section 3.2.3).

The user should always check the RESULTS table of in the output text file (Computation > View Output Text) to make sure that the correct flexural stiffness (listed in the 'EI' column) was used for the wall section in the analyses.

3.2.7.8 Uniform Distributed Load from Soils at the Top of Backfill

The user can specify a uniformly-distributed load that is permanent (eg, from extra soils) at the top of the backfill side. The resulting lateral pressures from this entry of surcharge loads are plotted under "Graphics > Earth Pressure Combined" with the legend "from Uniform Distributed Loads" (see Section 4.3.12).

Please notice that this entry is not available (not used) for user-specified active earth pressures (if the user selects Options > Earth-Pressure Options > Earth-Pressure Specified by User). When the earth pressure is specified by user the program assumes that the user already takes into account any additional uniform distributed load at the top of the backfill.

3.2.7.9 Uniform Lateral Distributed Load along Wall

The user can specify any applicable uniformly-distributed lateral load that is acting along the height of the wall. This entry must be inputted as pressure (units of Force/Area) acting perpendicular to the wall with positive value when applied outwards from the wall (towards the excavation) and negative value when acting towards the retained soil (inwards). The applicable load per unit height of wall is obtained by multiplying this entry times the value of Wa that was previously specified by the user (see Section 3.2.7.6).

This user-specified lateral pressure is plotted under "Graphics > Earth Pressure Combined" with the legend "from Uniform Distributed Loads" (see Section 4.3.12).

Please notice that this entry is not available (not used) for user-specified active earth pressures (if the user selects Options > Earth-Pressure Options > Earth-Pressure Specified by User). When the earth pressure is specified by user the program assumes that the user already takes into account any additional uniform distributed lateral load that is acting along the wall.

3.2.7.10 Slope of the Backfill, Beta1 (deg.)

The user can specify the slope angle of the backfill soil as indicated in the graphical reference. The Beta1 angle, β1, must be equal to the default of zero or a positive number (in degrees). Please notice that this entry is not available (not used) for user-specified active earth pressures (if the user selects Options > Earth-Pressure Options > Earth-Pressure Specified by User). When the earth pressure is specified by the user, the program assumes that the entries already takes into account any slope of the backfill.

3.2.7.11 Slope of the Excavation Ground, Beta2 (deg.)

Users may specify the slope angle of the excavation ground as indicated in the graphical reference. The Beta2 angle, β2, must be equal to the default of zero or a positive number (in degrees). A slope on the ground surface of the excavation introduces a modification (reduction) of the upper lateral soil response (*p-y* curves) in the excavation side. However, this modification is only included for soil layers and not for rock. There is no solid research that describes how the *p-y* curves of rock should be altered based on the slope angle. In addition, some rock may have seams and the direction of those seams with respect to the direction of loading will also have a large influence on their lateral response. Users should have extreme caution when evaluating slopes of rock.

3.2.7.12 Modification Factor for Active Earth Pressure

The user may specify a modification factor to increase or decrease the internally-generated earth pressure. Notice that this factor affects the active earth pressure that is generated internally by the program due to the soil behind the wall, due to any water differential and also due to any user-specified distributed loads (from Section 3.2.7.8). Please notice that this entry is not available (not used) for user-specified active earth pressures (if the user selects Options > Earth-Pressure Options > Earth-Pressure Specified by User).

3.2.8 Soil Layer Data

This dialog box allows the user to enter the different types of soils that will be used for the automatic generation of active earth pressure and soil resistance (*p-y*) curves. Program PYWALL will generate the soil-resistance curves below the excavation level automatically if the user leaves the default check mark on the submenu Options > Generate p-y Curves.

An illustration of the Soil Layers window for an example application is shown in Figure 3.34. The Add Row button always adds new rows at the end, after all existing rows, and the Delete Row button deletes the row where the cursor is currently located.

The maximum depth of the bottom soil layer should be limited to no more than 2 increments from the wall/pile tip or the data model may generate a warning/error message (i.e., bottom of last soil layer should be anywhere from wall/pile tip to two increments below pile tip). Although small, some variations in wall response will occur if the model uses very large distances between wall/pile tip and the bottom of last soil layer.

Descriptions for the data needed under each column in the Data > Soil Layer Data submenu are listed below.

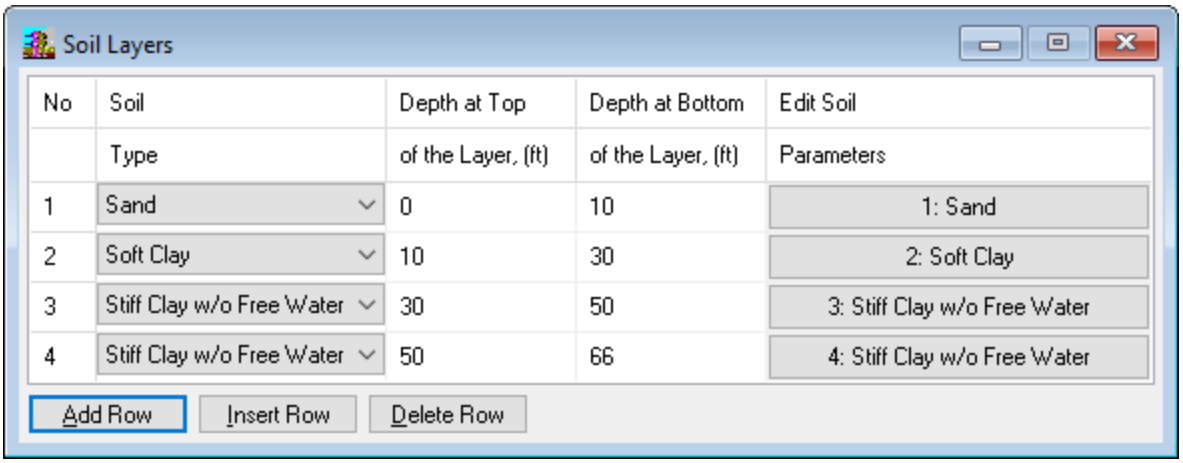


Figure 3.34 Data > Soil Layer Data

No

This is a sequential number that is provided for each soil layer. This number is automatically provided by the program as new rows of soil layers are added. The maximum number of soil layers that may be used in a program run is limited to 10. The soil layer should start at the top of the backfill behind the wall and count down one by one with sequential order.

Soil Type

There are currently eight internal types of soils that may be specified for program PYWALL. The user specifies the desired soil type using a dropdown list with the following choices:

Code Number	Internal Soil Type
1	Soft Clay
2	Stiff Clay with Free Water
3	Stiff Clay w/o Free Water
4	Mod. Stiff Clay w/o Free Water
5	Sand
6	Silt (c-phi soil)
7	Strong Rock (qu > 1000 psi)
8	Weak Rock

Depth at Top and at Bottom of the Layer

Values for the depth at the top and bottom of each soil layer are entered according to the soil profile, sequentially from grade at the top of retained soil to below the tip of the wall. The depth, always entered in positive values, is measured from the top of the wall (origin of vertical coordinates, always positive downwards).

Most models will use zero for the Depth at Top of the Layer in the first layer of the model. However, if the user wants to model a wall that sticks up a distance above a retained soil, the value of Depth at Top of the Layer for the top layer will be a number greater than zero.

The maximum depth of the bottom soil layer should be limited to no more than 2 increments from the wall/pile tip or the data model will generate a warning/error message (i.e., bottom of last soil layer should be anywhere from wall/pile tip to two increments below pile tip). Although small, some variations

in wall response will occur if the model uses very large distances between wall/pile tip and the bottom of last soil layer.

Edit Soil Parameters

The Edit Soil Parameters buttons allow the user to define soil or rock properties for each layer. A new dialog box will pop on the screen when the user selects any button under the column Edit Soil Parameters. The program requires different input parameters for each type of soil and rock. Therefore, the data column in the sub-window that appears varies with the selection made by the user in the Soil Type drop-down list.

In general, each dialog box that comes up under the Edit Soil Parameters button consists of entry fields for the top of the layer and the bottom of the layer. The user may enter different parameters for the top and the bottom of each layer and the program will linearly interpolate the data for any point between those two depths. The following sections will provide general description on data needed for each soil type.

3.2.8.1 Soil Data for Clay Layers

A sample window for all three different clay soils (Soft Clay, Stiff Clay with Free Water and Stiff Clay w/o Free Water) is shown in Figure 3.35. The soil type Stiff Clay with Free Water is used to represent soil conditions where stiff clay is the top layer and there is water above the ground line. If the stiff clay is below a sand layer the user should select Stiff Clay w/o Free Water even though the water table may be in the sand layer above the clay. The required properties for all clay layers are explained further below.

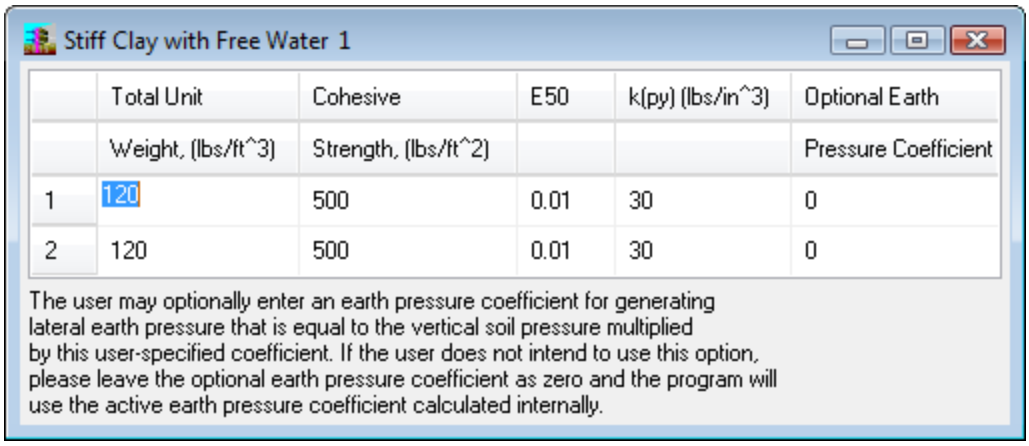


Figure 3.35 Sample Data Screen for Clays

Total Unit Weight

This column entry corresponds to values of total unit weight (submerged weight is calculated based on separate entry for depth of water table, see Section 3.2.7.3). Values for the top and bottom of the sand layer are entered in standard units of force per unit volume (either lb/ft^3 or kN/m^3).

Cohesive Strength

This column represents the input values for the unconfined-undrained shear strength at the top and bottom of the clay layer. Ordinarily, these values are taken as one half of the compression strength of samples obtained from unconsolidated-unconfined triaxial tests.

Epsilon 50 (E50) Strain

Values of strain at 50% of the maximum stress ϵ_{50} for clays and/or silts at each soil depth are entered in standard units of strain. Reference values are provided in Table 3.2 and Table 3.3.

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

Table 3.2 Values of E50 for Clays

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

Table 3.3 Values of E50 for Stiff Clays

K (py) Values

This is the value for the constant k used in the equation $E_s = k\,x$. This constant is in units of force per cubic length and depends on the type of soil and lateral loading imposed to the wall. It has two different uses: i) to define the initial (maximum) value of E_s on internally-generated *p-y* curves of stiff clays with free water and/or sands; and ii) to initialize the E_s array for the first iteration of analysis. Suggested values of the parameter k used for clays are given in Table 3.4.

Optional Earth-Pressure Coefficient

The user may optionally enter an earth-pressure coefficient so in this soil layer the program will generate a lateral earth pressure that is equal to the vertical soil pressure multiplied by this user-specified coefficient. In models where this new feature is not needed the user can leave the optional earth-pressure coefficient as zero or one and the program will use the active earth pressure coefficient that is calculated internally.

The optional, user specified, earth-pressure coefficient may be preferred in certain clay soils where the Rankine active earth pressure becomes close or equal to zero or negative when using the following equation (since $K_a = 1$):

$$P_a = K_a \gamma H - 2c\,H\sqrt{K_a} \tag{3.1}$$

Notice that the user-specified, earth-pressure coefficient (K_u) is used internally in similar manner as the at-rest earth pressure coefficient K_0:

$$\sigma_u = K_u \gamma H \tag{3.2}$$

where

σ_u = lateral earth pressure,
K_u = user-specified earth-pressure coefficient,
γ = soil unit weight,
H = the distance below the ground surface.

Clay Type	Avg. Undrained Shear Strength c_u	Static	Cyclic
Soft	1.74-3.47 psi 250-500 psf 12-24 KPa	30 pci 8,140 KPa/m	
Medium	3.47-6.94 psi 500-1000 psf 24-48 KPa	100 pci 27,150 KPa/m	
Stiff	6.94-13.9 psi 1000-2000 psf 48-96 KPa	500 pci 136,000 KPa/m	200 pci 54,300 KPa/m
Very Stiff	13.9-27.8 psi 2000-4000 psf 96-192 KPa	1000 pci 271,000 KPa/m	400 pci 108.500 KPa/m
Hard	27.8-55.6 psi 4000-8000 psf 192-383 KPa	2000 pci 543,000 KPa/m	800 pci 217,000 KPa/m

Table 3.4 Soil-Modulus Parameter k(py) for Clays

3.2.8.2 Soil Data for Sand Layers

A sample window for sand layers, as recommended by Reese et al., 1974, is shown in Figure 3.36. The required properties for sand layers are explained further below.

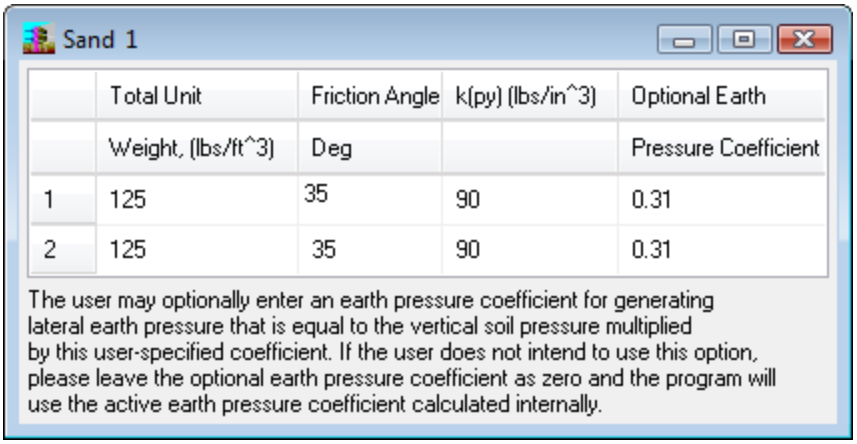


Figure 3.36 Sample Data Screen for Sands

Total Unit Weight

This column entry corresponds to values of total unit weight (submerged weight is calculated based on separate entry for depth of water table, see Section 3.2.7.3). Values for the top and bottom of the sand layer are entered in standard units of force per unit volume (either lb/ft^3 or kN/m^3).

Friction Angle

This column entry corresponds to values of the internal angle of friction (also known as friction or ϕ angle) for the top and bottom of the sand layer. The values of Friction Angle are entered in standard units of degrees.

K (py) Values

This is the value for the constant k used in the equation $E_s = k\,x$. This constant is in units of force per cubic length and depends on the type of soil and lateral loading imposed to the wall. It has two different uses: i) to define the initial (maximum) value of E_s on internally-generated p-y curves of stiff clays with free water and/or sands; and ii) to initialize the E_s array for the first iteration of analysis. Suggested values of the parameter k used for sands are given in Table 3.5.

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

Table 3.5 Soil-Modulus Parameter k(py) for Sands

Optional Earth-Pressure Coefficient

The user may optionally enter an earth-pressure coefficient so in this soil layer the program will generate a lateral earth pressure that is equal to the vertical soil pressure multiplied by this user-specified coefficient. In models where this new feature is not needed the user can leave the optional earth-pressure coefficient as zero and the program will use the active earth pressure coefficient that is calculated internally.

Notice that the user-specified, earth-pressure coefficient (K_u) is used internally in similar manner as the at-rest earth pressure coefficient K_0:

$$\sigma_u = K_u \gamma H \qquad (3.3)$$

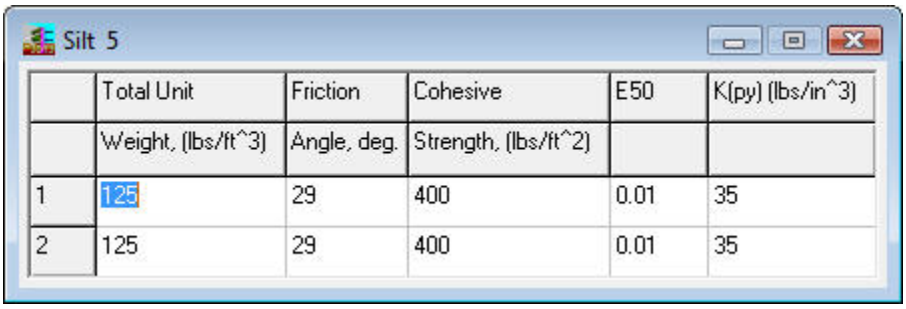
where

σ_u = lateral earth pressure,
K_u = user-specified earth-pressure coefficient,
γ = soil unit weight,
H = the distance below the ground surface.

The optional earth-pressure coefficient is thus helpful for users that want to apply equivalent fluid pressure as load in the wall.

3.2.8.3 Soil Data for Silt Layers

A sample window for silt layers is shown in Figure 3.37. The required properties for silt layers are explained further below.



	Total Unit Weight, (lbs/ft^3)	Friction Angle, deg.	Cohesive Strength, (lbs/ft^2)	E50	K(py) (lbs/in^3)
1	125	29	400	0.01	35
2	125	29	400	0.01	35

Figure 3.37 Sample Data Screen for Silt

Total Unit Weight

This column entry corresponds to values of total unit weight (submerged weight is calculated based on separate entry for depth of water table, see Section 3.2.7.3). Values for the top and bottom of the sand layer are entered in standard units of force per unit volume (either lb/ft^3 or kN/m^3).

Friction Angle

This column entry corresponds to values of the internal angle of friction (also known as friction or ϕ angle) for the top and bottom of the silt layer. The values of Friction Angle are entered in standard units of degrees.

Cohesive Strength

This column represents the input values for the unconfined-undrained shear strength at the top and bottom of the silt layer. Ordinarily, these values are taken as one half of the compression strength of samples obtained from unconsolidated-unconfined triaxial tests.

Epsilon 50 (E50) Strain

Values of strain at 50% of the maximum stress ϵ_{50} for clays and/or silts at each soil depth are entered in standard units of strain. Reference values were provided earlier in Table 3.2 and Table 3.3.

K (py) Values

This is the value for the constant k used in the equation $E_s = k\,x$. This constant is in units of force per cubic length and depends on the type of soil and lateral loading imposed to the wall. It has two different uses: i) to define the initial (maximum) value of E_s on internally-generated *p-y* curves of stiff clays with free water and/or sands; and ii) to initialize the E_s array for the first iteration of analysis. Suggested values of the parameter k used for clays were previously given in Table 3.4. Suggested values of the parameter k used for sands were previously given in Table 3.5.

Notice that usage of high K(py) values are expected only for cases of "cemented" sands (calcareous soils). For all other cases, the value that is selected for K(py) should be based on the friction angle of the sand since the criteria is for soils that are primarily granular in nature. If the soil has low friction angle and

relatively large cohesive strength but it is not classified as cemented sand or calcareous the user may choose instead one of the clay criteria or perhaps weak rock.

Optional Earth-Pressure Coefficient

The user may optionally enter an earth-pressure coefficient so in this soil layer the program will generate a lateral earth pressure that is equal to the vertical soil pressure multiplied by this user-specified coefficient. In models where this new feature is not needed the user can leave the optional earth-pressure coefficient as zero and the program will use the active earth pressure coefficient that is calculated internally.

Notice that the user-specified, earth-pressure coefficient (K_u) is used internally in similar manner as the at-rest earth pressure coefficient K_0:

$$\sigma_u = K_u \gamma H \qquad (3.4)$$

where

σ_u = lateral earth pressure,
K_u = user-specified earth-pressure coefficient,
γ = soil unit weight,
H = the distance below the ground surface.

The optional earth-pressure coefficient is thus helpful for users that want to apply equivalent fluid pressure as load in the wall.

3.2.8.4 Soil Data for Strong Rock

A sample window for strong rock layers is shown in Figure 3.38. The required properties for strong rock layers are explained further below.

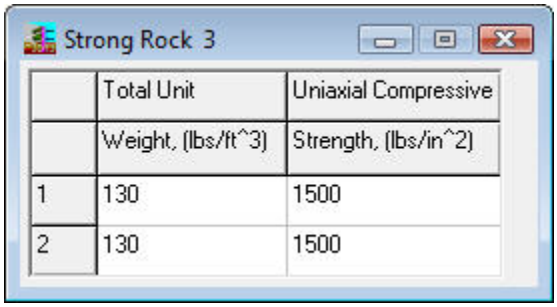


Figure 3.38 Sample Data Screen for Strong Rock

Total Unit Weight

This column entry corresponds to values of total unit weight (submerged weight is calculated based on separate entry for depth of water table, see Section 3.2.7.3). Values for the top and bottom of the sand layer are entered in standard units of force per unit volume (either lb/ft^3 or kN/m^3).

Uniaxial Compressive Strength

This entry allows the user to specify the uniaxial compressive strength for rock layers. It should be noted that lbs/in^2 (psi) is used for English units since high values are expected for rock material.

3.2.8.5 Soil Data for Weak Rock

A sample window for weak rock layers is shown in Figure 3.39. The required properties for weak rock layers are explained further below.

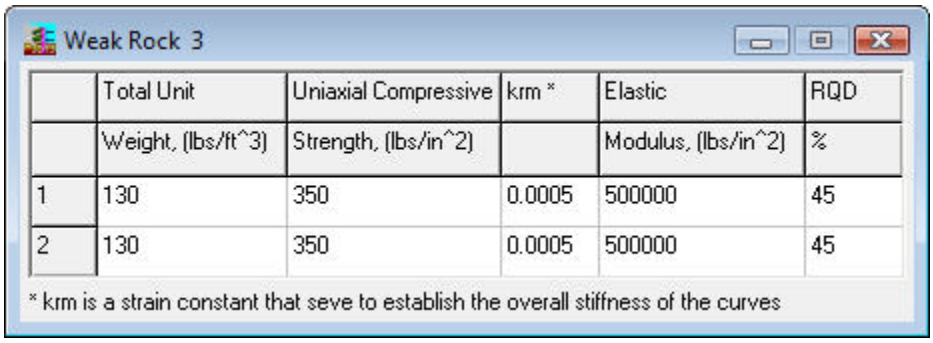


Figure 3.39 Sample Data Screen for Weak Rock

Total Unit Weight

This column entry corresponds to values of total unit weight (submerged weight is calculated based on separate entry for depth of water table, see Section 3.2.7.3). Values for the top and bottom of the sand layer are entered in standard units of force per unit volume (either lb/ft^3 or kN/m^3).

Uniaxial Compressive Strength

This entry allows the user to specify the uniaxial compressive strength for rock layers. It should be noted that lbs/in^2 (psi) is used for English units since high values are expected for rock material.

krm

This is a constant, ranging from 0.0005 to 0.00005, that serves to establish the overall stiffness of the curves for weak rock. More information regarding this parameter may be available in the accompanying Technical Manual and mentioned references.

Elastic Modulus

This is the initial Young's modulus of the rock mass. The elastic modulus obtained from the rock core needs to be adjusted by the geotechnical engineer for a representative Young's modulus for the rock mass.

RQD (Percentage)

Rock quality index, RQD, is needed for each rock layer. The user can enter different RQD values for the top and the bottom of each layer and the program will interpret linearly during computation.

3.2.9 Cyclic Model Data

This dialog box allows the user to enter the number of cycles that the program should use to compute the corresponding cyclic *p-y* curves. Notice that only some soils have a cyclic p-y criteria and only a few of those are influenced by the number of cycles. This entry is enabled if the user placed a check mark on the submenu Options > Use Cyclic Model for p-y Curves.

3.2.10 Distributed Active Earth Pressure

This dialog box allows the user to enter a special type of distributed earth pressure or loading that should be applied to the wall. This entry is enabled if the user placed a check mark on the submenu Options > Earth-Pressure Options > Earth-Pressure Specified by User (see Section 3.3.2).

The user can input the distributed earth pressure based on other recommendations from technical literature. The user may input up to thirty different values for laterally distributed load, which are placed in units of load per unit depth.

It should be noted that earth pressure should take into account the width of each contribution area for the analysis (pressure is for every *Wa* as defined in Section 3.2.7.6). The user must enter values in increasing magnitudes of depth. The program interpolates linearly the values of laterally distributed loads between specified depths. At least two entries (two depths) of distributed lateral loads are thus needed. An illustration of a typical window screen for Distributed Active Earth Pressure is shown in Figure 3.40.

If applicable, the effects from any surcharge loads (see Section 3.2.11) must also be taken into account in the user-specified loading.

Notice that a distributed active earth pressure or loading that is entered by the user may need to be extended below the excavation level to account for the vertical pressure from the backfill soil. The extension below excavation may be constant or decreasing to zero at the tip of the pile/wall unit. Users are encouraged to try both and use the most conservative loading (which depends on the type of lateral movement of the modeled pile/wall unit).

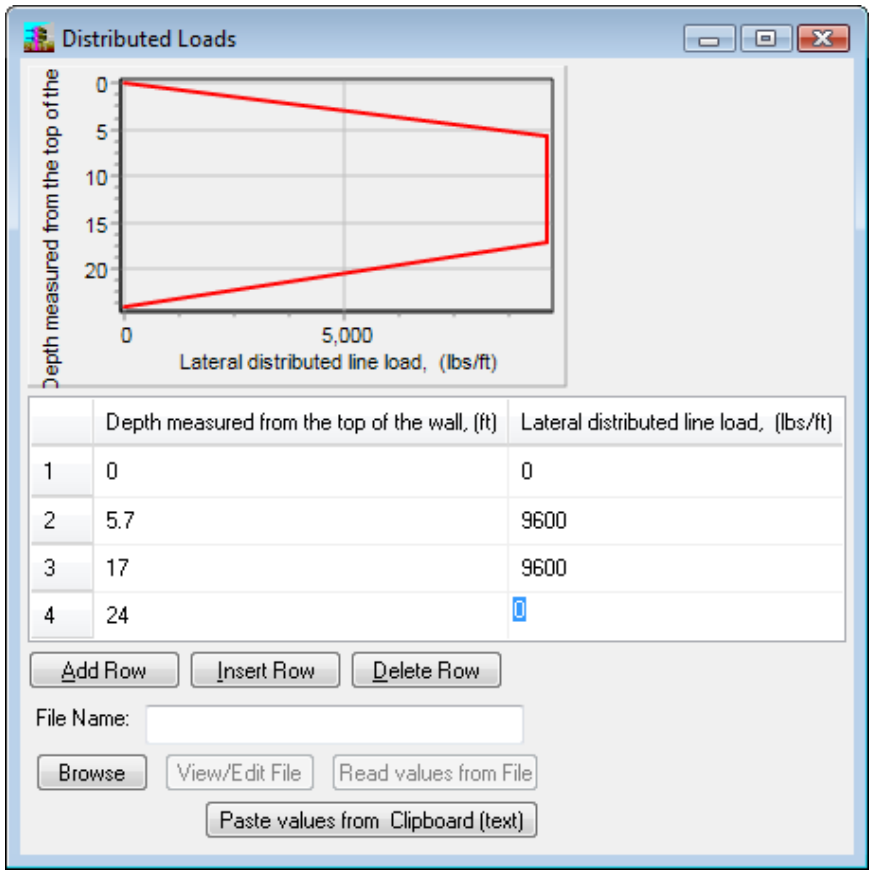


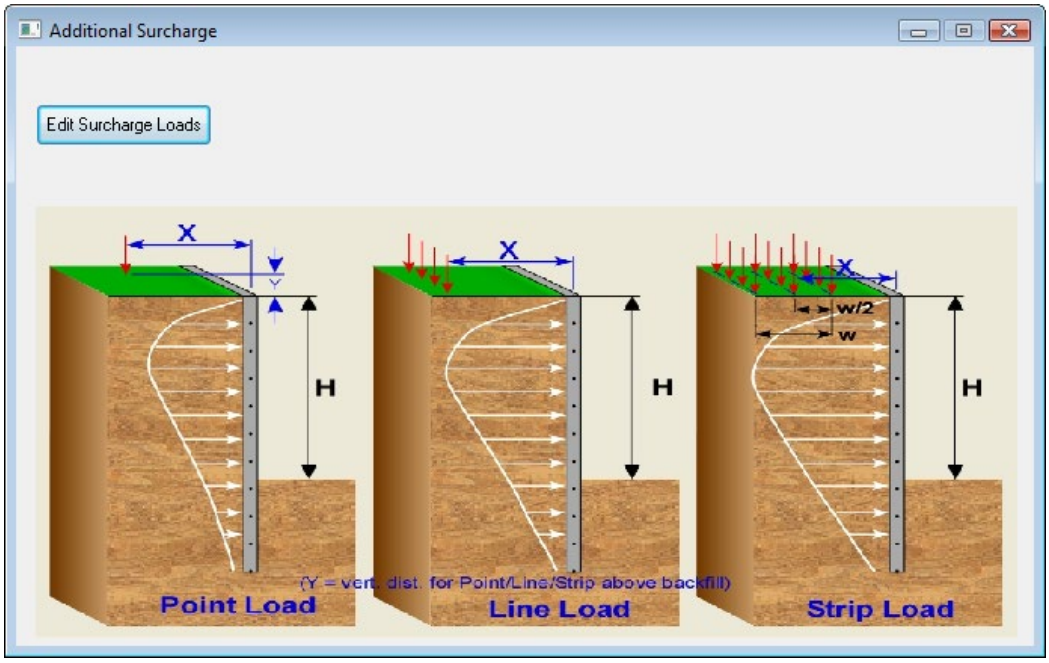
Figure 3.40 Data > Distributed Active Earth Pressure

3.2.11 Additional Surcharge Loads

This dialog box allows the user to enter additional surcharge loads on or nearby the wall. Notice that this submenu item will be enabled if the user placed a check mark on the submenu Options > Apply Additional Surcharge Loads (see Section 3.3.9). The resulting lateral pressures from entries of additional surcharge loads are plotted under "Graphics > Earth Pressure Combined" with the legend "from Additional Surcharge Loads" (see Section 4.3.12).

