Oasys

YOUR IDEAS BROUGHT TO LIFE

Xdisp

Alp

Help Guide

Oasys
YOUR IDEAS BROUGHT TO LIFE

13 Fitzroy Street
London
W1T 4BQ
Telephone: +44 (0) 20 7755 3302
Facsimile: +44 (0) 20 7755 3720

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: http://www.oasys-software.com/

Alp

Printed: February 2020

Table of Contents

Part I About Alp 8

1 General Program Description ... 8
2 Program Features ... 8
  Sample Files ... 9
3 Components of the User Interface ... 10
4 Preferences ... 10

Part II Step by Step Guide 14

Part III Step by Step Example 18

1 Creating the File and Setting the Units ... 18
2 General Data Information ... 19
3 Convergence Data ... 20
4 Partial Factors ... 21
5 Soil Data ... 22
6 Sections ... 22
7 Pile Properties ... 23
8 Groundwater ... 24
9 Applied Loads and Displacements ... 25
10 Restraints ... 25
11 Surcharges ... 26
12 Analysis and Results ... 26

Part IV Method of Analysis 32

1 General ... 32
  Elastic-Plastic Soil Model ... 33
  P-Y Soil Model ... 35
  Pile Stiffness ... 44
  Applied Lateral Loads and Displacements ... 45
  Restraints ... 45
  Partial Factors ... 45
  Pushover Analysis ... 46
  Modification Factors ... 47
  Moment vs Curvature ... 47

Part V Input Data 50

1 Assembling Data ... 50
  Opening the program ... 50
  Titles ... 55
2 Input Explorer ... 56

3 Data Input ... 57
Units ... 57
General Data ... 59
Convergence Control Parameters ... 61
Partial Factors ... 62
Soil Data ... 63
Modification Factors ... 66
Moment Curvature ... 67
Sections ... 68
Pile Properties/Node Levels ... 76
Groundwater ... 78
Pushover Data ... 79
Applied Loads and Soil Displacements ... 80
Restraints ... 81
Surcharges ... 82

Part VI Output 84

1 Output Explorer ... 84
2 Analysis and Data Checking ... 84
3 Text Output ... 85
4 Graphical Output ... 88
Set Exact Scale ... 89
5 Charts ... 89
6 Toolbars ... 90
Standard Toolbar ... 90
Alp Toolbar ... 90
Graphics Toolbar ... 91

Part VII Programming Interface 94

1 COM Automation ... 94
COM Export Functions ... 94

Part VIII List of References 102

1 References ... 102

Part IX Alp 106

1 Brief Technical Description ... 106

Part I

1 About Alp

1.1 General Program Description

Alp (Analysis of Laterally Loaded Piles) is a program that predicts the pressures, horizontal movements, shear forces and bending moments induced in a pile when subjected to lateral loads, bending moments and imposed soil displacements.

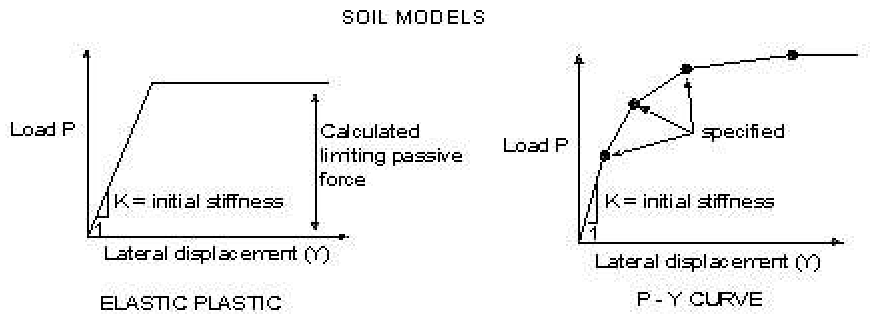
The pile is modelled as a series of elastic beam elements. The soil is modelled as a series of non interactive, non-linear "Winkler type" springs. The soil load-deflection behaviour can be modelled either assuming an Elastic-Plastic behaviour, or by specifying or generating load-deflection (i.e. P-Y) data. Two stiffness matrices relating nodal forces to displacements are developed. One represents the pile in bending and the other represents the soil.

1.2 Program Features

The main features of the problem analysed by **Alp** are summarised below and represented diagrammatically.

- The **geometry** of the pile is specified by a number of nodes, which may be specified directly by the user or generated automatically based on the elevation of soil boundaries, loads, restraints and displacements.
- The positions of these nodes are expressed in terms of reduced level. Pile stiffness is constant between nodes, but may change at nodes. Three methods of modelling the **soil** are available.

1. Elastic-Plastic
2. Specified P-Y curves
3. Generated P-Y curves

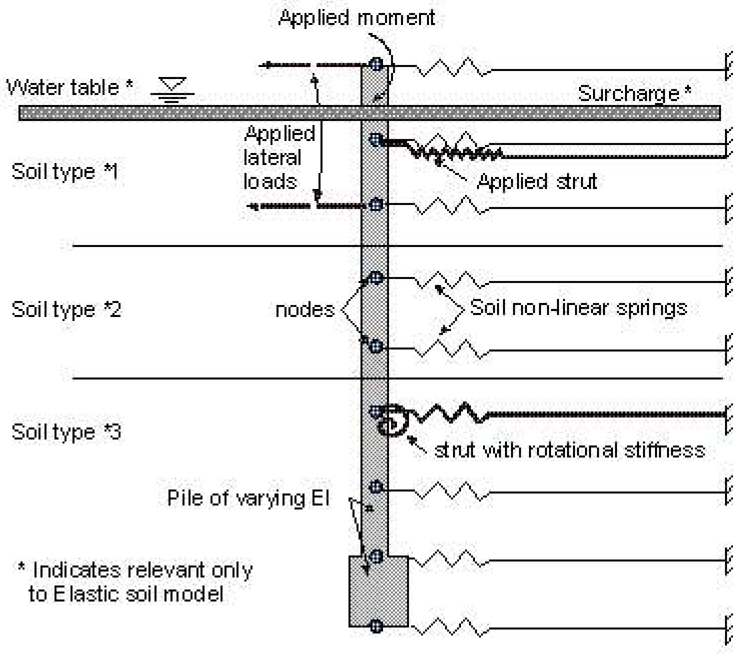


Elastic-Plastic

The program generates the load deflection behaviour at each node from the soil data, see Elastic-Plastic soil model. Soil strata are horizontal and their boundaries occur midway between node levels.

P-Y

The user may **specify** the load deflection behaviour directly in the form of specified load-deflection points (P-Y). The program linearly interpolates between these points. If P-Y data is to be **generated**, then the appropriate method for Generated P-Y Curves is used for each soil.



- **Applied lateral loads, bending moments and soil displacements** are applied at nodes. To obtain the deflection behaviour of the pile, an option is provided which enables these loads and/ or soil displacements to be applied in a specified number of increments.
- **Restraints** acting at nodes with both an axial and rotational stiffness can be specified.
- **Water pressures** can be specified and may be either 'hydrostatic' or 'piezometric' (i.e. non-hydrostatic). This option is not applicable to the specified P-Y curves.
- **Surcharges** may be specified at any level. This option is not applicable to the specified P-Y curves.
- **Partial Factors** may be applied to both loads and soil strength parameters.

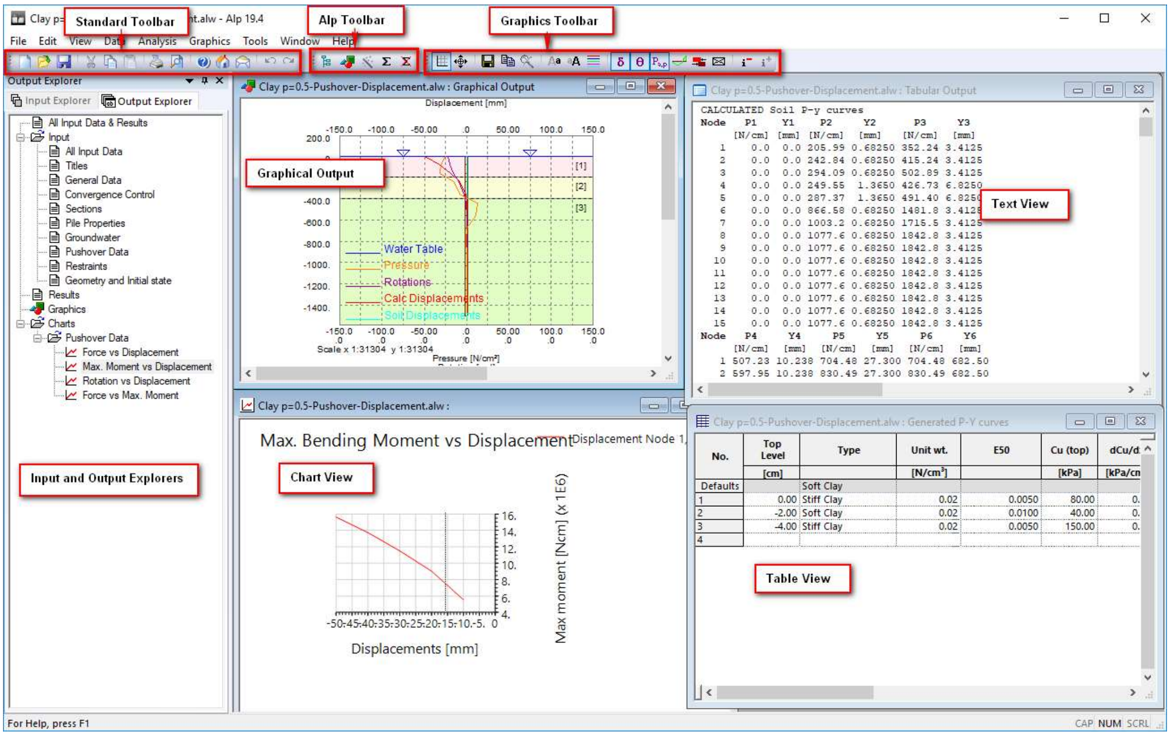
1.2.1 Sample Files

Sample files are provided during the installation process. These demonstrate **Alp**'s features. By default they are installed in the folder 'C:\Program Files\Oasys\Alp n\Samples', where n indicates the version of the program. These files may be opened and inspected in **Alp** in order to become familiar with the typical input data that is required to create an **Alp** model.

File Name	**Brief Description**
Alp_Elastic-plastic.alw	Example file having Elastic-plastic soil model
Alp_Specified_P-Y.alw	Example file having Specified P-y Curves soil model
Alp_Generated_P-Y.alw	Example file having Generated P-y Curves soil model

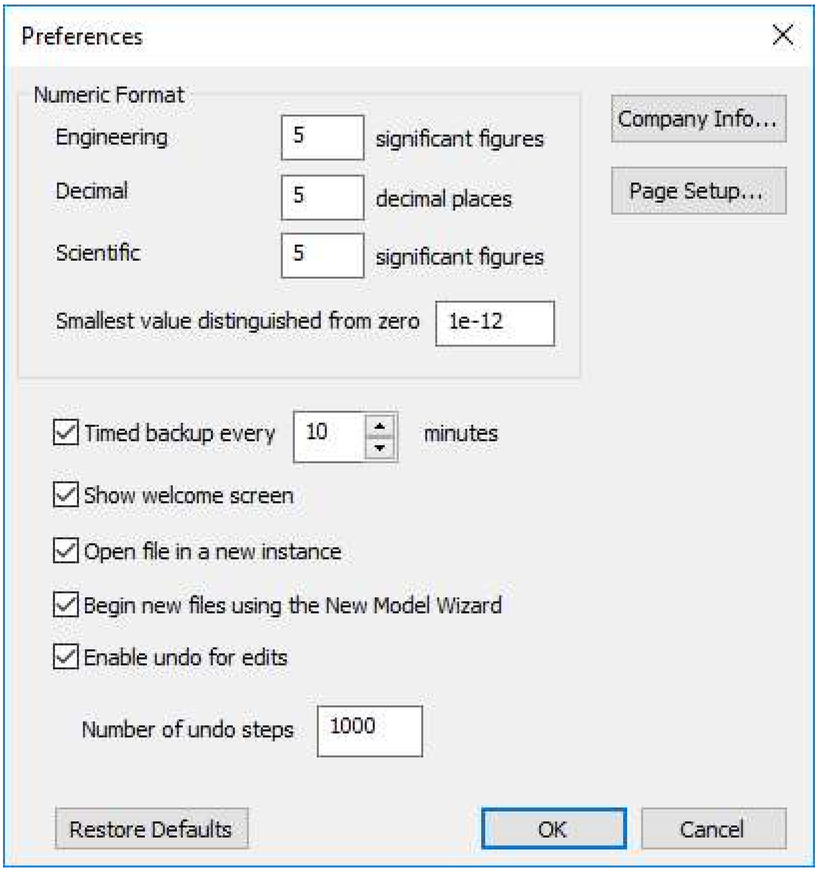
1.3 Components of the User Interface

The principal components of Alp's user interface are the Gateway (split into Input and Output Explorer tabs), Table Views, Graphical Output, Tabular Output, toolbars, menus and input dialogs. Some of these are illustrated below.



1.4 Preferences

The Preferences dialog is accessible by choosing Tools | Preferences from the program's menu. It allows user to modify settings such as numeric format for output, show welcome screen, option for new model wizard, print parameters and company information. These choices are stored in the computer's registry and are therefore associated with the program rather than the data file. All data files will adopt the same choices.



Numeric Format controls the output of numerical data in the Tabular Output. The Text Output presents input data and results in a variety of numeric formats, the format being selected to suit the data. Engineering, Decimal, and Scientific formats are supported. The numbers of significant figures or decimal places, and the smallest value distinguished from zero, may be set by the user.

Restore Defaults resets the Numeric Format specifications to program defaults.

A time interval may be set to save data files automatically. Automatic saving can be disabled if required by clearing the "**Save file..**" check box.

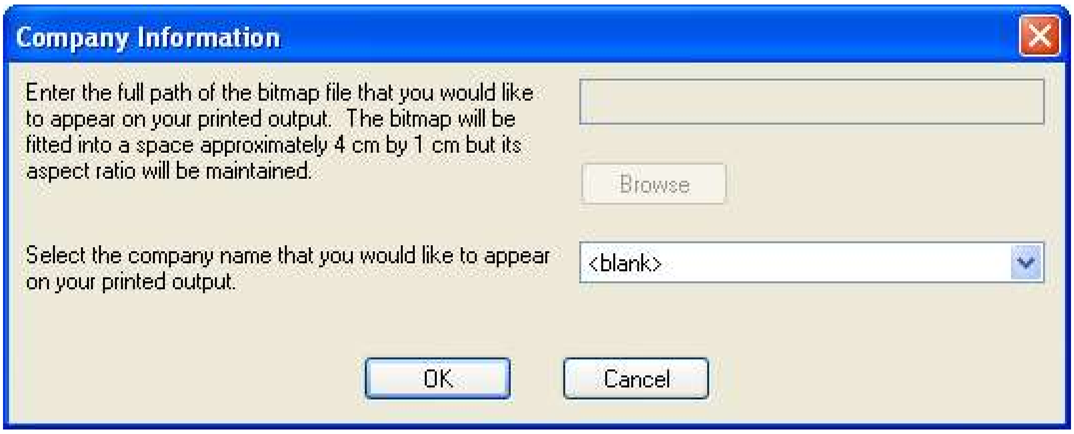
Open file in a new instance It gives an option to the user to open an existing file or to create a new file in a new instance of Alp.

Show Welcome Screen enables or disables the display of the Welcome Screen. The Welcome Screen will appear on program start-up, and give the option for the user to create a new file, to open an existing file by browsing, or to open a recently used file.

Begin new files using New Model Wizard It gives an option to the user to create a new file using new model wizard. For more details click here.

Company info The company information button in the preferences dialog box allows external companies to specify the bitmap and Company name that they would like to appear the top of the printed output.

Enable undo for edits enables the undo/redo option for changes made in table views.



To add a bitmap enter the full path of the file. The bitmap will appear fitted into a space approximately 4cm by 1cm. The aspect ratio will be maintained.
Note! For internal Arup versions of the program the bitmap option is not available.

Page Setup Opens the Page Setup dialog allowing the style of output for printed text and graphics to be selected.
If 'Calculation Sheet Layout´ is selected the page is formatted as a calculation sheet with details inserted in the page header. If `Logo´ is selected the company logo is inserted in the top left corner of the page. If `Border´ is selected this gives a border but no header information. If `Clipped´ is selected the output is clipped leaving a space for the logo. This has no effect on text output.

Part II

2 Step by Step Guide

The following provides a comprehensive guide through the menu options to help new users to the **Alp**. The requirements for data input are listed and linked to relevant sections of the main manual.

Please read the Data Input sections before attempting to create a new file.

Follow the New Model Wizard options to create the data file to ensure that the basic settings for a model are correct before any data is generated, and to ensure that sufficient data has been supplied in order to perform an analysis with minimum input data.

If not using New Model Wizard, then follow the below steps to create the data file.

No	Operation	Link	From Program Menu	From Input/ Output Explorer (Double click on below item)
1	On the Start-up screen select the option to "Create a new data file".	Opening the Program	File | New	--
2	Add the general file information into the Titles view.	Titles	Data | Titles	Input | Titles
3	Select the required Units for data entry and presentation of the calculations using the Global Data | Units option. The frequency of automatic file saving can also be set here.	Units	Data | Units	Input | Units
4	Select the soil model, load, displacement increment,number of increments and load cases.	General Data	Data | General Data	Input | General Data
5	Where applicable, enter partial factor set to be used in analysis.	Partial Factors	Data | Partial Factors	Input | Partial Factors
6	Enter data for sections to be used.	Sections	Data | Sections	Input | Sections
7	Enter node levels and pile properties.	Pile Properties	Data | Node Levels	Input | Node levels
8	Enter soil data for the selected type of soil model.	Soil Data	Data | Soil Data	Input | Soil data
9	Enter the ground water data.(if any)	Groundwater	Data | Groundwater	Input | Groundwater
10	Enter applied loads and soil displacements at any node.	Applied Loads and Soil Displacements	Data | App. Loads and Disp.	Input | Applied loads and displacements.
11	Enter restraints for the nodes.	Restraints	Data | Restraints	Input | Restraints
12	Enter surcharges (if any).	Surcharges	Data | Surcharges	Input | Surcharges
13	Enter control data for convergence.	Convergence Control Parameters	Data | Convergence Control	Input | Convergence control
14	Now Analyse the data, warning/ error messages are shown when the data is not consistent.	Analysis	Analysis | Analyse	--
15	After analysis results can be viewed by double clicking the Results leaf in the Output Explorer.	Text Output	View | Tabular Output	Output | Results
16	The graphical output shows a graphical representation of the pile and it's cross-section and the displacement, pressure,bending	Graphical Output	View | Graphical Output	Output | Graphics

moment and shear force diagrams.

Part III

3 Step by Step Example

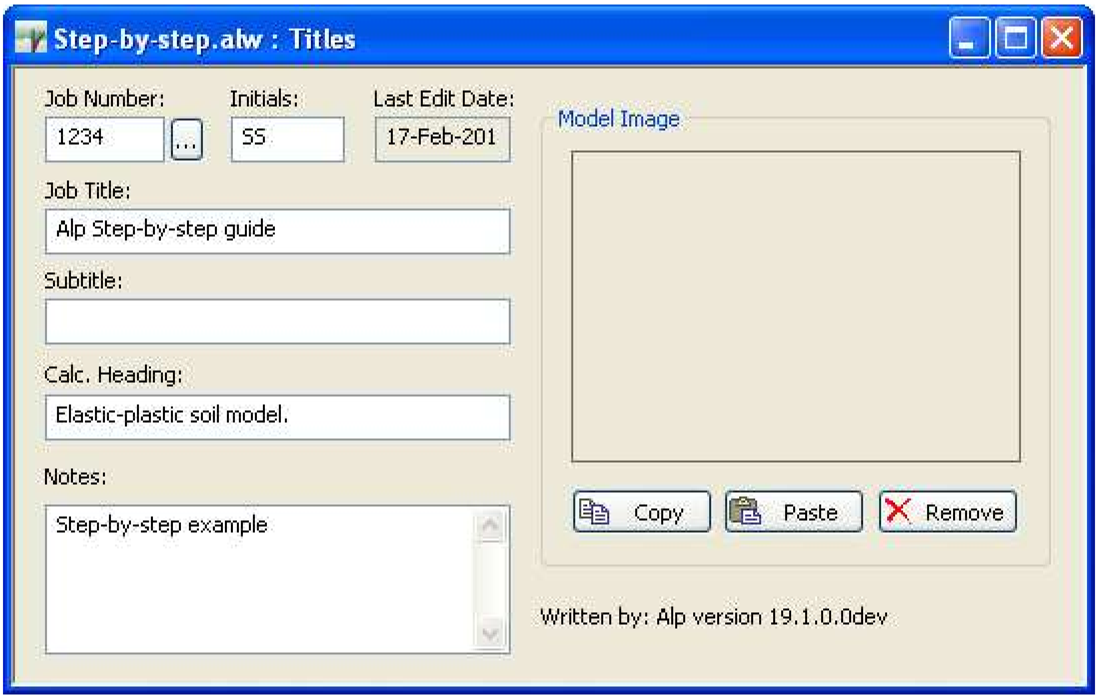
This guide provides a beginner's introduction to the use of Alp for the analysis of laterally loaded piles.