This feature is not available if the user chooses Options > Earth-Pressure Specified by User (see Section 3.2.10) since that case assumes that all loads are taken into account in the user-specified loading.

The program will allow the user to enter up to 20 different surcharge loads (Figure 3.41), using modified forms of the Boussinesq equations for earth-pressure calculation. The earth pressures generated from these loads are internally added by the program to those earth pressures from the soil. The required inputs for Additional Surcharge Loads are explained further below.



Surcharge Loads

	Load Types	Load Magnitude (SEE NOTES)	Distance, X (in)	Width of Strip Load, W (in)	Distance, Y (in)	Modification Factor for Surcharge Loads
1	Strip Load	2.08	246	228	60	1

Add Row | Insert Row | Delete Row

The unit for point load is lbs or kN
The unit for line load is lbs/in. or kN/m
The unit for strip load is lbs/in^2 or kN/m^2
For strip load, Distance X goes from the wall to the middle of the strip width

Figure 3.41 Data > Additional Surcharge Loads

Load Types

The user can specify the desired surcharge-load type using a drop-down list, which includes: (1) Point Load, (2) Line Load, and (3) Strip Load.

Point Load

For a Point Load selection, the user may enter under the Load Magnitude column a point/concentrated load on the back of the wall as indicated on the graphical reference. The unit is Force (F) and either lbs (when using English units) or kN (when using SI units). The PYWALL program analyzes the critical *Wa* width (defined in Section 3.2.7.6) of wall immediately in front of the specified concentrated load.

Line Load

For a Line Load selection, the user may enter under the Load Magnitude column a line load on the back of the wall as indicated on the graphical reference. The specified line load is assumed to be extended infinitely parallel to the wall. The unit is Force per unit Length (F/L) and either lbs/in (when using English units) or kN/m (when using SI units).

Strip Load

For a Strip Load selection, the user may enter under the Load Magnitude column a strip load on the back of the wall as indicated on the graphical reference. The specified strip load is assumed to be extended infinitely parallel to the wall. The unit is Force per unit Area (F/L^2) and either lbs/in^2 (when using English units) or kN/m^2 (when using SI units).

Distance X (between the Wall and the Load)

This entry is for the distance between the wall and the surcharge loads (use the distance to the middle of the width for a strip load) as indicated on the graphical reference. This input is required for every load type.

Width of the Strip Load, W

This entry is for the width of the strip loads as indicated on the graphical reference. This entry is required only for strip loads (it is not used for the other load types).

Distance Y

This entry is necessary only if the defined load (either Point, Line or Strip) is at a vertical distance Y above (positive number) or below (negative number) the top of the retaining wall, as indicated on the graphical reference (only shown in the reference for point load but applicable to all three loading types). A sample application using this variable is in Figure 3.1 for Example 3 in the Examples Manual. In the majority of models, users will select the default of zero for the Distance Y parameter. This input may be used for every load type.

Modification Factor for Surcharge Loads

Users may specify in this entry a factor that is applicable to all the load entries in the row. This may be useful when users are trying to use LRFD concepts into the PYWALL model. The default value of 1 produces no modifications to the entered loads on the applicable row.

3.2.12 Concentrated Loads on the Wall

The user may enter concentrated loads at any depth of the wall. The program allows the user to enter up to 50 different concentrated loads (Figure 3.42). This entry is enabled only if the user placed a check mark under Options > Apply Concentrated Loads. There are three different external forces/loads that may be specified as concentrated loads at the wall: Lateral Load, Bending Moment and Axial Load.

As a general guideline, the hypothetical example included in the screen capture on Figure 3.42 assumes a concentrated lateral load of 1500 lbs applied at 20 feet from the top of the wall plus a concentrated lateral load of 2000 lbs applied at 25 feet from the top of the wall.

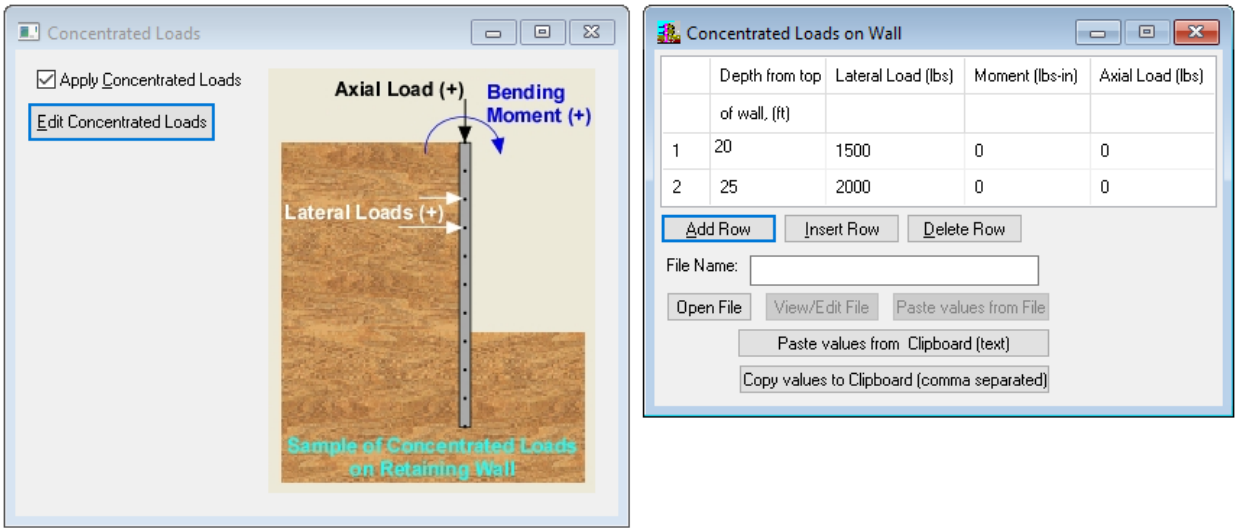


Figure 3.42 Data > Concentrated Load on the Wall

The concentrated loads on the wall represent loads encountered at every *Wa* (see Section 3.2.7.6) distance along the length of the wall. If external forces (such as a tieback at a nodal point) are separated by a distance *Wt* along the length of the wall, the concentrated load in the PYWALL model should be computed as *(Tieback Force) x (Wa/Wt)*. If the concentrated loads on a model occur at every unit increment of wall length (either every foot or every meter) then the input may be computed by *Load x (Wa/Unit Length)*.

Please notice that, the PYWALL program treats the soil response above the excavation level as the active earth pressure (loading) by assuming the wall always moves outwardly from the retained side. The portion of the wall above the excavation level may move inwardly (into the retained soil) due to large prestressing forces from tiebacks (or large negative concentrated loads). In that case, the soil response may change from the active earth pressure as applied load to large soil resistance against the inward movement of the wall. To consider the soil-structure interaction above the excavation level based on the direction of wall movement the user must select Options > Earth-Pressure Options > Generate Earth Pressure Based on Soil Movement. The other earth-pressure options will not consider the change of the soil resistance due to any inward movement of the wall.

Depth from top of wall

This is the distance in units of Length, always positive, from the top of the wall to the point of application of each concentrated load (each row).

Lateral Load

This is selected to specify values of applied lateral load (P, in units of Force) on the wall. The lateral load is considered positive when applied towards the excavation, as shown in the graphical reference.

Bending Moment

This is selected to specify the value of moment couple on the wall (M, in units of Force*Length). The bending moment is considered positive when applied clockwise, as shown in the graphical reference.

Axial Load

This is selected to specify the value of tension or compression loads on the wall (Q, in units of Force). The axial load is considered negative for tension loads and positive for compression loads, as shown in the graphical reference.

3.2.13 Tieback (Ground Anchor) Data

This is the recommended option to enter the tieback or ground anchor on the retaining wall. The program allows the user to enter up to 20 different tiebacks (Figure 3.43). This submenu item is enabled if the user placed a check mark on the submenu Options > Apply Tieback (Ground Anchor). Clicking the "Enter Tieback data" button will bring a table (shown in Figure 3.44) for entering various parameters related to tiebacks.

A check mark for the Consider Zero Bending Moment (AASHTO Hinge Model) instructs the program to follow simplifying numerical recommendations contained in FHWA-IF-99-015, from June 1999 (see List of References). This method allows for simple numerical models to determine approximate tieback forces by assumption of zero bending moments at all tieback locations except on the first (uppermost) level. This method can be useful when users are comparing results from PYWALL with hand computations.

Users may place a checkmark under the Use Wales to evaluate the moment and shear distribution on horizontal elements that may be located between tiebacks. The program assumes that the wales are loaded uniformly between tiebacks (unit load equal to tieback reaction divided by tieback spacing). Users may choose if the wales are Simply Supported or on a Continuous Span. See Section 4.7 in Technical Manual for computational references.

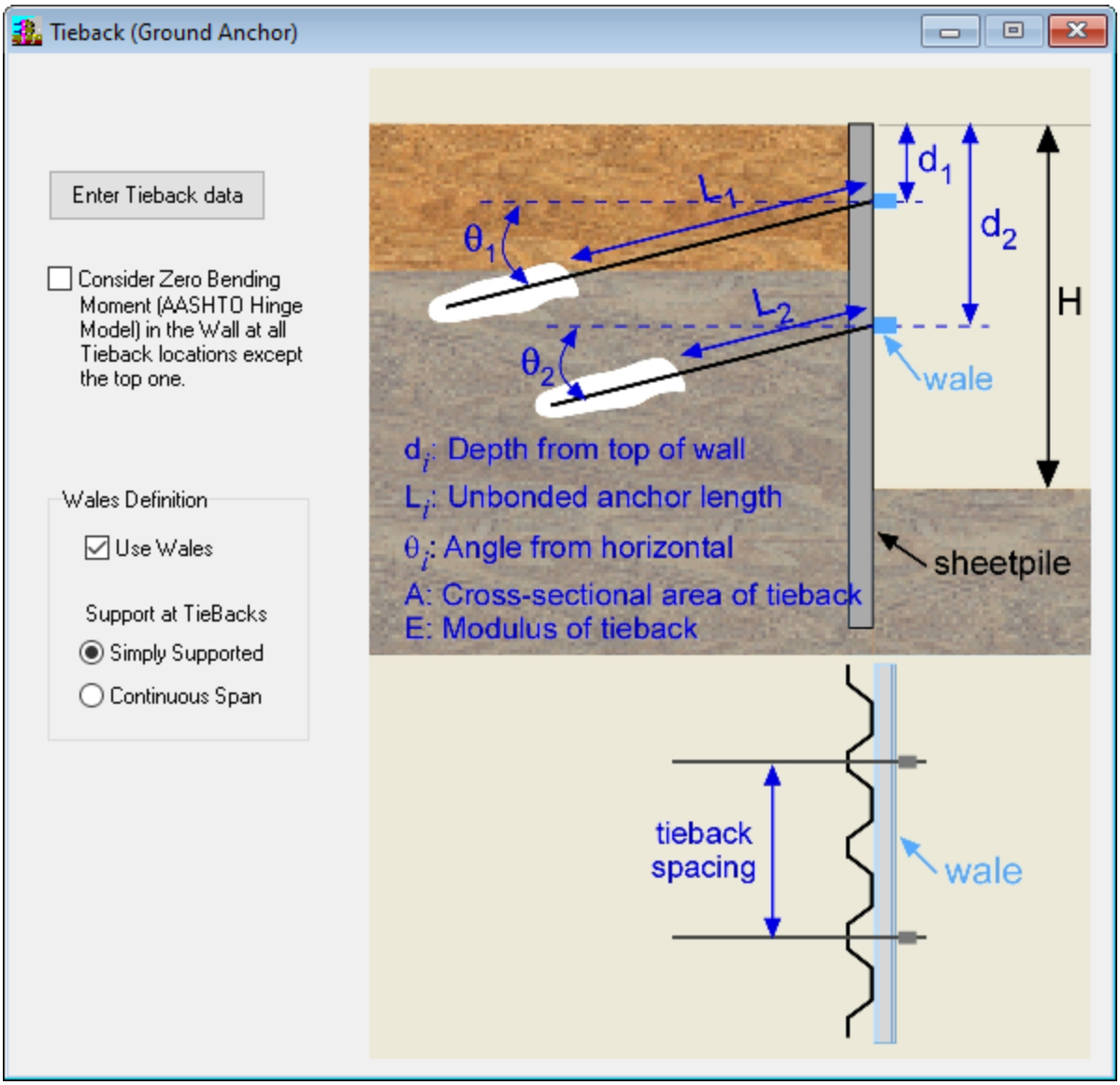
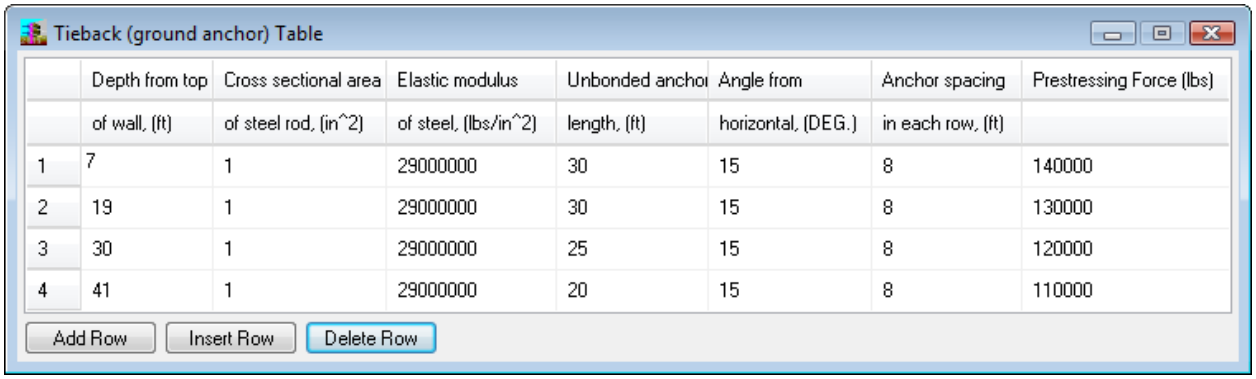


Figure 3.43 Data > Tieback (Ground Anchor) Data



	Depth from top of wall, (ft)	Cross sectional area of steel rod, (in^2)	Elastic modulus of steel, (lbs/in^2)	Unbonded anchor length, (ft)	Angle from horizontal, (DEG.)	Anchor spacing in each row, (ft)	Prestressing Force (lbs)
1	7	1	29000000	30	15	8	140000
2	19	1	29000000	30	15	8	130000
3	30	1	29000000	25	15	8	120000
4	41	1	29000000	20	15	8	110000

Figure 3.44 Sample Data > Tieback > Enter Tieback data

Depth from top of wall

This entry allows users to specify the location (depth) of each row of tiebacks on the wall. The unit is ft (English Units) and meter (S.I. unit) and start with zero at the top of the wall with increases in the downwards direction.

Cross sectional area of steel rod

This entry is for the cross-sectional area of the steel rod of the tieback. The program will use this steel area to compute the axial stiffness of the tieback or anchor element. The unit is in^2 (English Units) and m^2 (S.I. unit).

Elastic modulus of steel

This is selected to specify values for the modulus of elasticity of the tieback or ground anchor. The unit is lb/in^2 (English Units) and kN/m^2 (S.I. unit).

Unbonded anchor length

This is selected to specify the value of unbounded length of the tieback. The program will use this data for calculating the elastic shortening and axial stiffness of the tieback. The unit is ft (English Units) and meter (S.I. unit).

Angle from horizontal

This is selected to specify the angle of the tieback or ground anchor from the horizontal plane (always positive between 0 and 90 degrees). The program will use this data for converting the anchor stiffness (load capacity) from the axial direction of the tieback to the perpendicular direction against the wall that is needed for internal computations in PYWALL. The unit is degrees.

Anchor spacing in each row

This is selected to specify the horizontal spacing between two tiebacks or ground anchor in the same elevation. The unit is ft (English Units) and meter (S.I. unit).

Prestressing Force

The user can enter here any prestressing force (always positive or zero) that is applied on each tieback level during installation. The unit is lbs (English Units) and kN (S.I. unit).

Please notice that the PYWALL program cannot model a temporary state that represents the equilibrium that is reached after initial application of the prestressing force. The PYWALL program model represents a state of the wall after all loads are applied, including the initial prestressing force and those resulting from the retained soil or active loads (and/or others) specified by the user.

The PYWALL program treats the soil response above the excavation level as the active earth pressure (loading) by assuming the wall always moves outwardly (positive deflections). However, a portion of the wall above the excavation level may move inwardly (negative deflections) due to large prestressing forces from tiebacks. In those cases/stages, the soil response changes from the assumed active earth pressure to a different case of applied loads resisting large soil resistance against the inward movement of the wall. For those cases/stages, the estimation of forces and movement will be better made with usage of Options > Earth-Pressure Options > Generate Earth Pressure Based on Soil Movement.

3.2.14 Strut/Raker Data

This is the recommended option to enter the compression elements (struts or rakers) on the retaining wall. The program allows the user to enter up to 20 different struts/rakers (Figure 3.45). This submenu item is enabled if the user placed a check mark on the submenu Options > Apply Strut/Raker. Clicking the "Enter Strut/Raker data" button will bring a table (shown in Figure 3.46) for entering various parameters related to struts/rakers. See Section 4.7 in Technical Manual for computational references.

A check mark for the Consider Zero Bending Moment (AASHTO Hinge Model) instructs the program to follow simplifying numerical recommendations contained in FHWA-IF-99-015, from June 1999 (see List of References). This method allows for simple numerical models to determine approximate strut/raker forces by assumption of zero bending moments at all tieback locations except on the first (uppermost) level. This method can be useful when users are comparing results from PYWALL with hand computations.

Users may place a checkmark under the Use Wales to evaluate the moment and shear distribution on horizontal elements located between struts/rakers. The program assumes that the wales are loaded uniformly between struts/rakers (unit load equal to strut/raker reaction divided by strut/raker spacing). Users may choose if the wales are Simply Supported or on a Continuous Span.

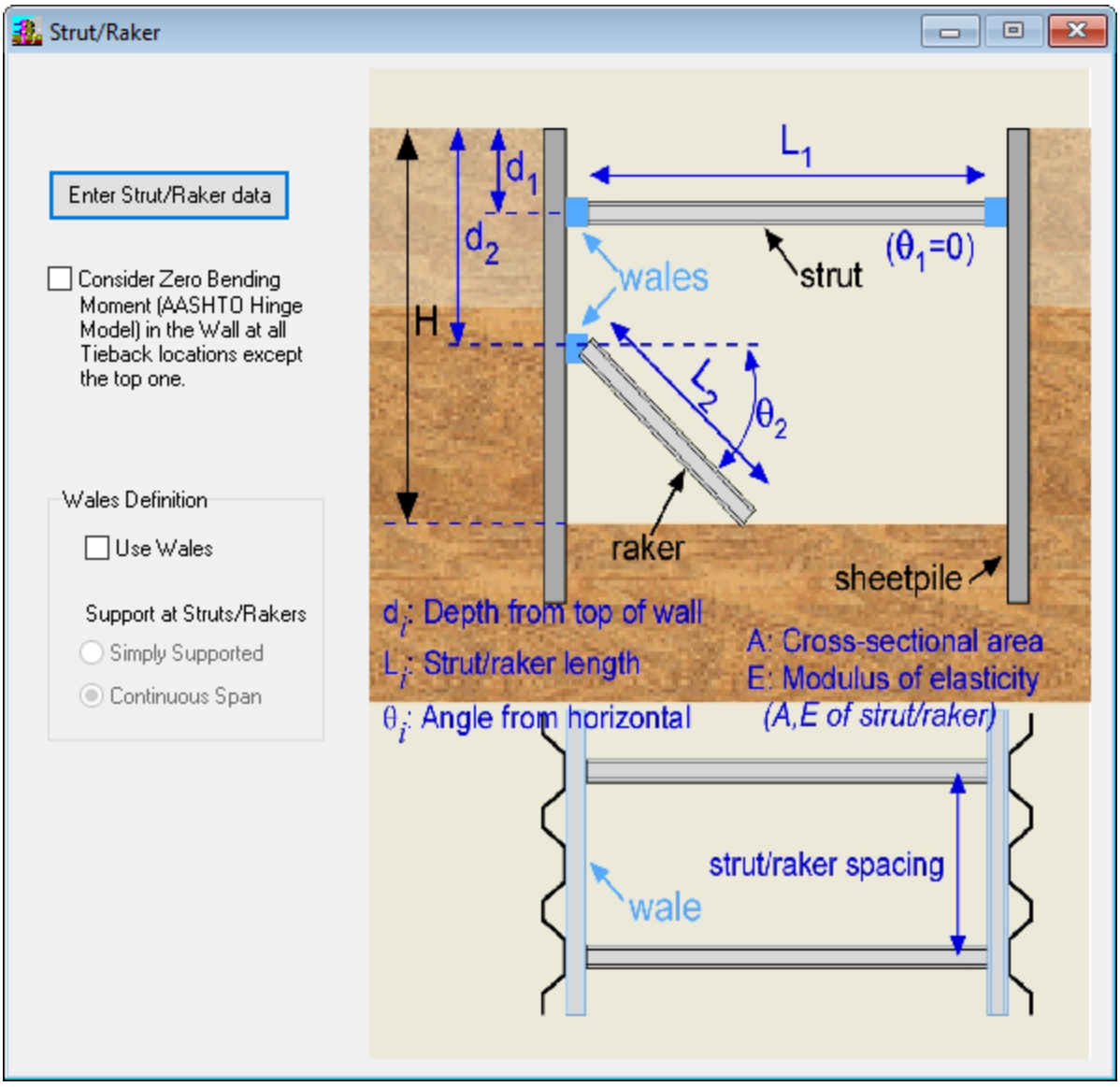
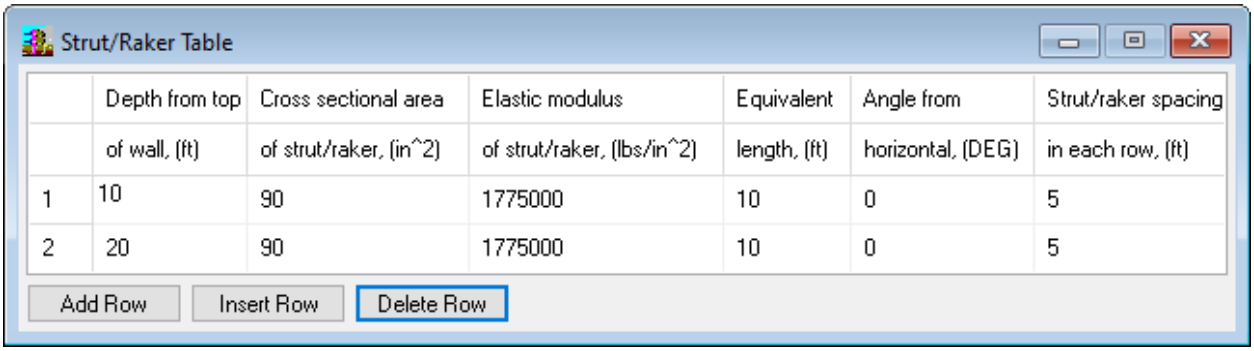


Figure 3.45 Data > Strut/Raker Data



	Depth from top of wall, (ft)	Cross sectional area of strut/raker, (in^2)	Elastic modulus of strut/raker, (lbs/in^2)	Equivalent length, (ft)	Angle from horizontal, (DEG)	Strut/raker spacing in each row, (ft)
1	10	90	1775000	10	0	5
2	20	90	1775000	10	0	5

Figure 3.46 Data > Strut/Raker Data > Enter Strut/Raker data

Depth from top of wall

This entry allows users to specify the location (depth) of each row of tiebacks on the wall. The unit is ft (English Units) and meter (S.I. unit) and start with zero at the top of the wall with increases in the downwards direction.

Cross sectional area of strut

This entry is for the cross-sectional area of the strut or raker. The program will use this steel area to compute the axial stiffness of the compression element. The unit is in^2 (English Units) and m^2 (S.I. unit).

Elastic modulus of steel

This is selected to specify values for the modulus of elasticity of the strut/raker. The unit is lb/in^2 (English Units) and kN/m^2 (S.I. unit).

Equivalent length

This is the length of the strut/raker (L_1 or L_2 in Figure 3.45). The program uses this data for calculating the elastic shortening and axial stiffness of the strut/raker. The unit is ft (English Units) and meter (S.I. unit).

Angle from horizontal

This is selected to specify the angle of the raker from the horizontal plane (θ_2 in Figure 3.45). This value is always positive and between 0 and 90 degrees. The program will use this data for converting the anchor stiffness (load capacity) from the axial direction of the raker to the perpendicular direction against the wall (which is needed for internal computations in PYWALL). The unit is degrees.

Strut/Raker spacing in each row

This is selected to specify the horizontal spacing between two struts or rakers in the same elevation. The unit is ft (English Units) and meter (S.I. unit).

3.2.15 Boundary Condition on Deflection or Slope

The user may enter a boundary condition of specified deflection or slope at any depth of the wall. The program allows the user to enter up to 20 different boundary conditions. This dialog box is enabled only if the user placed a check mark on the submenu Options > Apply Restrains on Deflection or Slope. The required inputs for Boundary Condition on Deflection or Slope are explained further below.

A sample window screen for the Data > Boundary Conditions on Deflection or Slope menu is included in Figure 3.47. As a general guideline, the hypothetical example included in the screen capture on Figure 3.47 assumes that a deflection (lateral movement) of 0.1 inches (in the direction towards the excavation) is imposed at the top of the wall, and a deflection of 0.5 inches (towards excavation) along with a slope of 0.0001 radians (clockwise direction) are imposed at a distance of 10 feet from top of wall.

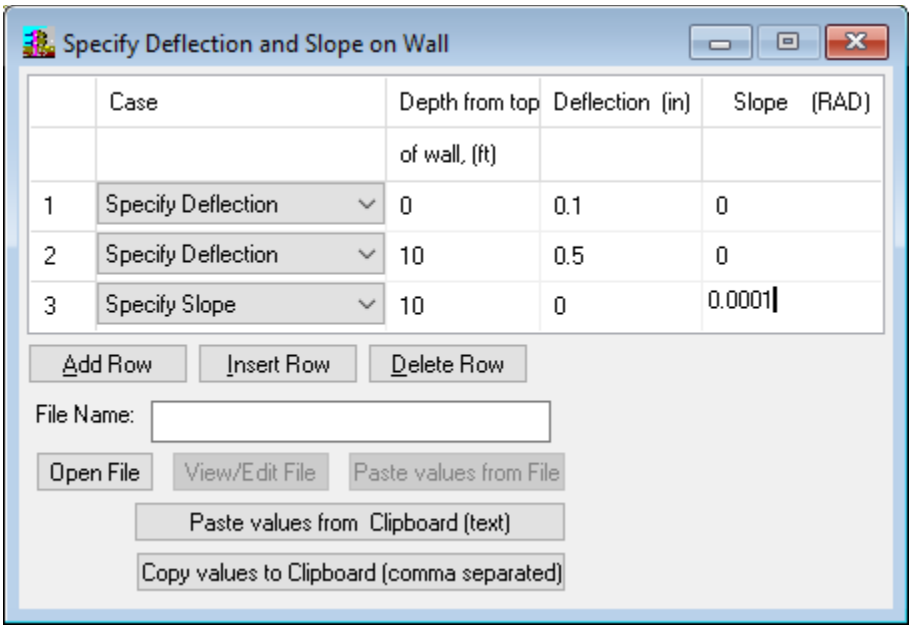


Figure 3.47 Data > Boundary Conditions on Deflection or Slope

Case

The user can specify either deflection or slope for the boundary condition.

Depth from top of wall

This is the distance in units of Length, always positive, from the top of the wall to the point of application of each concentrated load (each row).

Deflection

This entry allows users to specify the value for any imposed deflection (lateral movement) in units of Length (either inches or mm) at any depth on the wall. Positive values are in the direction towards the excavation (to the right).

Slope (radians)

This entry allows users to specify the value for any imposed slope in units of radians at any depth on the wall. Positive values are assumed to be in the clockwise direction.

3.2.16 Lateral Spring and Rotational Restraints

The user may enter different types of lateral springs and rotational restraints (constant values) at any depth of the wall. This option may be used for modeling tiebacks or struts. The Data > Lateral Springs and Rotational Restraints is included in Figure 3.48.

This dialog is enabled if the user places a check mark under Options > Apply Lateral Springs and Rotational Restraints. The restraints must be inputted in ascending order of depths from top of wall.

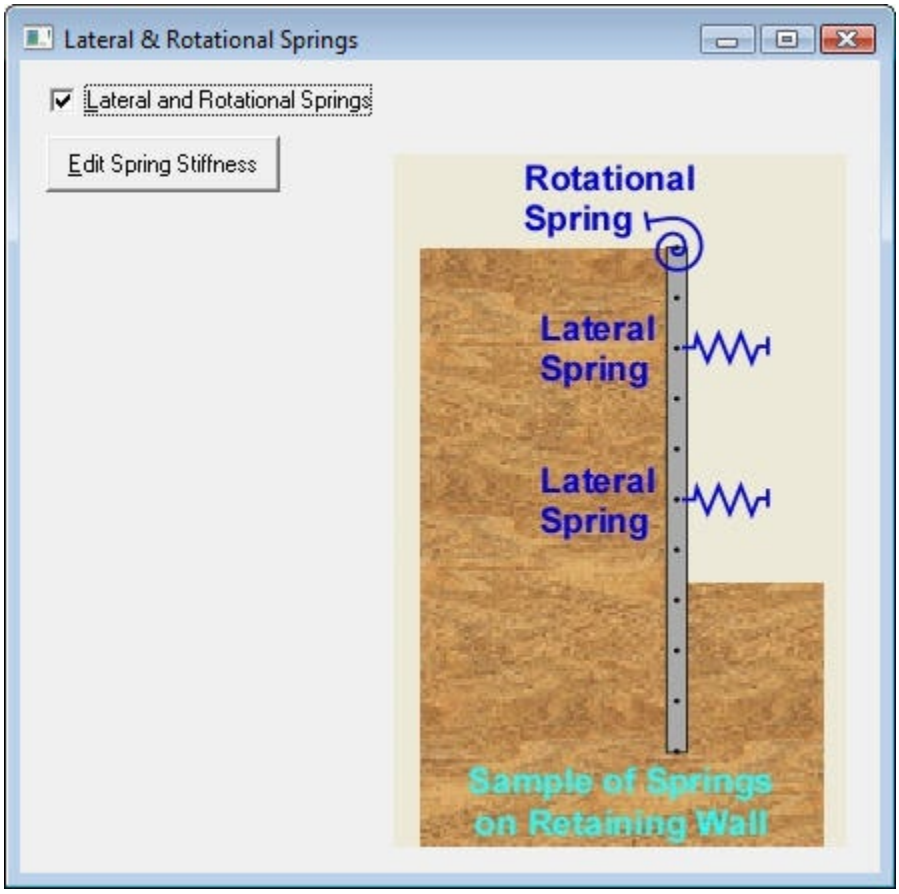


Figure 3.48 Data > Lateral Springs and Rotational Restraints

The Edit Spring Stiffness button opens a window labeled Lateral Spring and Rotation Spring, shown in Figure 3.49. As a general guideline, the hypothetical example included in the screen capture on Figure 3.49 assumes that two lateral springs of constant stiffness are applied at 5 and 10 feet from top of wall.