A sample file is created and analysis is performed. The various stages of its data input are illustrated.

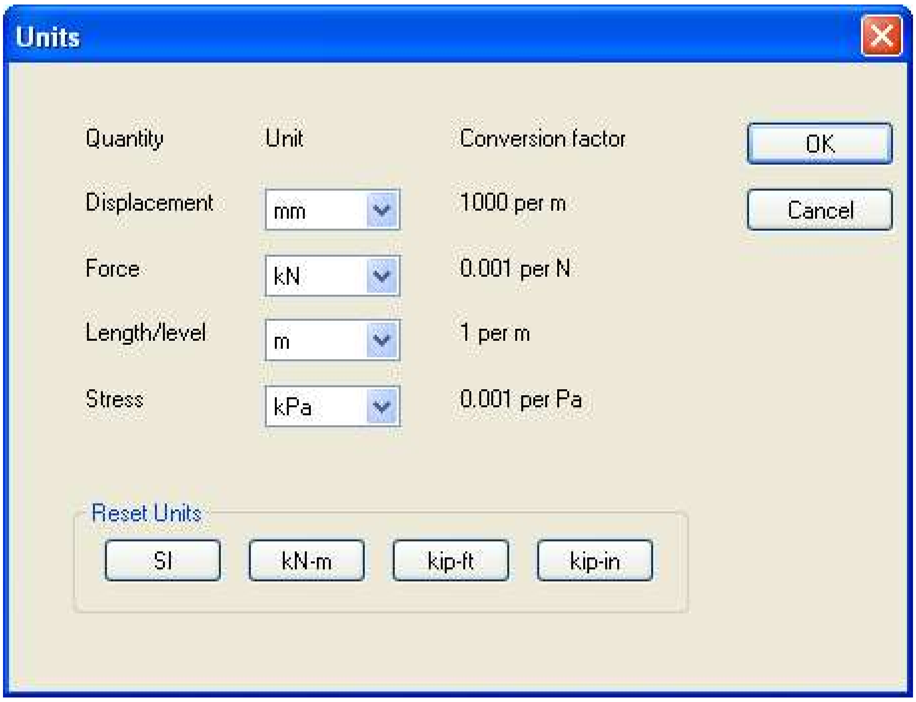
3.1 Creating the File and Setting the Units

Open Alp and create a new model by clicking the “New” button, or click “File” on the program's menu and select “New”. The main screen will open and the program automatically creates an "empty" set of data.

All the project identification data should be entered in the Titles dialog. The data entered here is reproduced in the title block at the head of all printed information for the calculations, so it is useful to provide enough details about the project.



The user can specify the units for entering the data and reporting the results via the Units dialog. This dialog can be accessed via Data | Units and via the Input Explorer. The user can set each base unit individually or switch between various standard sets of units. The standard sets provided are SI, kN-m, kip-ft and kip-in. For this example kN-m is used.



Note : If "Begin new files using the New Model Wizard" option is checked in the Preferences dialog (accessible via Tools | Preferences), the New Model wizard will open when new file is created. The New Model Wizard contains only basic data which is required for creation of a file. The New Model Wizard is not used in this example.

3.2 General Data Information

All general data, i.e. the type of soil model selected, the load case and the number of load and displacement increments are input via the “General Data” dialog. This dialog is accessible via Data | General Data from the program's menu or from the Input Explorer.

There are three soil models available. The Soil Data table view displays the input parameters that are appropriate to the type of soil model that is chosen.

“Elastic-plastic” is the soil model that is most commonly used, and is the soil model used in this example.Other sample files are available illustrating the use of other soil models.
“Factor on soil E value” is a reduction factor on the Young’s Modulus of the soil. The default value is 0.8. This default is used for this example.

The “Number of Increments” indicates the number of steps in which the specified load and/or displacement is applied. If the user is only interested in the results after the whole load and/or displacement has been applied, this can be left at the default value of 1.

The user must then select whether it is the load to be applied in increments and/or the soil deflection to be increased in increments, using the radio buttons. In this example the default selection, (Loads Only) is used.

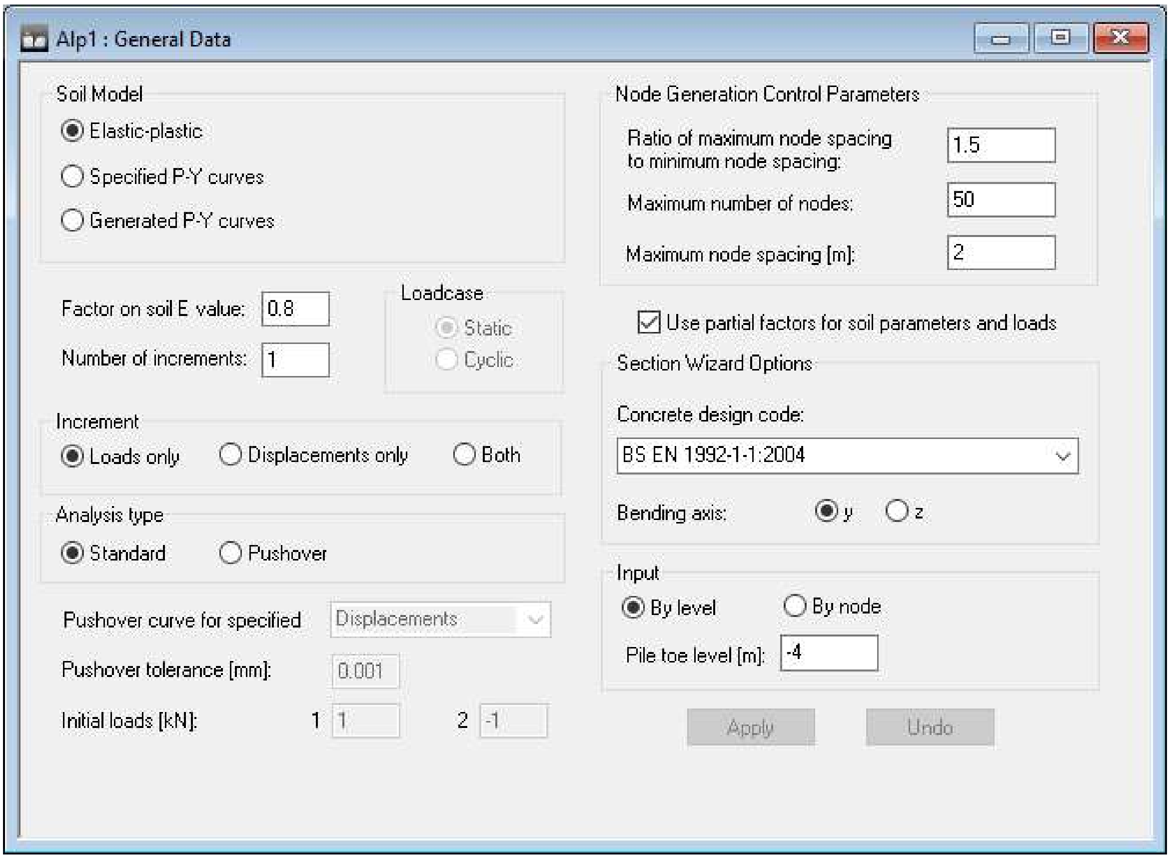
Note: If only loads or displacements are incremented then the others act to the full value throughout the analysis.

The "Input" option allow the user to choose between inputting data by level or by node. Using the level based option the user defines the level of the loads, soil boundaries, displacements and restraints, and Alp then generates a set of nodes to be used in the final analysis. Using the node based option to user explicitly specifies the level of each node to be used in the analysis. If selecting input by level, the user must set the pile toe level using the text box provided here. For this example set the toe level to -4m.

"Node Generation Control Parameters" can be used to ensure limit both the total number of nodes and the maximum variation in node spacings.

The user can use the check-box to activate partial factors. Checking the box will make the partial factors dialog visible in the gateway, and also shows options for partial factors in the soil (drained and undrained) and load tables (to differentiate between permanent and variable loads, and favorable and unfavorable loads).

The "Section Wizard Options" allow the user to set the concrete design code (this dictates the available options for concrete mix, and associated Young's Modulus) and the bending axis, where the user is using the wizard to generate pile width and EI values.

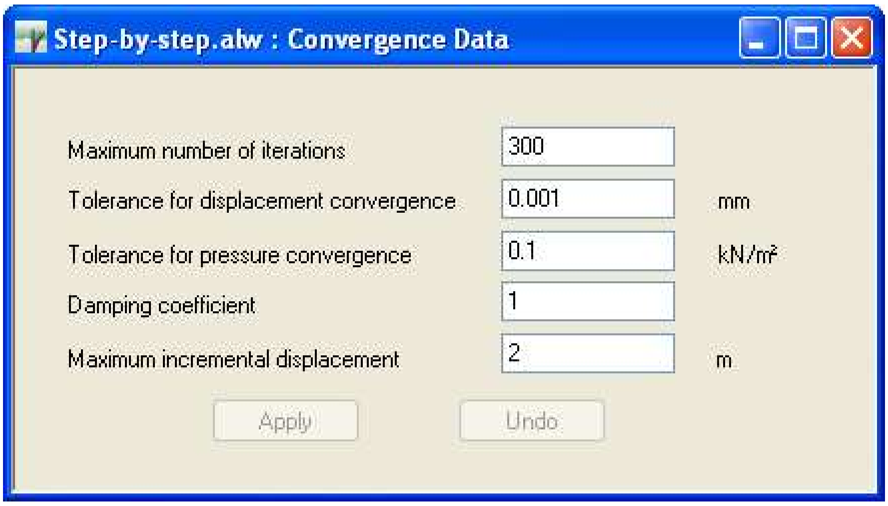


3.3 Convergence Data

Default values are provided by the program for the convergence parameters. The user can modify them.

This dialog can be accessed via Data | Convergence control or via the Input Explorer.

For this example the default values are used.

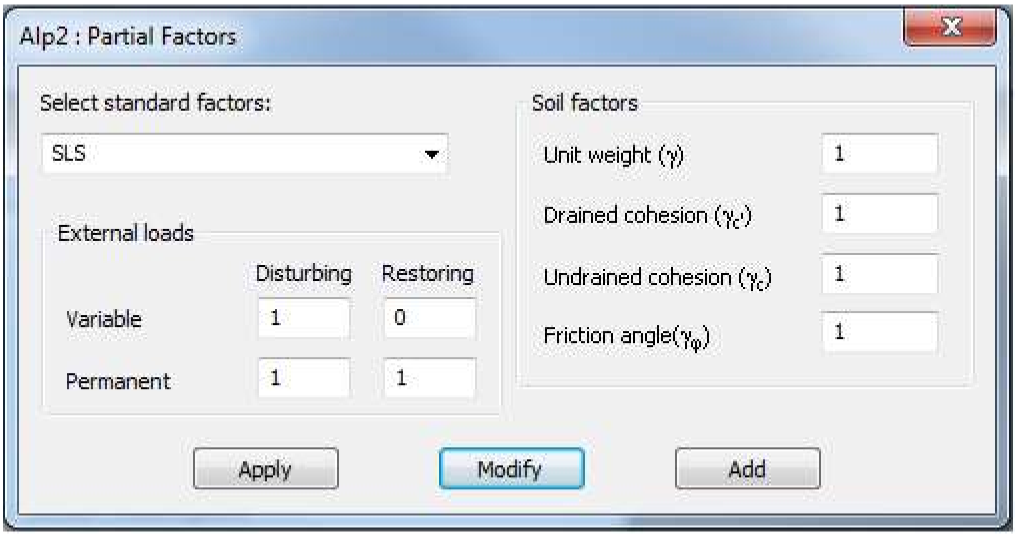


3.4 Partial Factors

The partial factor dialog box can be used to apply partial factors to soil strength and loads. Three standard partial factor sets are included as a default (SLS, BS EN 1997-1:2004 DA1 C1 and DA1 C2) these can be selected but not modified by the user. Additional partial factor sets can be created by the user by typing the name into the drop down box, inputting factors into the relevant text boxes (input 1 where no factor is required) and then clicking "Add". User specified partial factor sets can be modified by selecting the partial factor set to be modified using the drop down box, amending the name and factors as required then clicking on "Modify". Partial factor sets created or modified by the user are saved by the application, and will be available when creating or editing other models in the future.

To apply a partial factor set to the model, select the required set using the drop down box and click "Apply".

Note: If changing the name and/or factors it is necessary to click on the "Modify" or "Add" buttons to save the new partial factor set before clicking "Apply" to use the set in the model.

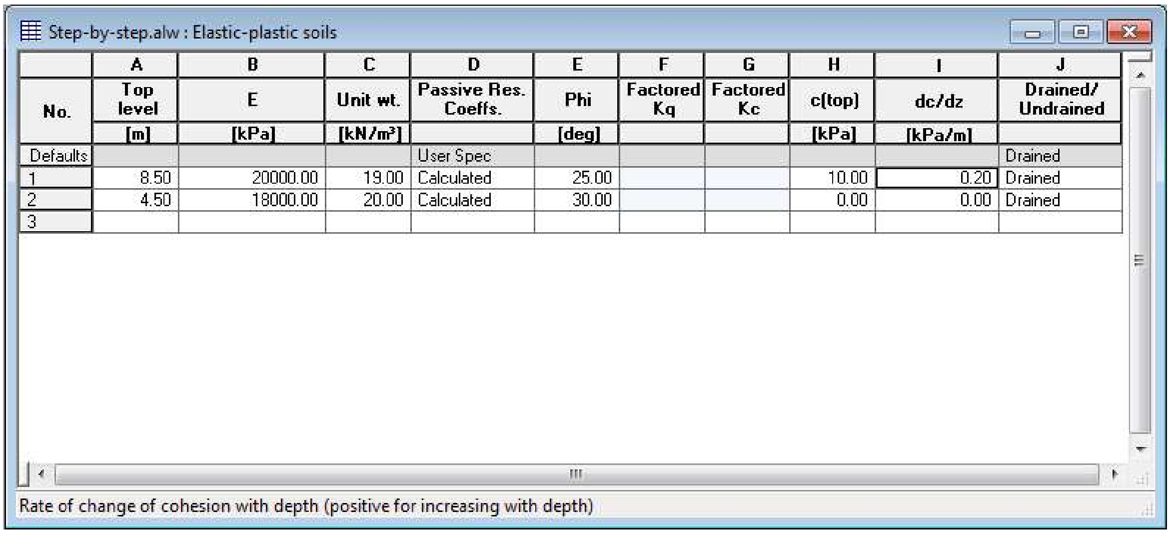


3.5 Soil Data

The parameters in this table view are governed by the soil model selected in the General Data dialog. This is accessible via Data | General Data or via the Input Explorer.

The example considered here has two different soil layers, the first layer runs from 8.5m to 4.5m and the second layer runs from 4.5m to depth. For both these layers the program is requested to calculate the Kq and Kc values; Hence the "Calculated" option is selected for "Passive Res Coeffs". If the user wants to enter the Kq and Kc values then the option to be selected is "User spec".

Note: if the user has selected input by node, then Top Node will be shown in column A. The soil surface is assumed to be midway between the top node given in the table, and the node immediately above. Where the user specifies the uppermost node as the top node, the ground surface is assumed to be level with the node.



Step-by-step.alw : Elastic-plastic soils

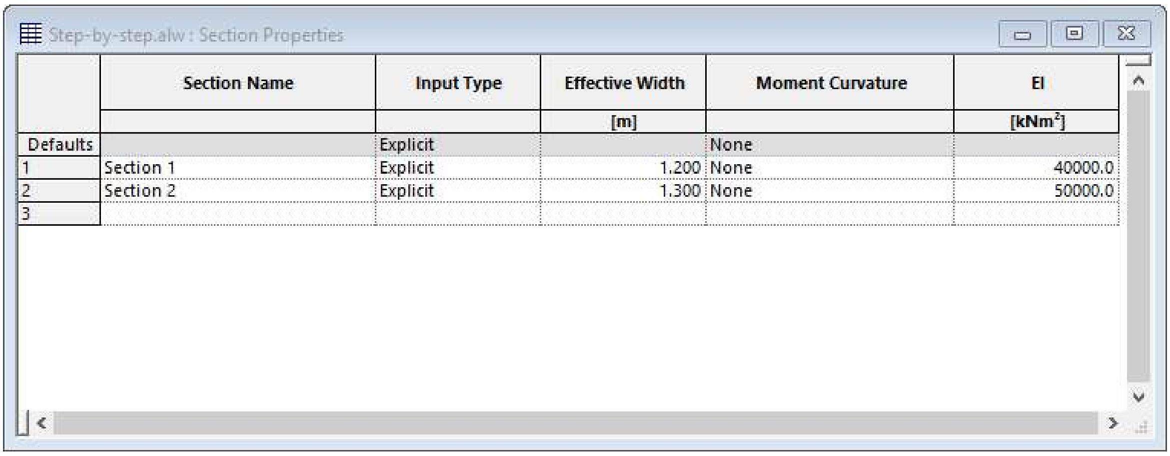
	A	B	C	D	E	F	G	H	I	J
No.	Top level	E	Unit wt.	Passive Res. Coeffs.	Phi	Factored Kq	Factored Kc	c(top)	dc/dz	Drained/ Undrained
	[m]	[kPa]	[kN/m³]		[deg]			[kPa]	[kPa/m]	
Defaults				User Spec						Drained
1	8.50	20000.00	19.00	Calculated	25.00			10.00	0.20	Drained
2	4.50	18000.00	20.00	Calculated	30.00			0.00	0.00	Drained
3										

Rate of change of cohesion with depth (positive for increasing with depth)

3.6 Sections

Sections to be used in the pile are specified in the "Section Properties" table. For each section that the user wishes to create they must type in a section name. The input type is then selected using the drop down list, "Explicit" allows the user to directly specify the pile width and EI values to be used in the model, "Generated" opens up a section wizard that calculates the width and EI values.

Two sections are used in this example, as shown in the figure.



	Section Name	Input Type	Effective Width	Moment Curvature	EI
			[m]		[kNm^2]
Defaults		Explicit		None	
1	Section 1	Explicit	1.200	None	40000.0
2	Section 2	Explicit	1.300	None	50000.0
3					

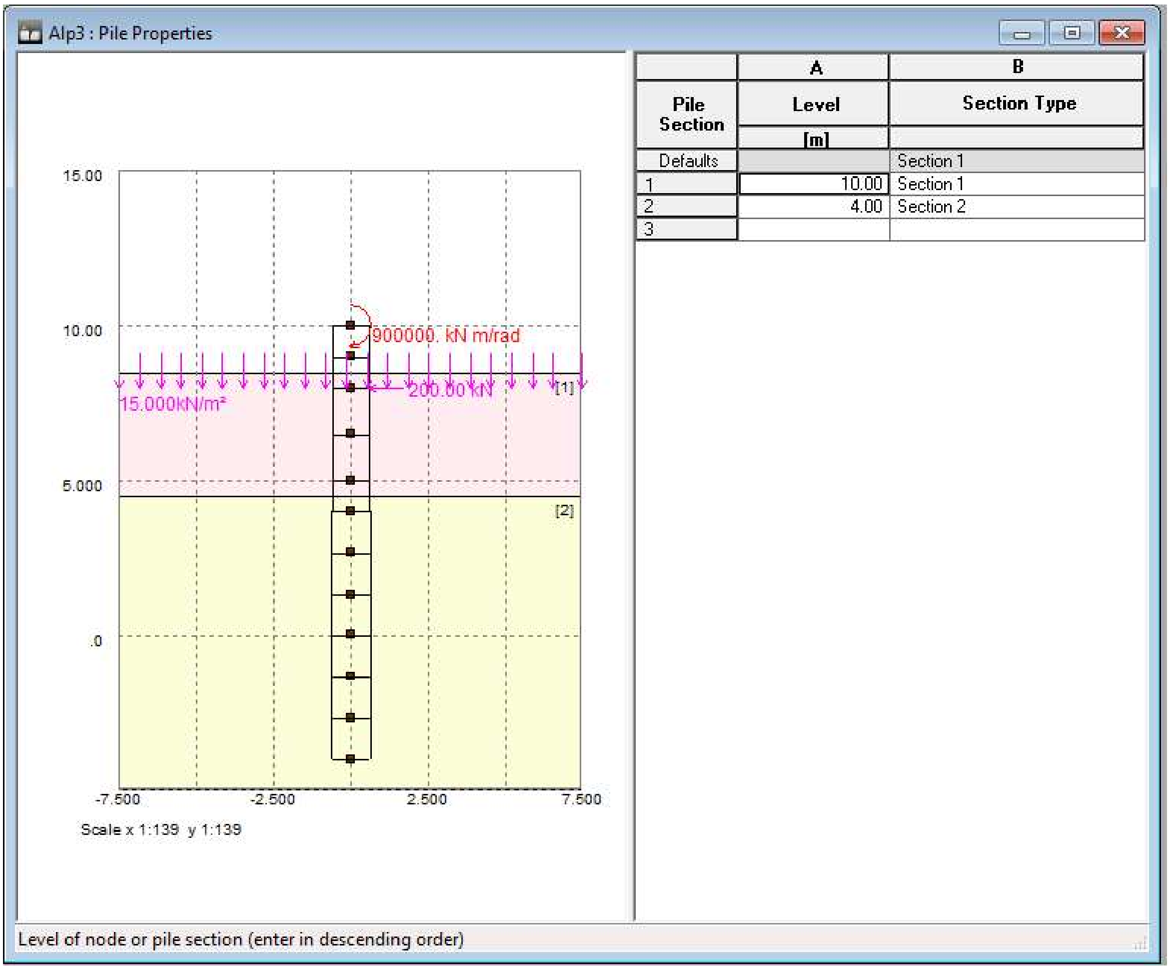
3.7 Pile Properties

The pile properties page allows the user to specify the top level of the pile and the level of any change in section. If the user selects the input "by node" option in the General Data dialog box the location of each node forming the pile can be input directly, this also allows the user to input the node levels using the Node Level table.