The required inputs for each Lateral Spring and Rotation Spring are explained further below.

Depth from top of wall

This is the distance in units of Length, always positive, from the top of the wall to the point of application of each lateral and/or rotational spring (each row).

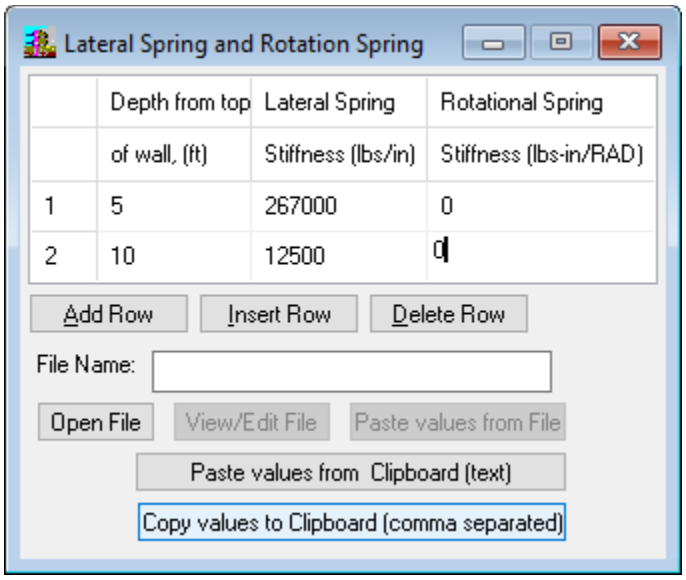


Figure 3.49 Sample screen for Edit Spring Stiffness

Lateral Spring Stiffness

This entry allows users to specify the value for a constant lateral spring stiffness at the selected distance from top of wall. The units are lbs/in for English units and kN/m for S.I. units.

The user may enter here the stiffness of a tieback or a bracing strut as a spring constant. However, it should be noted that the resistance of a tieback may not be well represented as a linear-elastic spring model. For an improved representation, the user can enter elastic-plastic resistance curves using the Options > Apply User-Specified p-y Curves (External) and entering the curves under the Data > Sets of External p-y Curves menu. The most recommended method for tiebacks, however, is to select Options > Apply Tieback (Ground Anchor) and then enter tieback information under Data > Tieback (Ground Anchor).

Rotational Spring Stiffness

This entry allows users to specify the value for a constant rotational spring stiffness at the selected distance from top of wall. The units are in lbs-in/rad for English units and kN-m/rad for S.I. units.

3.2.17 Modification Factors for p-y Curves

This window allows the user to enter modification factors for soil resistance (p) and/or lateral movement of the wall unit (y) at specified depths. This window is enabled only if the user placed a check mark on the submenu Options > Apply External p-y Modification Factors.

The user is provided with a couple of options under the Data > Modification Factors for p-y Curves shown in Figure 3.50.

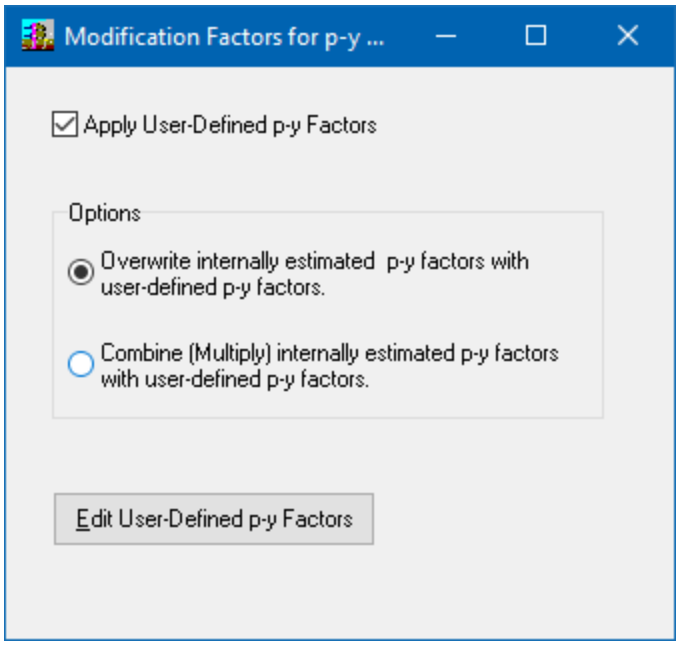


Figure 3.50 Data > Modification Factors for p-y Curves

By default (no check mark under Options > Apply External p-y Modification Factors or under Apply User-Defined p-y Factors in Figure 3.50), the PYWALL program will calculate internally and apply *p*-multipliers as necessary to account for reductions due to closely-spaced wall units (this reduction is also known as reduction for "group effects" and explained in Section 3.6 of the Technical Manual). The internal computations of those reduction factors are based on the Wa and Wp values defined by the user in the Data > Wall Properties window screen.

If the user chooses to Overwrite internally estimated p-y Factors with user-defined p-y factors in Figure 3.50 then the factors entered by the user in the Edit User-Defined p-y Factors button must account for any "group effects" mentioned earlier plus for any additional reduction (or increases) purposes. Such combination of factors can be done by multiplying all applicable modification factors.

If the user chooses to Combine (Multiply) internally estimated p-y Factors with user-defined p-y factors in Figure 3.50 then the factors entered by the user in the Edit User-Defined p-y Factors button will be multiplied to any "group effects" mentioned earlier that may be calculated internally by the PYWALL program.

The *p*-multipliers that are used by the program (either calculated internally or inputted externally by the user) are printed in the output text file for verification purposes under the section labeled `P-Y CURVES DATA` at various soil depths and for excavation and backfill sides.

A maximum of 30 entries of modification factors for *p-y* curves may be used in an analysis. The program allows the input of modification factors for any depths of the soil profile. The *p-y* modification factors only apply to *p-y* curves that are internally generated by the program. If the user requests the printout of internally-generated *p-y* curves, outputted curves (in text and graphics) will already include the changes produced by the specified *p-y* modification factors.

A sample window for the Data > Modification Factors for p-y Curves menu is included in Figure 3.51. As a general guideline, the hypothetical example included in the screen capture on Figure 3.51

assumes that a *p*-multiplier of 0.95 is applied from the top of the wall to a depth of 42 feet while a *y*-multiplier of 1.0 is applied at the top of the wall and this multiplier increases linearly to 1.5 at the depth of 42 feet.

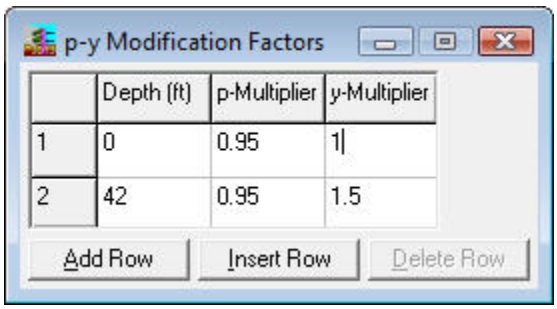


Figure 3.51 Sample Modification Factors for p-y Curves

Depth

These values represent the depths (starting from top of wall) where modification factor for *p-y* curves are being specified. Intermediate values of *p-y* modification factors located between two defined depths are obtained by linear interpolation of the specified factors. It is therefore necessary to have at least two entries of modification factors. The rows of *p-y* modification factors must be inputted in ascending order of depths.

p-Multiplier

The *p*-multiplier values may be larger or smaller than one. However, in most cases these values are smaller than one to account for group effect of closely-spaced piles/drilled shafts or for strength reduction factors for LRFD design requirement (if such is required by the applicable code). The program will calculate and apply the group reduction for the wall internally as a default in this version. If the user accepts the internally-generated group reduction factors, the *p*-multipliers can be assigned for other purposes such as the strength reduction factors based on LRFD design requirement. If the user would like to apply the group reduction factor externally, the user has the option of entering the data for *p*-multipliers directly. Furthermore, If the user would like to consider the group reduction factors as well as strength reduction factors based on LRFD design requirement, the user need to combine those two reduction factors together (multiplying the factors) and then enter the data.

A large reduction in *p*-values (and/or increase of *y*-values) may also be used to ignore lateral transfers in any soil layer below the excavation (e.g., for liquefiable layers of sand). Do not leave the *p*-multiplier as zero if the *y*-multiplier is specified, or vice versa, do not leave the *y*-multiplier as zero if the *p*-multiplier is specified.

y-Multiplier

The *y*-multiplier values may be larger or smaller than one. However, in most cases these values are larger than one to account for long-term deformation in clay. A large increase in *y*-values (and/or reduction of *p*-values) may also be used to ignore lateral transfers in liquefiable layers of sand (or any soil layer where lack of lateral load transfer is desired). Do not leave the *y*-multiplier as zero if the *p*-multiplier is specified, or vice versa, do not leave the *p*-multiplier as zero if the *y*-multiplier is specified.

3.2.18 Sets of External p-y Curves

The user may enter specific relationships of soil resistance (p) and lateral movement of the pile (y) at user-specified depths. This window is inactive under default conditions, because the program defaults to an automatic internal generation of soil-resistance (p-y) curves. However, sometimes the user may prefer to enter different external p-y curves. These cases usually arise when local data for the soil response are available from instrumented tests or when the user wants to use this option to model nonlinear behavior of tiebacks or struts with springs.

This window is enabled if the user places a check mark on the submenu Options > Apply User-Specified p-y Curves (External). External p-y curves must be inputted in ascending order of depths. A sample window for the Data > Sets of External p-y Curves menu is included in Figure 3.52. The required inputs for Sets of External p-y Curves are explained further below.

Depth from top of wall

This represents the distance from top of wall where each external p-y curve is being specified. Entries (rows) must be inputted sequentially in ascending depths from top of wall.

For depths located within the excavation height, the values of the user-specified p-y curves are used as nonlinear springs placed at each specified location. This feature may be used by users to simulate the nonlinear deformation of a tieback or strut on anchored walls.

For depths below the excavation level, intermediate values of the external p-y curves located between two specified nodes (depths) are obtained by linear interpolation of the specified curves. It is therefore necessary to have at least two nodal entries of external p-y curves when those are below the excavation level.

Curve Sets

This is a sequential number and name provided to independent buttons that are clicked to access the input data for each set of external p-y curves.

Multiplier for p-value and Multiplier for y-value

These are multipliers that can be applied to user-inputted p and y values. These multipliers allow users to easily check influence from multiplying factors for usage during parametric studies or studies of sensitivity.

The required inputs for each External p-y Curve are explained further below.

Point

This is a sequential number that is provided to each point of the soil-resistance curve that is being defined. This number is automatically provided by the program as new rows are added or inserted. The maximum number of points in each soil-resistance (p-y) curve is limited to 17 and the minimum limited to 2.

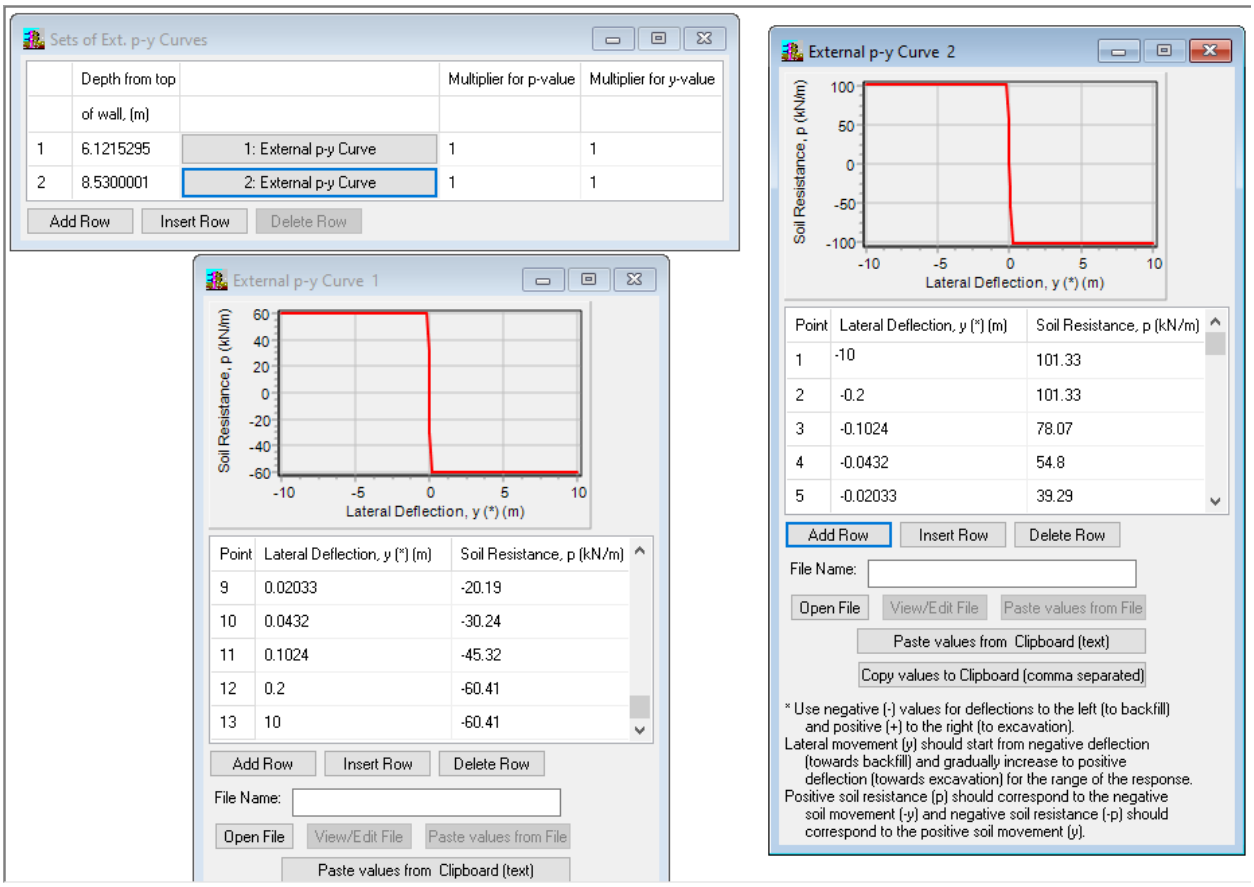


Figure 3.52 Data > Sets of External p-y Curves

Lateral Deflection, y

Values of lateral movement must be inputted in units of length, starting from negative deflections and gradually increasing to positive deflections. As explained in the screen notes, negative values of deflection (-y) are entered for movements towards the backfill (towards the left) and positive values of deflection (+y) are entered for movements towards the excavation (towards the right).

Soil Resistance, p

Values of load transfer corresponding to the related deflection must be inputted in units of load per increment length. A positive soil resistance (+p) corresponds to a negative deflection (-y) and a negative soil resistance (-p) corresponds to a positive deflection (+y).

3.3 Options Menu

The menu contains parameters that set preferences for each analytical run or problem description. The user can pick up the particular item of interest simply by clicking the optional items listed in Figure 3.53. Once a check mark is placed on the preferred option item, the associated entry fields for input data will be activated under Data menu accordingly.

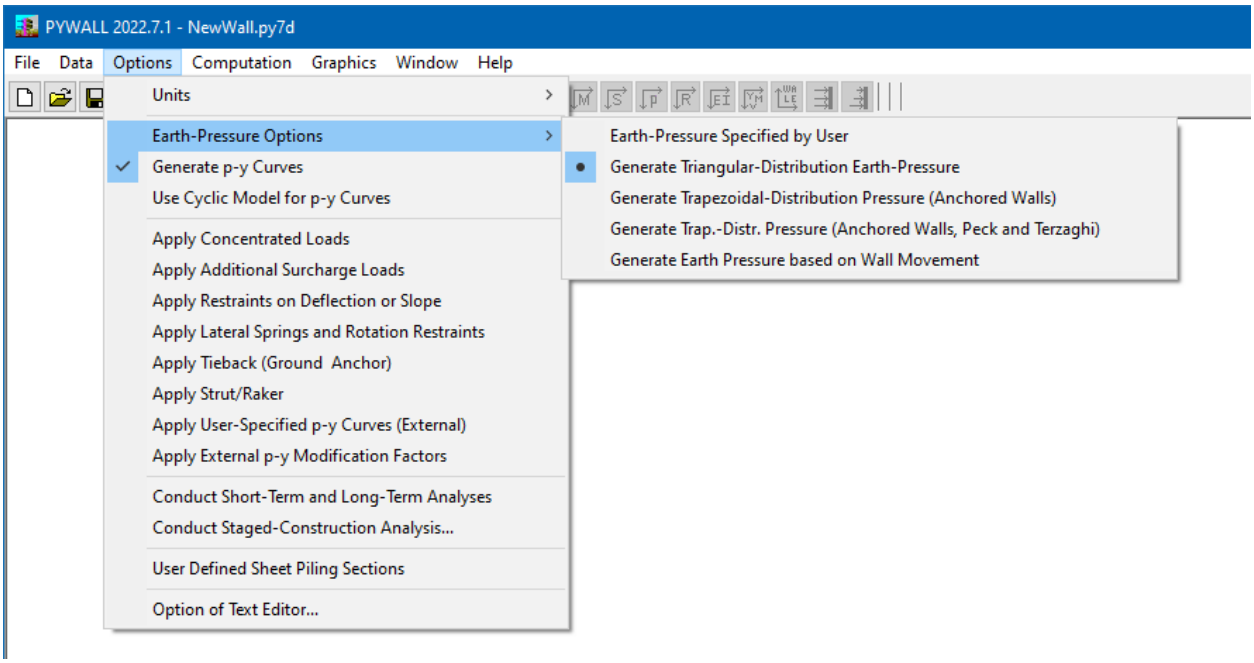


Figure 3.53 Sample Options Menu

3.3.1 Units

This submenu option provides the user with a choice of specifying input data in English Units or S.I. Units. For English Units, the program uses lbs, ft, inch, psf, psi, and pcf dependent on the common practices. For example, the program will use lb/ft^3 for unit weight, but lb/in^2 for Young's modulus. Please refer to the unit caption shown at each entry field when English units are selected. For S.I. Units, the program consistently uses kN for force and meter for length.

The user may change from one system of units to the other as many times as desired. The program will automatically convert all the appropriate input that was already specified by the user before the time of selection of a different system of units. Small variations of the inputs may be noticed due to significant figures used in the conversions if multiple changes of units are performed. The user must always check that the input data has been converted appropriately.

3.3.2 Earth-Pressure Options

This submenu option provides the user with a choice of internal computations or input of active earth pressures.

Earth-Pressure Specified by User

When this entry is selected, the submenu Data > Distributed Active Earth Pressure will be enabled for the user to enter manually the earth pressure as a function of depth (must be calculated by the user externally from the program). Read more information in Section 3.2.10.

Generate Triangular-Distribution Earth-Pressure

With this selection, the program will generate active earth pressures using Rankine's equations for triangular distribution. This selection will enable the entries for Wall Configuration and Soil Layer Data under the Data menu and disable the entry for Data > Distributed Active Earth Pressure.

Generate Trapezoidal-Distribution Pressure (Anchored Walls, FHWA)

This selection instructs the program to generate active earth pressures based on the trapezoidal distribution recommended by FHWA for temporary braced and anchored excavations. The method that is used by the program internally is described in Section 2.7.2 of the PYWALL Technical Manual. The program internally chooses the distribution profile and the applicable equation depending on the soil type that is inputted by the user on the retained side (see notes in Technical Manual). Any Optional Earth-Pressure Coefficient (Section 3.2.8) that is inputted by the user on any retained soil layer will be ignored when using this method.

This selection will enable the entries for Wall Configuration and Soil Layer Data under the Data menu and disable the entry for Data > Distributed Active Earth Pressure.

Generate Trap.-Distr. Pressure (Anchored Walls, Terzaghi and Peck)

This selection instructs the program to generate active earth pressures based on the trapezoidal-distribution recommended by Terzaghi and Peck for mixed or uniform layers of sand or clay in temporary braced and anchored excavations. The method that is used by the program internally is described in Section 2.7.1 of the PYWALL Technical Manual. The program internally chooses the distribution profile and the applicable equation depending on the soil type that is inputted by the user on the retained side (see notes in Technical Manual). Any Optional Earth-Pressure Coefficient (Section 3.2.8) that is inputted by the user on any retained soil layer will be ignored when using this method.

This selection will enable the entries for Wall Configuration and Soil Layer Data under the Data menu and disable the entry for Data > Distributed Active Earth Pressure.

Generate Earth Pressure Based on Wall Movement

In this case, the program will generate earth pressures based on the wall movement as described in Section 3.9 of the PYWALL Technical Manual (and also in Example 12). The earth pressure above the excavation level may not be in the active state if the wall is pushed back into the backfill by prestressing forces in tieback anchor(s). The earth pressure will approach the 'at rest' state if the wall does not post any lateral movement. When the wall starts to deflect into the backfill soil due to prestressing forces on tieback anchors, resistance may increase toward the maximum passive earth pressure. The earth pressure in the active state, at rest, and passive state is determined by the movement of the wall.

This selection will enable the entries for Wall Configuration and Soil Layer Data under the Data menu and disable the entry for Data > Distributed Active Earth Pressure.

3.3.3 Generate p-y Curves

This optional parameter is checked by default. With this option the program will automatically generate soil resistance (*p*-*y*) curves based on the recommended criteria for soil types and layers selected by the user.

If the user takes out this check mark (does not ask the PYWALL program to generate internal p-y curves) then the dialog screens for Wall Configuration and Soil Layer Data under the Data menu are both turned off. Also, the user must enter the active earth pressures (Data > Distributed Active Earth Pressure) and user-specified *p*-*y* curves (Data > Sets of External p-y Curves).

3.3.4 Use Cyclic Model for p-y Curves

If this option is selected, the PYWALL program will be instructed to use cyclic *p-y* curves where applicable. The menu Data > Cyclic Model Data is enabled when a check mark is placed on this option. Read more information in Section 3.2.9.

Notice that only some soils have a cyclic *p-y* criteria and only a few of those are influenced by the number of cycles. Additional field tests under cyclic loading are needed in order to improve the understanding on soil degradation for most soil types.

3.3.5 Options for Continuous Walls

This selection provides users with controls related to the response of continuous walls (either secant or tangent walls). However, this menu selection is enabled for all models, since soldier-pile walls that are spaced very closely may behave like continuous walls and the user may decide to evaluate both conditions. The user's engineering judgment will be needed, since the response as soldier pile or as continuous wall may depend on the pile spacing as well as the 'arching' capabilities of the retained soils.

The dialog screen for Options > Options for Continuous Walls… is shown in Figure 3.57. The different inputs are defined below.

3.3.5.1 Reference Diameter

The entry for Reference Diameter for Continuous Wall Model represents the diameter (in units of Length, inch or meter) that is used to compute the lateral deformation ('*y*' values) for the '*p-y*' curves below excavation level. This is applicable to all the *p-y* soil criteria that use diameter to compute the '*y*' responses, such as clay, sand and silt). Notice that the value of Wp (Section 3.2.7.5) entered by the user is used to generate the '*p*' component of the *p-y* curves below excavation level.

A value of 12 inches (0.3048 m) is set as default for reference diameter in the initial release of PYWALL v2025 since it is the value that was used internally in previous releases of PYWALL. However, the most appropriate value for reference diameter may arguably need to be different. In the future, a careful evaluation of all load tests used for the various sand and clay criteria may be needed to determine a more applicable averaged value for reference diameter. Also, further research is needed in this area along with comparisons with load tests and/or measurements from actual continuous walls. The use of a larger 18-in or even 24-in reference diameter may provide more conservative results of deflection and bending moments for most models.

3.3.5.2 Transition Between Continuous Wall Model and Group-Pile Model

With a checkmark on this optional input, the user can control the extent of the transition zone between continuous wall and soldier-pile behaviors. Group-Pile Model is a generalized soldier pile model where the wall response is controlled by the single pile subjected to the reductions from size (diameter) and spacing of neighboring piles (known as 'group-pile' multiplier).

This option was not available in PYWALL releases prior to v2025, where there was no 'transition zone' but rather a jump right at 1.1 S/D (where S is for center-to-center pile spacing and D is for pile diameter). For compatibility with previous releases and models, the Use Transition Between Continuous Wall Model and Group-Pile Model is unchecked and defaulted to 1.1 for Start and End.

As a default for new models, this option is checked and suggests to use a smoother nonlinear transition zone between 1.1 S/D (Start of Transition Zone) and 1.5 S/D (End of Transition Zone).

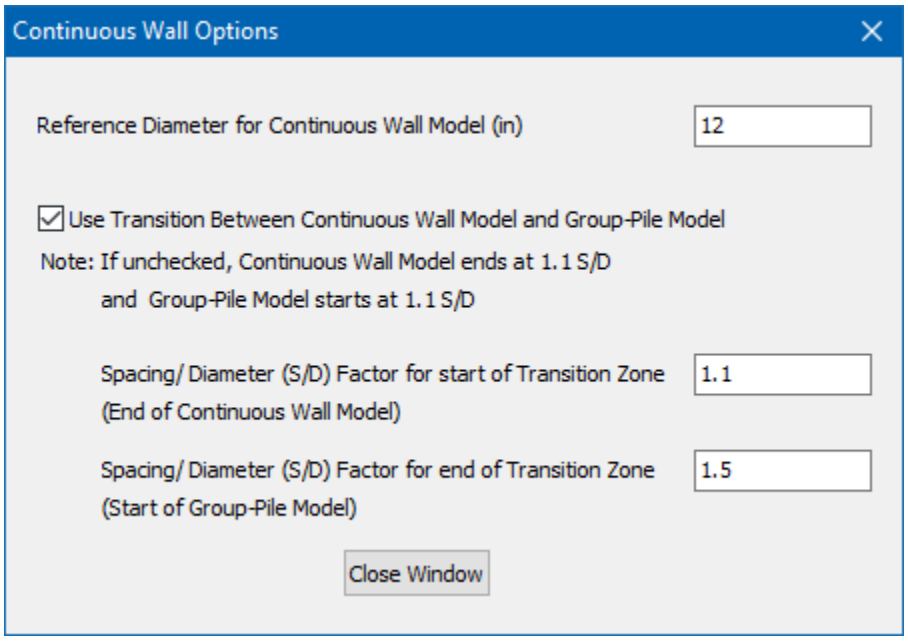


Figure 3.54 Dialog screen for 'Options for Continuous Walls …'

3.3.6 Options for Anchored Walls (FHWA)…

This entry is enabled only if the user selected Options > Earth-Pressure Options > Generate Trapezoidal-Distribution Pressure (Anchored Walls, FHWA) earlier introduced in Section 3.3.2. The resulting dialog box, shown in Figure 3.55, provides the user with a few controls regarding the computation of active-earth pressures based on the trapezoidal distribution recommended by FHWA for temporary braced and anchored excavations. This method is described in Section 2.7.2 of the PYWALL Technical Manual.

3.3.6.1 Depth to Potential Failure Surface for Soft to Medium Clays

This entry represents the value for variable 'D' in the referenced graphics (in units of Length, feet or meter). By default, PYWALL uses a value of zero but it can be changed based on the user's judgement. This is only applicable to models where the entire retained height is formed by a single or multiple layers of soft clay (no sand, no stiff clay).

3.3.6.2 Maximum Pressure Coefficient for Stiff to Hard Clays

This entry represents the factor 'C_m' in the referenced graphics. By default, PYWALL uses a value of 0.3 but it can be changed based on the user's judgement. This is only applicable to models where the entire retained height is formed by a single or multiple layers of clay (no sand, at least one layer of stiff clay).

3.3.6.3 Uniform Pressure

The user may choose one of two methods to incorporate the active earth pressure generated by the uniform surcharge pressure specified as Uniform Distributed Load at the top of backfill side under Data > Wall

Properties plus any uniform lateral load acting along the wall that was specified as Uniform Lateral Distributed Load along Wall (multiplied by Wa) under Data > Wall Properties.

The default selection of Convert Uniform Pressure into Equivalent Trapezoidal computes any uniform pressure and converts it to a trapezoidal configuration.

The selection of Add Uniform Pressure to Equivalent Trapezoidal simply adds any uniform pressure to the trapezoidal configuration.

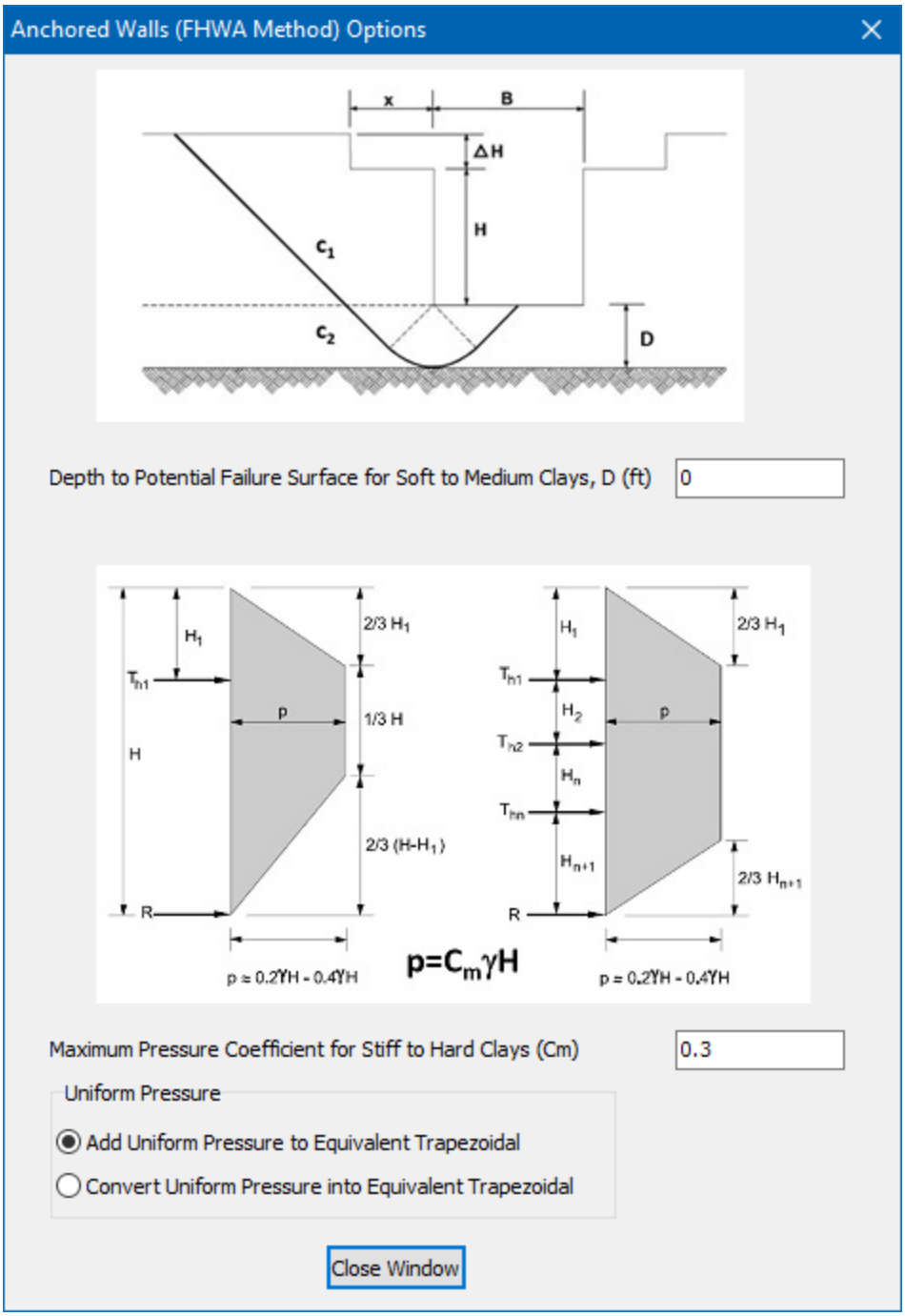


Figure 3.55 Sample dialog screen for 'Options for Anchored Walls (FHWA) …'

3.3.7 Options for Anchored Walls (Terzaghi and Peck)…

This entry is enabled only if the user selected Options > Earth-Pressure Options > Generate Trap.-Distr. Pressure (Anchored Walls, Terzaghi and Peck) earlier introduced in Section 3.3.2. The resulting dialog box, shown in Figure 3.56, provides the user with a few controls regarding the computation of active-earth pressures based on the trapezoidal distribution recommended by Terzaghi and Peck for temporary braced and anchored excavations. This method is described in Section 2.7.1 of the PYWALL Technical Manual.

3.3.7.1 Depth to Potential Failure Surface for Soft to Medium Clays

This entry represents the value for variable 'D' in the referenced graphics (in units of Length, feet or meter). By default, PYWALL uses a value of zero but it can be changed based on the user's judgement. This is only applicable to models where the entire retained height is formed by a single or multiple layers of soft clay (no sand, no stiff clay).

3.3.7.2 Maximum Pressure Coefficient for Stiff to Hard Clays

This entry represents the factor 'C_m' in the referenced graphics. By default, PYWALL uses a value of 0.3 but it can be changed based on the user's judgement. This is only applicable to models where the entire retained height is formed by a single or multiple layers of clay (no sand, at least one layer of stiff clay).

3.3.7.3 Uniform Pressure

The user may choose one of two methods to incorporate the active earth pressure generated by the uniform surcharge pressure specified as Uniform Distributed Load at the top of backfill side under Data > Wall Properties plus any uniform lateral load acting along the wall that was specified as Uniform Lateral Distributed Load along Wall (multiplied by Wa) under Data > Wall Properties.

The default selection of Convert Uniform Pressure into Equivalent Trapezoidal computes any uniform pressure and converts it to a trapezoidal configuration.

The selection of Add Uniform Pressure to Equivalent Trapezoidal simply adds any uniform pressure to the trapezoidal configuration.

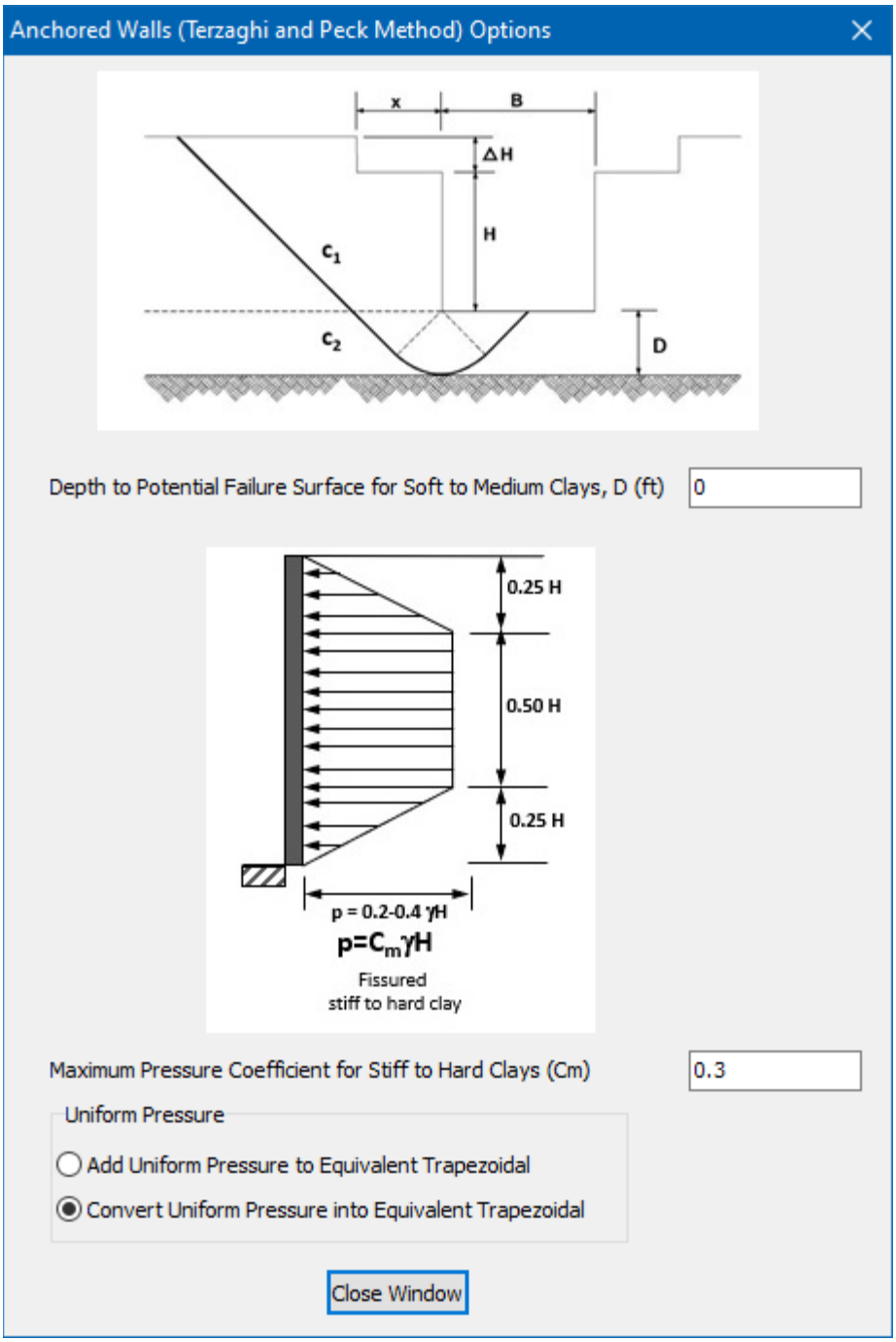


Figure 3.56 Sample dialog screen for 'Options for Anchored Walls (Terzaghi and Peck) …'

3.3.8 Apply Concentrated Loads

If this option is selected, the user can enter up 20 sets of lateral loads, bending moment, and axial loads on nodal points of the wall. The menu Data > Concentrated Loads on the Wall is enabled when a check mark is placed on this option. Read more information in Section 3.2.12.

3.3.9 Apply Additional Surcharge Loads

If this option is selected, the program will allow the user to enter up to 20 different surcharge loads, using modified forms of the Boussinesq equations for earth-pressure calculation. The user can enter up to 20 different sets of surcharge loads and the program will add the earth pressure generated from each surcharge load. The menu Data > Additional Surcharge Loads is enabled when a check mark is placed on this option. Read more information in Section 3.2.11.

3.3.10 Apply Restraints on Deflection or Slope

If this option is selected, the user can enter up 20 sets of restraints on deflection or slope at nodal points of the wall. The menu Data > Boundary Condition on Deflection or Slope is enabled when a check mark is placed on this option. Read more information in Section 3.2.15.

3.3.11 Apply Lateral Springs and Rotational Restraints

If this option is selected, the user will apply one or more specific spring constants or lateral springs and rotational springs to simulate the tiebacks or struts on the nodal points. The user can enter up to 20 different sets of data. The menu Data > Lateral Springs and Rotational Restraints is enabled when a check mark is placed on this option. Read more information in Section 3.2.16.

3.3.12 Apply Tieback (Ground Anchor)

If this option is selected, the user can enter up 20 sets of tiebacks (ground anchors) at any location of the wall. The menu Data > Tieback (Ground Anchor)Data is enabled when a check mark is placed on this option. Read more information in Section 3.2.13.

3.3.13 Apply Strut/Raker

If this option is selected, the user can enter up 20 sets of tiebacks (ground anchors) at any location of the wall. The menu Data > Strut/Raker Data is enabled when a check mark is placed on this option. Read more information in Section 3.2.14.

3.3.14 Apply User-Specified p-y Curves (External)

If this option is selected, the user will specify arbitrary soil response (*p-y*) curves externally from the program (these *p-y* curves are calculated by the user using other methods and entered into the program). The menu Data > Sets of External p-y Curves is enabled when a check mark is placed on this option. Read more information in Section 3.2.18.

3.3.15 Apply External p-y Modification Factors

This is selected if the user wants to enter modification factors for soil resistance (*p*) and/or lateral movement of the pile (*y*) at specified depths (instead of letting the program perform internal computations of these factors). If the user would like to apply the strength reduction factors based on LRFD design requirements, p-multipliers can be assigned for that purpose. The menu Data > Modification Factors for p-y Curves is enabled when a check mark is placed on this option. Read more information in Section 3.2.17. This feature is not available if the user chooses Options > Earth-pressure specified by User since that case assumes that all loads are taken into account in the user-specified loading.

3.3.16 Conduct Short-Term and Long-Term Analyses

If this option is selected, the program will allow the user to enter different soil parameters and earth pressures for short-term and long-term conditions in one model run (data file). After a successful run, the program will present the response curves (deflection, bending moment, shear and reaction versus depth) for both long and short term together for convenient comparison between those two conditions.

A check mark on this option enables the user to enter separate soil properties and separate active-earth pressures for short-term and long-term conditions. This option cannot be combined along with the option to conduct staged-construction analysis.

3.3.17 Conduct Staged-Construction Analysis

With this selection, the user can conduct analysis at different construction stages. However, the program will not trace the stress history of the soil and structure at each construction stage. Instead, the program will use the earth pressure, restrained boundary conditions and soil resistance curves (*p*-*y* curves) at each stage as the initial conditions with which to evaluate the behavior of the system without considering any effects due to the stress history.

A selection of this option provides the screen in Figure 3.57 where the user should check the appropriate box and enter the total number of construction stages (minimum of 2 and maximum of 8). After entering the number of construction stages and selecting the OK button the user should notice a couple of changes (see Figure 3.58): i) a new menu option labeled Stage Construction is now visible in the menu and ii) new speed buttons are visible below the menu screen.

The Stage Construction menu lists each of the stages that the user defines for the construction sequence. The user must first select Stage 1 (place a check mark on Stage 1) and define the complete PYWALL model for such stage (selecting all appropriate inputs under Data and Options menu).

After defining the model for the first stage the user should select Stage Construction > Stage 2 (place a check mark on Stage 2) to define the model for the second construction phase (selecting all appropriate inputs under Data and Options menu). For construction stages beyond the first one, the user may select Stage Construction > Copy Previous Stage which instructs the program to copy all inputs from the immediately previous stage to the one that is selected, which is helpful for many inputs that may be similar.

If desired, users may access the Stage Construction menu to select any desired Stage that was defined and access again the Stage Construction menu to Delete Current Stage (to delete whichever stage the user selected previously).

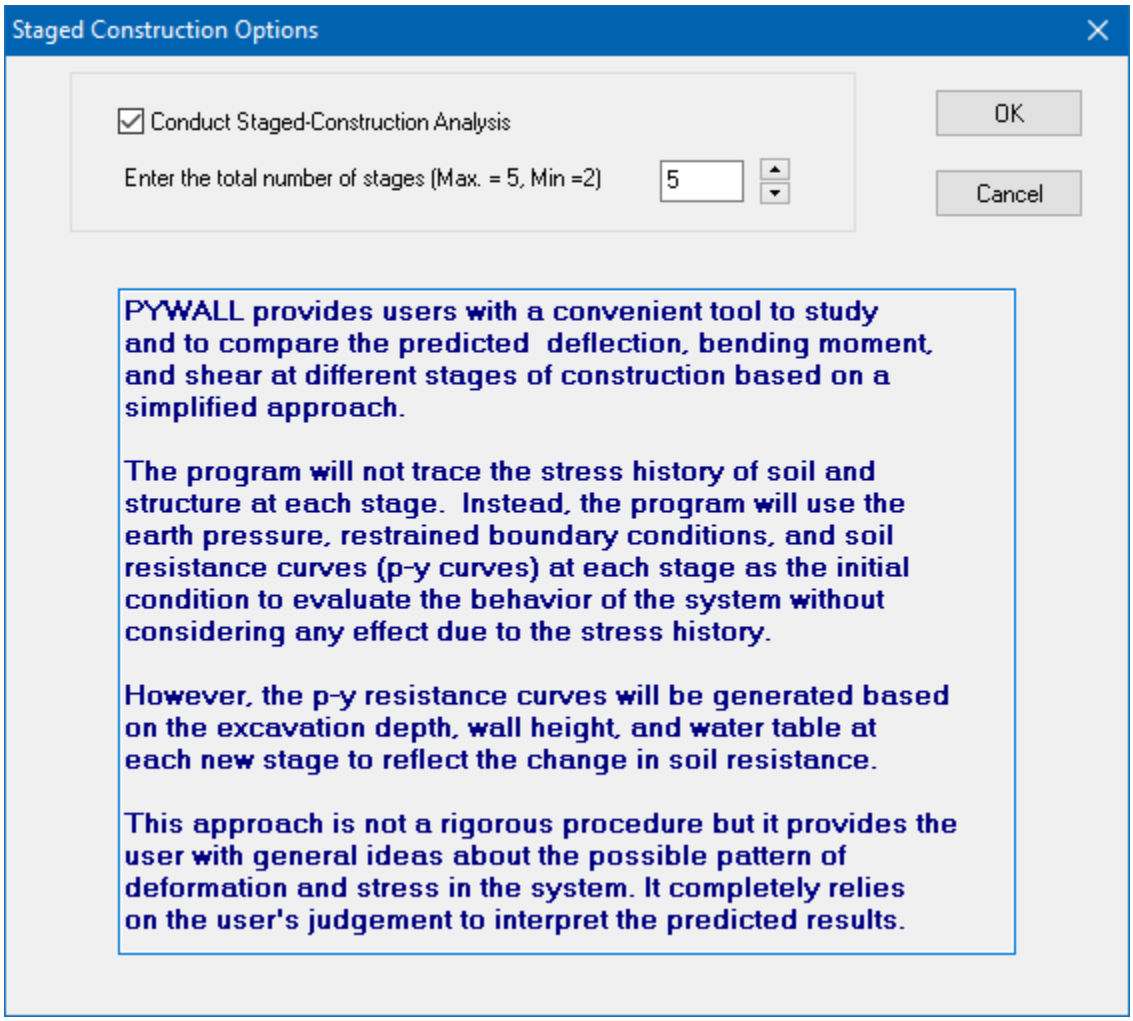


Figure 3.57 Staged Construction Options menu

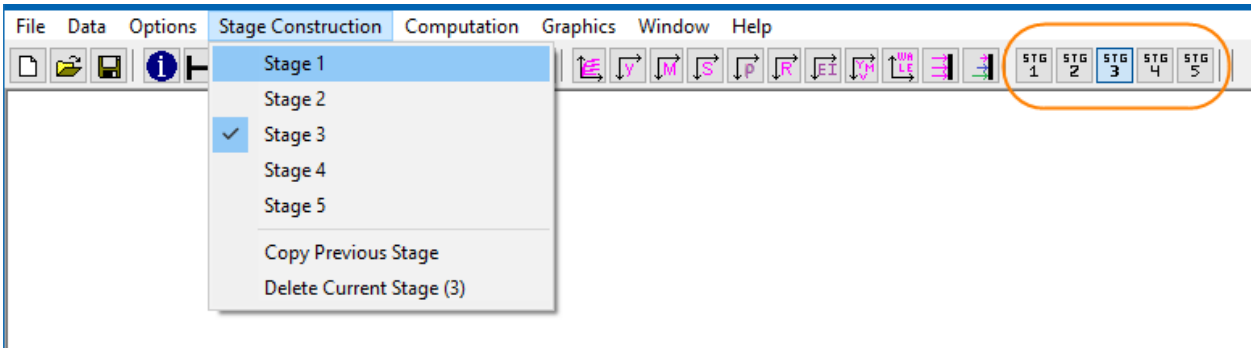


Figure 3.58 Stage Construction Menu and Speed Buttons

3.3.18 User Defined Sheet Piling Sections…

This selection will provide a dialog box similar to the one in Figure 3.59. When first used, the "Select File" line will point the user to a file in the original installation directory (default is *SheetPiling_User_in.csv* for models in English Units and *SheetPiling_User_mm.csv* for models in SI Units). Notice that some users may not be able to save/modify that file in the "*Program Files*" folder (depending

on their access rights provided by their IT or Admin). Users should be able to select the Browse button and save their user-defined sheet piling table to any other directory (where they have proper rights). Users can choose almost any file name but the extension must be of the type "*csv*".

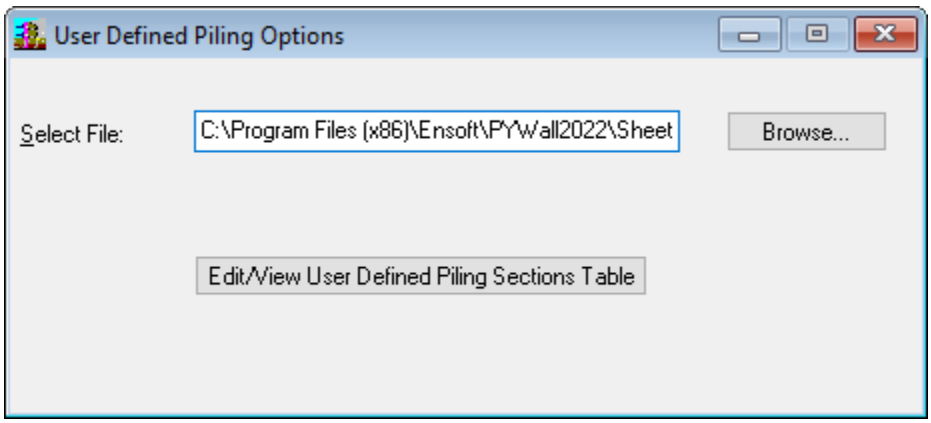


Figure 3.59 Sample Options > User Defined Sheet Piling Sections...

Selecting the Edit/View User Defined Piling Sections Table button provides the screen in Figure 3.60 where the user defines the name and properties of their own custom sheet piling shapes. The various entries are defined below.

Applicable entries are in the same units (English or SI) as the PYWALL model that is being prepared. If the user needs to switch to different units, close the dialog boxes and select the desired units in Options > Units then come back to this menu. Users should save their custom sheet pile entries in the units of their interest.

Type

This drop-down menu defines the basic shape of the sheet piling section that is being custom defined by the user. There are four basic shape options: Flat, Z, PAN, U which are shown in Figure 3.61.

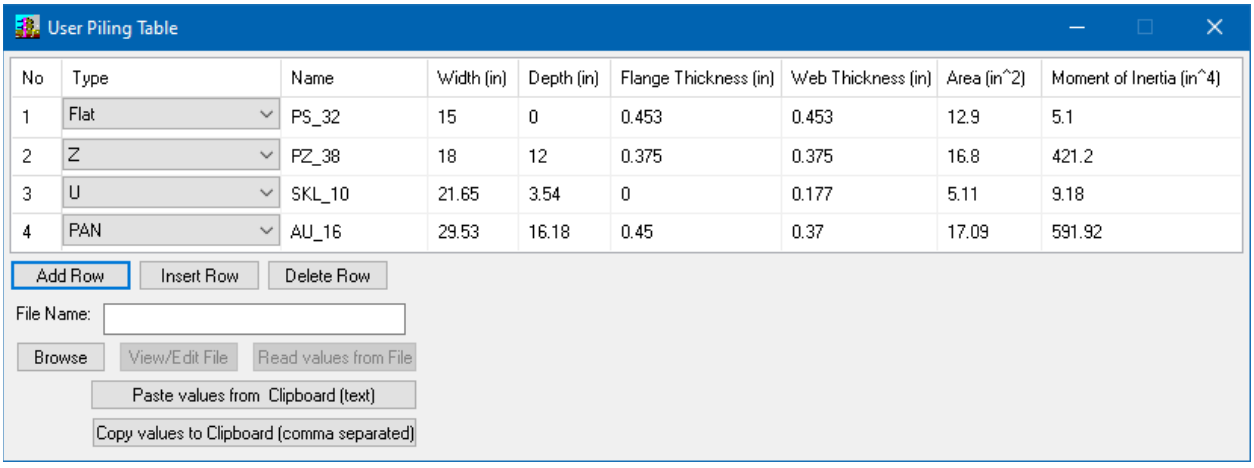
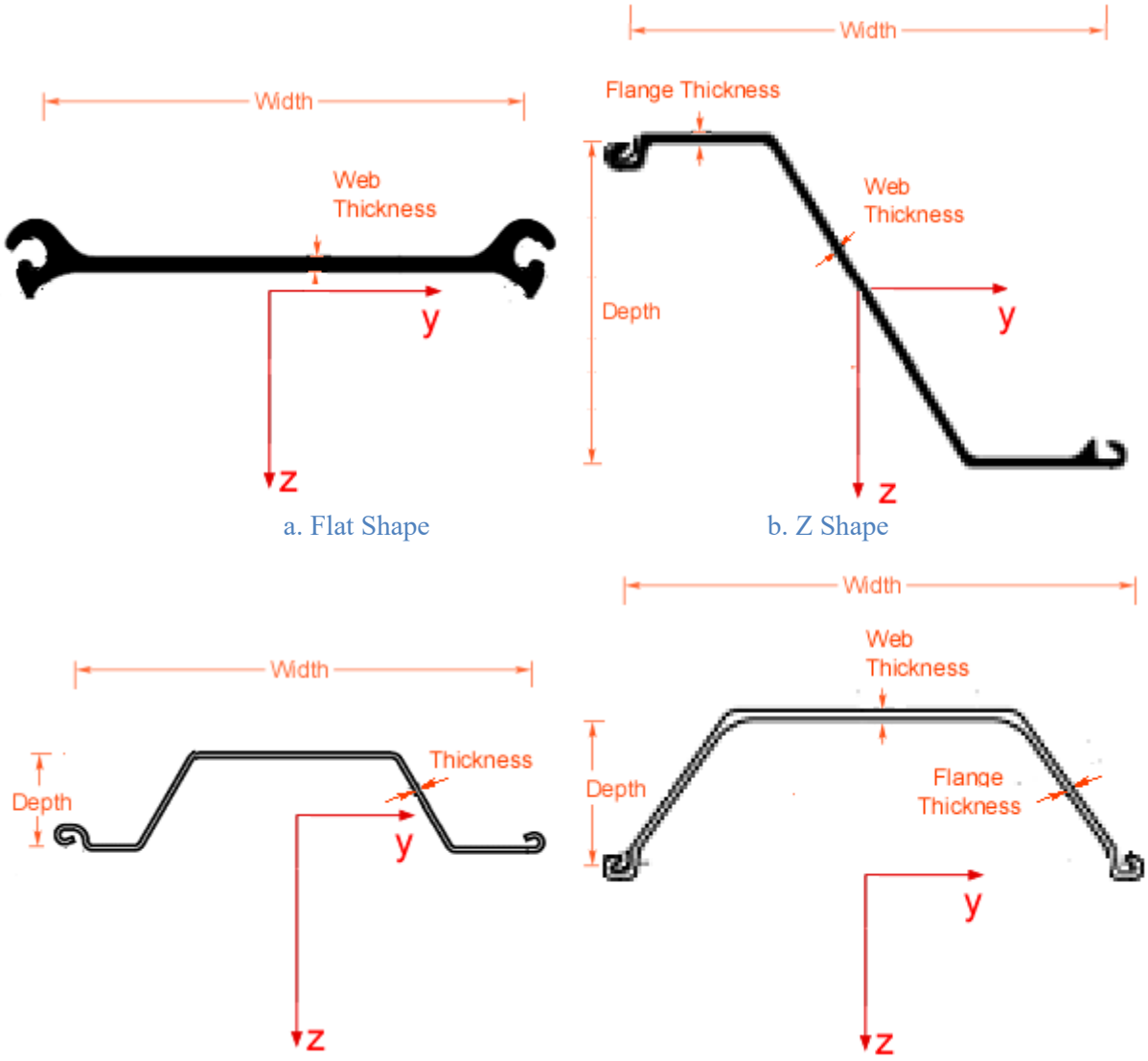


Figure 3.60 Menu for User-Defined Sheet Piling Section



a. Flat Shape b. Z Shape

c. PAN Shape d. U Shape

Figure 3.61 Basic Sheet Piling Shape Options

Name

In this entry users can enter any alphanumeric entry that identifies the custom section that is being defined. Samples are shown in Figure 3.60.

Width

These entries are inputted in units of inches or mm and should represent the dimension that are referenced in Figure 3.61 for the various shape options.

Depth

These entries are inputted in units of inches or mm and should represent the dimension that are referenced in Figure 3.61 for the various shape options (not used for the Flat shape).

Flange Thickness

These entries are inputted in units of inches or mm and should represent the dimension that are referenced in Figure 3.61 for the various shape options. These entries are the thickness of the inclined components of the shape options. Notice that the Flange Thickness is not very obvious for the referenced PAN shape (Figure 3.61c). This entry is not used for the Flat shape.

Web Thickness

These entries are inputted in units of inches or mm and should represent the dimension that is referenced in Figure 3.61 for the various shape options. These entries are the thickness of the horizontal components of the shape options. Notice that the Web Thickness is not very obvious for the referenced PAN shape (Figure 3.61c).

Area

These entries are inputted in units of inches to the square or mm to the square and represent the cross-sectional area of the custom sheet pile section that the user is defining. This is a necessary entry for all four shape options.

Moment of Inertia

These entries are inputted in units of inches to the fourth power or mm to the fourth power and represent the moment of inertia of the custom sheet pile section that the user is defining. This is a necessary entry for all four shape options. The moment of inertia that is entered by the user must be defined for the working axes of the shape option.

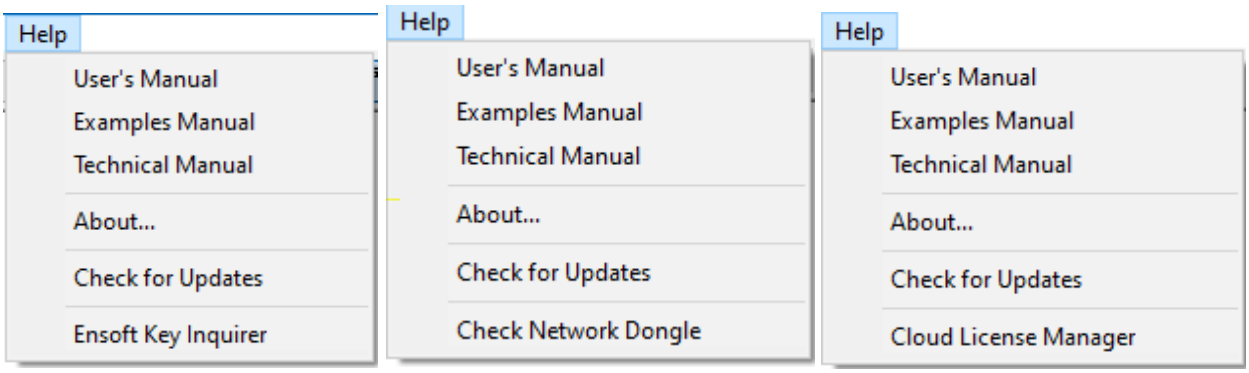
3.3.19 Option of Text Editor

The user should input in this box the complete path and command line for the preferred text editor or word processor that will be used to examine and print the input, output, and processor plain text files created by the program. As a default, the command line is used to operate the standard Microsoft Notepad text editor that is included with most Windows installations. However, there may be some instances when the output files are too large for Notepad and a different text editor or word processor should be used. In those cases, it is suggested to use Microsoft WordPad or similar.

3.4 Help Menu

This menu allows for access to three manuals that are installed with PYWALL: User's Manual, Examples Manual, and Technical Manual. All three manuals are copyrighted and also protected against modifications or copying of its content. These manuals are distributed in standard PDF files and should be available for any PDF reader that is installed in the user's computer.

The Help menu, shown in Figure 3.62, also provides access to slightly different utilities/information according to the type of license that is installed in each computer. The various entries in the Help menu are described below.



a. Single-User License b. Client on Local Network License c. Cloud Subscription License

Figure 3.62 Help Menu

3.4.1 User's Manual

The Help > User's Manual calls the default PDF-file reader that is installed in the user's computer and opens the PDF file of the PYWALL User's Manual. The User's Manual for PYWALL is contained in the *PYWall 2025 Users Manual.pdf* file that is installed with the program.

3.4.2 Examples Manual

The Help > Examples Manual calls the default PDF-file reader that is installed in the user's computer and opens the PDF file of the PYWALL Examples Manual. The Examples Manual for PYWALL is contained in the *PYWall 2025 - Examples Manual.pdf* file that is installed with the program.