It is usual for 15-30 nodes to be used in an analysis.

Input by Level

To input the pile location by level enter the elevation at the top of the pile in column A, and select the section type in the table.

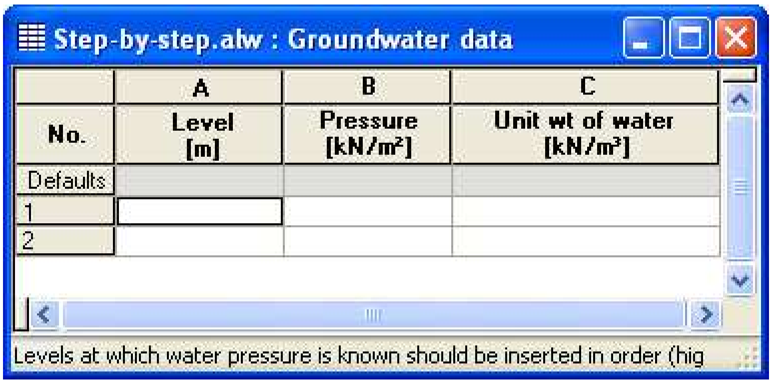


Input by Node

Where the user has selected the "By node" method for input, the level of each node, and the pile section below each node is input via the Node Levels table view. This input can be given in tabular form. The changes made to either of the forms are reflected in the other.

3.8 Groundwater

There is no Groundwater data for the problem being considered here. If there is any groundwater data then it should be input in the groundwater table which can be accessed by Data | Groundwater or via the Input Explorer.


Step-by-step.alw : Groundwater data

No.	A Level [m]	B Pressure [kN/m²]	C Unit wt of water [kN/m³]
Defaults			
1			
2			

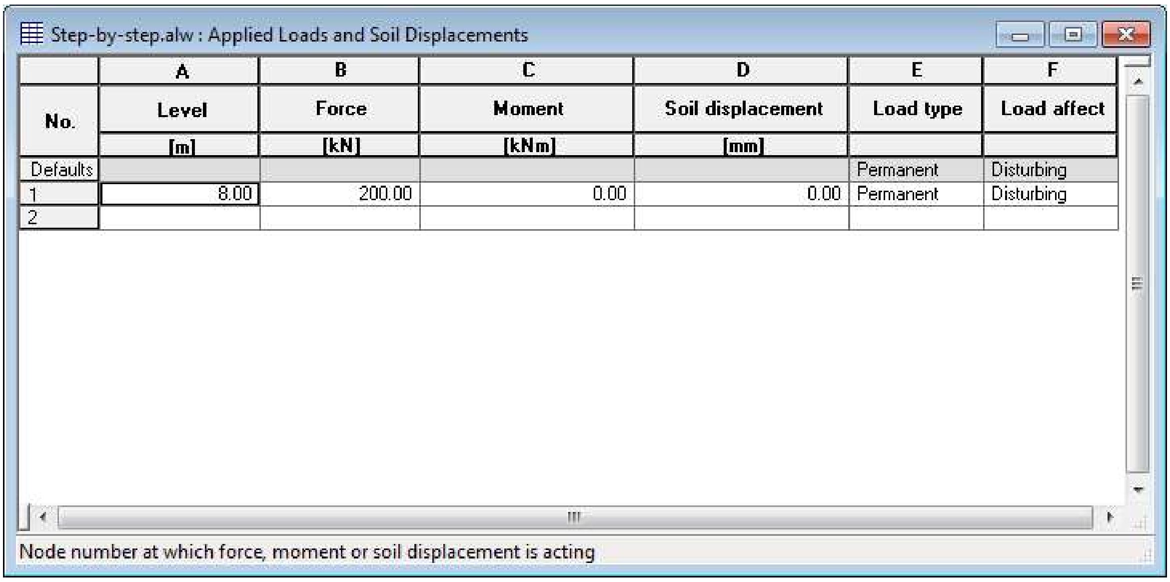
Levels at which water pressure is known should be inserted in order (hig

3.9 Applied Loads and Displacements

Loads, moments and lateral soil displacements at a node/level can be specified via the Applied Loads and Displacements table. This is accessed via Data | Applied loads and displacements from the program's menu or via the Input Explorer.

A lateral load of 200 kN is applied at a level of 8 m for this problem.

Note : Positive loads act from right to left. Positive moments are clockwise.


Step-by-step.alw : Applied Loads and Soil Displacements

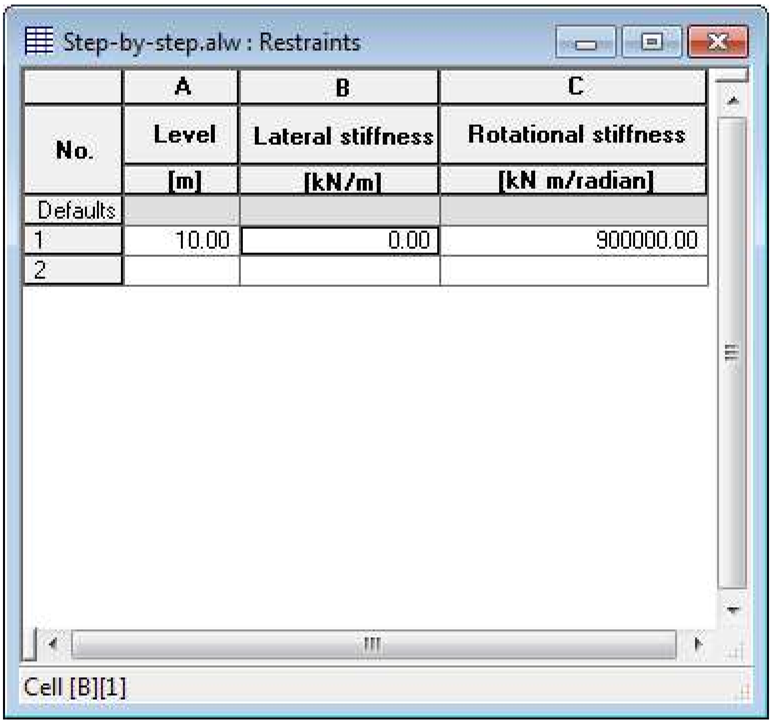
No.	A Level [m]	B Force [kN]	C Moment [kNm]	D Soil displacement [mm]	E Load type	F Load affect
Defaults					Permanent	Disturbing
1	8.00	200.00	0.00	0.00	Permanent	Disturbing
2						

Node number at which force, moment or soil displacement is acting

3.10 Restraints

Lateral and Rotational stiffness can be specified at any node/level via the Restraints table. This is accessible via Data | Restraints or via the Input Explorer.

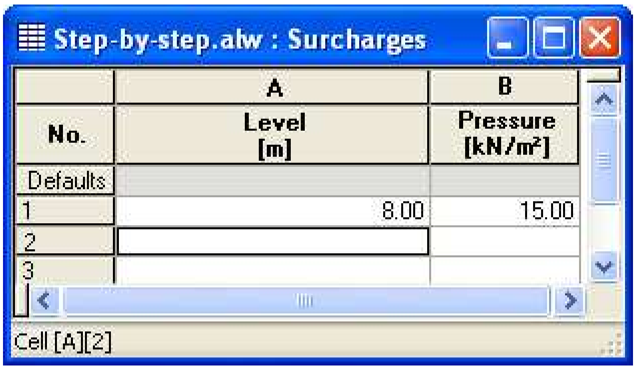
In this example a high rotational stiffness of 900000 kN/m is applied to the top of the pile (10m) in order to fix the node.



3.11 Surcharges

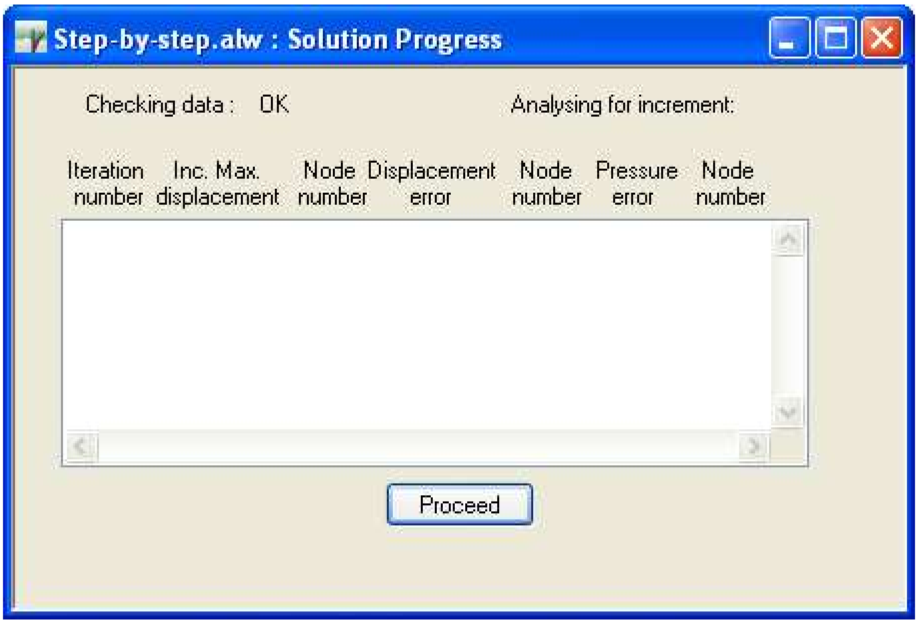
Surcharges can be provided at any level within or on the soil. The surcharges table is accessible via Data | Surcharges or via the Input Explorer.

The example considered here has a surcharge of 15 kN/m ^{2} at a level of 8 m.



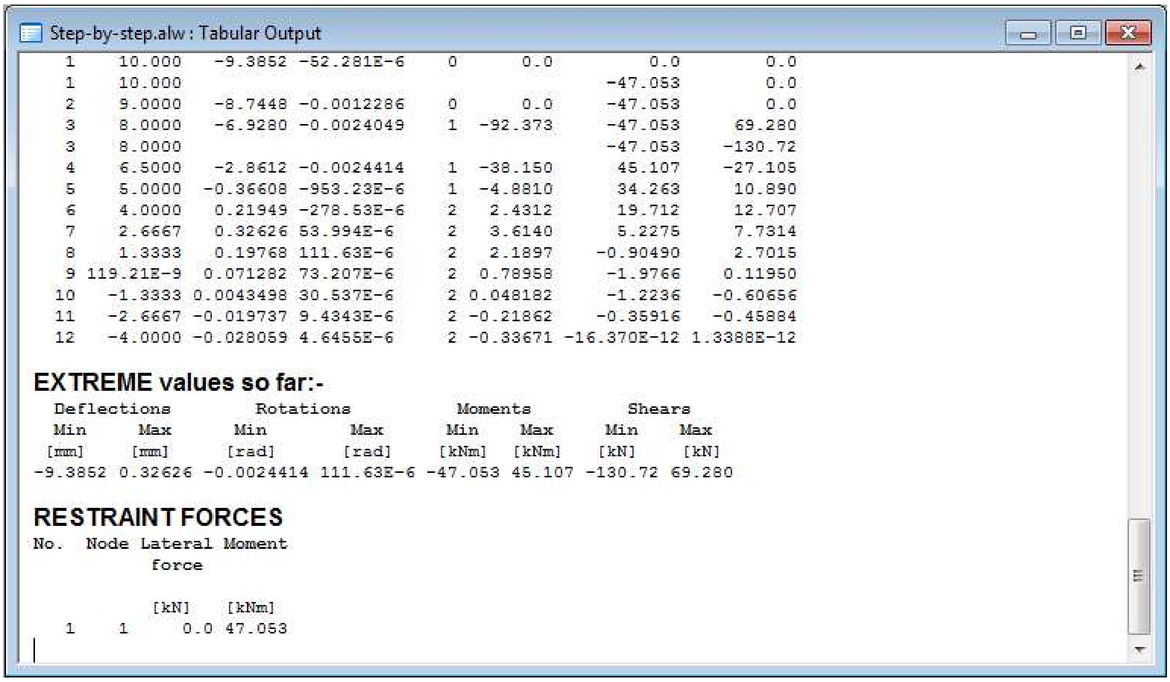
3.12 Analysis and Results

The model is now ready to be analysed. Click the Analyse button (Σ) on the Alp toolbar or choose "Analysis" on the menu bar and select "Analyse". The "Solution Progress" window will appear. Alp will automatically check the data for input errors. If no errors are found, "Checking data: OK" will appear in the top left corner of the window.



If errors are found, Alp will not allow the analysis to proceed until they are corrected. To run the analysis click “Proceed”.

Once the analysis has completed access the text output of results via the Output Explorer. This will contain the information about deflection, rotation, bending moment and shear at each node. All input data can also be presented in the tabular output



Step-by-step.alw : Tabular Output

1	10.000	-9.3852	-52.281E-6	0	0.0	0.0	0.0
1	10.000					-47.053	0.0
2	9.0000	-8.7448	-0.0012286	0	0.0	-47.053	0.0
3	8.0000	-6.9280	-0.0024049	1	-92.373	-47.053	69.280
3	8.0000					-47.053	-130.72
4	6.5000	-2.8612	-0.0024414	1	-38.150	45.107	-27.105
5	5.0000	-0.36608	-953.23E-6	1	-4.8810	34.263	10.890
6	4.0000	0.21949	-278.53E-6	2	2.4312	19.712	12.707
7	2.6667	0.32626	53.994E-6	2	3.6140	5.2275	7.7314
8	1.3333	0.19768	111.63E-6	2	2.1897	-0.90490	2.7015
9	119.21E-9	0.071282	73.207E-6	2	0.78958	-1.9766	0.11950
10	-1.3333	0.0043498	30.537E-6	2	0.048182	-1.2236	-0.60656
11	-2.6667	-0.019737	9.4343E-6	2	-0.21862	-0.35916	-0.45884
12	-4.0000	-0.028059	4.6455E-6	2	-0.33671	-16.370E-12	1.3388E-12

EXTREME values so far:-

Deflections		Rotations		Moments		Shears	
Min [mm]	Max [mm]	Min [rad]	Max [rad]	Min [kNm]	Max [kNm]	Min [kN]	Max [kN]
-9.3852	0.32626	-0.0024414	111.63E-6	-47.053	45.107	-130.72	69.280

RESTRAINT FORCES

No.	Node	Lateral force [kN]	Moment [kNm]
1	1	0.0	47.053

The text output for the input data and results can be accessed via the Output Explorer.

To view the graphical output, click the Graphical Output button () or click “View” in the menu bar and select “Graphical Output” or double click the appropriate leaf on the Output Explorer.

Different results can be shown on the graphical output using the buttons on the graphics toolbar, such as:

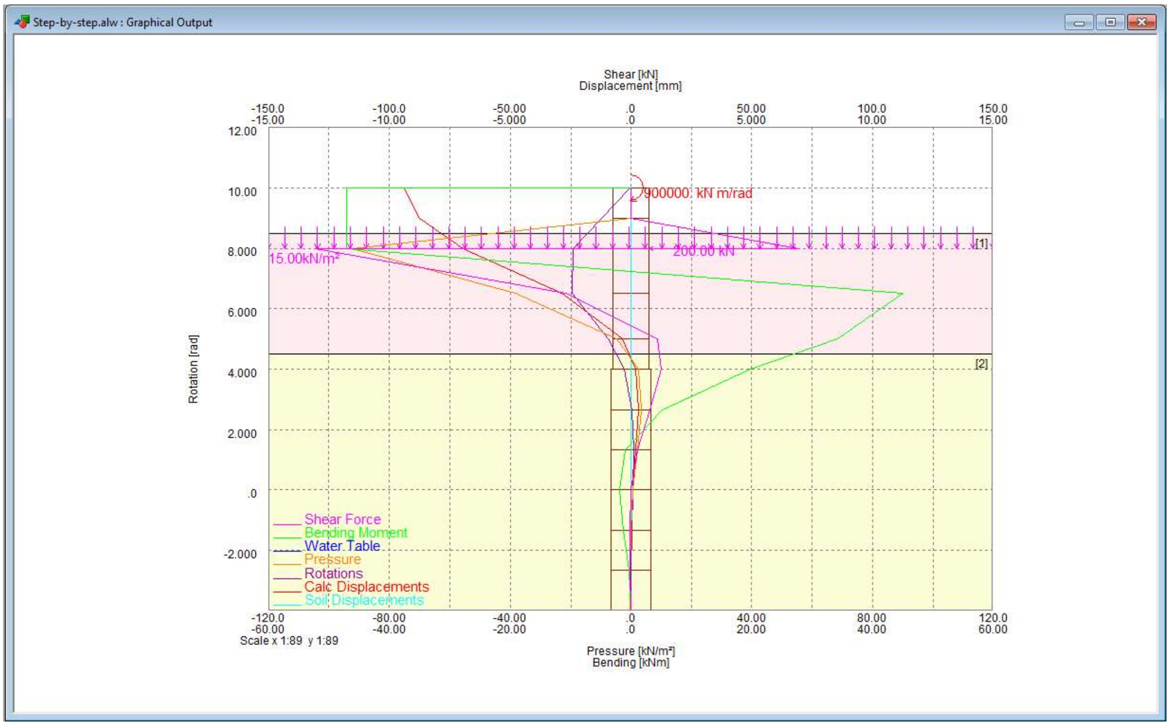
 deflection

 rotation

 pressure

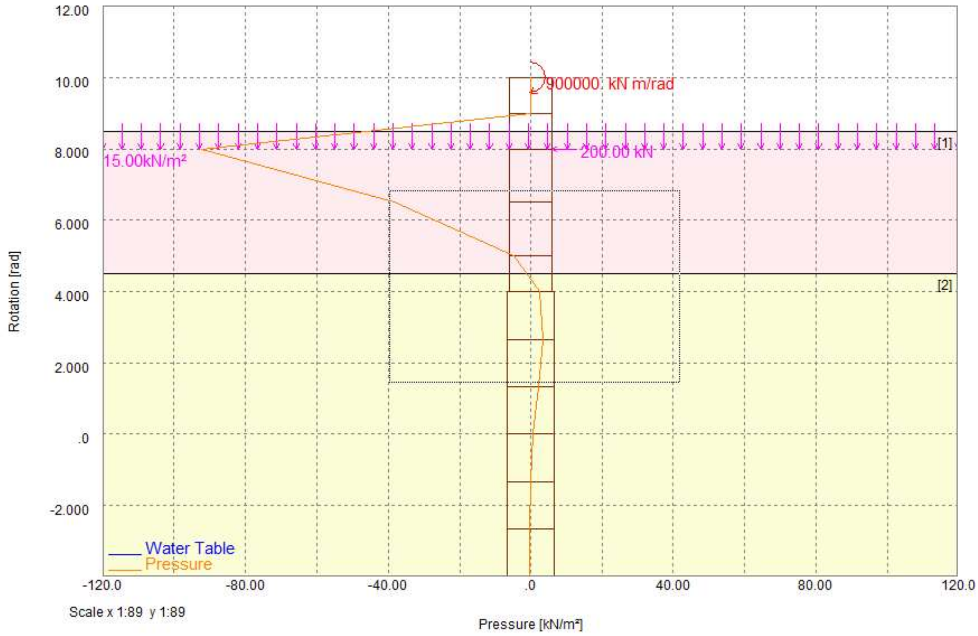
 bending moment

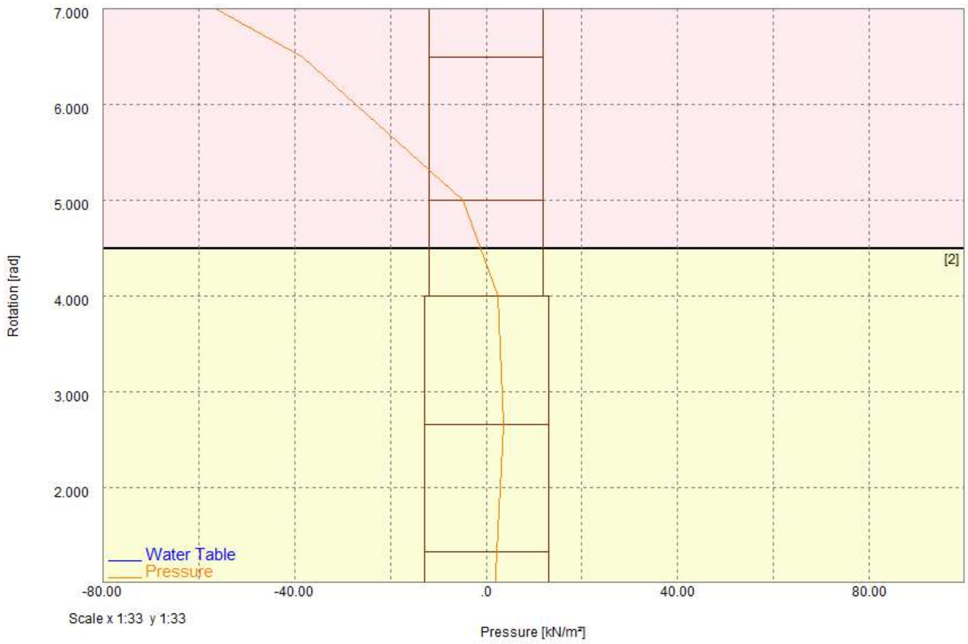
 shear



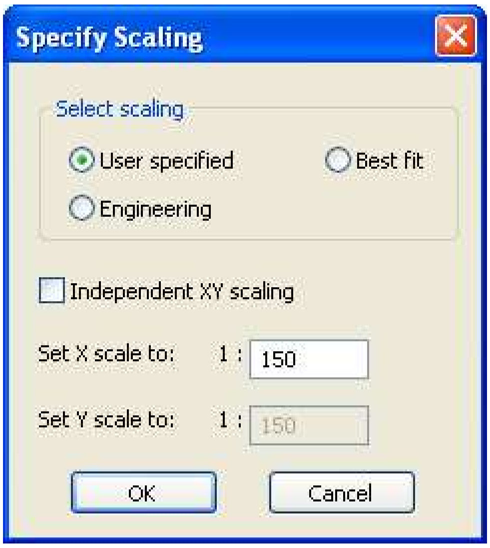
If the number of increments specified in the General Data dialog is more than one, the user can view the output for each increment by switching between next increment () and previous increment () buttons on the Graphics toolbar. The number of increments for this example is only one. Hence the increment buttons are deactivated.