3.4.3 Technical Manual

The Help > Technical Manual calls the default PDF-file reader that is installed in the user's computer and opens the PDF file of the PYWALL Technical Manual. The Technical Manual for PYWALL is contained in the *PYWall 2025 Technical Manual.pdf* file that is installed with the program.

3.4.4 About PyWall

This command provides a dialog describing the program version, date, and methods for accessing technical support. Other information about the program licensing and maintenance expiration date, program version, and program release date are also shown. A sample of this dialog box is provided in Figure 3.63.

3.4.5 Check for Updates

The Help > Check for Updates submenu will start up the default Internet browser and report the user of the current program version and maintenance expiration date as well as any new update that is available for downloading from Ensoft's website (*www.ensoftinc.com*). Users shall be able to download, install and run, in full mode and without the intervention from Ensoft, any PYWALL program release/update that was made prior to their maintenance expiration date. Notice that some anti-virus programs may require the user to grant permission to PYWALL to open the Internet browser program.

3.4.6 User Information

In this entry, the user may enter any information that personalizes their PYWALL installation. This information is requested when the user first starts the program after a fresh installation. A sample of this dialog box is provided in Figure 3.64.

The information in this entry is included near the beginning of all output files that are generated by this particular installation of PYWALL. As an example, the first section of a sample output file is provided in Figure 3.65.

3.4.7 Ensoft Key Inquirer

This submenu is only available on single-user installations of the software product (it is not available for users running a network version of the product). This utility is used to check for licensing information on the USB Key that is connected to a working USB port of the computer in use. It provides information such as serial number, maintenance expiration date, etc. This utility is also necessary for a remote update and renewal of the license for this product. This utility is installed in the selected installation directory with the following filename: *ensoft_key_inquirer_v5.exe* (may change with time).

3.4.8 Check Network Dongle

This submenu is only available on client computers of network versions of the software product (it is not available for users running a single-user version of the product). This utility helps to check the licensing and to troubleshoot communications with a license manager (or software server). This utility is very useful in case of problems in clients while trying to run the licensed software. This utility is installed in the selected installation directory with the following filename: *CheckNetworkDongle_ipv_511.exe* (may change with time).

3.4.9 Cloud License Manager

This is only available on cloud subscription installations of the software product (it is not available for users with USB Key based installations, such as single-user or network licenses). A sample dialog screen for this utility is shown in Figure 3.66.

The Account tab allows to Logout so as to prevent other users of this computer from logging into PYWALL with the same Ensoft Cloud user account. After a Logout, starting the program, or running new computations or saving a file will require a new Ensoft Cloud account login procedure with the required <username> and <password>.

The License tab allows users to Checkout Seat for a specified period of time (Duration) in order to be able to use the license without internet access (this is also referenced as 'hard checkout' of a license). By default, the maximum Duration is limited to 15 days by Ensoft. However, the account 'Admin' may choose a lower maximum Duration for any License. After the expiration, the checked-out License will return to the system automatically. However, the user may return the checked-out License earlier by selecting the Release Seat button.

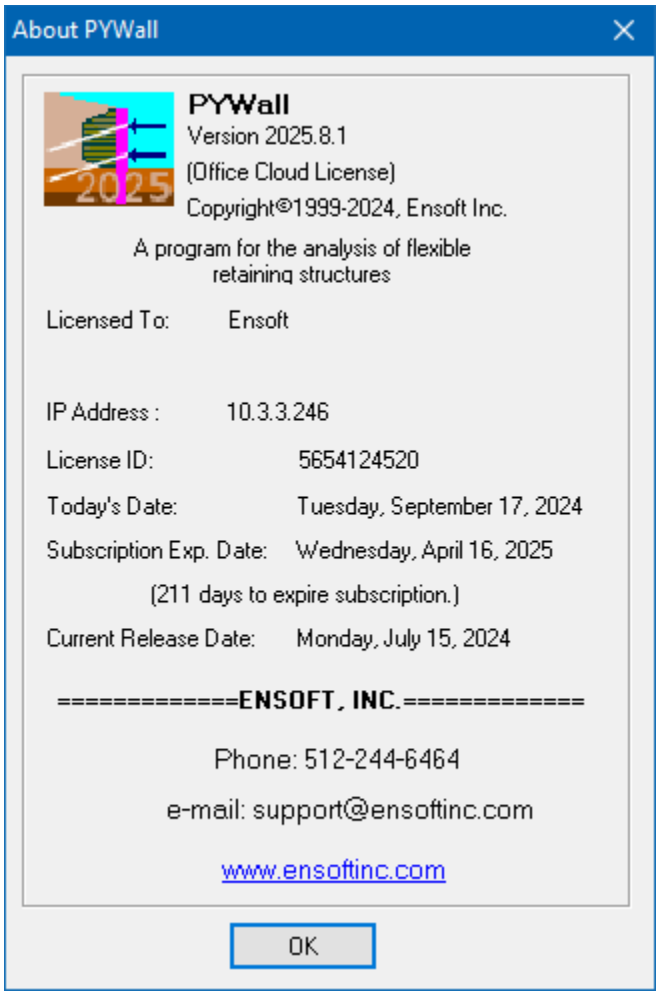


Figure 3.63 Help > About Sample Dialog Box

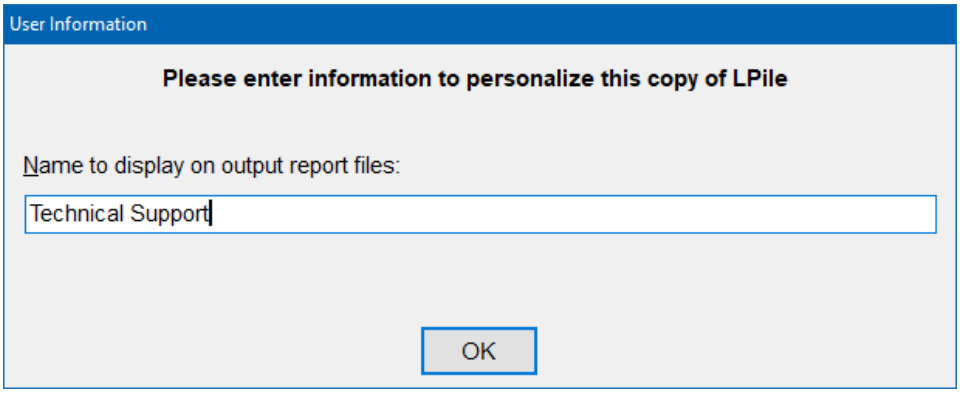


Figure 3.64 Sample Dialog Box for User Information

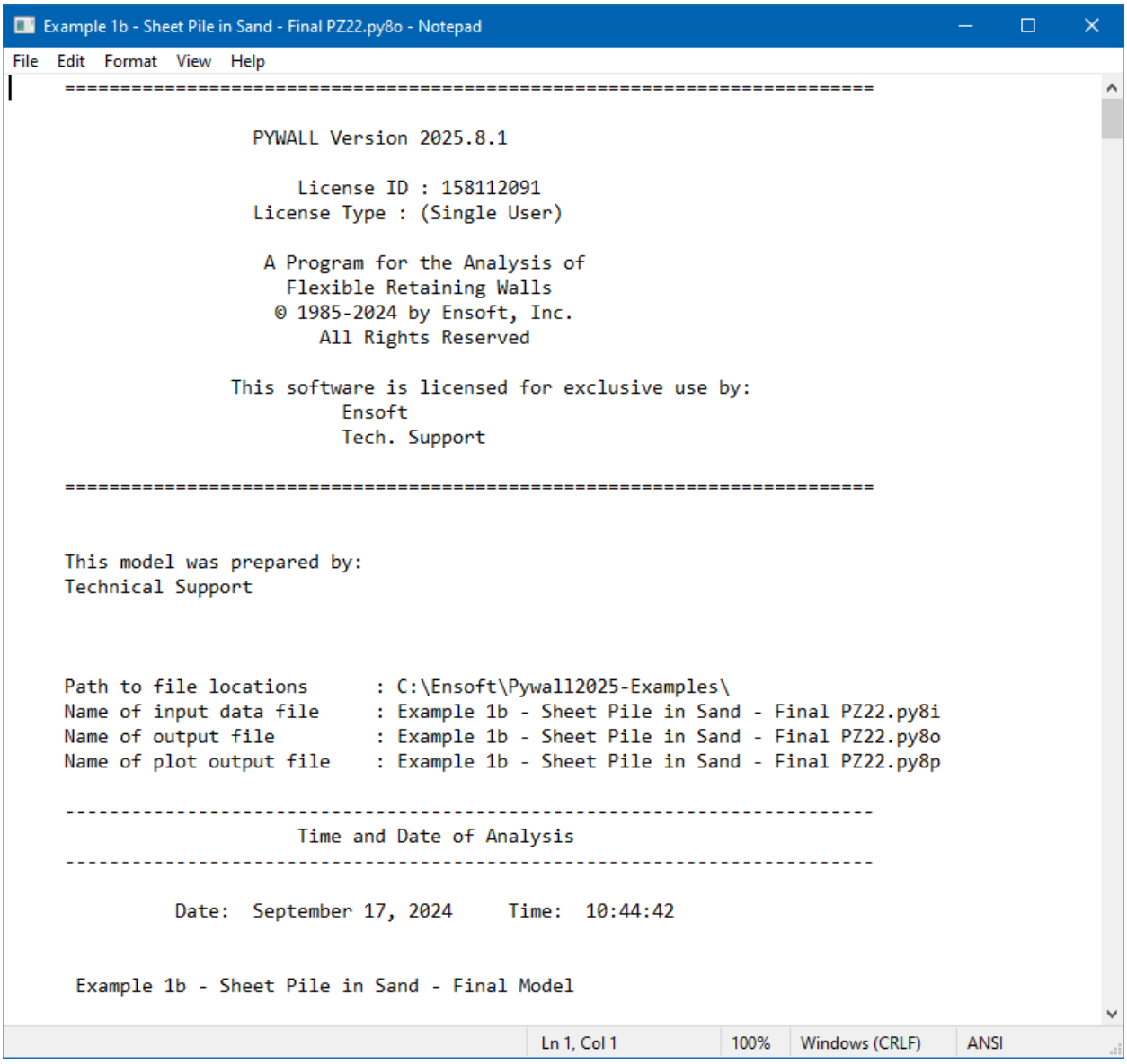


Figure 3.65 Sample Heading from Output File

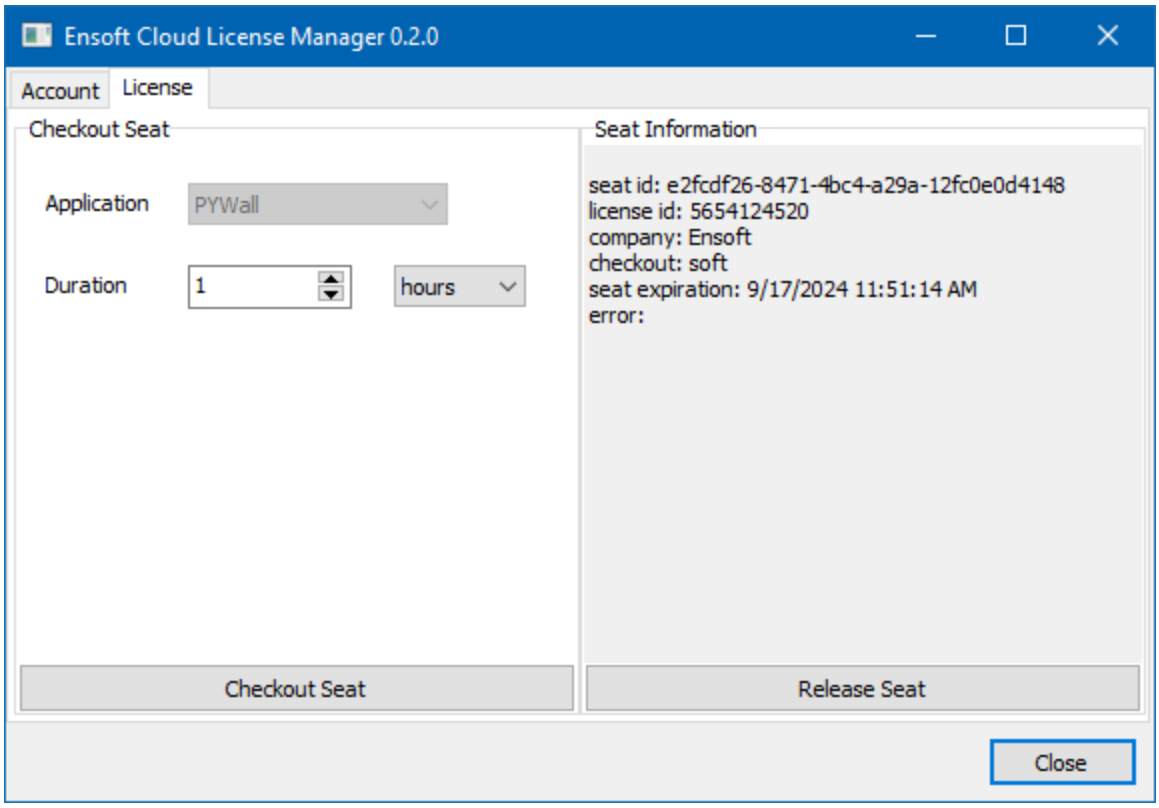


Figure 3.66 Sample Dialog Box for Ensoft Cloud License Manager

CHAPTER 4. Program Execution and Output Reviews

4.1 Introduction

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

4.2 Computation Menu

This menu option is selected to execute the program using the parameters that were saved in the input-data file. Within the options contained under this menu, shown in Figure 4.1, there are commands that facilitate the reviews of the text files produced for storing output results and processor notes. In addition, the user may select options of graphical representations of the modeled wall geometry, soil layers and some of the output. Detailed description of the submenu options contained under the Computation menu are explained in the following topics.

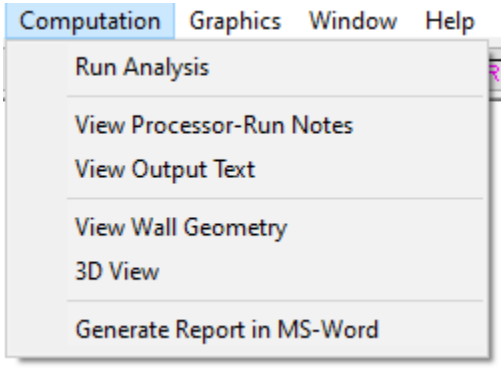


Figure 4.1 Computation Menu

4.2.1 Run Analysis

An input file, after preparation or modification, must be saved to disk before selecting the Computation > Run Analysis submenu option, which executes the analytical portion of program PYWALL.

The PYWALL program always first automatically saves the input data file before performing computations. If the model was never saved then the program will prompt the user to save the data file. When saving a data file, the PYWALL program will automatically add an extension of the type **.py6d* to the name of the input data file. The program is not able to save data files that are compatible to older versions/releases of PYWALL.

When the execution process is finished, the active command is returned to the main PYWALL program with a screen indicating that computations are completed. The user should always check the output text file for any error messages that were produced during computations.

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

4.2.2 View Processor-Run Notes

This submenu option is used to edit an intermediate text file that is automatically produced during each analytical run. This file only may include notes produced during the processing of the input data. This submenu option becomes active after new data files have been saved to disk and executed, or when opening previously-executed data files.

This submenu automatically invokes the word processor or text editor specified in Control Options... under the Data menu. The default setting is to use the utility program named *notepad.exe* provided by Microsoft Windows©.

Files containing processor-run notes are automatically saved to disk with the same file name as the input-data file but with the extension **.py8r*. Use of the Microsoft Notepad© for editing the processor- run notes for Example Problem 1 is shown in Figure 4.2.

Observation of the notes produced during a processor run may become helpful to debug a data file that did not produce a successful run. A successful run usually produces a file of processor-run notes containing similar lines of text as those in Figure 4.2.

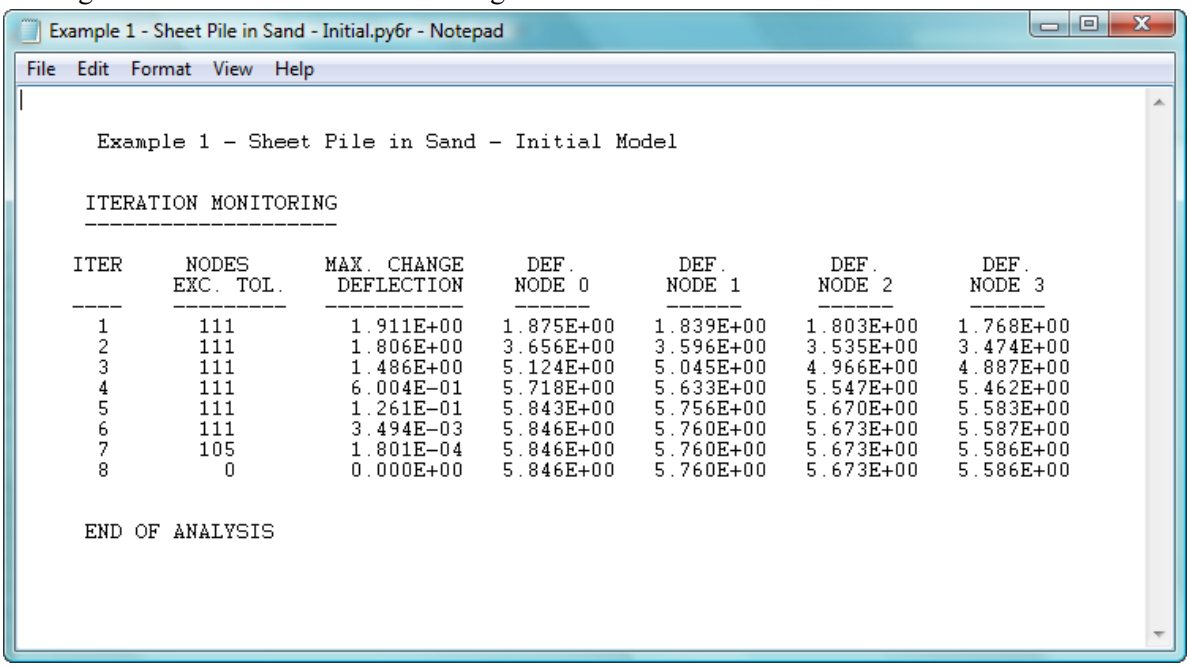


Figure 4.2 Sample processor-run notes for Example Problem 1 in Microsoft Notepad©

4.2.3 View Output Text

This submenu option is used to edit the output-text file that is automatically produced during each successful analytical run. This command becomes active after new data files have been saved to disk and successfully executed, or when opening previously-executed data files.

The submenu automatically invokes the word processor or text editor that was previously specified in Control Options... under the Data menu. The default setting is to use the utility program named *notepad.exe* provided by Microsoft Windows©.

Output files are automatically saved to disk with the same file name as the input-data file but with the extension **.py8o*. Use of Microsoft Notepad© for editing the output file for Example Problem 1 is shown in Figure 4.3.

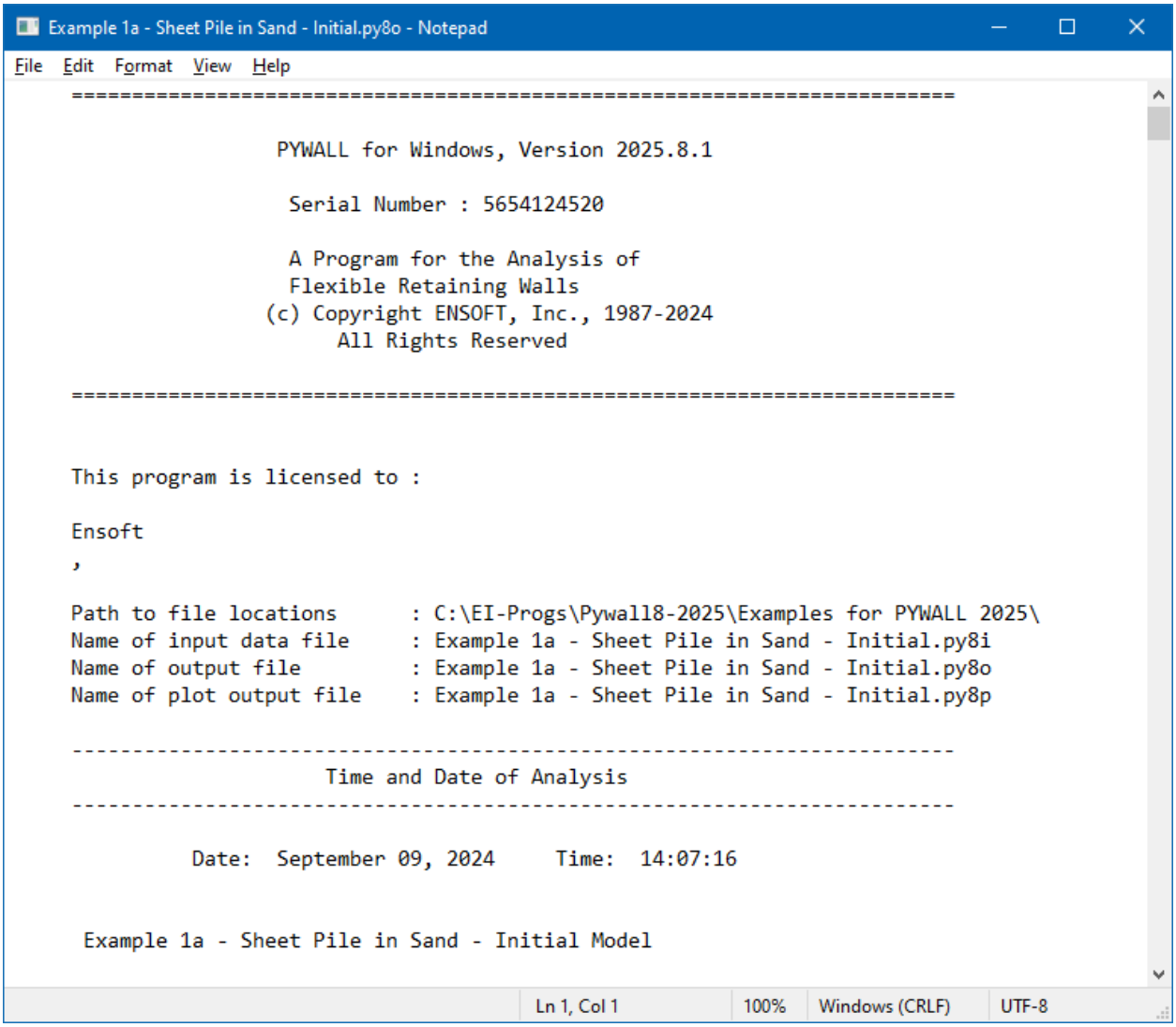


Figure 4.3 Sample output-text file of Example Problem 1 in Microsoft Notepad©

4.2.4 View Wall Geometry

The Computation > View Wall Geometry submenu option provides a graphical screen that displays an elevation view of the wall geometry and soil layers along with their various depths. The graphics also includes symbols to represent tiebacks and struts as well as other fixities and restraints in the wall. On the right side of the graphics, it displays a simple plan view of the modeled wall.

This screen is displayed whenever the user opens an existing model with valid data. The position of the cursor in the screen provides an approximate indication of the depth (from top of wall/ground surface at backfill side to the cursor position) in the status menu at the bottom of the program dialog box. This command option becomes active after data of Drilled-Shaft Properties and Soil Layer Data have been entered under the Data menu, or when opening previously-executed data files.

A sample graphics of the View Wall Geometry command option is shown in Figure 4.4. Clicking the Print button on the graphics screen send the observed graphics to the default Windows printer; without any other prompt.

Double clicking on various locations in the new View Wall Geometry opens applicable data windows, such as the following:

- Data > Soil Layer Data opens when double clicking in any soil layer.
- Data > Tieback Data opens when double clicking in any tieback.
- Data > Strut/Raker Data opens when double clicking in any strut/raker.
- Data > Wall Properties opens when double clicking in the vertical wall element.
- Data > Wall Configuration opens when double clicking to the right of the vertical wall, above the excavation line, or on the water line in backfill or excavation side (when available), or on the slope above the backfill (if available) or on the excavation slope (if available).
- Data > Cross Sections opens when double clicking in the top view of the wall.

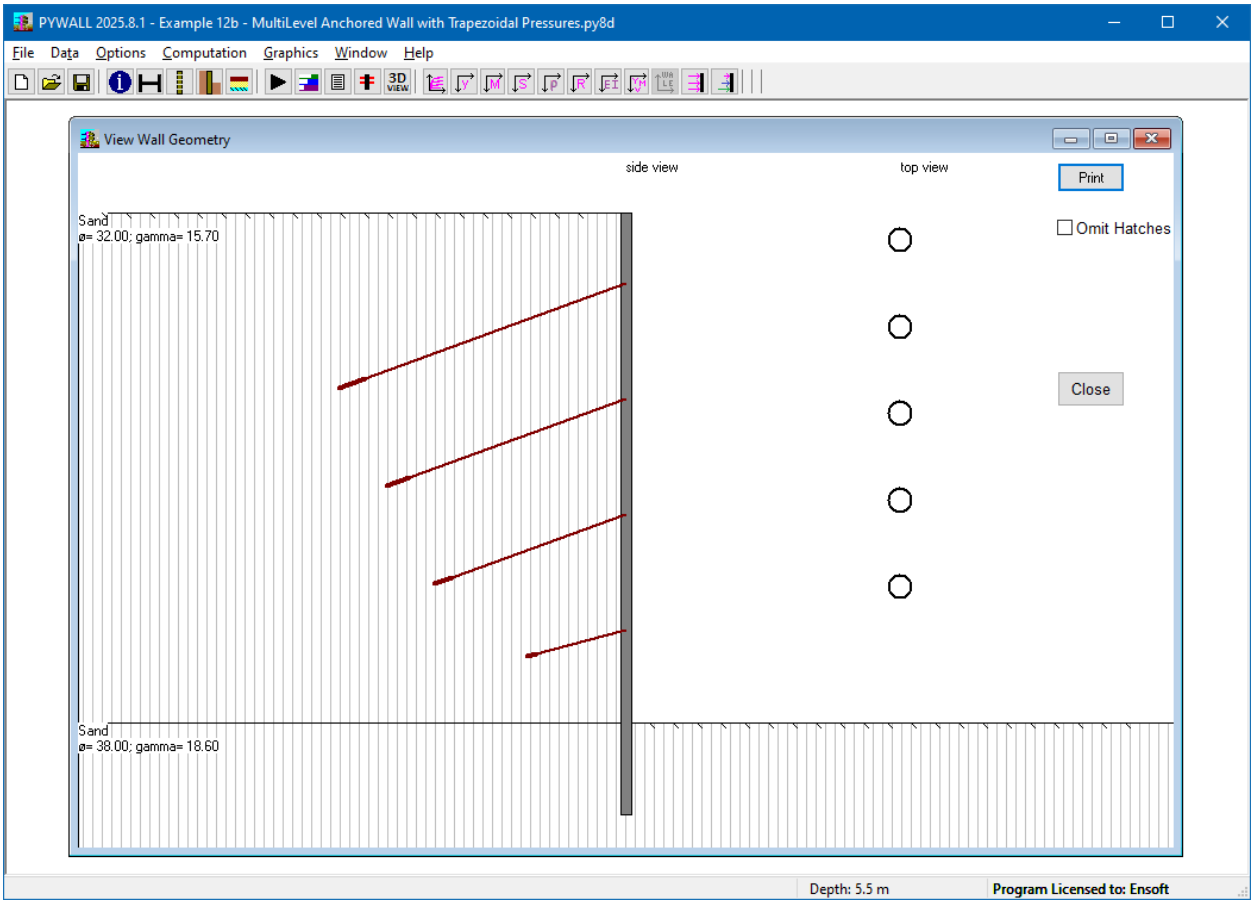


Figure 4.4 Sample View Wall Geometry

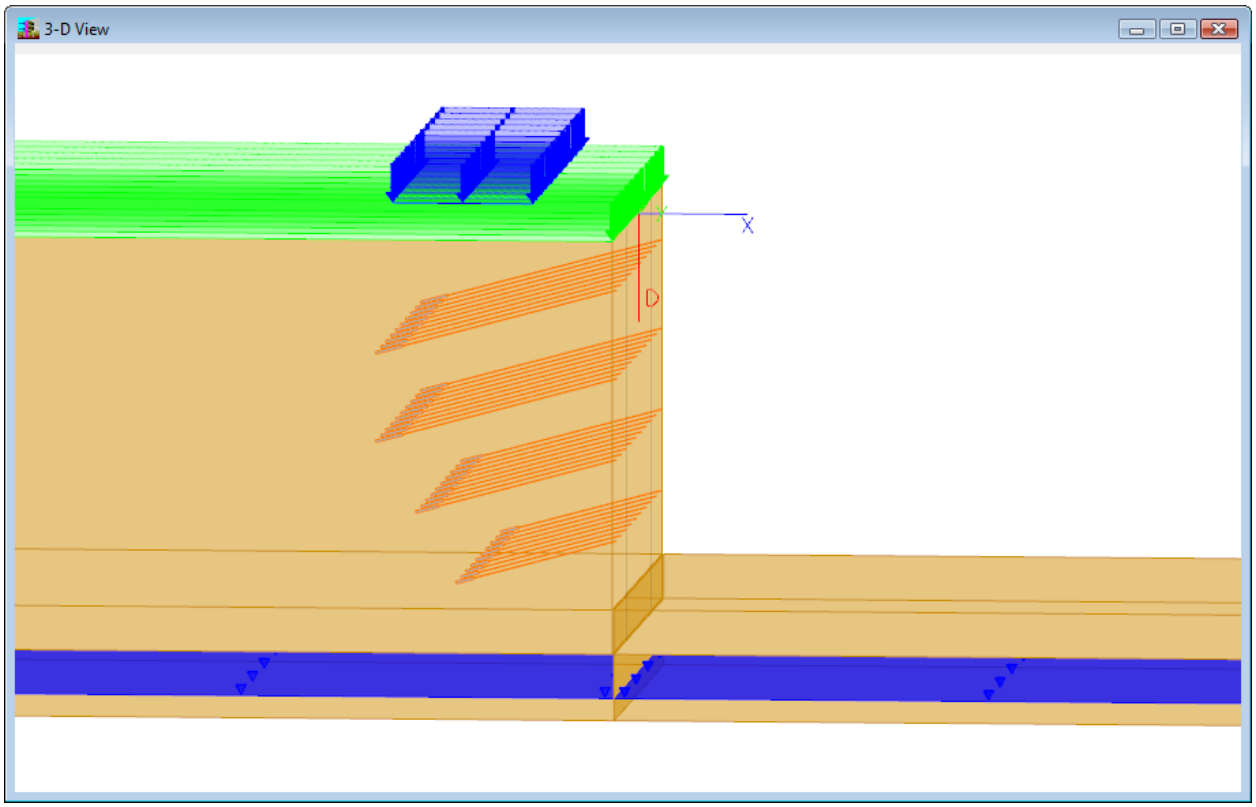
4.2.5 3D View

The Computation > 3D View submenu option provides a three-dimensional graphical screen that displays an elevation view of the wall geometry, soil layers along with their various depths, surcharge loads, tiebacks, and other boundary conditions. This program feature can be very helpful to visualize and verify the modeled wall geometry and layout as shown in the two samples on Figure 4.5.