It is possible to zoom in to areas of the graphical output by left clicking and dragging a square over the area you wish to view.





Clicking on the "Set Scale" button () allows the user to change the scale at which the output is displayed. The "User defined" radio button allows the precise scale to be specified.

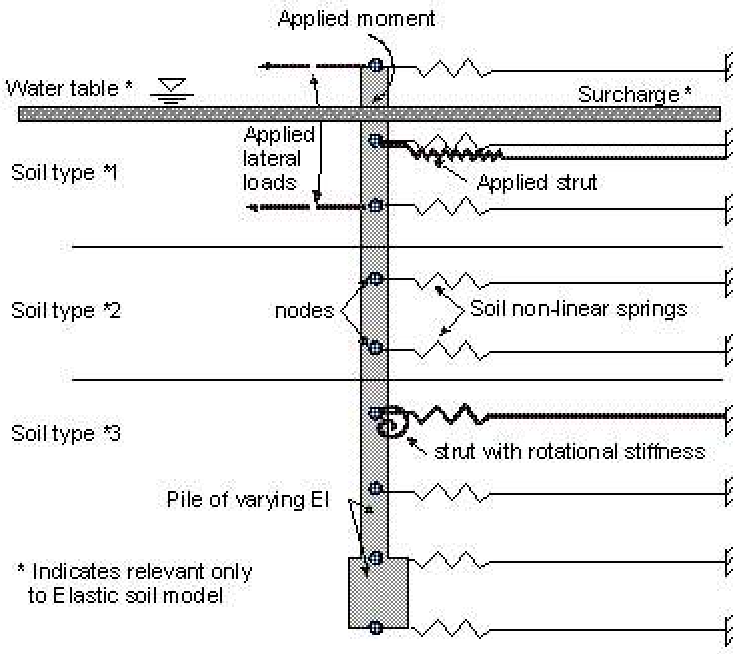

Specify Scaling
Select scaling
User specified
Best fit
Engineering
Independent XY scaling
Set X scale to:
1 :
150
Set Y scale to:
1 :
150
OK
Cancel

Part IV

4 Method of Analysis

4.1 General

The numerical representation of the problem analysed by **Alp** is shown below.



The pile is modelled as a series of elastic beam elements joined at the nodes. The soil is modelled as a series of non-interactive non-linear "Winkler type" springs connected at the nodes. Only **horizontal** forces can be transmitted between the soil and the nodes, and these forces are directly related to the earth pressures.

The analysis comprises the following steps:

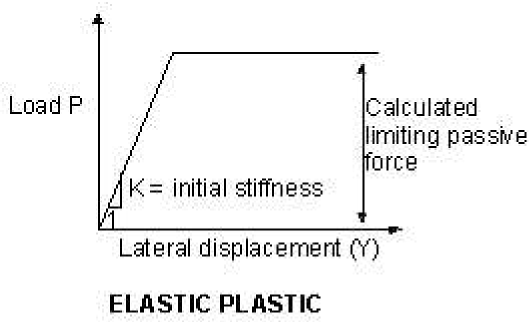
1. The stiffness matrices representing the soil and the pile are assembled. In the P-Y model, the maximum spring stiffness is used.
2. These matrices are combined, together with any stiffnesses representing the action of struts, to form the overall stiffness matrix.

For each applied load or displacement increment:

3. If there are applied soil displacements, the program calculates the load corrections. These load corrections are the loads applied to each soil spring such that the soil, in the absence of the pile, moves the specified amount.
4. The incremental nodal displacements are calculated from the nodal forces acting on the overall stiffness matrix assuming linear elastic behaviour.
5. The change in earth pressure from that of the previous load increment is found by multiplying the incremental nodal displacements by the soil stiffness matrix (and subtracting the load corrections if appropriate).
6. The earth pressures are compared with the soil load displacement behaviour. If the passive limit is infringed, in the elastic plastic model a set of nodal correction forces is calculated. In the P-Y method the passive limit is set to the appropriate displacement (Y). These forces are used to restore earth pressures to within the strength limits. A new set of nodal forces is then calculated.
7. Steps (3) to (6) are repeated until convergence is achieved.
8. Total nodal displacements, earth pressures, strut forces and pile shear forces and bending moments are calculated.

4.1.1 Elastic-Plastic Soil Model

The load displacement curve assumed for the elastic-plastic soil model is shown below.



The elastic spring constant is calculated from the expression:

$K = (EhE_{fact})$ kN/m

where:

E	=	Young's modulus of the soil
h	=	distance between the midpoint of the elements immediately above and below the node under consideration
E_{fact}	=	factor generally taken between about 0.6 and 1.0 (see Broms (1972) and Poulos (1971)).

A typical value of E_{fact} for clay is 0.8. It is recommended that users read the relevant references.

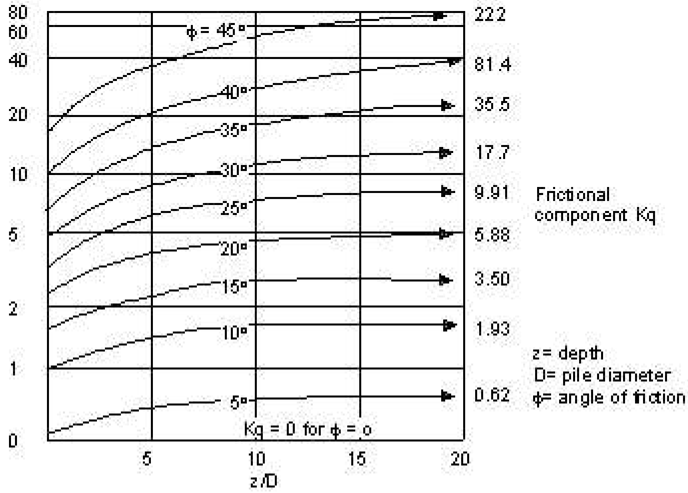
The soil stiffness matrix contains the values of K at each node along the diagonal with all other terms equal to zero.
The passive limiting force (F_p) is calculated from the following expression:

$Fp = (K_q \sigma'v + cK_c)hD$

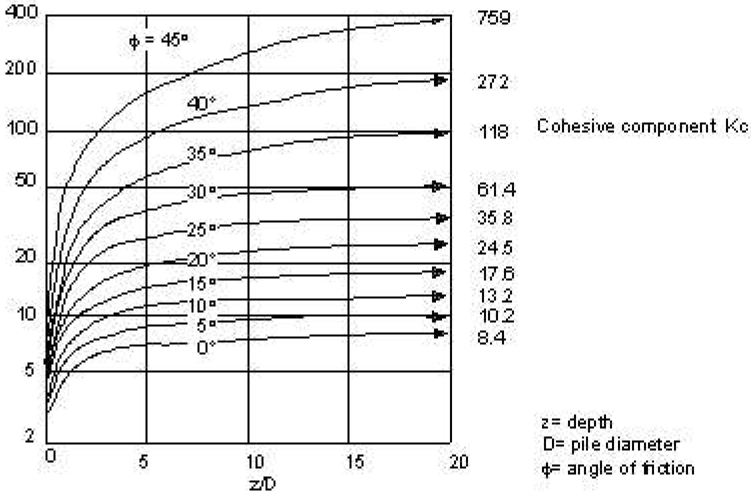
where:

K_q	=	Passive resistance coefficient for the frictional component of the soil.
$\sigma'v$	=	Vertical effective stress at the node under consideration.
K_c	=	Passive resistance coefficient for the cohesive component of the soil
c	=	Cohesion.
h	=	Distance between the midpoints of the elements immediately above and below the node under consideration.
D	=	Pile diameter.

Passive Resistance Coefficients (Tomlinson, 1986)



Values of Kq established by Brinch Hansen (1961)



Values of Kc established by Brinch Hansen (1961)

For a cohesionless soil K_q can be taken = 3*K_p where K_p is the Rankine coefficient of passive pressure. This was the value adopted by Broms (1964).

The vertical effective stress (σ'_v) at a node at level z is calculated as:

$$\sigma'_v = \int_z^{z_s} \gamma \, dz - u + \sigma_{su}$$

where:

γ = Unit weight of a stratum
z_s = Surface level
u = Prescribed pore pressure
σ_{su} = Sum of the vertical pressures of all surcharges above the level z

Factors for calculating coefficient of modulus variation (n_h) for cohesionless soil (after Tomlinson,1981)

Relative density		Loose	Medium dense	Dense
n_h for dry or moist soil (Terzaghi, 1955)	MN/m³ tons/ ft³	2.5 7	7.5 21	20 56
n_h for submerged soil (Terzaghi, 1955)	MN/m³ tons/ ft³	1.4 4	5 14	12 34
n_h for submerged soil (Reese et al, 1956)	MN/m³ tons/ ft³	5.3 15	16.3 46	34 94

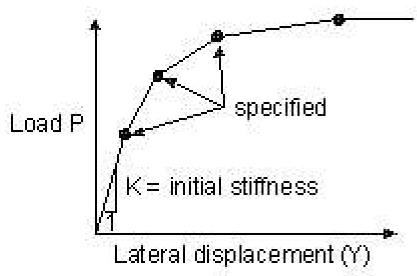
Other observed values of n_h are as follows:

- Soft normally-consolidating clays: 350 to 700kN/m³ (1 to 2 tons/ft³).
- Soft organic silts: 150kN/m³ (0.5 tons/ft³).

$E_{fact} = n_h.z/E$ where z = depth

4.1.2 P-Y Soil Model

A P-Y curve represents the lateral load-deflection behaviour of a discrete layer of soil. This is achieved either by the specification of points that define a piece-wise linear curve:



P - Y CURVE

or by the specification of the equation that describes the curve.

The curve is **independent** of the shape and stiffness of the pile. The curves used for each of the three soil model types are described below.

4.1.2.1 Specified P-Y Curves

The user may specify the P-Y data to represent the load deflection of the soil.

The load and corresponding deflection is input **at each node** to provide a vertical profile of behaviour down the side of the pile.

4.1.2.2 Generated P-Y Curves

Lateral soil resistance deflection (P-Y) curves may be constructed for '**Soft clay**', **'Stiff clay'**, and '**Sand**' strata for both '**Static**' and '**Cyclic**' load cases as detailed below:

Soft

P-Y curves for soft clay are calculated using the method established by Matlock (1970).
The ultimate resistance (P_u) of Soft clay increases from $3c_u$ to $9c_u$ as the depth X increases from 0 to X_R according to:

$$P_u = D\left\{3c_u + \sigma'_v + J\frac{c_u X}{D}\right\} \qquad \text{For } X \le X_R \quad (1)$$

$$P_u = 9c_u D \qquad \text{For } X \ge X_R \quad (2)$$

P_u	=	ultimate soil resistance per unit length
c_u	=	undrained shear strength
σ'_u	=	vertical effective stress
D	=	pile diameter
J	=	dimensionless empirical constant (0.5 for Soft clays)
X	=	depth below soil surface
X_R	=	depth below soil surface to bottom of reduced resistance zone

If c_u is constant with depth, equations (1) and (2) are solved simultaneously to give:

(3)

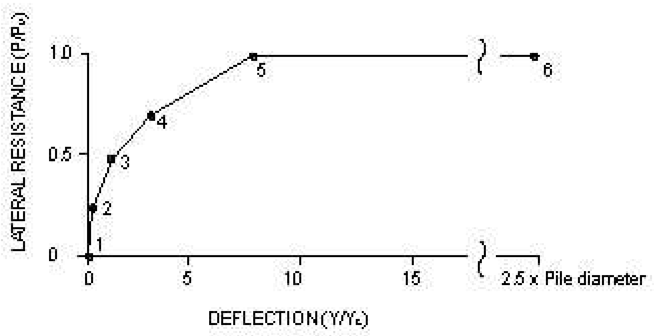
If c_u varies with depth, equations (1) and (2) are both solved at each depth, until equation (2) is less than equation (1) to give X_R.

The P-Y curve for short-term '**Static** ' load cases is then generated for the following points:

P/P_u	Y/Y_c
0	0
0.29	0.2
0.50	1.0
0.72	3.0
1.00	8.0
1.00	∞ (2.5D)

where:

P	=	Soil resistance per unit length
Y	=	Lateral deflection
Y_c	=	2.5 x E50 x D
E50	=	Strain at one-half the maximum stress for an undrained triaxial compression test.



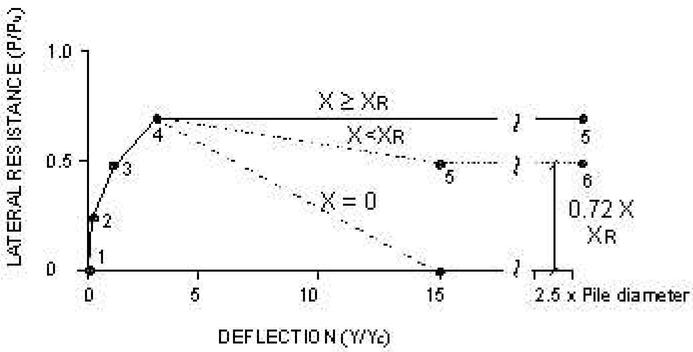
P-Y curve for Soft Clay (Static)

If no direct laboratory data is available suggested values of E50 for Soft to Firm clays are as follows (after Sullivan et al, 1980):

Consistency	**E50**
Soft	0.020
Firm	0.010

The P-Y curves for the '**Cyclic** ' case are generated from the following points:

P/P_u	**$X \geq X_R$**	**Y/Y_c**	**P/P_u**	**$X < X_R$**	**Y/Y_c**
0		0	0		0
0.29		0.2	0.29		0.2
0.50		1.0	0.50		1.0
0.72		3.0	0.72		3.0
0.72		∞ (2.5D)	$0.72X/X_R$		15.0
			$0.72X/X_R$		∞ (2.5 D)



P-Y curve for Soft Clay (Cyclic)

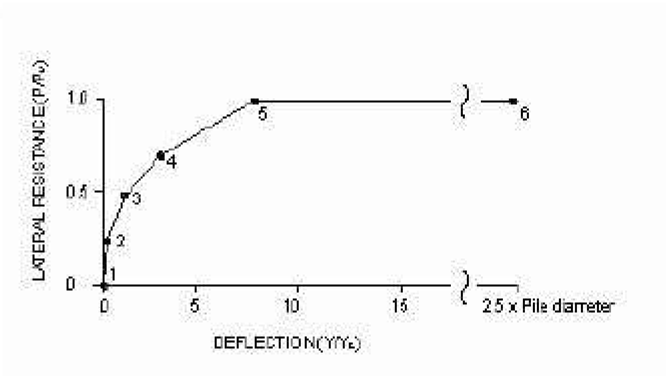
Stiff Clay

The API RP2A 18th Edition (1989) and 21st Edition (2000) recognise that stiff clays have important non-linear stress strain relationships and are generally more brittle than soft clays. However, the API references give no P-Y data. Alp therefore uses the following.

The ultimate soil resistance of stiff clay is calculated using equations (1) and (2) as before, but assuming J = 0.25.

The P-Y curve for short-term '**Static**' load cases is then generated for the following points; i.e. the same values as for soft clay.

P/P_u	Y/Y_c
0	0
0.29	0.2
0.50	1.0
0.72	3.0
1.0	8.0
1.0	∞(2.5D)



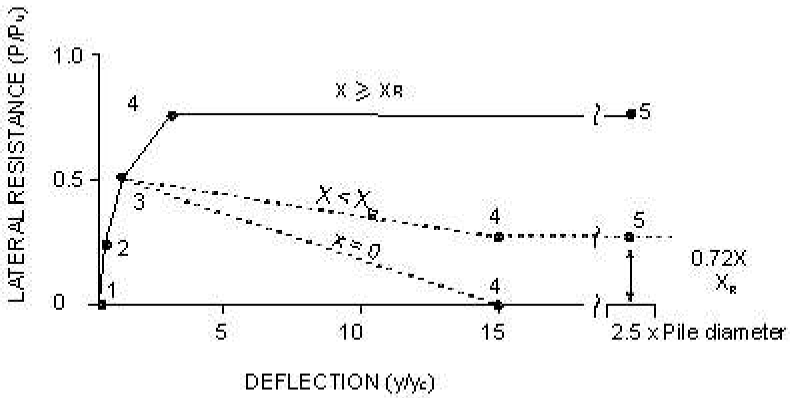
P-Y Curve for Stiff Clay (Static)

If no laboratory data is available suggested values of E50 for stiff to hard clay are as follows (after Sullivan et al, 1980):

Consistency	E50
Stiff	0.005
Hard	0.004

The P-Y curves for '**Cyclic**' are generated for the following points:

P/P_U	$X \geq X_R$	Y/Y_C	P/P_U	$X < X_R$	Y/Y_C
0		0	0		0
0.29		0.2	0.29		0.2
0.50		1.0	0.5		1.0
0.72		3.0	$0.72X/X_R$		15.0
0.72		∞ (2.5D)	$0.72X/X_R$		∞ (2.5D)



P-Y Curve for Stiff Clay (Cyclic)

Sand

Two choices of P-Y curve for sand are available:

- Reese et al (1974); and
- API RP2A 21st edition (2000).

Descriptions of the two methods are given below.

Reese et al (1974)

If the Reese et al has been selected, then the P-Y curves for sand are calculated using the method established by Reese et al. (1974).

The ultimate resistance of sand varies from a value determined by equation (4) at shallow depths to a value determined by equation (5) at deep depths. The depth of transition (X_t) is determined by comparing the value of each equation at the specified depths.

The ultimate resistance of sand at **shallow** depths is determined according to:

Equation 4

$$P_u = A\sigma'_v\left[\frac{K_0 X \tan\phi' \sin\beta}{\tan(\beta-\phi')\cos\alpha} + \frac{\tan\beta}{\tan(\beta-\phi')}(D + X\tan\beta\tan\alpha) + K_0 X\tan\beta(\tan\phi'\sin\beta - \tan\alpha) - K_a D\right] \quad (4)$$

and the ultimate resistance of sand at **deep** depths is determined according to:

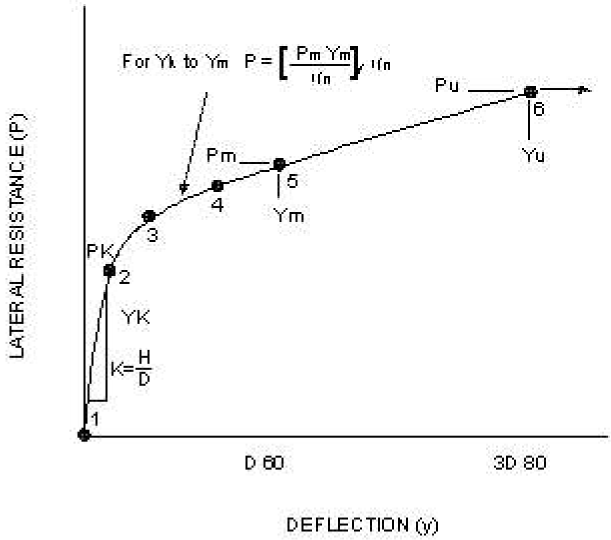
Equation 5

$$P_u = AD\left[k_a\sigma'_v(\tan^8\beta - 1) + k_0\sigma'_v\tan\phi'\tan^4\beta\right] \quad (5)$$

P_u = ultimate resistance per unit length
A = Empirical adjustment factor which accounts for differences in static and cyclic behaviour (see Reese et al, 1974)
σ'_v = effective vertical overburden pressure
X = depth below soil surface
K_0 = coefficient of earth pressure at rest
ϕ' = angle of internal friction of sand
β = $45°+\phi'/2$

α = $\phi'/2$
D = pile diameter
K_a = Rankine minimum active earth pressure coefficient [$\tan^2(45°-\phi'/2)$]

The lateral soil resistance-deflection (P-Y) relationship for sand is non-linear and is represented by a four-segment curve. If $Y_k > Y_m$, point 2 is omitted and the curve becomes linear between (0,0) and ($Y_k > P_m$).



P-Y Curve for Sand (Static/Cyclic)

The values of points u, m and k are computed as follows:

u: P_u = Equation 4 at depths $\le X_t$
Equation 5 at depths $\ge X_t$
Y_u = 3D/80
m: P_m = (B/A)P_u
Y_m = (1/60)D
where B = Non-dimensional empirical adjustment factor to account for difference in static and cyclic behaviour (see Reese et al, 1974).
k: P_k = (X/D)$K_1 Y_k$
Y_k =
where N =
k_1 = Initial soil modulus

The P-Y curve between points k and m is a parabola with intermediate points calculated from:

$$P = \left[\frac{P_m}{Y_m^{\frac{1}{n}}}\right] Y^{\frac{1}{n}}$$

The following values of k1 for submerged sand are typical for static and cyclic loading (Reese et al, 1974).

Relative Density	k_1 ($MN/m^2/m$)
Loose	5.43
Medium	16.29
Dense	33.93

API RP2A 21st Edition (2000)

If the API 21st Edition has been selected, then the P-Y curves for sand are calculated as described below.

The ultimate lateral bearing capacity for sand at shallow depths is calculated as:

$p_{us} = (C_1 \times H + C_2 \times D) \times \gamma \times H$

The ultimate lateral bearing capacity for sand at deep depths is calculated as:

$p_{ud} = C_3 \times D \times \gamma \times H$

where:

p_u = ultimate resistance (force/unit length), kN/m (s = shallow, d = deep)
γ = effective soil weight, kN/m^3
H = depth (m)
C_1, C_2, C_3 = coefficients determined from Figure 6.8.6-1 of the API RP2A 21st Edition
D = average pile diameter from surface to depth (m)

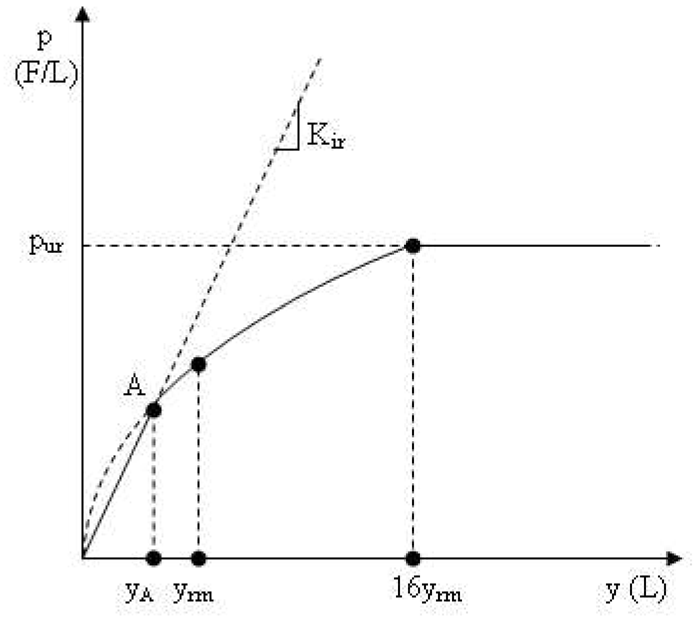
The lateral soil resistance-deflection (p-y) relationship is described by:

$$P = Ap_u \tanh\left[\frac{kH}{Ap_u} y\right]$$

where:

P = actual lateral resistance (kN/m)
A = factor to account for cyclic or static loading conditions (0.9 for cyclic loading, max(3.0 - 0.8H/D, 0.9) for static loading
k = initial modulus of subgrade reaction (kN/m^3) determined from Figure 6.8.7-1of the API RP2A 21st Edition
y = lateral deflection (m)

Weak Rock



P-Y Curve for Weak Rock

P-Y curves for weak rock are calculated using the method established by Reese (1997). The ultimate resistance (p_{ur}) of weak rock increases as the depth x_r increases from 0 to 3b according to:

$$p_{ur} = \alpha_r q_{ur} b\left(1+1.4\frac{x_r}{b}\right) \qquad \text{For } 0 \le x_r \le 3b \quad (1)$$

$$p_{ur} = 5.2\alpha_r q_{ur} b \qquad \text{For } x_r \ge 3b \quad (2)$$

p_{ur} = ultimate soil resistance per unit length
α_r = strength reduction factor
q_{ur} = compressive strength of rock
b = pile diameter
x_r = depth below rock surface

The lateral soil resistance-deflection (P-Y) relationship for weak rock is represented by a three-segment curve. The relationship is described by:

- $p = K_{ir} y$ For $y \le y_A$
- $p = \frac{p_{ur}}{2}\left(\frac{y}{y_{rm}}\right)^{0.25}$ For $y \ge y_A$ and $p \le p_{ur}$
- $p = p_{ur}$ For $y \ge 16\, y_{rm}$

where,

K_{ir} = $k_{ir}E_{ir}$

k_{ir} = $\left(100 + 400\,\frac{x_r}{3b}\right)$ For $0 \le x_r \le 3b$ (1)

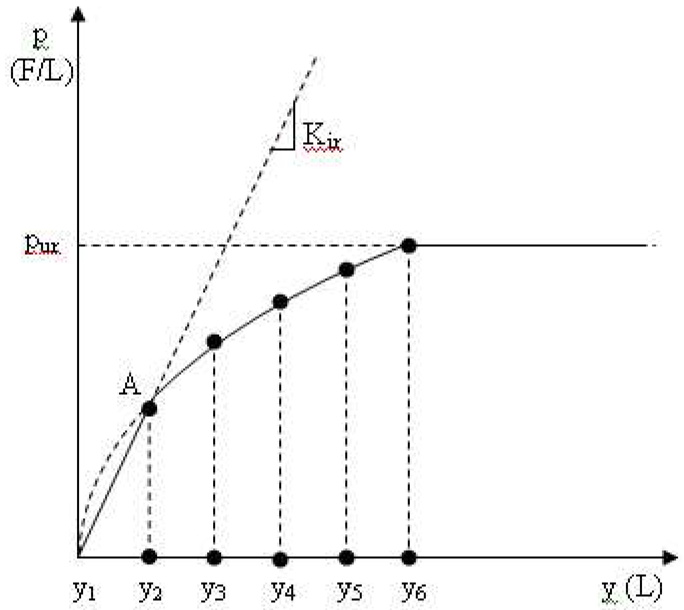
k_{ir} = dimensionless constant = 500 For $x_r \ge 3b$ (2)

y_{rm} = $k_{rm}b$

y_A = lateral deflection of Pile at point A = $\left[\frac{p_{ur}}{2(y_{rm})^{0.25}K_{ir}}\right]^{1.333}$

k_{rm} = dimensionless constant ranging from 0.0005 to 0.00005, that serves to establish overall stiffness of p-y curves

E_{ir} = initial modulus of rock

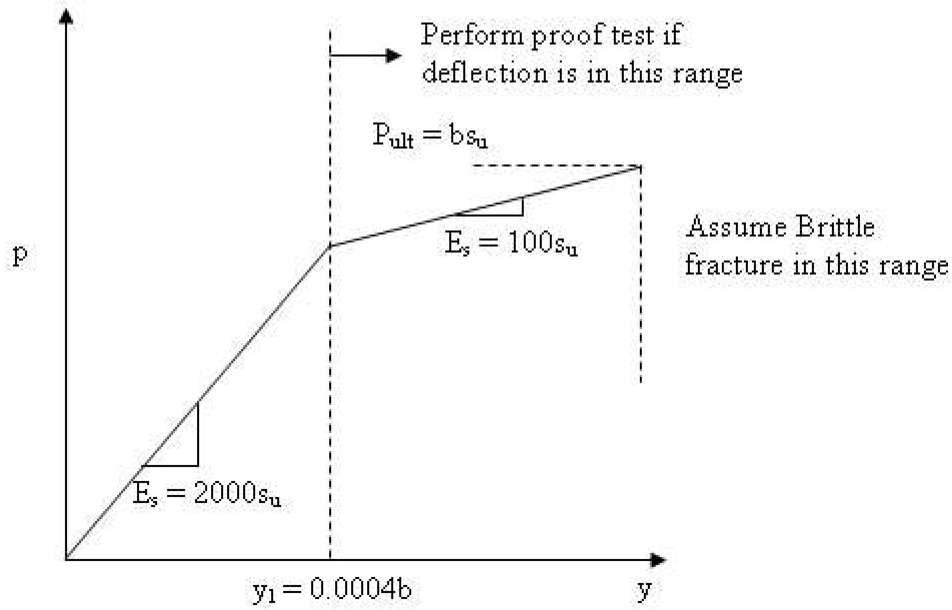


Six points representing the P-Y Curve for Weak Rock

In the program, the curve is represented as a series of straight lines formed between the following six points:

First Point (y_1) = 0 (origin)
Second Point (y_2) = y_A
Third Point (y_3) = y_2+((y_6 - y_2) / 4)
Fourth Point (y_4) = y_3+((y_6 - y_2) / 4)
Fifth Point (y_5) = y_4+((y_6 - y_2) / 4)
Sixth Point (y_6) = 16*y_{rm}

Strong Rock



P-Y curves for strong rock are calculated using the method published by Turner (2006). The ultimate resistance (P_{ult}) of strong rock is given by the expression:

P_{ult}	=	bs_u
b	=	pile diameter
s_u	=	half of the compressive strength of the rock

The lateral soil resistance-deflection (P-Y) relationship for strong rock is represented by two linear segments.

4.1.3 Pile Stiffness

The pile is modelled as a series of elastic beam elements, the stiffness matrix being derived using conventional methods from slope deflection equations.

Considering a single beam element of length L and flexural rigidity EI spanning between nodes A and B, the moments (M) and forces (P) at nodes A and B can be expressed as functions of the deflections and rotation at the nodes, i.e.

$$\begin{bmatrix} M_A \\ M_B \end{bmatrix} = \frac{6EI}{L^2}\begin{bmatrix} 1 & -1 \\ 1 & -1 \end{bmatrix}\begin{bmatrix} \delta_A \\ \delta_B \end{bmatrix} + \frac{2EI}{L}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}\begin{bmatrix} \theta_A \\ \theta_B \end{bmatrix}$$

$$\begin{bmatrix} P_A \\ P_B \end{bmatrix} = \frac{12EI}{L^3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} \delta_A \\ \delta_B \end{bmatrix} + \frac{6EI}{L^2}\begin{bmatrix} 1 & 1 \\ -1 & -1 \end{bmatrix}\begin{bmatrix} \theta_A \\ \theta_B \end{bmatrix}$$

where δ_A, δ_B and θ_A, θ_B represent the deflections and rotations at nodes A and B respectively referred to the neutral axis of the beam. The above equations can be rewritten in matrix form as:

$[M] = [A][\delta] + [B][\theta]$

and

$[P] = [C][\delta] + [A]^T[\theta]$

where [A], [B] and [C] are functions of the element lengths and flexural rigidity (EI), and [δ] and [θ] are the nodal horizontal displacements and rotations.

If there are no moments applied to the pile, [θ] can be eliminated to give

$[P] = [S][\delta]$

in which [S] is the pile stiffness matrix given by

$[S] = [C] - [A]^T[B]^{-1}[A]$

4.1.4 Applied Lateral Loads and Displacements

Lateral Loads

Lateral loads can be applied in the form of horizontal forces and bending moments. The forces can be defined explicitly or in the form of soil displacements.

Lateral loads and bending moments can be applied at any node. The applied loading moment loads [M] are converted into a series of lateral loads

$[P] = [M][A]^T[B]^{-1}$.

Soil Displacements

Soil displacements can be specified at any node. Between nodes when soil displacements are specified the program calculates an applied displacement using linear interpolation. Above the highest node and below the lowest node the respective value is assumed to be constant.

Increments

The program allows for the loads and soil displacements to be applied in increments. This can be done in the following combinations;

- loads only
- displacements only
- or both loads and displacements.

If both loads and displacements are incremented both are applied gradually. If only one is incremented the other is kept constant at the full value.

4.1.5 Restraints

Restraints with a specified axial and rotational stiffness can be specified at any node. The additional stiffness supplied to the pile by restraints is catered for in the program by adding the moment stiffness to the diagonal of matrix [B] described in Pile stiffness and by adding the lateral stiffness to the relevant position on the diagonal of the overall stiffness matrix.

4.1.6 Partial Factors

Partial factors can be set by the user to adjust the loads and soil strength parameters.

Load Factors

The load (L) used in the analysis is calculated from the design load (Ld) input by the user, using the relevant partial factor (γ_L) that can vary for permanent/variable and favourable/unfavourable loads.

$L = Ld \times \gamma_L$

Soil Strength Factors

The design soil parameters are modified as shown below.

The unit weight (W) used in the analysis is calculated from the design unit weight (Wd) input by the user, using the relevant partial factor (γ) such that

$W = Wd/\gamma$

The drained cohesion (C') used in the analysis is calculated from the design drained cohesion (C'd) input by the user, using the relevant partial factor ($\gamma_{c'}$) such that

$C' = C'd/\gamma_{c'}$

The undrained shear strength (Cu) used in the analysis is calculated from the design undrained shear strength (Cud) input by the user, using the relevant partial factor (γ_c) such that

$Cu = Cud/\gamma_c$

The shear angle (φ) used in the analysis is calculated from the design shear angle (φd) input by the user, using the relevant partial factor (γ_φ) such that

$\varphi = \tan^{-1}(\tan(\varphi d)/\gamma_\varphi)$

The ultimate compressive strength of rock (Qur) used in the analysis is calculated from the design ultimate compressive strength (Qurd), using the undrained shear strength partial factor (γ_c) such that

$Qur = Qurd/\gamma_c$

4.1.7 Pushover Analysis

The pushover analysis option in Alp allows the user to identify the response of the pile to a range of specified loads or pile displacements.

Specified Load

For pushover curves with specified loads the same analysis is undertaken as during the standard analysis. However the analysis is undertaken once for each load specified, with the specified load applied to the defined node for each analysis, with all results sets and a summary of key results retained.

Specified Displacement

For pushover curves with specified displacements the same fundamental analysis is undertaken as with that for the specified load. However, as the solver does not allow for the calculation of an unknown load based on a specified displacement, the solution is iterated to determine the correct load that results in that displacement.

The method used to determine the load for a specified displacement follows the following steps.

1. Calculate the displacement based on loads 1 and 2 specified by the user in the General Data dialog.
2. Check the displacement calculated based on the two iterations, if either matches the required displacement save the corresponding results and exit this process. Alternatively if either load case has failed to analyse alert the user and exit the process.

3. Otherwise, determine the load for the next iteration. This is done by assuming a linear relationship between load and displacement based on the points derived for loads 1 and 2, and either interpolating or extrapolating to the required load.
4. Calculate the displacement for the next load iteration.
5. If the the calculation in step 4 fails, set a new load at the midpoint between the failed load and the nearest of the previous successful loads and repeat from the fourth step.
6. If the calculation succeeds check to see if the displacement is equal to the specified displacement +/- the specified tolerance, if equal save the results and exit the process.
7. If the calculation is not successful calculate the next load iteration. To do this assume a linear load/displacement relationship and where possible interpolate between previously estimated loads discarding the third result, where not possible extrapolate from the two nearest results discarding the third result.
8. With the two result sets retained go back to step 4.

Continue with the steps shown above until either a displacement is calculated within the tolerance of that specified by the user, or until the iteration limit of 100 iterations is reached.

Repeat the procedure above for each user specified displacement and store results.

Results

When running a pushover analysis, additional results are stored that capture key data from the process. Results are stored for each load/displacement specified and include:

- the load at the specified node/level,
- the displacement at the specified node/level
- the rotation at the displacement node/level
- the maximum moment in each section of the pile.

4.1.8 Modification Factors

P Modification factors may be applied to account for the effects of piles within a group, where piles may be ‘shadowed’ by other piles, which reduces the amount of load they attract via the P-y curve.

These factors are applicable only for Generated P-y curves and Elastic-Plastic soil models.

4.1.9 Moment vs Curvature

This feature allows the user to model non-linear bending stiffness of the pile. Each pile section can have a different Moment vs Curvature curve. When this option is active, the program iteratively computes the Bending Stiffness based on the Bending Moment at each node.

Part V

5 Input Data

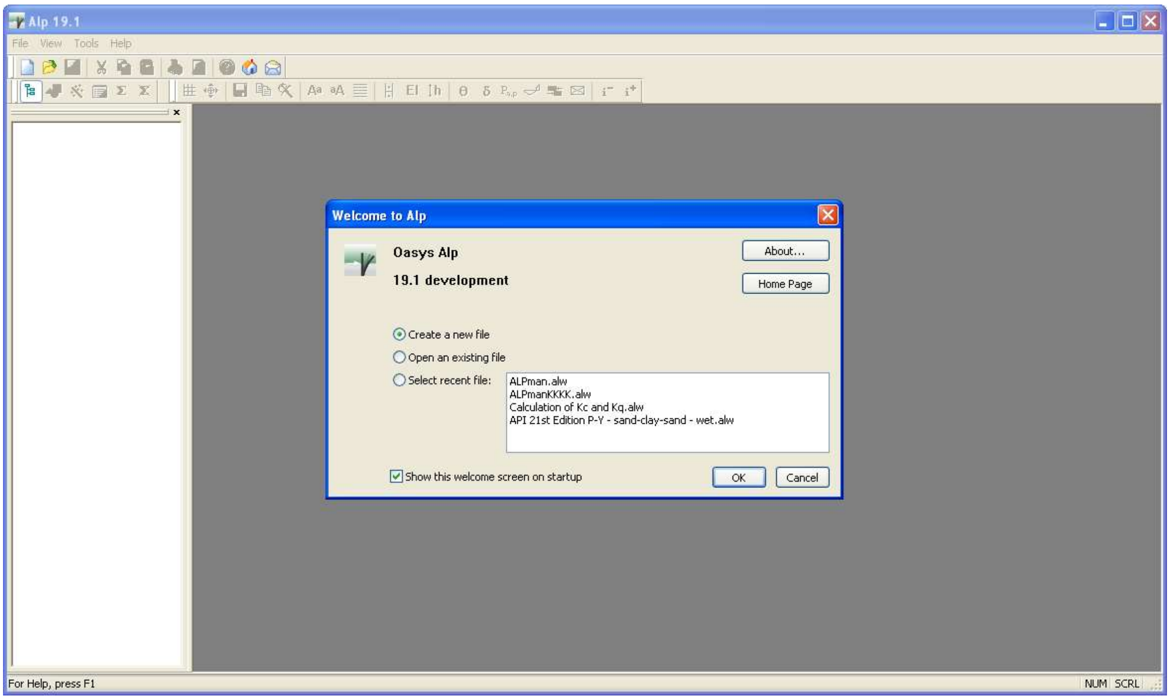
5.1 Assembling Data

It is best to make a sketch of the problem **before the computer is approached**. This should comprise a cross section of the proposed pile with the:

- ground surface
- location of each soil strata
- calculation method
- parameters of each material
- phreatic surface
- location of any piezometers
- location and magnitude of applied lateral loads, bending moments or soil displacements
- location of any restraints
- level of any vertical surcharges.

5.1.1 Opening the program

On selection of the **Alp** program the main screen will open.



This is the main screen within which all further data, graphics and results are entered and viewed. All further information appears in the smaller window, which is placed inside the main background screen.

To start a new project file select :

- File | New or
- the new file icon .

This will open a new Titles window and allow you to proceed. It is possible to open more than one data file at any one time. The file name is therefore displayed in the title bar at the top of each child window.

5.1.1.1 Intranet link and e-mails

To view the latest information regarding the **Alp** program or contact the support team click on the internet  or support team  buttons on the Start screen or select the options from the toolbar.

Note! Once in the program the Start screen can be re-accessed using Help | Show startup window.

List of information required and actions before contacting support team:

- Version of Alp (see top bar of program or Help | About Alp)
- Spec of machine being used.
- Type of operating system.
- Please pre-check all input data.
- Access help file for information.
- Check web site for current information.
- Should you report a program malfunction then please attempt to repeat and record process prior to informing the team.