When observing any model in Computation > 3D View the user should notice the new Show 3DView menu (also some new speed buttons) shown in Figure 4.6 that are used to modify the observed screen. The various menu options are described in the following sections. Some of the options in this menu may be grayed out, depending on the specific features of the observed model.

4.2.5.1 Show 3DView > Deformed Shape

This feature is sometimes useful to visualize a model's response. This may be particularly helpful in models with multiple tieback locations, such as the sample case shown in Figure 4.7.



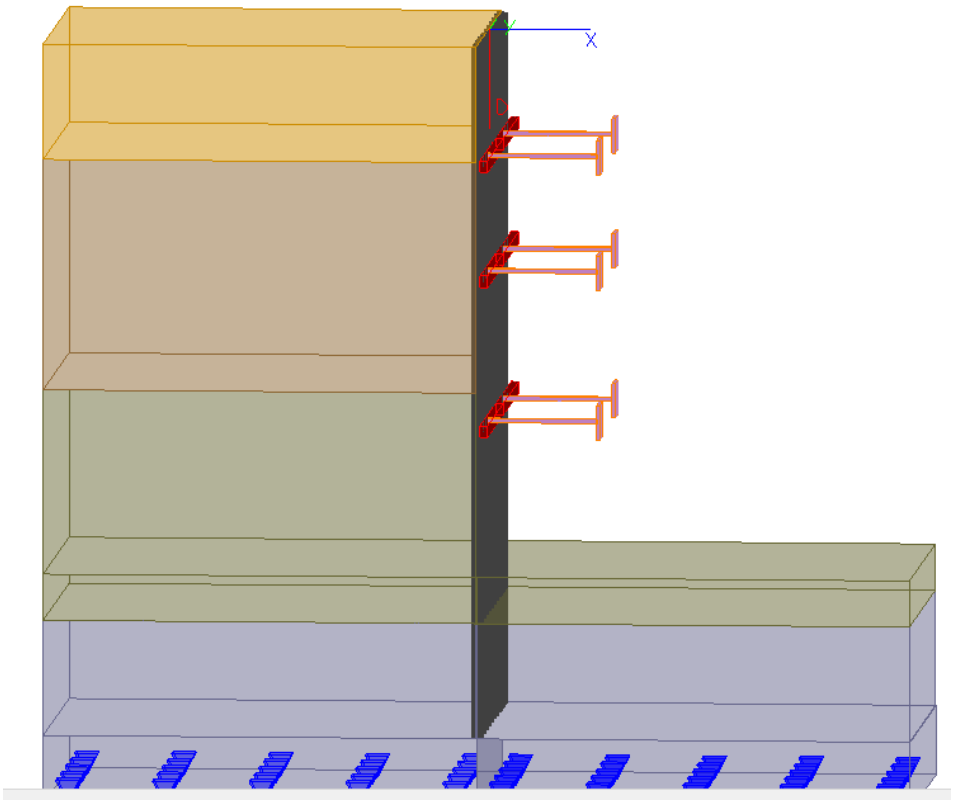


Figure 4.5 Sample Three-Dimensional Views of Wall Geometry and Configuration

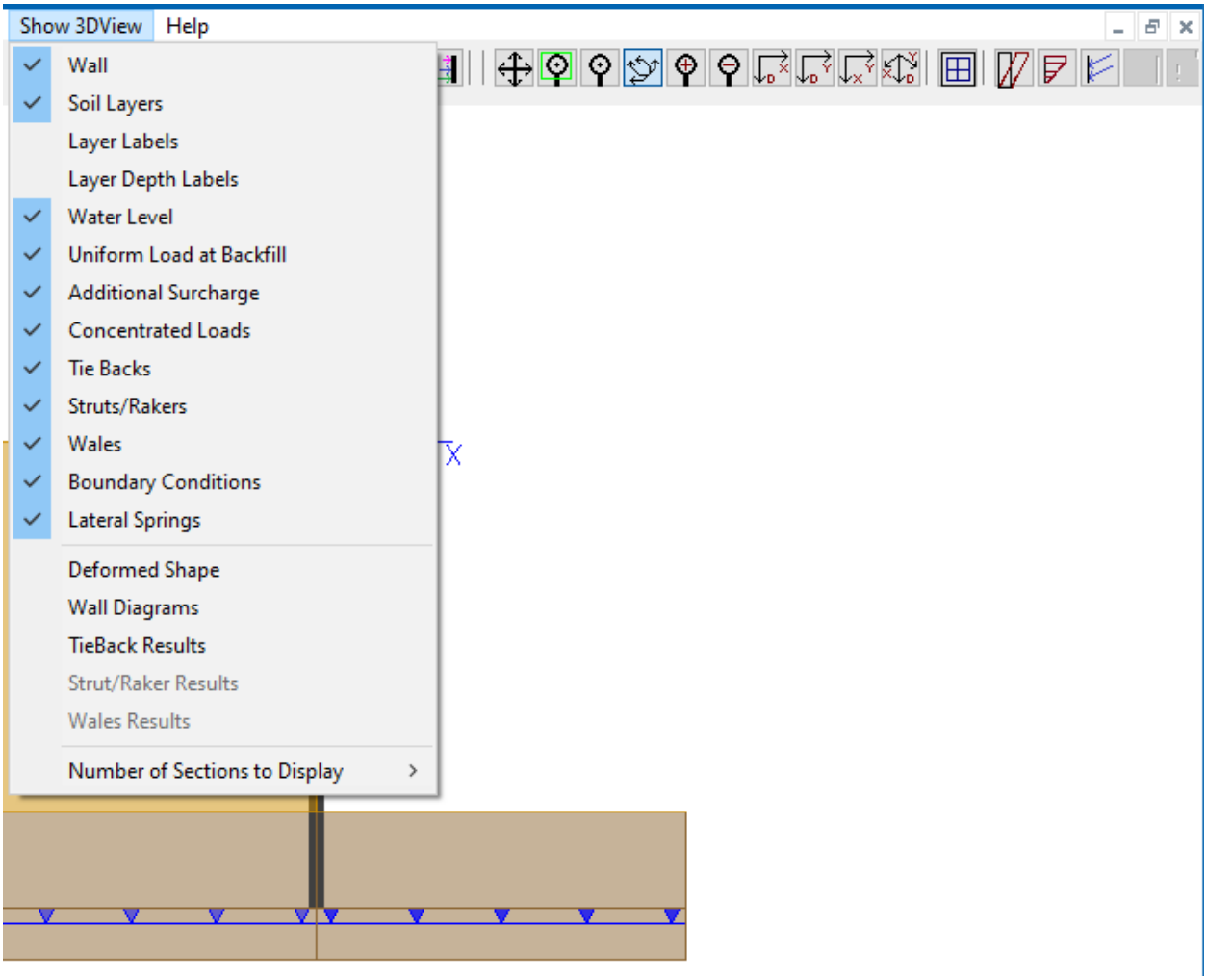


Figure 4.6 Show 3DView Menu

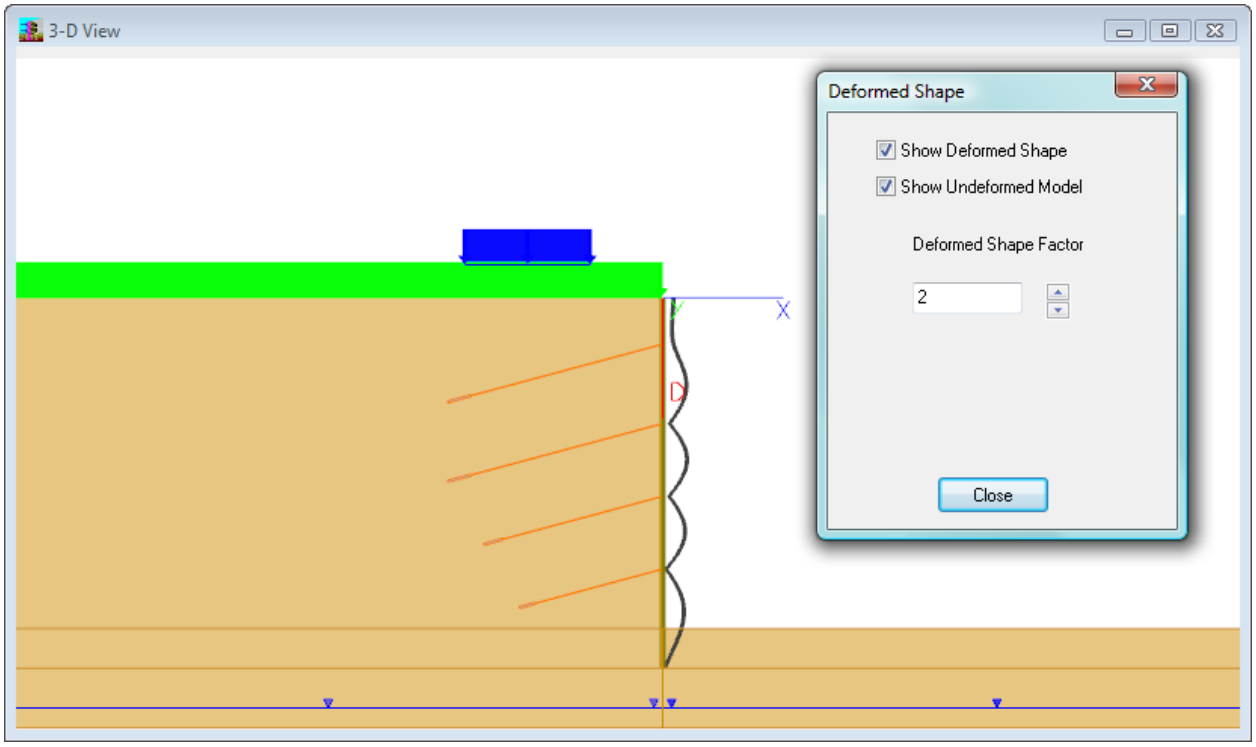


Figure 4.7 Sample Show 3DView > Deformed Shape

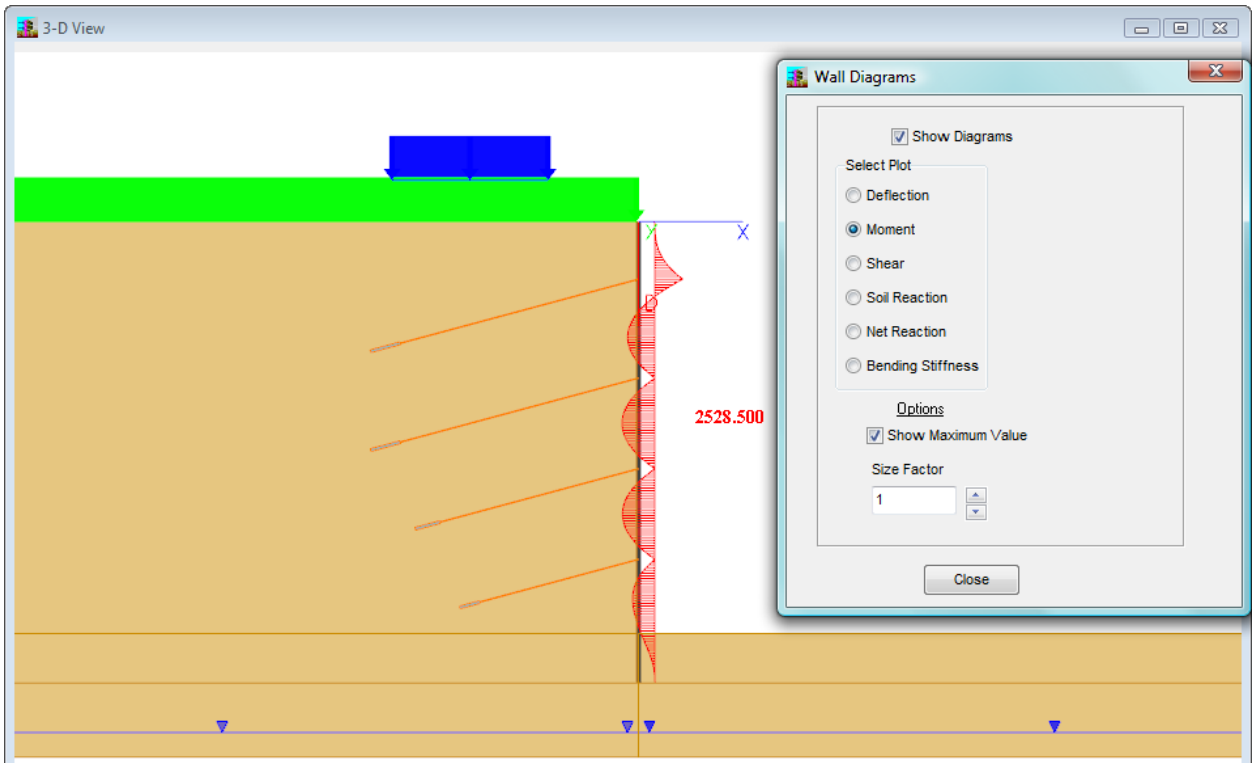


Figure 4.8 Sample Show 3DView > Wall Diagrams

4.2.5.2 Show 3DView > Wall Diagrams

Users may desire to plot some of the output results alongside the modeled wall and with some of the numerical results (maxima). With this feature the users can display a graphics like the sample in Figure 4.8. Users can display plots/maximum value of: Deflection, Bending Moment, Shear, Soil Reaction, Net Reaction, and Bending Stiffness. Units are not shown but the same as those in the corresponding plots under the Graphics menu.

4.2.5.3 Show 3DView > Tieback Results (Strut/Raker Results)

Output results of tieback responses alongside the modeled wall can be quickly observed with this feature (see sample in Figure 4.9). Users can display plots/values of: Horizontal Force per Station, Total Force per Station, Total Force at Tieback and Prestressing Force (Initial). The 'Total Force per Station' is the force per unit Width for Active Earth Press. Calculation, Wa (Section 3.2.7.6).

Units are in lbs for models in English Units and in kN for models in SI Units. These results can be helpful for the user's design, outside PYWALL, of the tiebacks (size and bonded lengths).

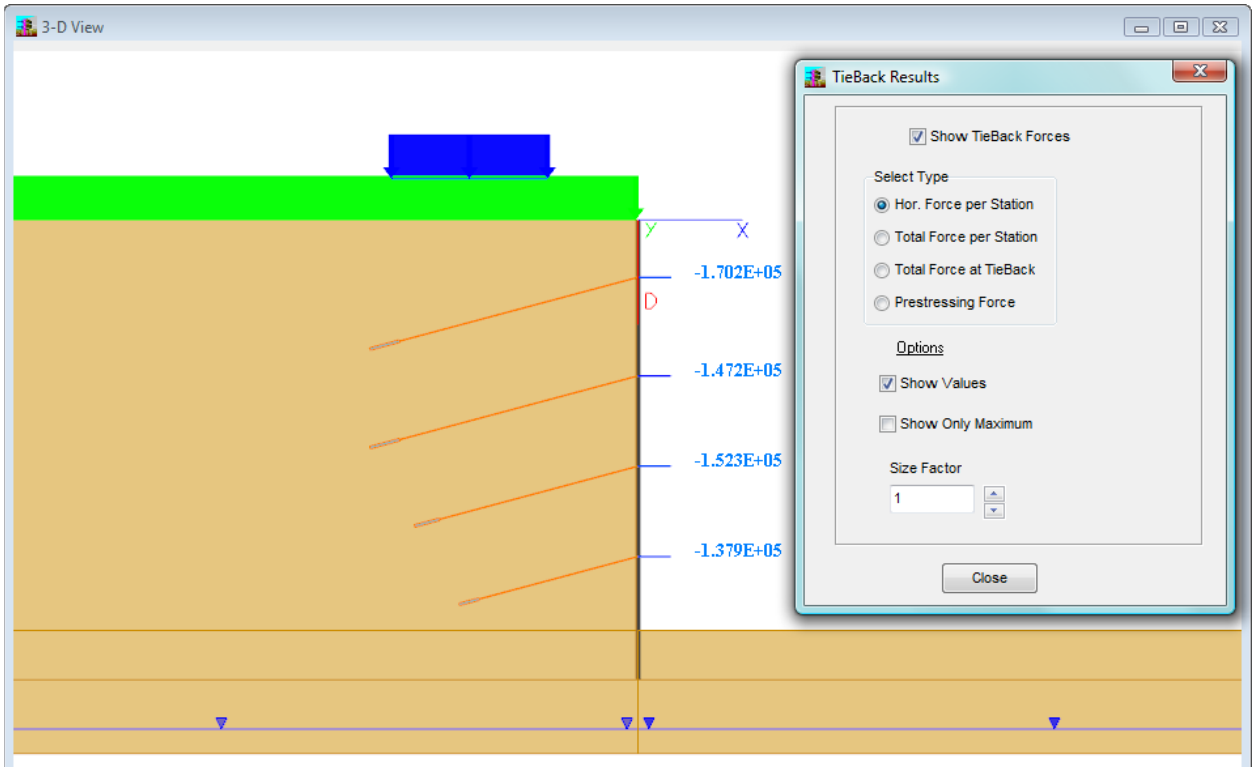


Figure 4.9 Show 3DView > Tieback Results

4.2.5.4 Show 3DView > Wales Results

Output results of the demand on the wales in the model can be quickly observed with this feature. The sample in Figure 4.10 was prepared with a rotated wall and hiding the wall elements. Users can select to display plots/values of: Equivalent Distributed Load, Shear and Bending Moment. These results can be helpful for the user's design/check of the wales.

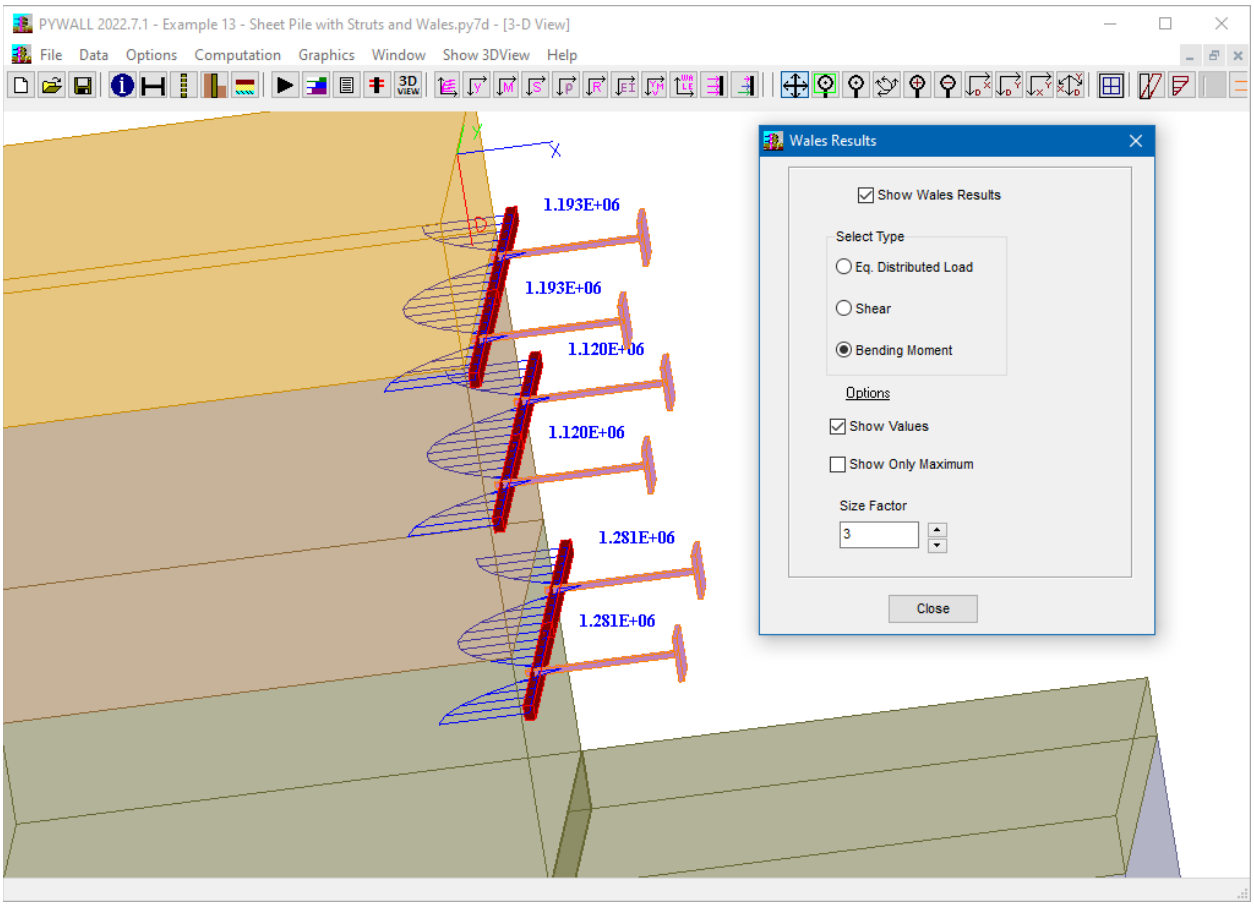


Figure 4.10 Show 3DView > Wales Results

4.2.6 Generate Report in MS-Word

This menu selection allows users to export input and output information for the PYWALL model along with optional data groups (see Figure 4.11) into a pre-formatted Microsoft Word file. Each generated report is provided with the following information:

- Software name, version and update.
- License information (serial number, company name and licensed office site – when applicable).
- Name and path of input/output data files.
- Date of computational run.
- Echo printing of various model data parameters.

In addition, in the dialog box users can select to include the following optional content:

Input > Model Schematic adds to the Word report a graphical representation with a Side View of the model.

Input > Wall Parameters adds Word tables with echo printing of the properties that were entered by the user for the model.

Output > Analysis Results adds Word tables with the computational results from the model.

Output > Graph plots adds a formatted plot of any applicable result plot that is also selected to include in the report.

Users may copy and paste the contents of any of the tables above from Word to a spreadsheet if they prefer different units or plot format. However, keep in mind that the separate Graphics > Export Plots to Excel described in the next section provides a simpler method for editing plots.

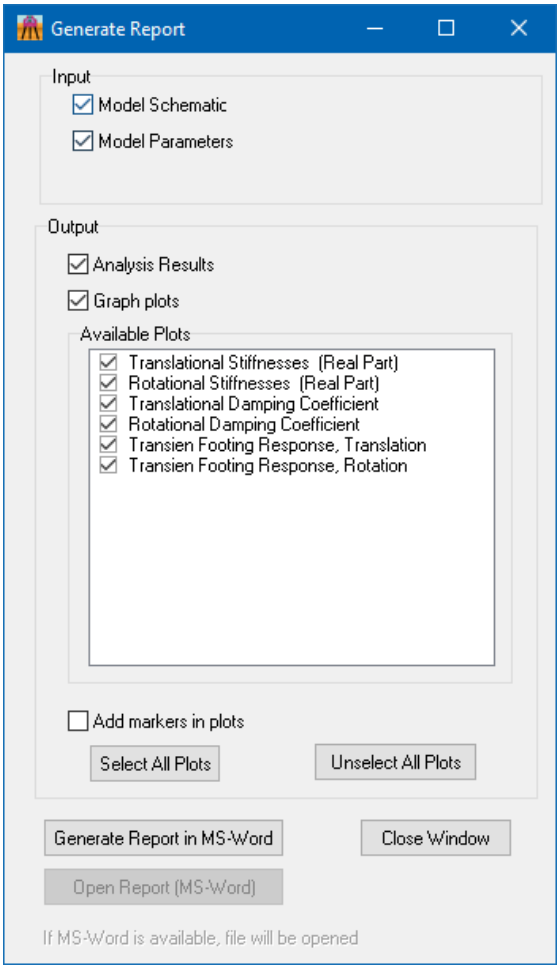


Figure 4.11 Sample Dialog box to Generate Report in MS Word

4.3 Graphics Menu

This menu option is selected to observe the different graphical representations of the program results contained in the output file. Submenu options contained under this menu are shown in Figure 4.12. Not all of the graphics may be enabled for observation, since active curves depend on the parameters that were selected by the user for the model. If all items under the Graphics menu are disabled (grayed out) it is likely that the model is incomplete, had run-time errors or was not yet computed (no results).

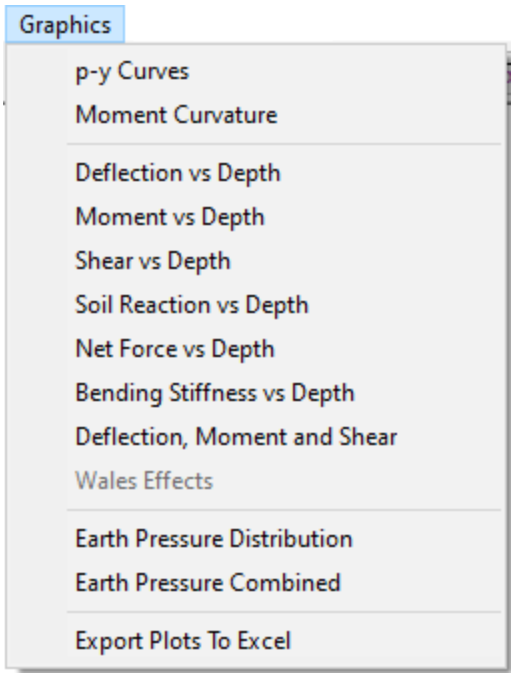


Figure 4.12 Graphics Menu

4.3.1 Mouse Commands

Table 4.1 describes mouse commands that become enabled automatically during all graphical observations of output curves.

Event	Description
Mouse Left Click	Zooms in on the graphics at the clicked position (up to five zoom levels)
Mouse Right Click	Zooms out from the graphics
Mouse Double Click on Legend	Turns the selected curve on/off at every click

Table 4.1 Mouse Commands in the Graphics Mode

4.3.2 Graphics > p-y Curves

PYWALL automatically generates graphical outputs of either internally-generated *p-y* curves in each soil layer or user-entered (external) *p-y* curves. For the case of internally-generated *p-y* curves, the curves are printed at the top, bottom and quarter depth of each soil layer that is specified by the user in the numerical model. If below excavation and if different than the depth of changes in soil layers, additional *p-y* curves are printed at depths above and below the water table.

The plotted *p-y* curves cover both the soil resistance at the backfill side and at the excavation side. A general sample of this graphics option is shown in Figure 4.13. It is noticeable that the soil resistance (*p*) at the backfill side (upper left quadrant) is higher than those at the excavation side (lower right quadrant) due to the difference in the overburden pressure.

The legend with depth references are as tabulated from top of wall. The *p-y* curves are generated according to the wall width *Wp* as defined in Section 3.2.7.5. The *p* values are in units of Force per unit Length (representing unit of depth) and the *y* values are in units of Length (representing unit of lateral movement).

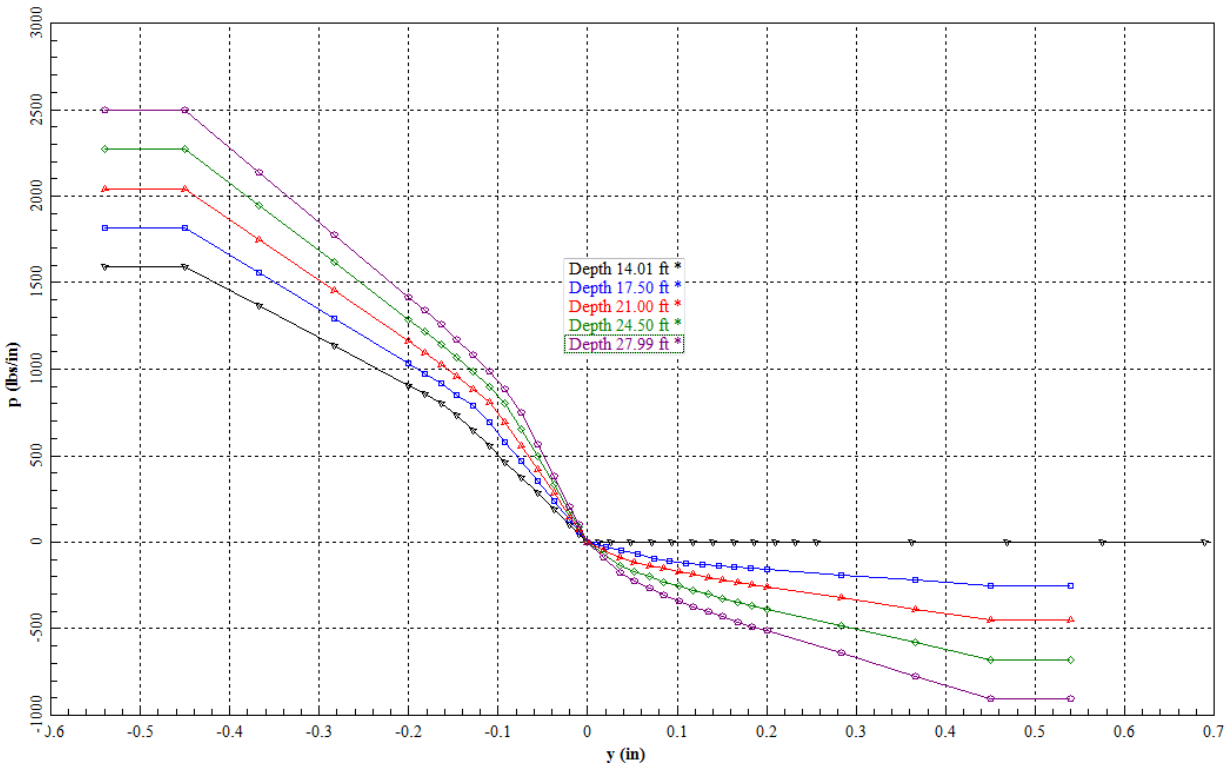


Figure 4.13 Graphics > p-y Curves

4.3.3 Graphics > Moment Curvature

The plots in this menu option are related to the response of the wall sections that were defined in the model. This menu is most helpful when using nonlinear sections since for other sections the graphs are all linear. A general sample of this graphics plot for a steel section is shown in Figure 4.14.