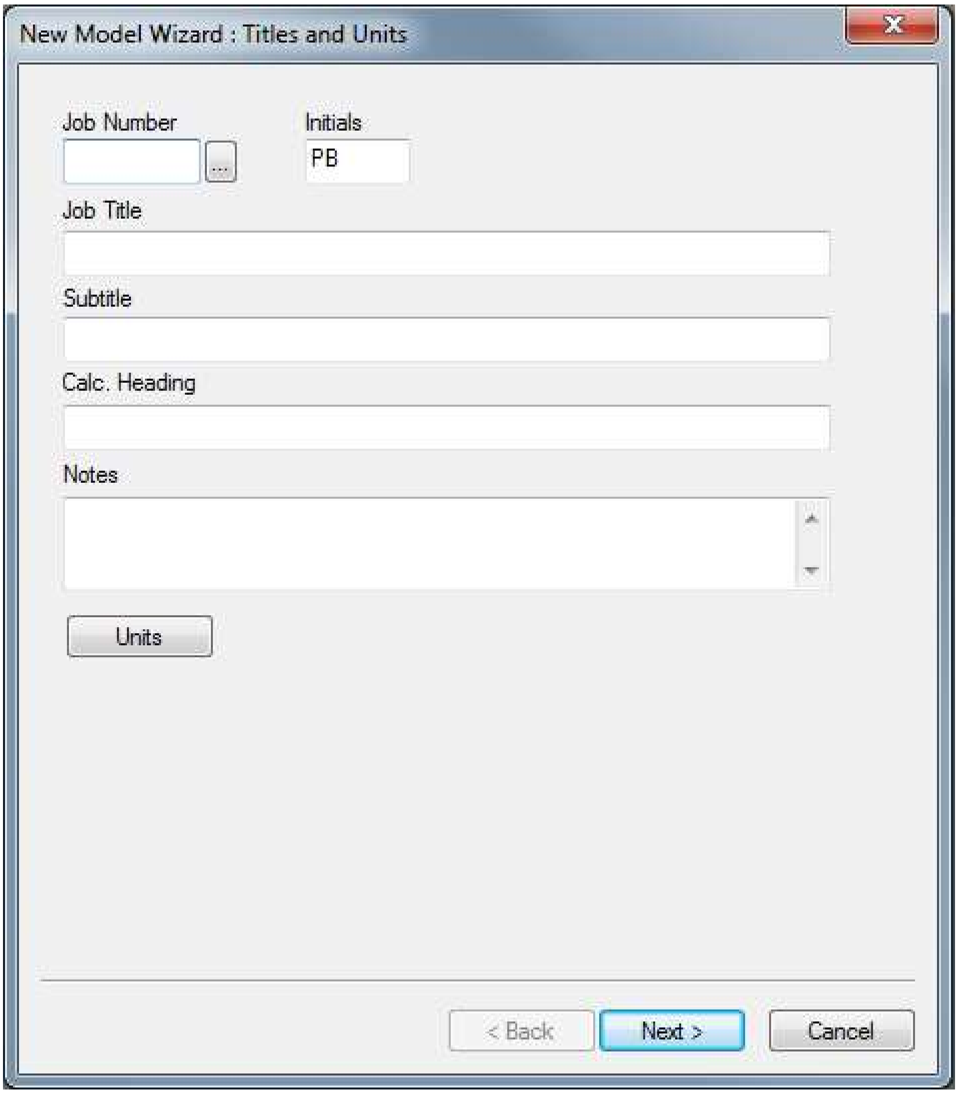
The web site aims to remain up to date with all data regarding the program and available versions. Should any malfunctions persist then the work around or fix will be posted on the web site.

5.1.1.2 New Model Wizard

If the program preferences have been set appropriately then the New Model Wizard is accessed by selecting the `File | New´(Ctrl+N) option from the main menu, or by clicking the 'New' button on the Alp toolbar. Cancelling at any time will result in an empty document.
The New Model Wizard provides a quick and easy method of the inputting a simple model that is sufficiently complete to perform an analysis. Additional more complex or detailed data may be added to the model once the New Model Wizard is completed.

5.1.1.2.1 New Model Wizard : Titles and Units

The first property page of the New Model Wizard is the Titles and Units window. This allows the user to enter the job details. By default the job details of the previous job are used.



Job Number
This is the job number, which can be any alphanumeric string. By clicking the drop-down button, the user can access the job numbers recently used.

Initials
The initials of the user used on printed output.

Job Title
The title of the job.

Subtitle
The subtitle that this model relates to.

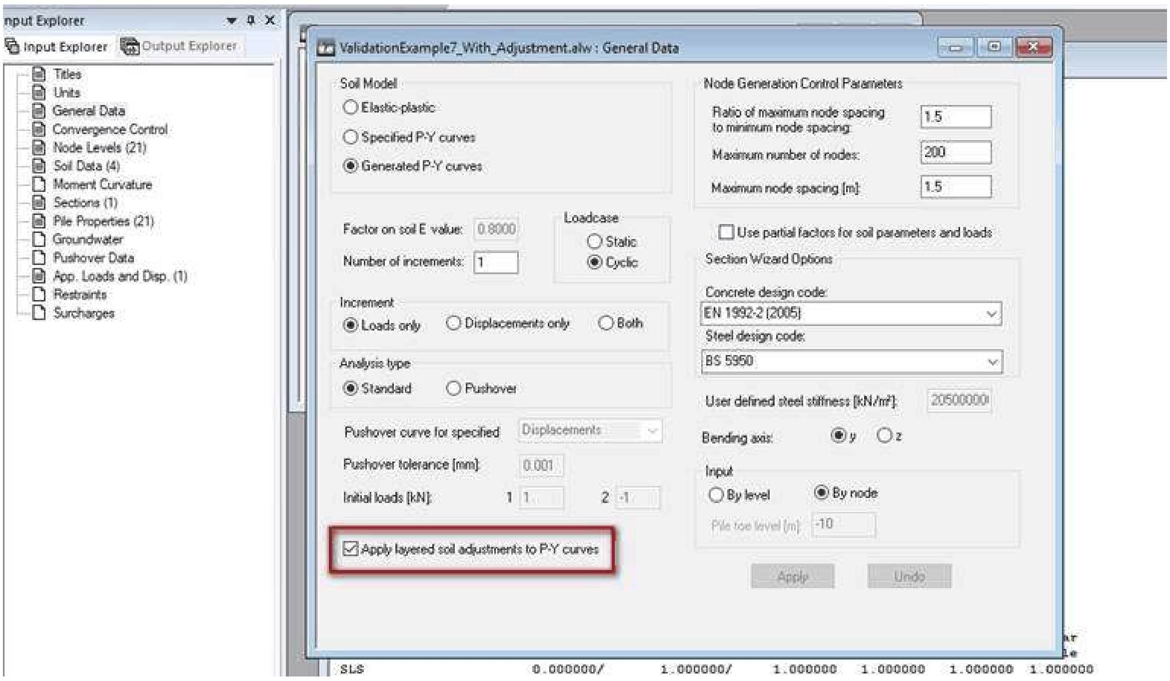
Calc Heading
Specific to this model.

The above items are reproduced in the title block at the head of all printed information for the calculations. The fields should therefore be used to provide as many details as possible to identify the individual calculation runs.

An additional field for Notes has also been included to allow the entry of a detailed description of the calculation. This is reproduced at the start of the data output.

Units
It is possible for user to enter the preferred units before entering the input data by clicking the Units button.

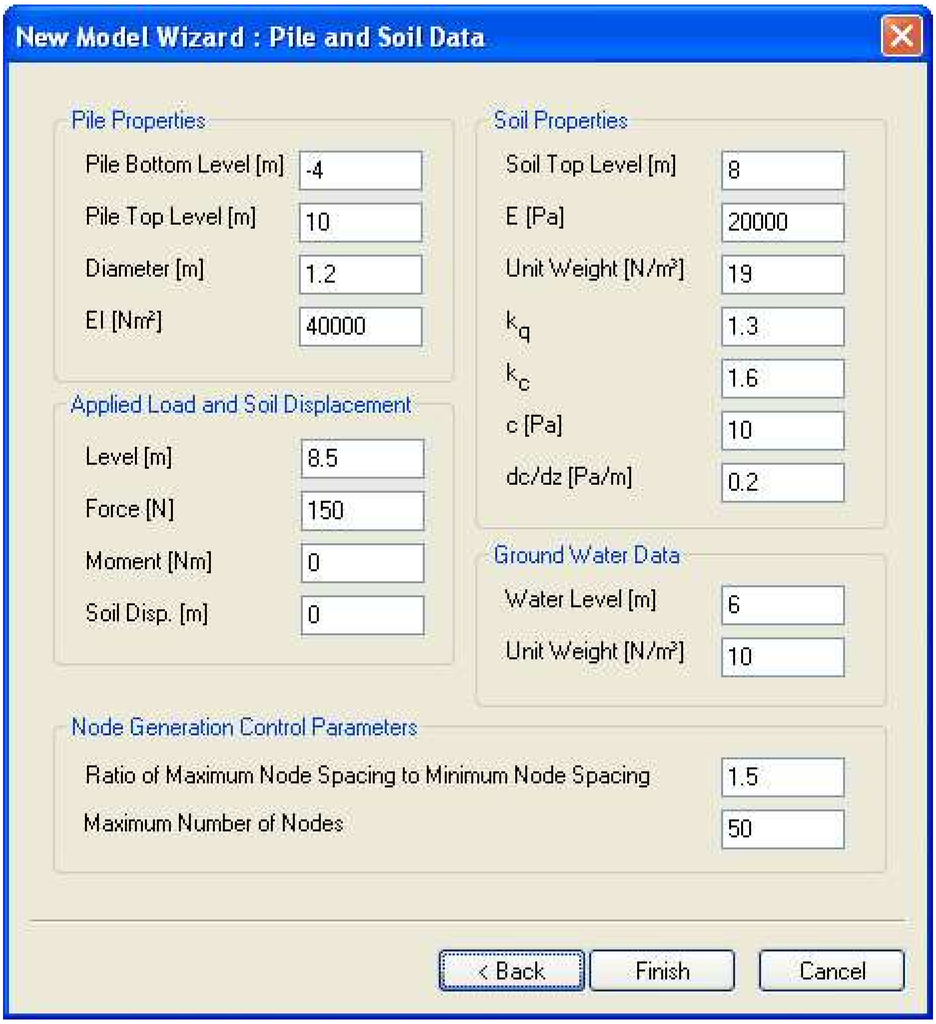
5.1.1.2.2 New Model Wizard : General Data



The general data is entered to define the outline of the problem and type of analysis to be carried out.
Only the Elastic-plastic soil model is available through the New model wizard.

The user can also choose to set a number of options for the model including the application of increments, to input data by node or level, whether or not to use partial factors, and whether to do a standard or pushover analysis. These dictate the available options when the initial model is created, and can be amended subsequently through the main General Data dialog.

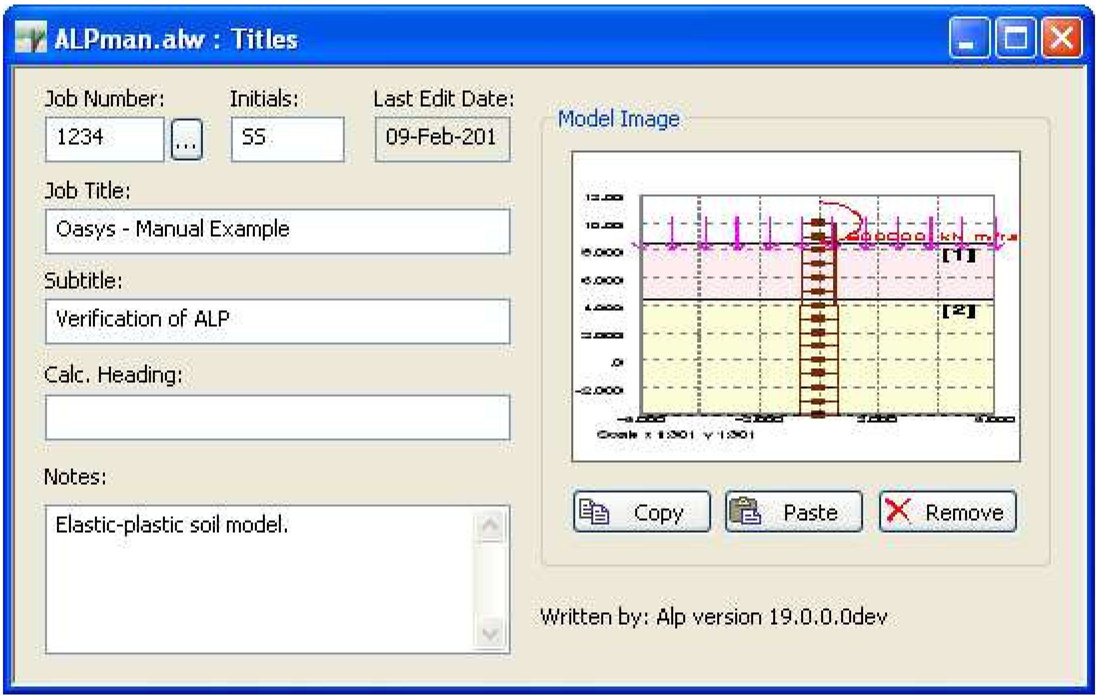
5.1.1.2.3 New Model Wizard : Pile and Soil Data



- Pile data, soil data, applied load, soil displacement, ground water data is entered through this page.
- Nodes and Pile properties are generated automatically by clicking the Finish button after entering data.
- Only a single soil layer and single pile with a constant diameter and properties is generated.

5.1.2 Titles

The first window to appear, for entry of data into **Alp**, is the Titles window. This view also accessible via Data | Titles from the program's menu.



This window allows entry of identification data for each program file. The following fields are available:

Job Number	allows entry of an identifying job number.
Initials	for entry of the users initials.
Date	this field is set by the program at the date the file is saved.
Job Title	allows a single line for entry of the job title.
Subtitle	allows a single line of additional job or calculation information.
Calculation Heading	this allows a single line for the main calculation heading.

The titles are reproduced in the title block at the head of all printed information for the calculations. The fields should therefore be used to provide as many details as possible to identify the individual calculation runs.

An additional field for **notes** has also been included to allow the entry of a detailed description of the calculation. This can be reproduced at the start of the data output by selection of notes using File | Print Selection, see the contents list for information on File Handling.

5.1.2.1 Titles window - Bitmaps

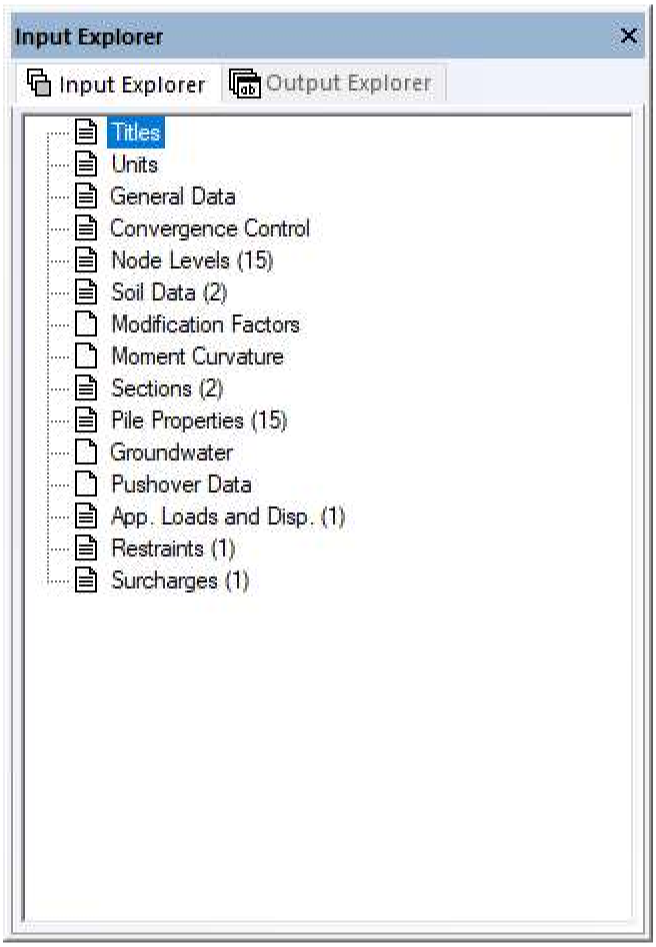
The box to the left of the Titles window can be used to display a picture beside the file titles. To add a picture place an image on to the clipboard. This must be in a RGB (Red/Green/Blue) Bitmap format. Select the Paste Bitmap button to place the image in the box.

The image is purely for use as a prompt on screen and can not be copied into the output data. Care should be taken not to copy large bitmaps, which can dramatically increase the size of the file.

To remove a Bitmap select the Remove Bitmap button.

5.2 Input Explorer

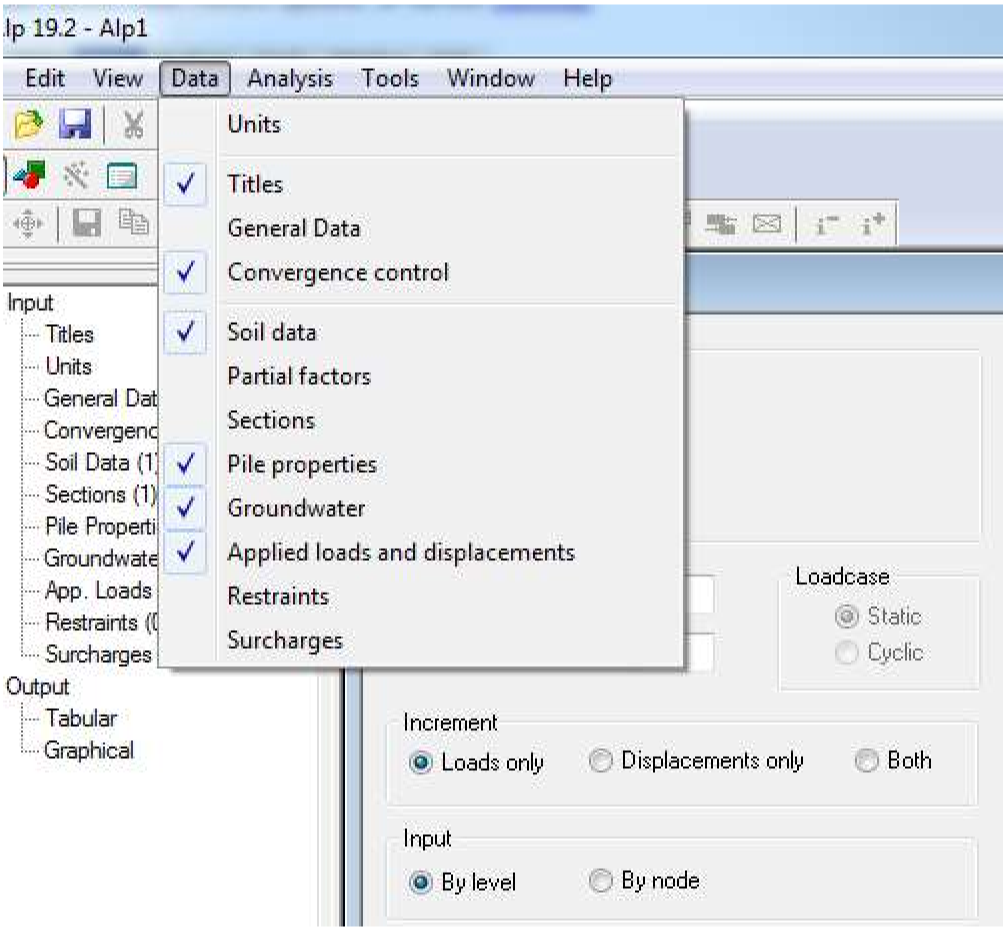
The Input Explorer is accessible from the 'View' menu, or via the 'Explorers' button on the Alp Toolbar. It allows access to all input data.



A context menu, to expand or collapse individual or all items, is available by right-clicking in the window.