Users can select to plot graphs of: Bending Moment vs Curvature, Bending Stiffness vs Curvature, Bending Stiffness vs Bending Moment and Axial Force-Bending Moment Interaction. These graphs are available for each wall section defined by the user, which are selected with the drop-down menu at bottom right labeled Select Section.

The interface allows users to export the viewed graph or all graphs to pre-formatted Microsoft Excel spreadsheets. The exported spreadsheets will contain tabs with formatted graph(s) and others with the corresponding tabular values (source).

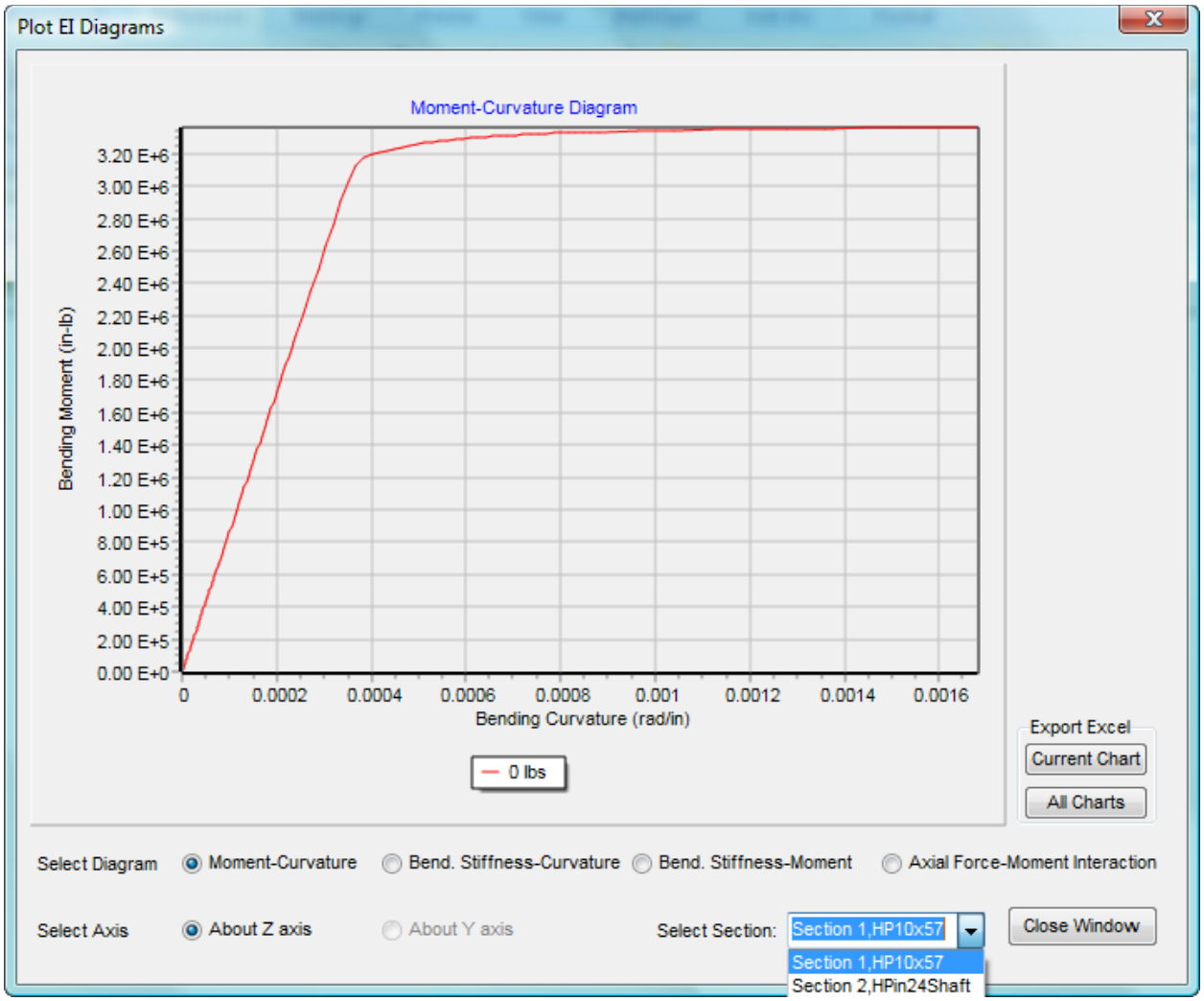


Figure 4.14 Sample Graphics > Moment Curvature for a Steel Section

4.3.4 Graphics > Deflection vs Depth

This menu selection displays a plot of the lateral deflections of the modeled wall versus depth (vertical distance from top of wall). This curve is automatically generated in all analytical runs. The number of points on the deflection curve is equal to the selected number of increments specified on the wall. Several curves may be displayed with this selection if the user inputs several load cases (eg., in models with staged construction or with short/long-term analyses). A general sample of this graphics plot is shown in Figure 4.15.

4.3.5 Graphics > Moment vs Depth

This graphics option shows a plot of the bending moment of the modeled wall unit versus depth. This curve is automatically generated in all analytical runs of a laterally-loaded wall. The number of points on the moment curve is equal to the selected number of increments specified on the wall. Several curves may be contained in this graphic if the user inputs several load cases (models with staged construction or with short/long-term analyses). A general sample of this graphics plot is shown in Figure 4.16. The bending moments are provided per wall unit as defined in Section 3.2.3.

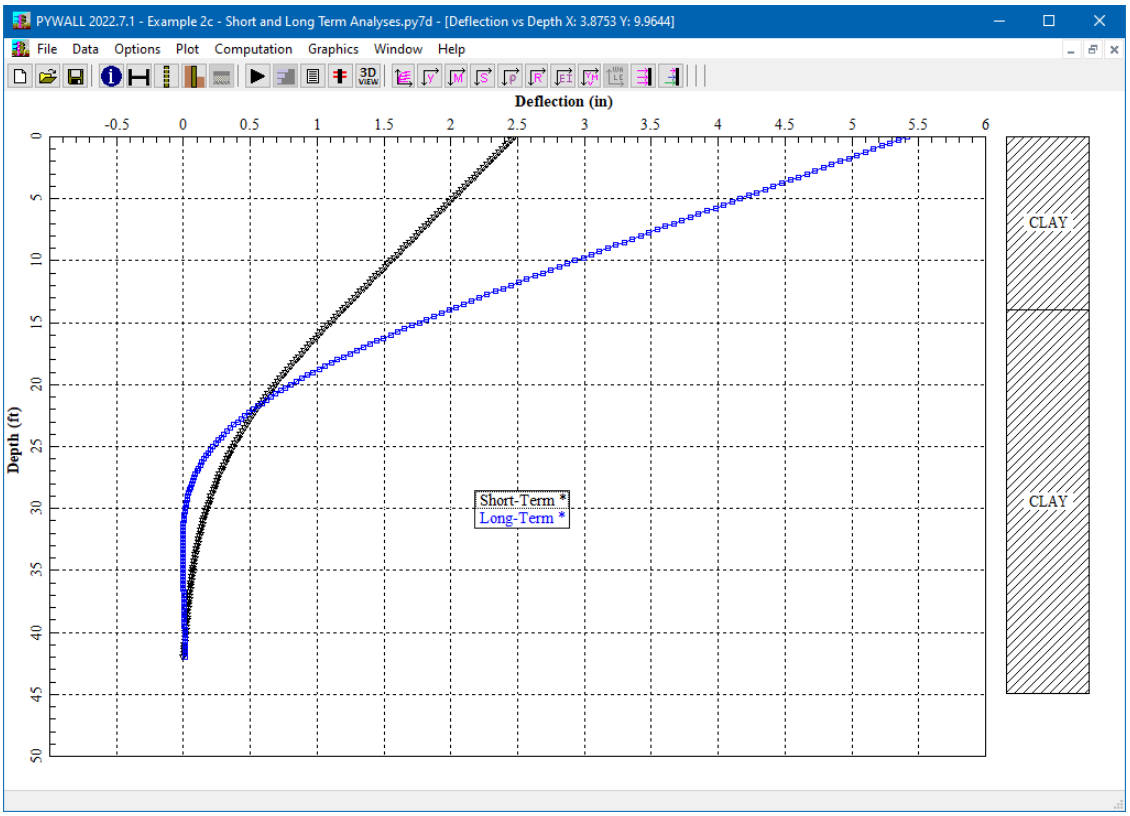


Figure 4.15 Sample Graphics > Deflection vs Depth

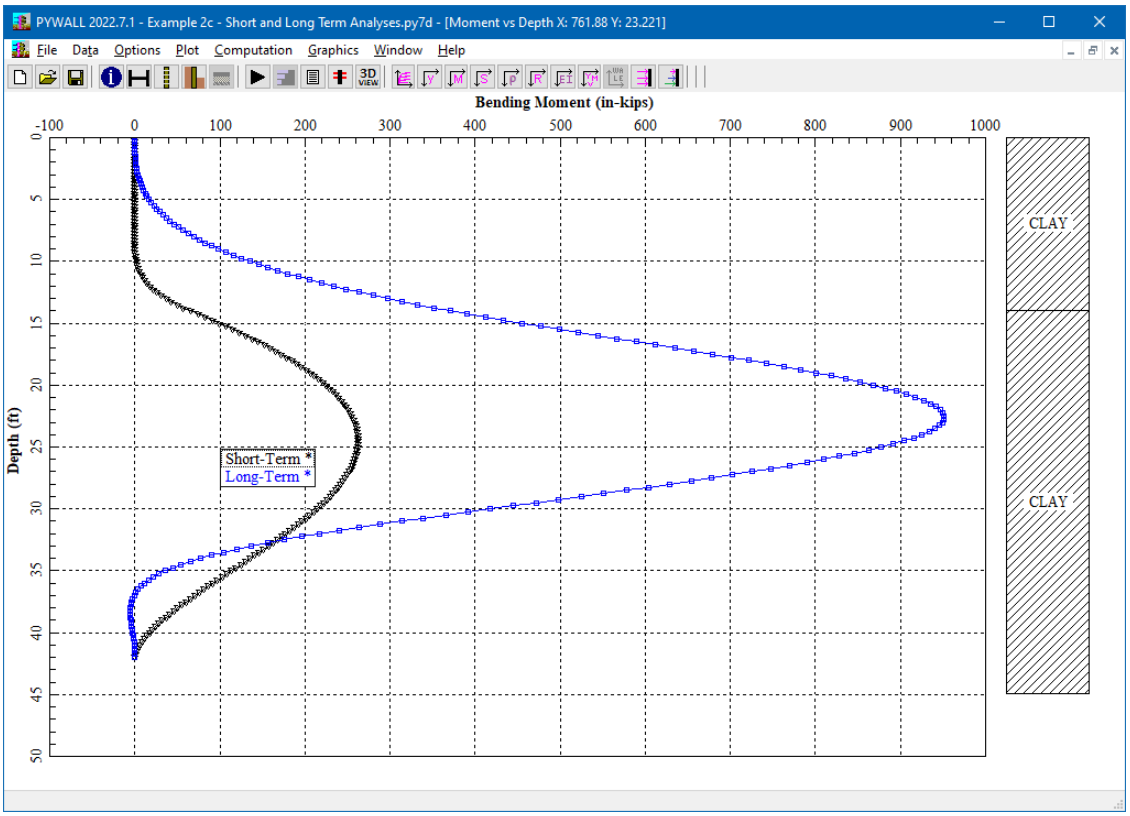


Figure 4.16 Sample Graphics > Moment vs Depth

4.3.6 Graphics > Shear vs Depth

This graphics option shows a plot of the shear of the modeled wall unit versus depth (measured from top of wall). This curve is automatically generated in all analytical runs of a laterally-loaded wall. The number of points on the shear curve is equal to the selected number of increments specified on the wall. Several curves may be displayed with this selection if the user inputs several load cases (models with staged construction or with short/long-term analyses). A general sample of this graphics plot is shown in Figure 4.17. The shear forces in this graph are provided per wall unit as defined in Section 3.2.3.

4.3.7 Graphics > Soil Reaction vs Depth

This graphics option shows a plot of the soil reaction (soil resistance or p value that is engage at each depth from the p-y curves) of the modeled wall versus depth (measured from top of wall). The soil reaction are in units of Force per unit Length (representing unit of depth) and provided per wall width W_p as defined in Section 3.2.7.5 when below the excavation and wall width W_a as defined in Section 3.2.7.6 when above the excavation.

The curve is automatically generated in all analytical runs of a laterally-loaded wall. The number of points on the soil reaction curve is equal to the selected number of increments specified on the wall. Several curves may be displayed with this selection if the user inputs several load cases (models with staged construction or with short/long-term analyses). A general sample of this graphics option is shown in Figure 4.18.

If the model uses the Options > Earth-Pressure Options > Generate Earth Pressure Based on Movement then the Soil Reaction may be an active pressure or a soil resistance, depending on the direction of movement of the wall. A sample result for the Soil Reaction vs Depth plot on a model with Generate Earth Pressure Based on Movement is included in Figure 4.19. In this particular example, the Soil Reaction below the 11-m excavation depth represent the soil resistance (or engaged p values) while the Soil Reaction above 11 m represent the pressures from the retained soil calculated by either K_a or K_p based on the direction and amount of movement, as explained in the Technical Manual (Section 3.9). When a sufficiently large tieback force is introduced then the soil pressure (or resistance to the tieback load) may be high (in the passive state) if the wall is moving towards the retained soil.

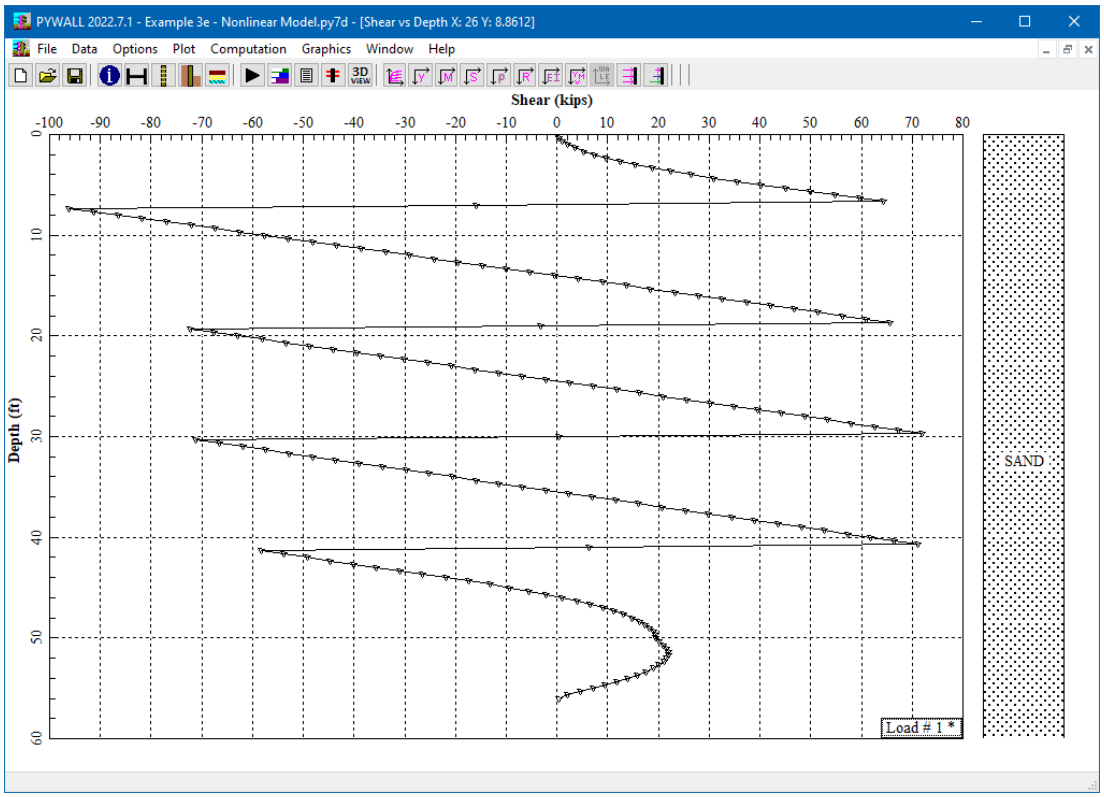


Figure 4.17 Sample Graphics > Shear vs Depth

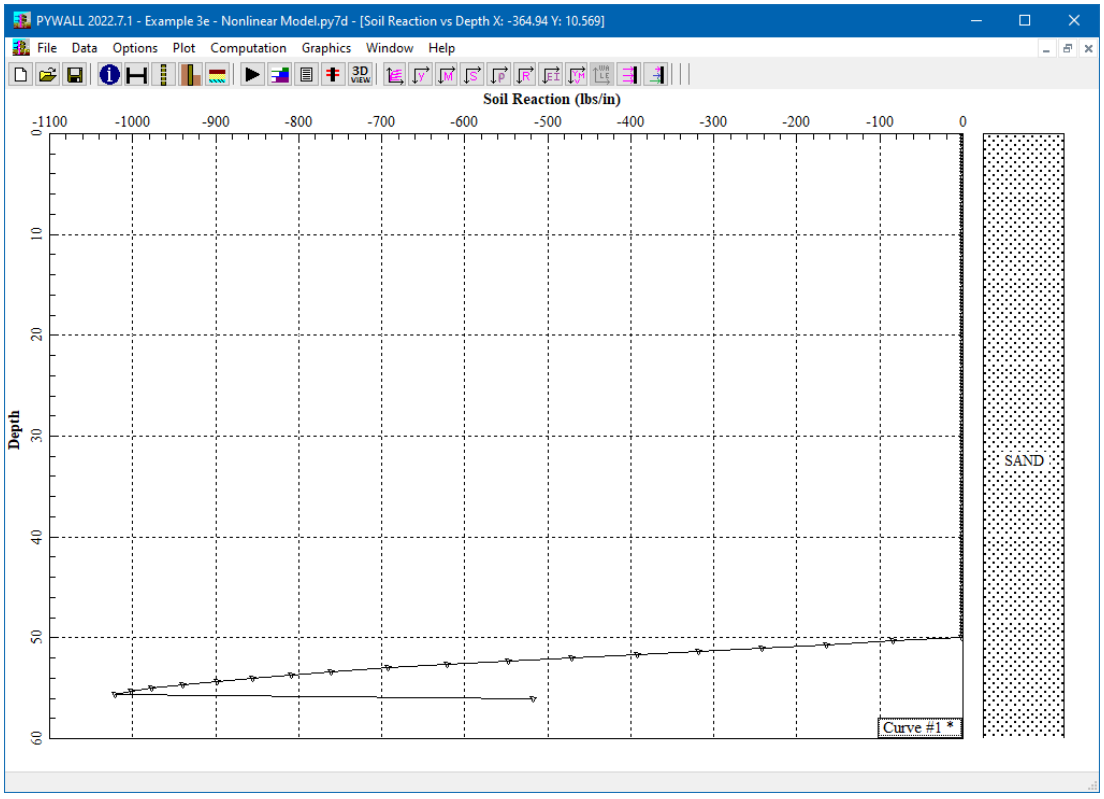


Figure 4.18 Sample Graphics > Soil Reaction vs Depth

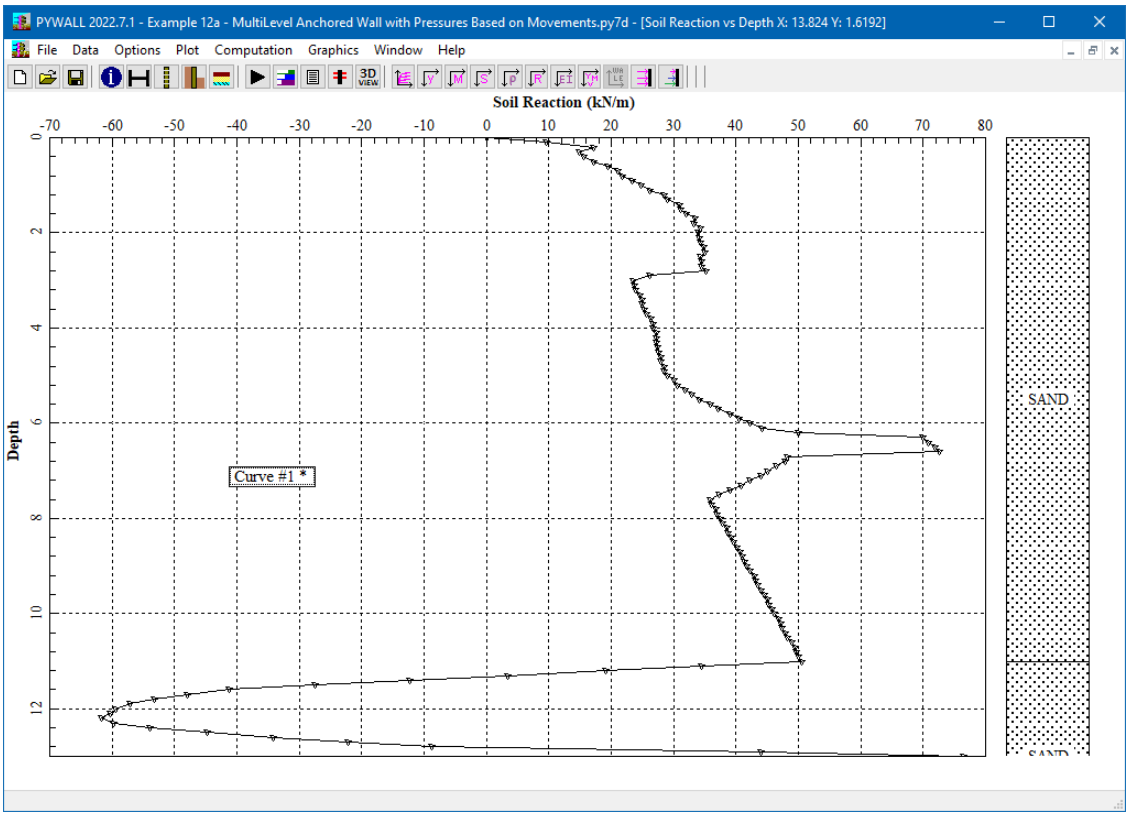


Figure 4.19 Sample Graphics > Soil Reaction vs Depth (Model with Earth Pressures Based on Wall Movements)

4.3.8 Graphics > Net Force vs Depth

This graphics option shows a plot of the net reaction force of the modeled wall versus depth from top of wall. The net reaction force is equal to the force that produces equilibrium at any node. The net forces in this graph are provided per wall unit as defined in Section 3.2.3 or per tieback (if one is present at the observed node).

The curve is automatically generated in all analytical runs of a laterally-loaded wall. The reaction is the sum of all external forces assigned at the nodal point, representing each wall increment. The net force may include the components from active earth pressure, soil resistance and tieback force depending on the location of the nodal point.

The number of points on the reaction curve is equal to the selected number of increments specified on the wall. Several curves may be displayed with this selection if the user inputs several load cases (models with staged construction or with short/long-term analyses). A general sample of this graphics option is shown in Figure 4.20. In this particular example, the tieback load (158.2 kips) at depth of 7 ft can be found by adding the absolute values of negative reaction (153,400 lbs) with the positive load from soil pressure (4,800 lbs). Notice that the tieback load calculated above is the total force per tieback in the horizontal direction so it should be increased if the tieback is placed at an angle from horizontal.

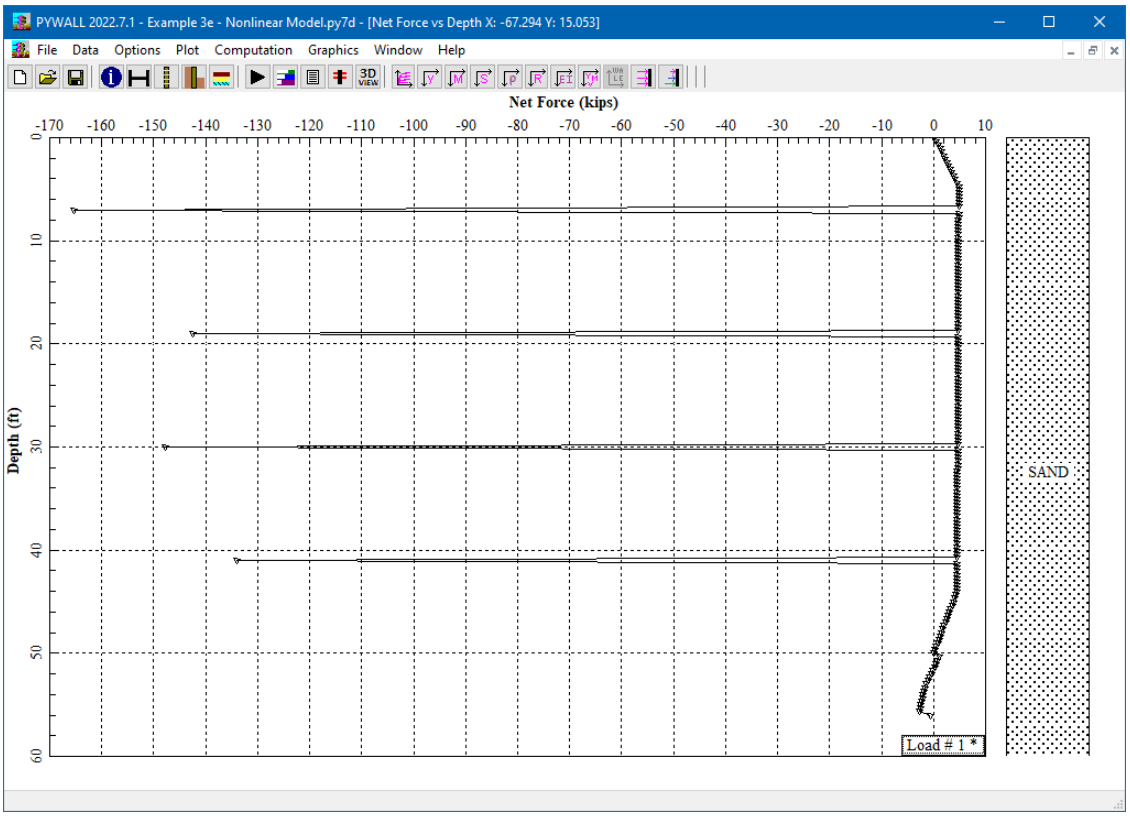


Figure 4.20 Sample Graphics > Net Force vs Depth

4.3.9 Graphics > Deflection, Moment, and Shear

This graphics option shows a combined plot of the deflection, bending moment, and shear of the modeled wall unit versus depth from top of wall. It is a convenient option because engineers are interested in comparison of those response curves together for a quick evaluation.

Several curves may be displayed with this selection if the user inputs several load cases (models with staged construction or with short/long-term analyses). A general sample of this graphics option is shown in Figure 4.21. The values of deflection, bending moment and shear are provided per wall unit as defined in Section 3.2.3.

4.3.10 Graphics > Wales Effect

This graphics option provides a set of plots of Equivalent Distributed Load, Shear and Bending Moment distributions versus Distance on the wales that are inserted in the model (as indicated by their "Depth" label in the legend). The Distance (in ft or m) represents the horizontal distance from one support (tieback or strut/raker) to the next. A general sample of this graphics is shown in Figure 4.22. See Section 4.7 in Technical Manual for computational references.