5.3 Data Input

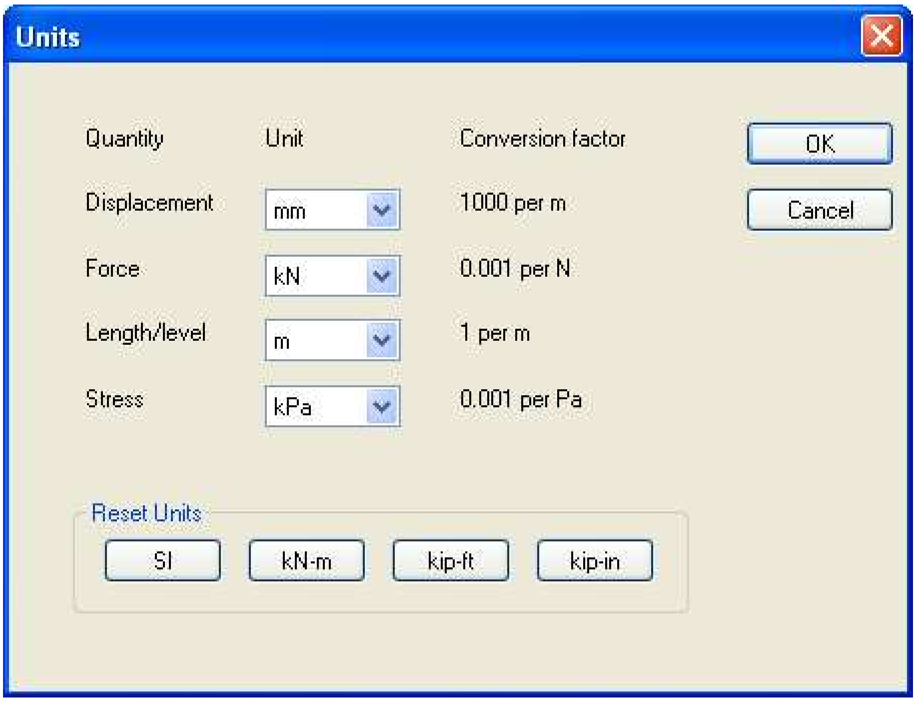
Data is input via the Data menu's options, or via the Gateway.



Once the data has been entered the program places a tick against that item in the menu list.

5.3.1 Units

This option allows the user to specify the units for entering the data and reporting the results of the calculations.



Units

Default options are the Système Internationale (SI) units - kN and m. The drop down menus provide alternative units with their respective conversion factors to metric.

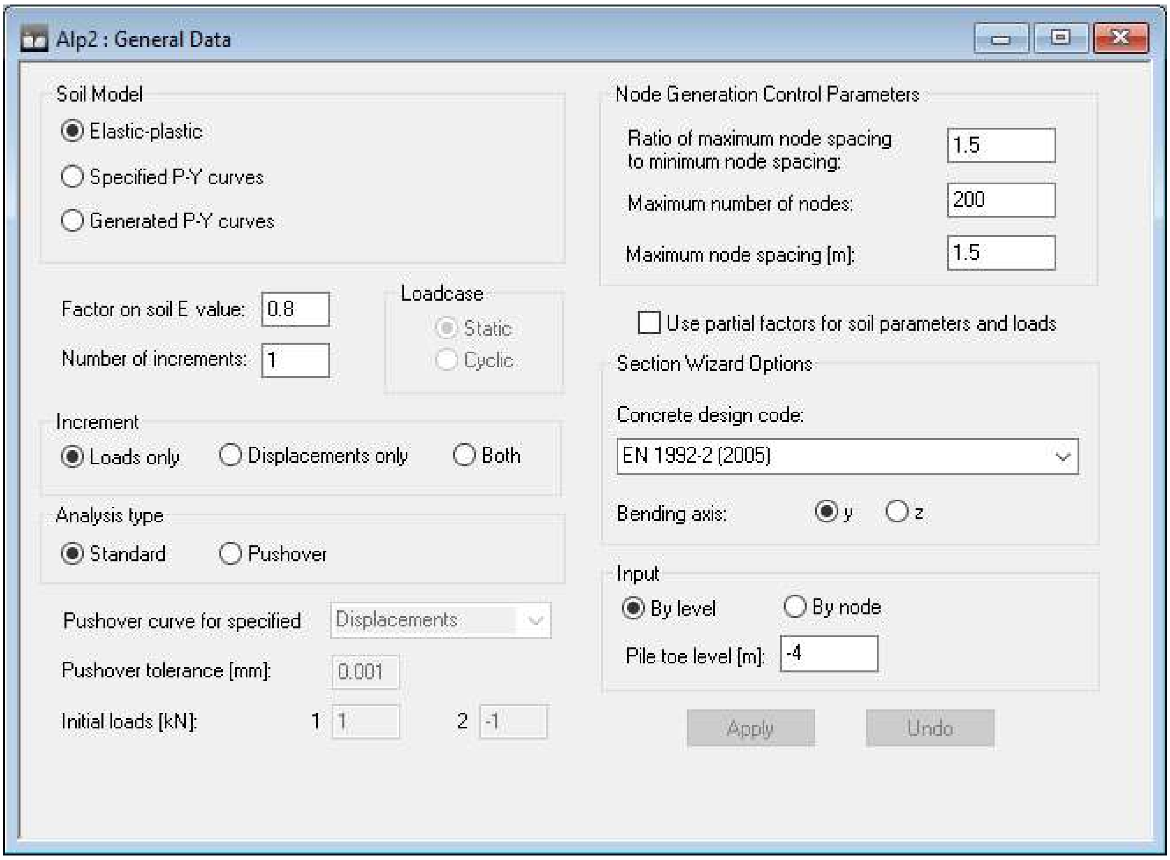
Standard sets of units may be set by selecting any of the buttons: SI, kN-m, kip-ft kip-in.

Once the correct units have been selected then click 'OK' to continue.

SI units have been used as the default standard throughout this document.

5.3.2 General Data

The following general data is entered to define the methods of solution.



Soil Model

Three soil models are available:

- Elastic Plastic
- Specified P-Y curves
- Generated P-Y curves with static or cyclic load cases

Selection of the required type amends the Soil Data table to the correct parameters, see Soil Data.

For the elastic-plastic model a factor on the E value, Efact, is required. This factor is discussed in Elastic-Plastic soil model. The default value is 0.8.

Loads and displacement increments

Applied loading can be defined explicitly or in the form of soil displacements. The actual values are specified in tabular form, see Applied Loads and Soil Displacements.

The user must specify the number of increments required to reach the applied load or displacement and then, if the number of increments is greater than one, whether the increments are applied to:

- 'Applied Loads ONLY' or
- 'Soil displacements ONLY' or
- 'Loads AND soil displacements'.

If only the loads or the displacements are incremented then the others act to the full value throughout the analysis.

Input and Node Generation Control Parameters

Data can be input by level, with a node set automatically generated. Using this option the user can directly specify the level of soil boundaries, loads, restraints and other features. The program will then automatically generate a set of nodes for the analysis. If input by level is selected the user will need to specify the pile toe level in the text box provided.

If the user chooses to input data by node, then the user can add each node explicitly. Loads and restraints are then assigned directly to nodes, and soil boundaries to the mid point between nodes.

Note: if input by level is chosen, the user should check that the generated node set is appropriate for the analysis. It is possible to switch from input by level to input by node after the nodes have been generated, and either add, delete or edit nodes.

Partial Factors

If partial factors are required, the partial factor option must be checked. This will make the partial factor dialog available, and also add partial factor options to the soil and load tables.

Section Wizard Options

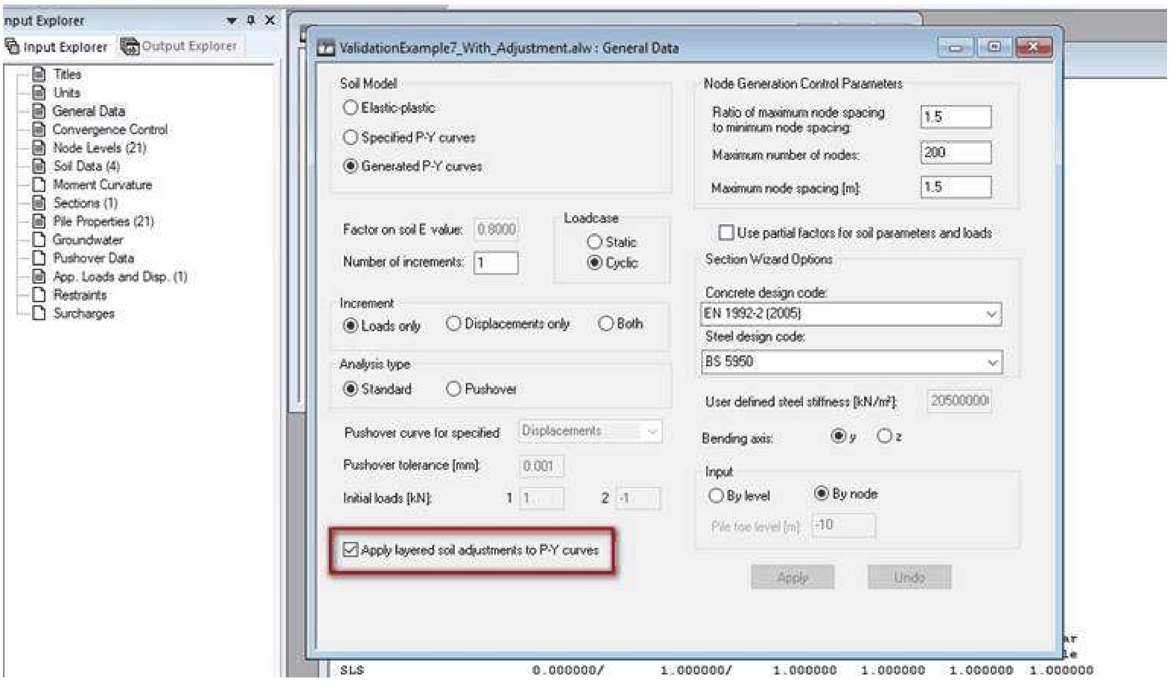
If using the section wizard to generate pile section properties, these options can be used to select which concrete design code and bending axis is considered by the wizard. The concrete design code is used to generate a list of concrete mixes specified by the code, and obtain the specified E value to calculate the section stiffness.

Analysis Type

Using the analysis type drop down box the user can select either a standard or pushover analysis. Where the user selects the pushover option they can they choose either to set specified loads or specified pile displacements to calculate the pushover curve. Additionally, for the specified displacements which use an iterative solution, the user can specify the tolerance for acceptance on the calculated displacement and the initial iteration loads.

Layered soil adjustments to P-y curves

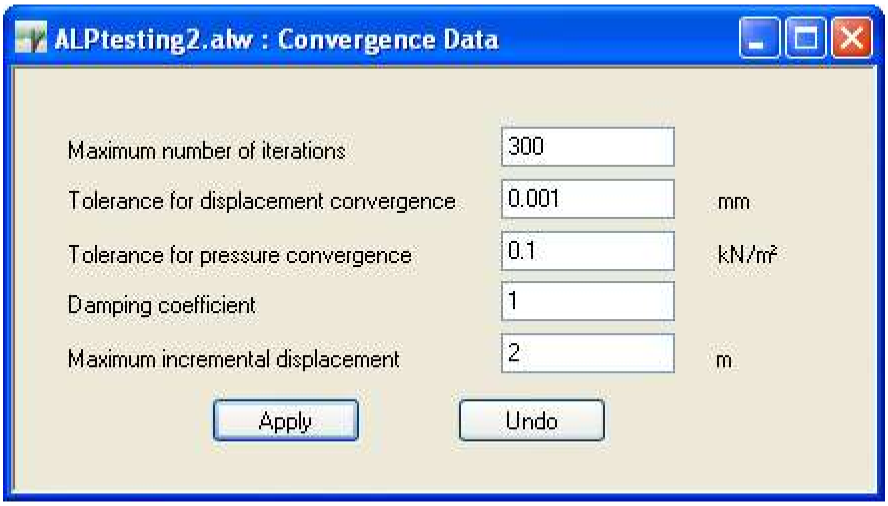
If this option is activated as shown below, the program calculates the equivalent soil layer depth based on the theory outlined in Georgiadis (1983).



These adjustments apply to ONLY “Generated P-y curves”, and not to “Elastic-plastic” or “Specified P-y curves” options. Also, in “Generated P-y curves” option, this feature is applicable ONLY to sand and clay soil types – not to strong rock and weak rock.

5.3.3 Convergence Control Parameters

The following convergence controls are required.



Number of iterations

The maximum number of iterations allowed by the user. The default value is 300.

Displacement error

The maximum displacement error in any one iteration. This represents the maximum change of displacement between successive iterations. The absolute error in the result will be considerably larger (typically by a factor of 10 to 100). The default value is 0.001mm.

Pressure error

The maximum error in pressure (i.e. how much the pressure at any node is below the active limit or in excess of the passive limit). This is an absolute value and the default value is 0.1kN/m².

Damping Coefficient

The damping coefficient used in the analysis. If convergence is slow this can be increased. If instability is apparent it may possibly be solved by reducing this. The default value is 1.0.

Incremental deflection

Maximum deflection in one increment. The default value is 2.0m.

Note! Convergence control parameters may be varied from the default values offered to improve the speed/accuracy of the solution, or to reduce the chance of numerical instability.

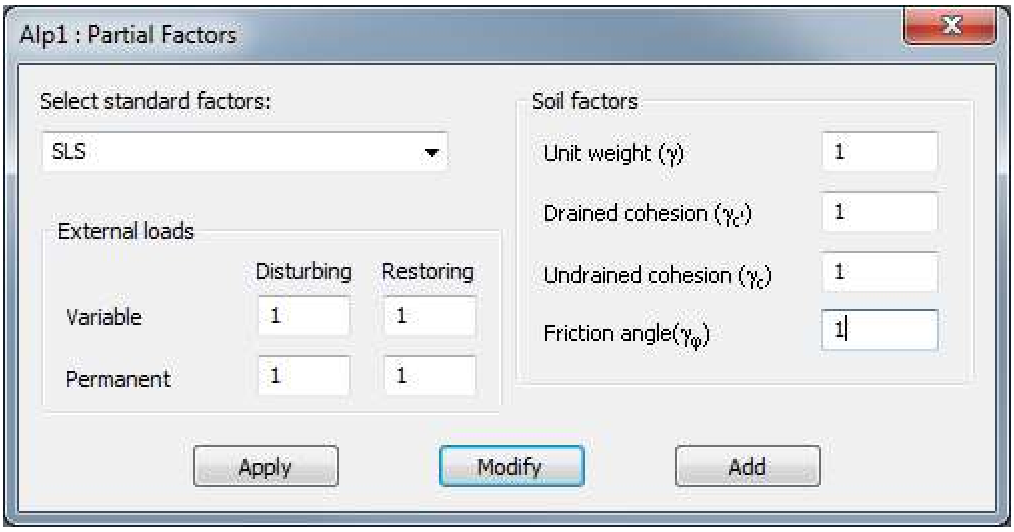
The program solution will iterate until the two tolerance conditions are satisfied, subject to the specified maximum number of iterations not being exceeded.

5.3.4 Partial Factors

Partial factors dependent on the method and on material parameters can be specified. A default set of partial factors is hard coded into the program and is not editable. This default set includes factors recommended by:
- Eurocode 7 Design Approach 1, combinations 1 and 2
- SLS (all values = 1.0)

Method partial factors can be applied to:
- favourable or unfavourable loads
- soil unit weight
- drained or undrained cohesion
- soil friction angle

User-specified factors can be added and will then be stored in an XML file that is included in Alp - if a data file containing user-defined values is sent to another user, the values will be extracted from the data and saved to the second user's XML file.



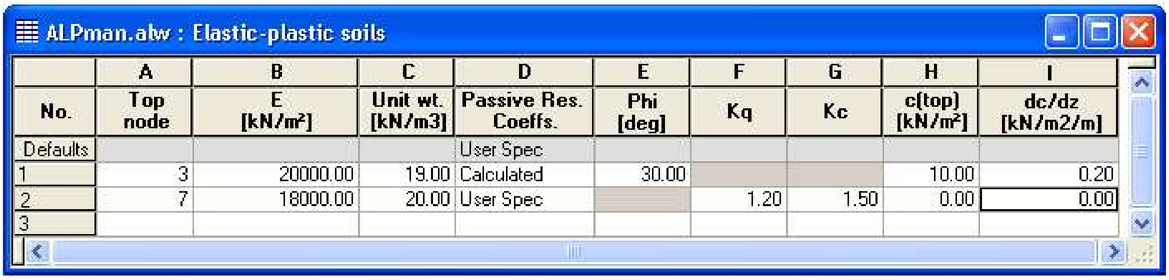
5.3.5 Soil Data

The table of input soil parameters is governed by the selection of soil model in the General Data, see General Data. Soil data can be entered in tabular form by selecting 'Soil Data' from the Data menu or the Gateway.

Note! Stratum boundaries occur midway between the top node and the node immediately above it, except when the top node coincides with the top of the pile.

5.3.5.1 Elastic-Plastic Data

Details of the elastic-plastic soil model are given in Elastic-Plastic soil model. The required data entry is as follows:



	A	B	C	D	E	F	G	H	I
No.	Top node	E [kN/m²]	Unit wt. [kN/m3]	Passive Res. Coeffs.	Phi [deg]	Kq	Kc	c(top) [kN/m²]	dc/dz [kN/m2/m]
Defaults				User Spec					
1	3	20000.00	19.00	Calculated	30.00			10.00	0.20
2	7	18000.00	20.00	User Spec		1.20	1.50	0.00	0.00
3									

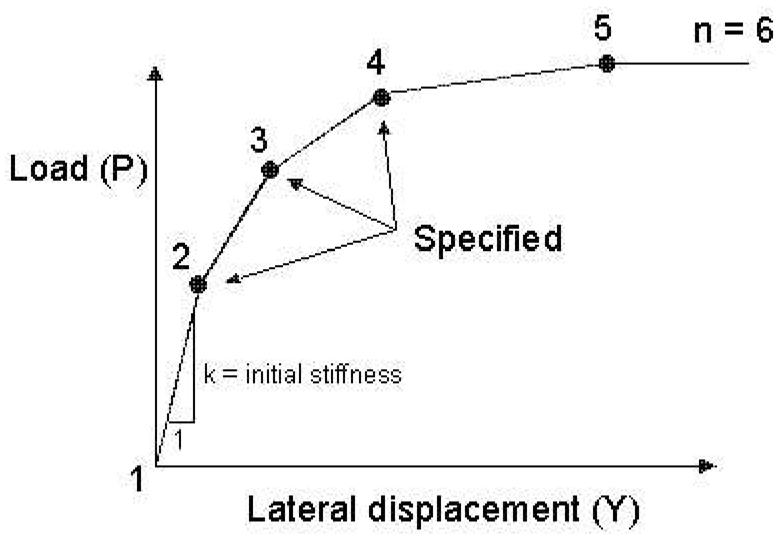
Top Node or **Top Level**	Top node (if input by node) or top level (if input by level) within this stratum
E	Young's modulus
Unit Weight	Bulk unit weight
Passive Res. Coeffs.	The user may input the soil friction (Phi) that is used to calculate the Passive Resistance Coefficients as established by Brinch Hansen (1961). Alternatively K_q and K_c can be overridden as per user's choice.
Phi	Internal soil friction phi can vary between 0.01 to 60 degrees
K_q	Passive resistance coefficient for the frictional component of the soil
K_c	Passive resistance coefficient for the cohesive component of the soil.

$c_{(top)}$	Cohesion
dc/dz	Rate of change of cohesion with depth

If partial factors are active the user will also need to use the drop down menu to indicate whether the analysis is drained or undrained for each stratum.

5.3.5.2 Specified P-Y Data

If P-Y curves are specified for each node, the coordinates (P_n, Y_n) are input as follows:



ALPtesting.alw : Specified P-Y curves

	A	B	C	D	E	F	G	H	I	J	K	L
Node	P1 [kN/m]	Y1 [mm]	P2 [kN/m]	Y2 [mm]	P3 [kN/m]	Y3 [mm]	P4 [kN/m]	Y4 [mm]	P5 [kN/m]	Y5 [mm]	P6 [kN/m]	Y6 [mm]
Defaults												
1	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
2	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
3	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
4	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
5	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
6	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
7	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
8	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
9	0	0.0	5	280.0	10	809.0	20	1715.0	0	0.0	0	0.0
10	0	0.0	11	280.0	17	809.0	47	1715.0	0	0.0	0	0.0
11	0	0.0	25	280.0	36	809.0	69	1715.0	0	0.0	0	0.0
12	0	0.0	2	97.0	5	128.0	8	209.0	0	0.0	0	0.0
13	0	0.0	9	97.0	12	128.0	15	209.0	0	0.0	0	0.0
14	0	0.0	16	97.0	19	128.0	21	209.0	0	0.0	0	0.0
15	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0	0	0.0
16												

User specified P-Y data at each node represents the load deflection behaviour of the soil to a distance of half the element above and below the node. The program assumes perfectly plastic behaviour for values of displacement in excess of the final specified displacement. A maximum of 6 specified points can be used with the first point being set at (0,0).

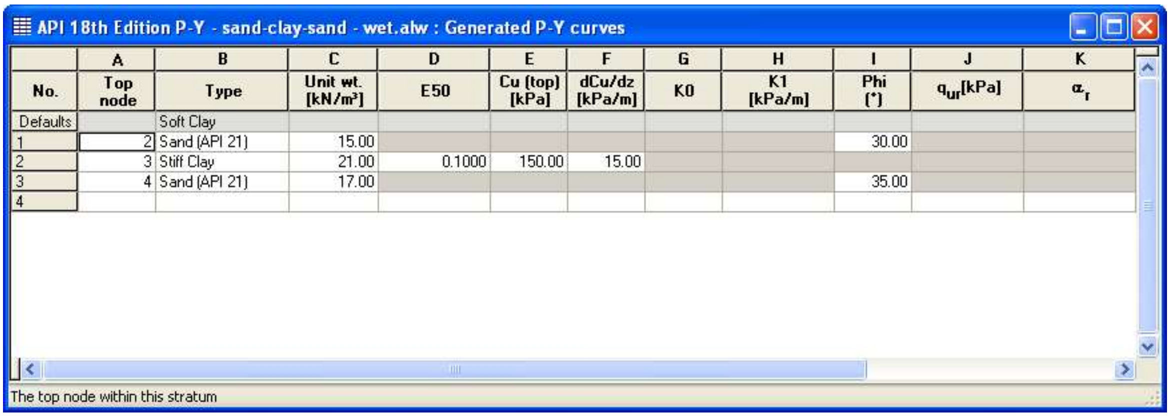
For nodes above the soil surface only the first point (0,0) is specified.