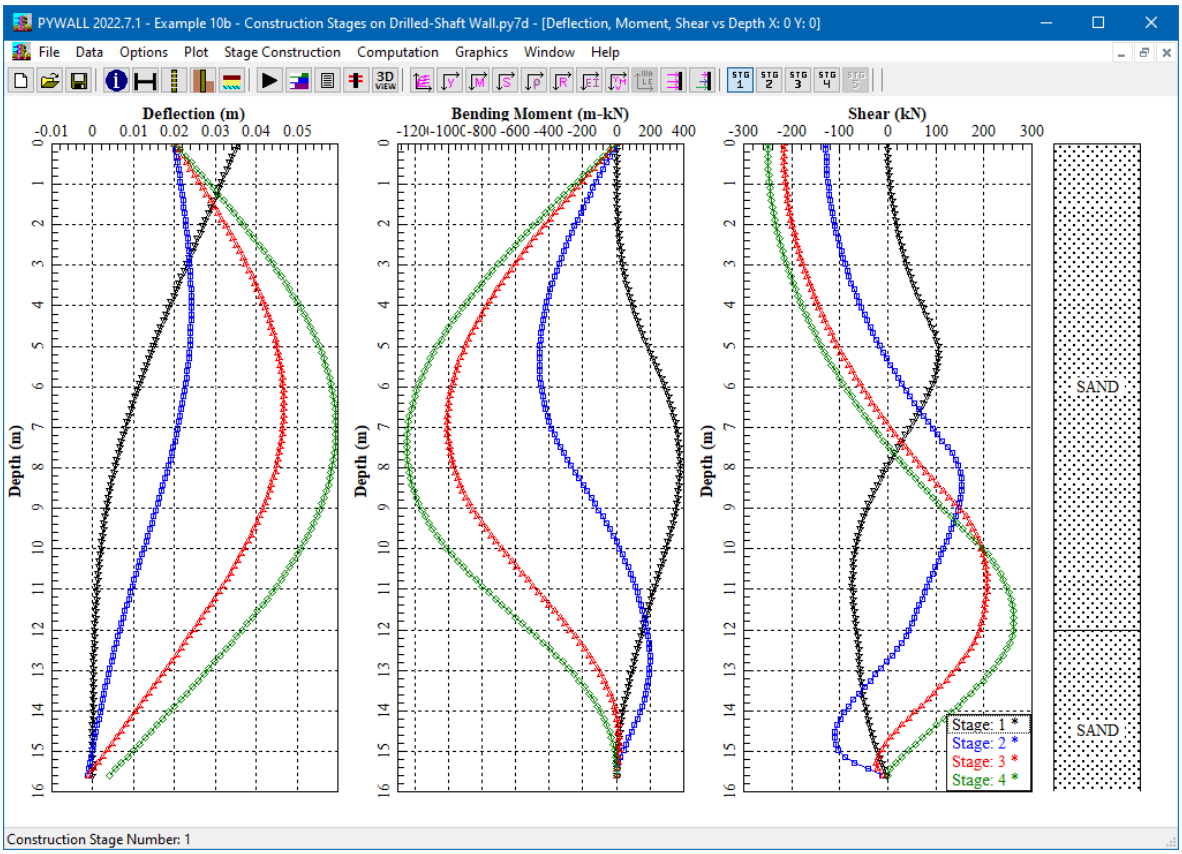


Figure 4.21 Sample Graphics > Deflection, Moment and Shear

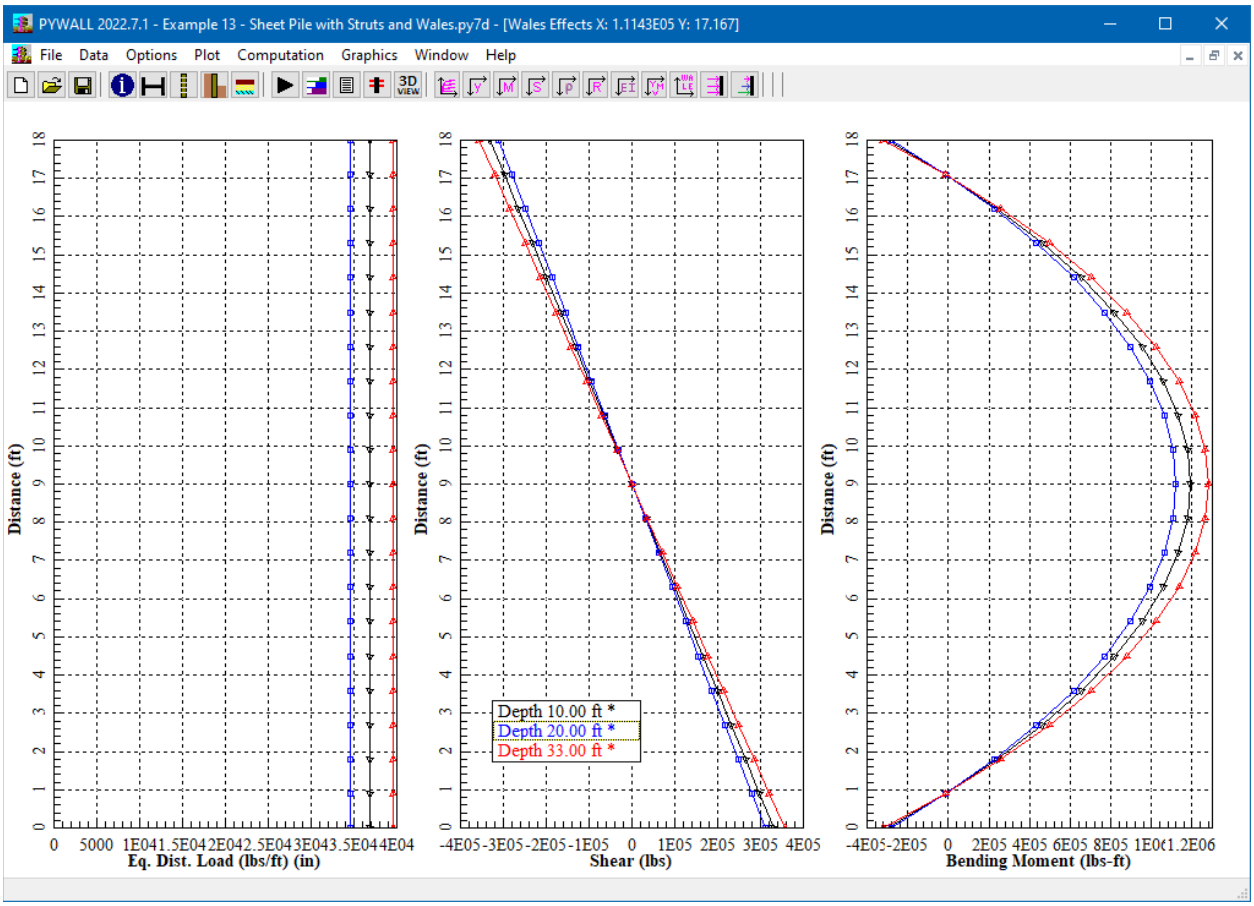


Figure 4.22 Sample Graphics > Wales Effect

4.3.11 Graphics > Earth Pressure Distribution

This graphics option shows a plot of the active earth pressures on the modeled wall unit versus depth from top of wall. The pressures in this graph are provided per wall unit as defined in Section 3.2.6.4. A sample of this graphics is shown in Figure 4.23. Multiple curves are displayed for models with staged construction or in models with short- and long-term analyses. The 'Distributed Load' in the horizontal axis is in units of load per unit height of wall and per wall unit as defined in Section 3.2.3.

If the user asks the program to generate triangular-distribution of active earth pressure, the curve provided in this menu is the active earth pressure developed by the program based on Rankine's equations. If the wall requires to use either single level or multi levels of tiebacks or struts for supporting the wall, the program will convert the active earth pressure curve generated based on Rankine's theory to a trapezoidal earth-pressure distribution based on the recommendations mentioned in the Technical Manual.

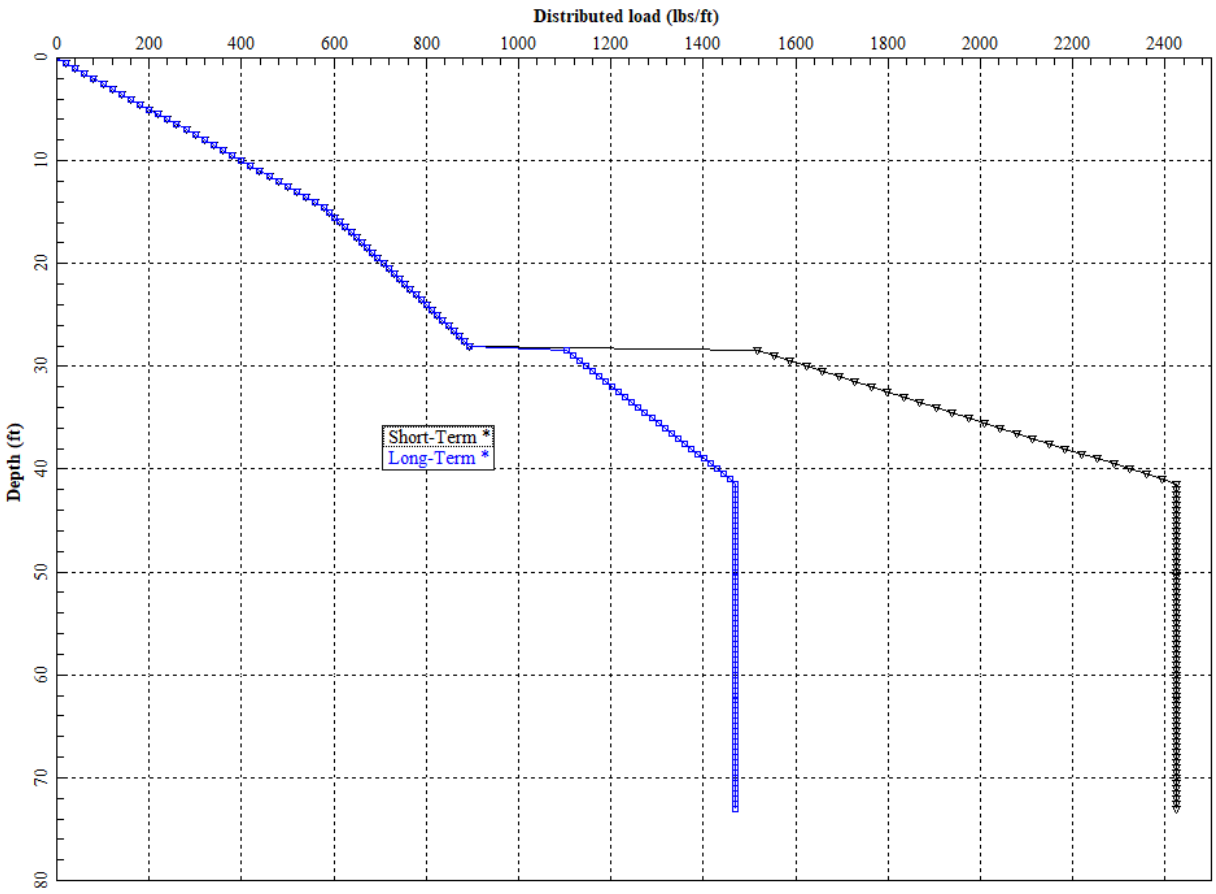


Figure 4.23 Sample Graphics > Earth Pressure Distribution

4.3.12 Graphics > Earth Pressure Combined

This graphics option provides individual plots of distributed load versus depth from top of wall of the various active earth pressures resulting from the soil layers, water and from all the surcharge loads defined on the modeled wall. A general sample graphics of this menu command is shown in Figure 4.24. The 'Distributed Load' in the horizontal axis is in units of load per unit height of wall and per wall unit as defined in Section 3.2.3.

The number of lines and associated legends that are displayed in this chart depends on the details/options that are used in each model. Possible lines/legends, some of which are shown in the sample of Figure 4.24, are the following:

"from Soils due to Effective Stress" (green colored) represents the active earth pressure generated from the soil layers.

"from Water" (blue colored) represents the active earth pressure generated from the water pressures.

"from Uniform Distributed Loads" (red colored) represents the active earth pressure generated by the uniform surcharge pressure specified as Uniform Distributed Load at the top of backfill side under Data > Wall Properties plus any uniform lateral load acting along the wall that was specified as Uniform Lateral Distributed Load along Wall (multiplied by Wa) under Data > Wall Properties.

"Equivalent Trapezoidal Pressure" (olive colored) represents the equivalent trapezoidal distribution for the applicable active pressures.

"from Add. Surch. Loads (Bouss.)" (purple colored) represents the active earth pressure generated by the surcharge loads based on the modified form of the elastic Boussinesq equations.

"Reduction due to Cohesion" (maroon colored) represents the reduction of active earth pressure due to retained cohesive soils (see Section 2.5 'Earth Pressures in Cohesive Soils' of the Technical Manual).

"Total Earth Pressure" (always black colored) represents the summation of all the applicable active pressures. Notice that Total Earth Pressure for trapezoidal distributions is not representing a simple summation of each pressure component but rather a conversion process to trapezoidal diagram (for some components) followed by a summation process.

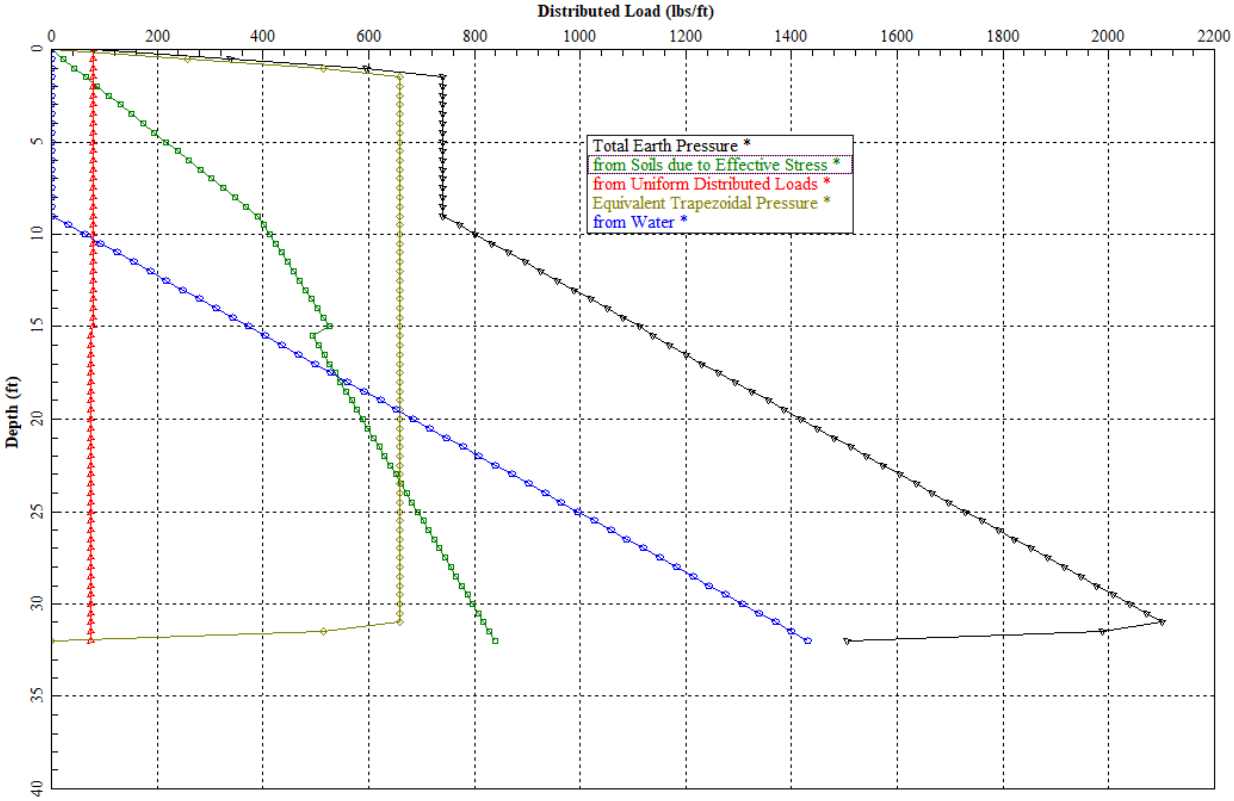


Figure 4.24 Sample Graphics > Earth Pressure Combined

4.3.13 Graphics > Export Plots to Excel

This graphics option allows to export formatted graphical data (such as data of *p-y* curve, deflection, shear, bending moment, soil reaction, and earth pressure) to Microsoft Excel files for convenient usage. A general sample of this export option is shown in Figure 4.25. The user should have the Microsoft Excel software installed in the user's computer in order to have this function to work. The purpose of this option is to provide a convenient tool in case the user would like to format the plots in certain ways for the engineering report.

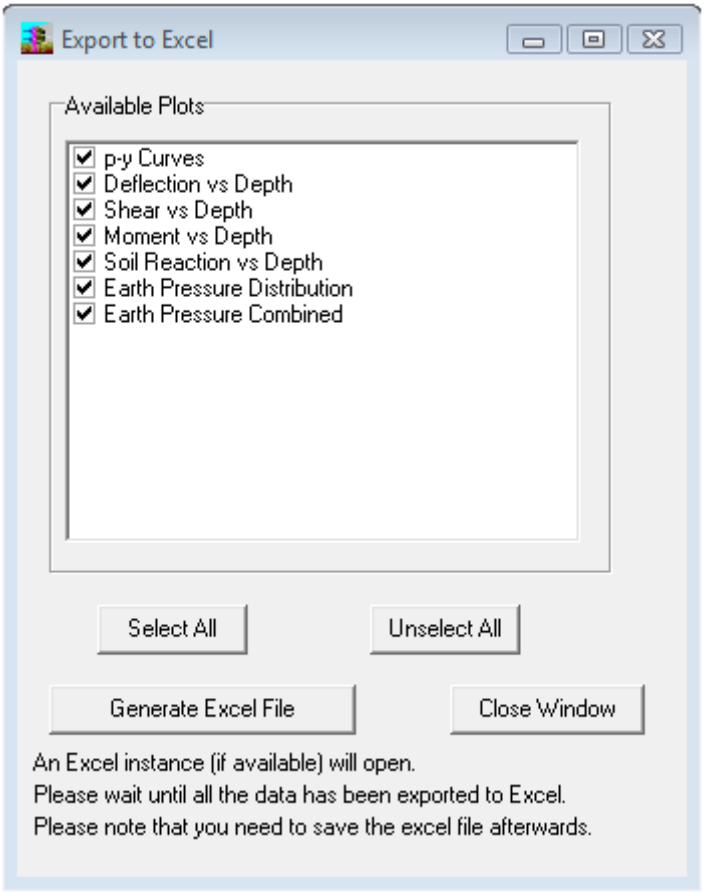


Figure 4.25 Options for "Export Plots to Excel"

4.4 Plot Menu

This menu command only appears in the top-menu bar while in the graphics mode. Command options contained in this menu, shown in Figure 4.26, are explained below.

Show Legend

This activates or deactivates the floating legend included in the graphics mode. The legend box can be moved with click-and-drag operations of the mouse to any position in the graphics screen. Despite its location in the screen, the legend is always located at the right-bottom corner of the printouts.

Show Soil Layer

This selection toggles the display of the soil stratigraphy on a vertical section to the right of the graph. This selection works in conjunction with Wall Geometry explained below.

Show Wall Geometry

This selection toggles the display of the wall geometry on a vertical section to the right of the graph. This selection is only enabled when the Show Soil Layer is also selected/check marked.

Show Markers

This menu provides the option to activate or deactivate the markers for each point of all the curves displayed in the active screen of the graphics mode. By default, the program automatically plots at every node (Every Increment), but the user may optionally select to show markers every 2, 3, 5, or 10 increments (nodes). The type of markers used in each curve is automatically selected by the program. Once activated, the markers are enabled for all visible curves of the active graphics screen.

Font Sizes

This menu provides the option to enlarge the default size of fonts used in plots. The user may optionally select to enlarge the font by 1.5, 2, 2.5, 3, or 4 times of the default size.

Line Width

This menu provides the option to enlarge the default width of lines used in plots. The user may optionally select to enlarge the width by 2, 4, 6, or 10 times of the default width.

Graph Title

The user may optionally add a title to the active graphics using this command. However, the parameters selected here are not saved with the output file and must be changed each time the file is opened.

Edit Legend

The user may change the text of the legend in the active graphics using this command. However, the modified names are not saved with the output file and must be changed each time the file is opened.

Active Curves

This command is used to active or de-active curves that are displayed on the screen.

Print...

This command is used to produce a hard copy of the active graphics screen. The Print sub-windows includes a drop-down menu that allows the user to select any of the installed Windows printers, allows for change of printer properties, and allows the selection of the total number of copies to be printed. The printed graphics is sized according to the margins that the user specifies in the Plot – Page Setup menu.

Page Setup

The user can here specify the printed size of the active-graphics screen. The size of the plot is based on the specified margins and selection of paper orientation. The parameters selected here are not saved with the output file and must be changed each time that the graphics is plotted.

Save to Disk...

This command is used to save the active-graphics screen to disk. The format of the saved file is as *bmp* graphics (bitmap file), which is the internal-file format used in Microsoft Windows©.

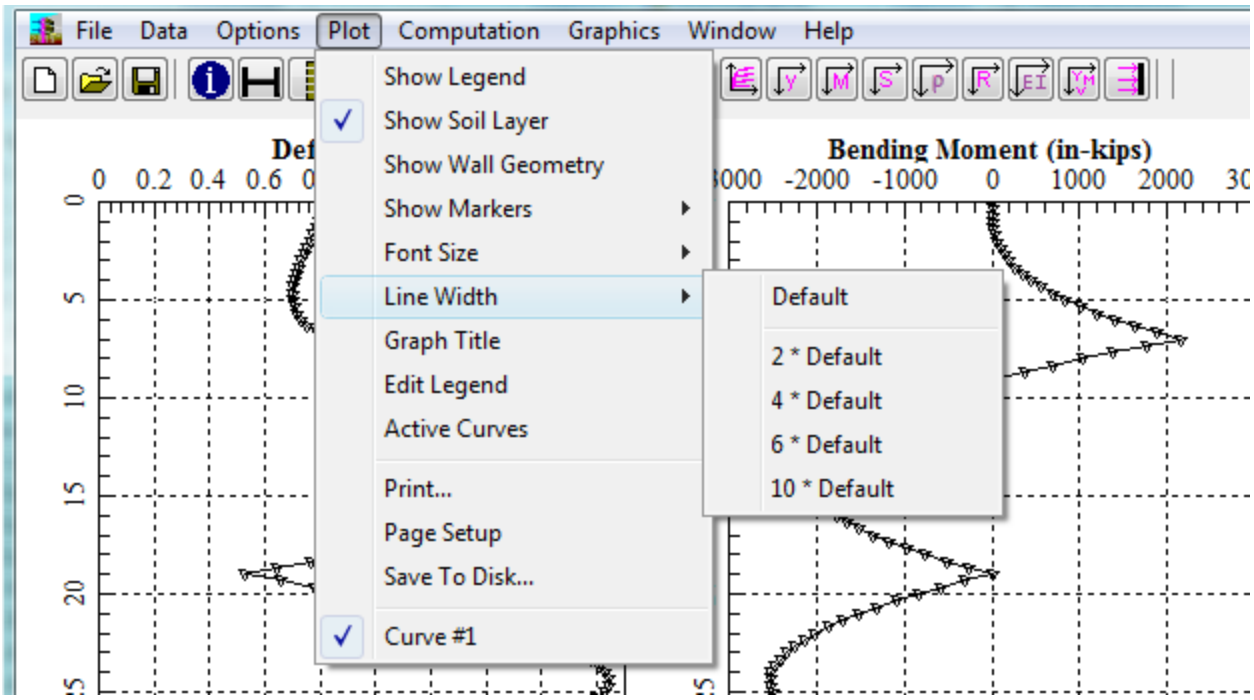


Figure 4.26 Sample Plot Menu

List of Technical References

FHWA-IF-99-015, Ground Anchor and Anchor Systems, Geotechnical Engineering Circular No. 4, Authors: Sabatini, P.J., Pass, D.G., and Bachus, R.C., U.S. Department of Transportation, Federal Highway Administration, Publication No. FHWA-IF-99-015, June 1999.

Haliburton, T. A., "Numerical Analysis of Flexible Retaining Structures," Proceedings Journal of The Soil Mechanics and Foundations Division, ASCE 94 (SM6), 1233-1252, Nov. 1968.

NCHRP Report 611, Seismic Analysis and Design of Retaining Walls, Buried Structures, Slopes and Embankments, Authors: Anderson, D.G. Martin, G.R., Lam, I., and Wang, J.N., Transportation Research Board, 2008.

Peck, R.B., W.E. Hanson and T.H. Thornburn, *Foundation Engineering*, 2nd Edition, Published by: Wiley, New York, 1974.

Reese L.C. and W.F. Van Impe, *Single Piles and Pile Groups Under Lateral Loading*, 2nd Edition, Published by: CRC Press/Balkema/Taylor and Francis Group, The Netherlands, 2011.

Roth, W.H., Dawson, E. M., Nesarajah, S., and Stirbys, A., Design Analysis & Field Verification of Alameda Corridor Retaining Walls, Foundations and Ground Improvement, Geotechnical Special Publication (GSP) 113, Edited by Thomas L. Brandon, ASCE, June 9-13, 2001.

Sabatini, P.J., Pass, D.G., and Bachus, R.C., Ground Anchor and Anchor Systems, Geotechnical Engineering Circular No. 4, U.S. Department of Transportation, Federal Highway Administration, Publication No. FHWA-IF-99-015, June 1999.

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 to the USER (company or individual) whose name is registered with ENSOFT and can only be installed in individual computers of employees of one specific physical and licensed site ("SITE") that is registered with ENSOFT. You may install and use the PRODUCT on any number of computers for employees of the licensed SITE but installations cannot be made for shared access of the PRODUCT over a network. The PRODUCT is fully operational only in the SITE computer that carries the appropriate PRODUCT's hardware security key ("KEY") that is used as software protection device.

1.1.2 Network Licenses

This software is licensed to the USER (company or individual) whose name is registered with ENSOFT and can only be accessed by the employees of the specific physical licensed SITE and within white-listed IP ranges registered with ENSOFT.

The most basic local network license for a SITE allows the USER to white list up to four unique /24 (unique third octets) IPv4 ranges for the employees of the licensed SITE. Broader access can be purchased if a network configuration requires the white listing of additional subnets/ranges for the employees of the licensed SITE.

The USER may install and use the PRODUCT on any computer in the licensed SITE that is within the white-listed IPv4 ranges registered with ENSOFT. Access to the PRODUCT from VPN Routers or Remote Desktop or similar may be implemented but must limit access to employees of the licensed SITE.

This Network License strictly prohibits the PRODUCT to be accessed by employees from office locations that are different than the licensed SITE and/or outside the white-listed IPv4 ranges. Employees from office locations other

than the registered SITE at ENSOFT are required to purchase additional licenses of the PRODUCT, even if the USER (company) name is the same and/or if the additional offices are located in the same city.

1.1.3 Cloud Licenses

LICENSE is made available to USER (company or individual) for a limited subscription time period, not sold. This software is licensed to the USER whose name is registered with ENSOFT and may only be accessed by an individual employee(s) of the USER, hereby referenced as "LICENSE USER", who has been explicitly authorized to use the LICENSE for the specified subscription time period. The LICENSE USER may install and use the PRODUCT on any computer owned or operated by LICENSE USER.

A LICENSE USER is a single individual that is required to register with ENSOFT in order to activate, access, or use the PRODUCT. The LICENSE USER access to the PRODUCT may not be shared with multiple individuals. You are responsible for anyone who obtains, accesses, or uses PRODUCT with your LICENSE and their compliance with the LICENSE agreement.

PRODUCT may require making automatic connections to the Internet intermittently or on a regular basis to validate the LICENSE, and your LICENSE subscription. ENSOFT may disable, terminate, or suspend your LICENSE if (a) your subscription has expired and has not been renewed; (b) you or your LICENSE USER are in breach of LICENSE agreement and fail to address such breaches within 30 days after written notice of the breach; (c) pose a security risk or adversely impact PRODUCT, its infrastructure, systems, or other users; (d) tamper, remove, disable or otherwise limit the effectiveness of any technical or security protections.

2. OTHER RIGHTS AND LIMITATIONS

2.1 EDUCATIONAL VERSION

If the PRODUCT was licensed with educational discounts, the use of the PRODUCT is strictly limited to academic or research activities of educators and/or students. Any other usage voids all the rights under this LICENSE. If in doubt, please contact ENSOFT to determine if you have a PRODUCT version that was licensed for educational purposes.

2.2 LIMITATIONS ON REVERSE ENGINEERING, DECOMPILATION, AND DISASSEMBLY

You may not reverse engineer, decompile, or disassemble the PRODUCT, except and only to the extent that such activity is expressly permitted by applicable law notwithstanding this limitation.

2.3 RENTAL

You may not rent, lease or lend the PRODUCT to third parties.

2.4 TRADEMARKS

This LICENSE does not grant you any rights in connection with any trademarks or service marks of ENSOFT.

2.5 SUPPORT SERVICES

ENSOFT shall provide you with limited support and maintenance services (“SUPPORT”) related to the PRODUCT during the initial year after purchase and for any additional SUPPORT term that the COMPANY elects to purchase for PRODUCT after the initial warranty period expires.

Use of SUPPORT is limited to the ENSOFT policies and programs described in the PRODUCT’s User’s Manual, in documentation on the ENSOFT web site, and/or in other materials provided by ENSOFT.

Any supplemental software code that may be provided to you as part of the SUPPORT or that is downloaded from the ENSOFT web site shall be considered part of the PRODUCT and subject to the terms and conditions of this LICENSE. With respect to technical information you provide to ENSOFT as part of the SUPPORT, ENSOFT may use such information for its business purposes, including for product support, development or advertisements. However, ENSOFT will not utilize such technical information in a form that personally identifies the USER.

2.6 SOFTWARE TRANSFER

The initial user of the PRODUCT may make a permanent transfer of this LICENSE and PRODUCT and only directly to an end user. This transfer must include all of the PRODUCT (including all component parts, the media and printed materials, any upgrades, this LICENSE, and the PRODUCT's appropriate hardware KEY). Such transfer may not be by way of consignment or any other indirect transfer. The transferee of such one-time transfer must agree to comply with the terms of this LICENSE, including the obligation not to further transfer this LICENSE and PRODUCT.

2.7 TERMINATION

Without prejudice to any other rights, ENSOFT may terminate this LICENSE immediately if you fail to comply with the terms and conditions of this LICENSE. In such event, you must destroy all copies of the PRODUCT and all of its component parts and provide evidence of such destruction in writing to ENSOFT within 3 business days.

2.8 INDEMNIFICATION

Within and during the terms of this LICENSE, ENSOFT warrants that the use of the initially provided PRODUCT does not infringe any patent, copyright, or trademark in the United States, and ENSOFT shall indemnify and hold USER harmless against any and all losses, damages and expenses, (including attorney's fees) which USER may sustain or incur as a result of a breach of this warranty.

2.9 CONFIDENTIAL INFORMATION

ENSOFT and the USER may have access to certain proprietary information and materials of the other that are confidential and of substantial value to the respective party, which value would be impaired if such information were disclosed to third parties ("CONFIDENTIAL INFORMATION"). ENSOFT and the USER agree that neither shall disclose any CONFIDENTIAL INFORMATION to any third party and shall take every reasonable precaution to protect CONFIDENTIAL INFORMATION.

The provisions of this section shall not apply to any information which (i) is or becomes available to the public other than by breach of the LICENSE agreement by the receiving party, (ii) is rightfully received by receiving party from a third party without confidential limitations, (iii) is independently developed by receiving party's employees without access to CONFIDENTIAL INFORMATION, or (iv) is known to the receiving party without any restriction on its use or disclosure prior to first receipt of it from the disclosing party.

3. COPYRIGHT

All title and intellectual property rights in and to the PRODUCT (including but not limited to any images, photographs, animations, video, audio, music, and text that may be incorporated into the PRODUCT), the accompanying printed materials, and any copies of the PRODUCT are owned by ENSOFT. All title and intellectual property rights in and to the content which may be accessed through use of the PRODUCT is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This LICENSE grants you no rights to use such content. All rights not expressly granted are reserved by ENSOFT. This PRODUCT is protected by the United States Copyright Law and International Copyright Treaty.

4. SOFTWARE DISCLAIMER

Although the PRODUCT has been used with apparent success in many analyses, new information is developed continuously and new or updated PRODUCT releases may be written from time to time. All users are requested to inform ENSOFT immediately if any errors are found in the PRODUCT. As modifications, updates, or new versions are produced, the latest codes are posted on ENSOFT's web site and made available to all visitors for free downloading.

No warranty, expressed or implied, is offered as to the accuracy of results from ENSOFT's PRODUCT. The PRODUCT should not be used for design unless caution is exercised in interpreting the results and independent

calculations are available to verify the general correctness of the results. Users are assumed to be knowledgeable of the information in the printed documentation that is distributed with the digital media. Users are assumed to recognize that the input parameters, 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.