The units for the P-Y curves are:

P = force/unit length of pile
Y = displacement

5.3.5.3 Generated P-Y Data

Guidance on the choice of suitable curves for particular soils for both static and cyclic load cases is given in Generated P-Y curves.



	A	B	C	D	E	F	G	H	I	J	K
No.	Top node	Type	Unit wt. [kN/m³]	E50	Cu (top) [kPa]	dCu/dz [kPa/m]	K0	K1 [kPa/m]	Phi [°]	q_{ur}[kPa]	α_r
Defaults		Soft Clay									
1	2	Sand (API 21)	15.00						30.00		
2	3	Stiff Clay	21.00	0.1000	150.00	15.00					
3	4	Sand (API 21)	17.00						35.00		
4											

The loading type, 'static' or 'cyclic', must be specified in the General Data, see General Data. For each soil layer the top node or top level must be specified, for input by node and input by level settings respectively. Using the drop down menu, the soil must be defined as 'Soft clay', 'Stiff clay', 'Sand (API 18)', 'Sand (API 21)', 'Weak Rock' or 'Strong Rock' .

Note: Alp does not consider the loading type i.e. 'static' or 'cyclic' for soils defined as weak or strong rocks.

If either the 'Soft clay' or 'Stiff clay' option is selected, the following soil properties are required for each stratum:

Unit weight	Bulk unit weight
E50	Strain at half the maximum stress for an undrained triaxial test, see Generated P-Y curves
$c_{u(top)}$	Undrained shear strength at the top of the stratum
dc_u/dz	Rate of change of undrained shear strength with depth

If the 'Sand (API 18)' or 'Sand (API 21)' option is selected, the following soil properties are required for each stratum:

Unit weight	Bulk unit weight

K_0	Coefficient of earth pressure at rest
K_1	Initial soil modulus (k1), see Generated P-Y curves
Phi	Angle of internal friction

If the 'Weak Rock' option is selected, the following soil properties are required for each stratum:

Unit weight	Bulk unit weight
q_{ur}	Compressive strength of rock
α_r	Strength reduction factor
k_{rm}	Dimensionless constant, ranging from 0.0005 to 0.00005
E_{ir}	Initial modulus of rock

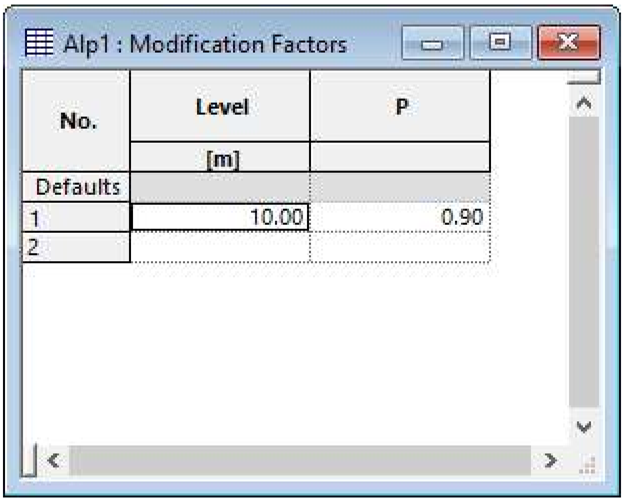
If the 'Strong Rock' option is selected, the following soil properties are required for each stratum:

Unit weight	Bulk unit weight
q_{ur}	Compressive strength of rock

If partial factors are active the user will also need to use the drop down menu to indicate whether the analysis is drained or undrained for each stratum.

5.3.6 Modification Factors

Data may be input in tabular form in this input table.

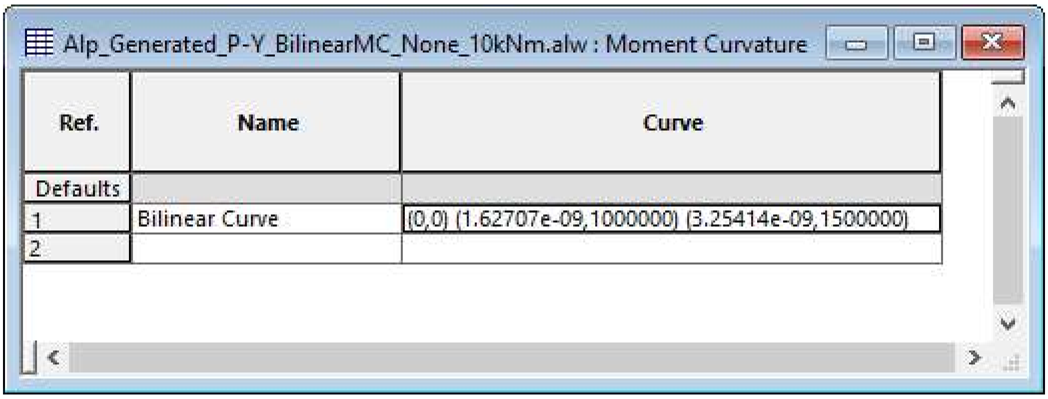


Level specifies the level.

P specifies the factor to be multiplied.

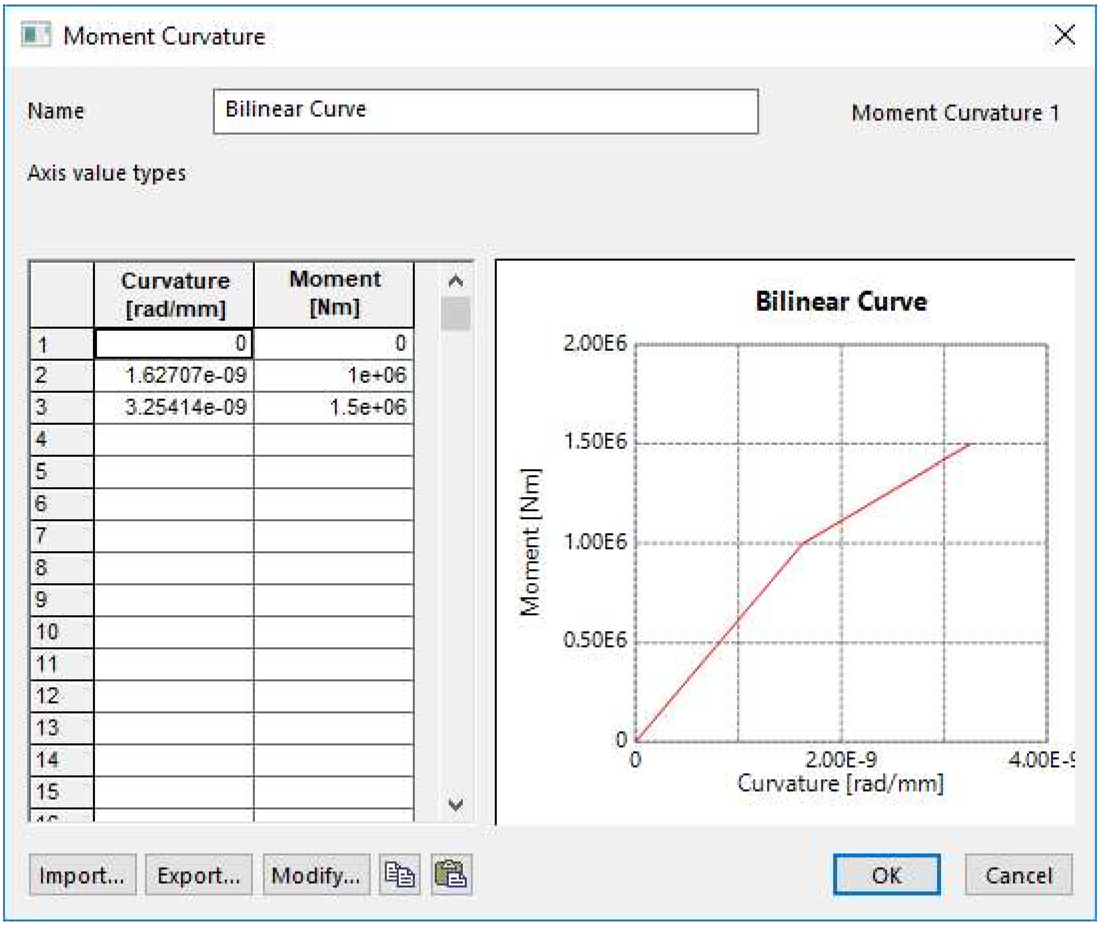
5.3.7 Moment Curvature

Data may be input in tabular form in this input table, or in dialog form by double-clicking within a cell or by clicking the 'Wizard' button on the Alp Toolbar.



Name specifies the name of the curve.

Curve specify the points of the curve.



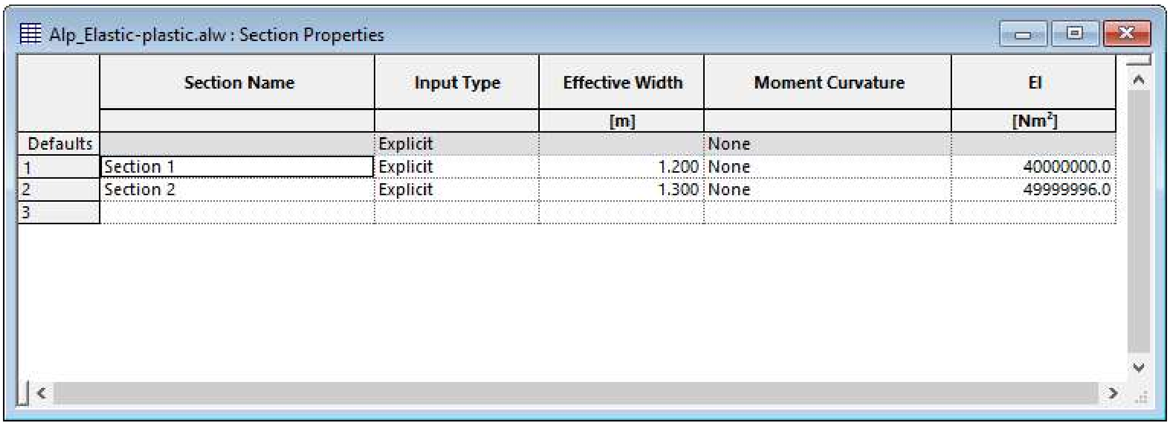
5.3.8 Sections

A range of pile sections can be entered by the user, these define the effective width/diameter and the flexural stiffness of the pile where the section is used.

Sections can either be entered explicitly or generated using the section wizard.

5.3.8.1 Explicit

Using the explicit method the section details are entered directly into the section properties table.



Alp_Elastic-plastic.alw : Section Properties

	Section Name	Input Type	Effective Width	Moment Curvature	EI
			[m]		[Nm²]
Defaults		Explicit		None	
1	Section 1	Explicit	1.200	None	40000000.0
2	Section 2	Explicit	1.300	None	49999996.0
3					

Section Name
Enter a unique name for the section.

Input Type
Using the drop-down list select 'Explicit' to type in the pile parameters.

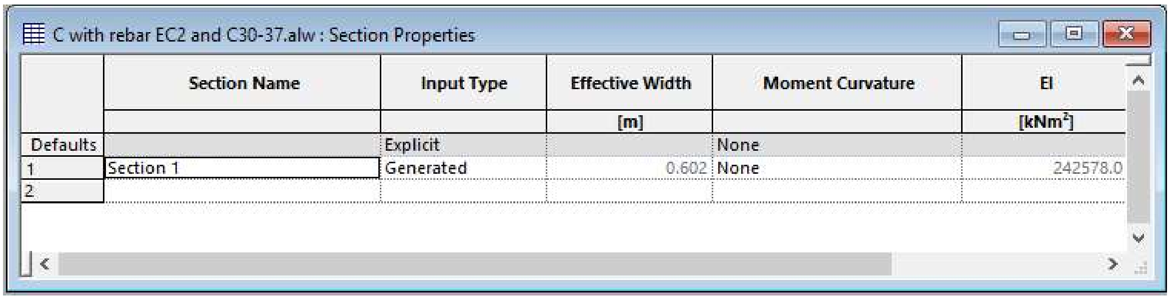
Effective Width
The effective width of the pile, i.e. the width perpendicular to the direction of loading.

Moment Curvature
Reference to the curve defined in Moment Curvature that is to be used for calculations.

EI
The pile flexural stiffness is typed into the relevant column if 'None' is selected as the Moment Curvature curve.

5.3.8.2 Generated

Using the generated method the section details are calculated using the section wizard.



C with rebar EC2 and C30-37.alw : Section Properties

	Section Name	Input Type	Effective Width	Moment Curvature	EI
			[m]		[kNm²]
Defaults		Explicit		None	
1	Section 1	Generated	0.602	None	242578.0
2					

Section Name
Enter a unique name for the section.

Input Type
Using the drop-down list select 'Generated' to type in the pile parameters. This will automatically open the section wizard.

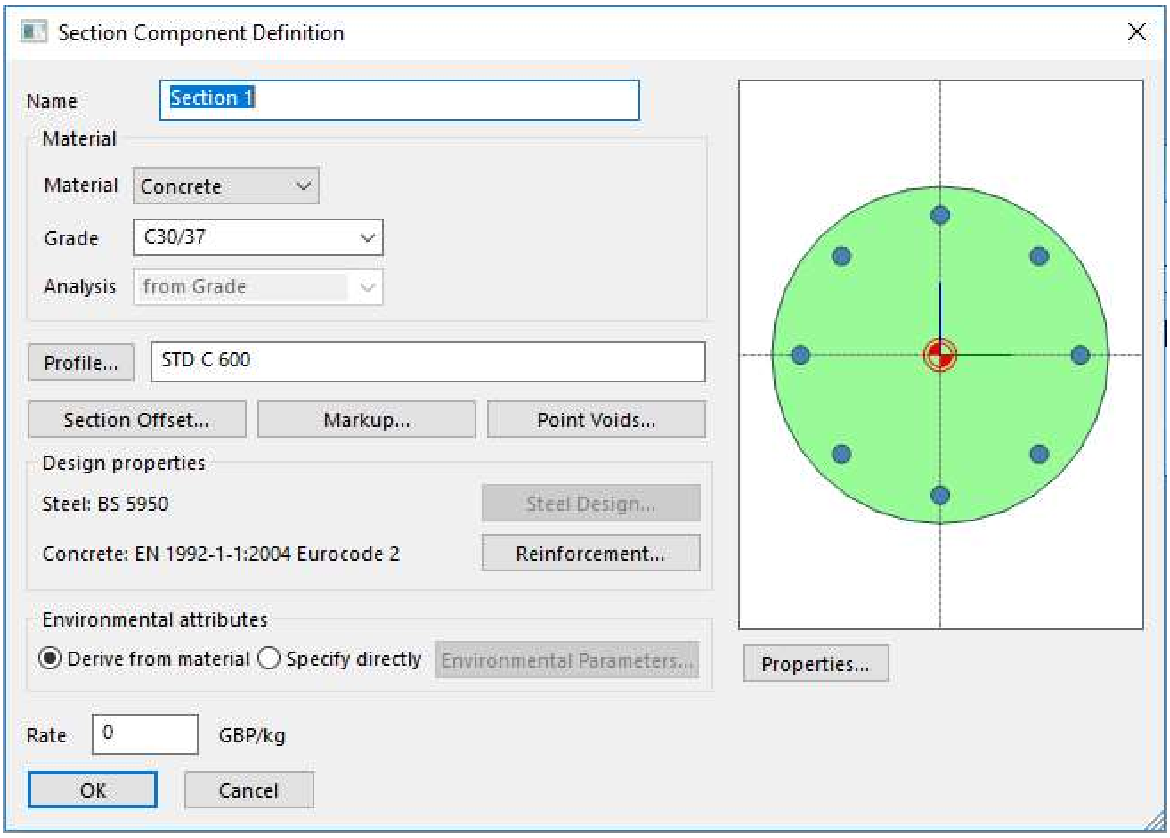
Effective Width
The effective width of the pile, this box is non-editable for 'Generated' entries and shows the value calculated by the wizard.

EI
The pile flexural stiffness is non-editable for 'Generated' entries and shows the value calculated by the wizard.

Editing Existing Sections
To edit an existing section right click anywhere within the relevant row then select 'Wizard' from the drop down menu. Alternatively select a cell within the relevant row and click on the wizard button . This will reopen the section wizard.

5.3.8.2.1 Section Wizard

The section wizard contains a number of options to allow the user to define the pile section. The section wizard is a shared object used by several Oasys software programs. Some of the options used by other programs are not applicable to Alp and have been deactivated (including the environmental attributes, modifiers, miscellaneous and compound section). The available options are summarised below.



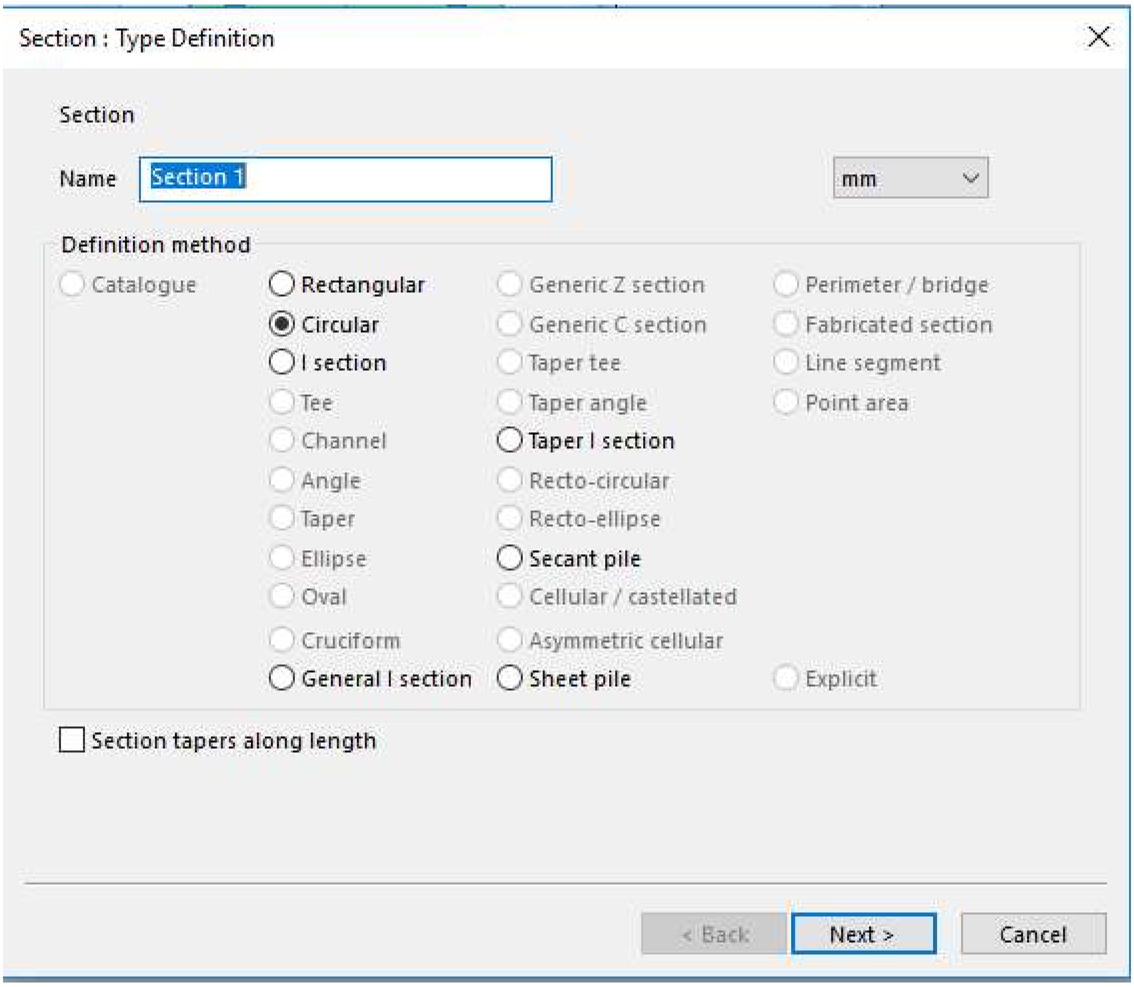
Member Type
The drop down box can be used to assign the type of section being used. This defaults to 'Pile' and it is recommended that this option is used. The 'General' member type can also be selected. This allows a wider range of section profiles to be selected, however, it does not calculate an

effective width for the pile.

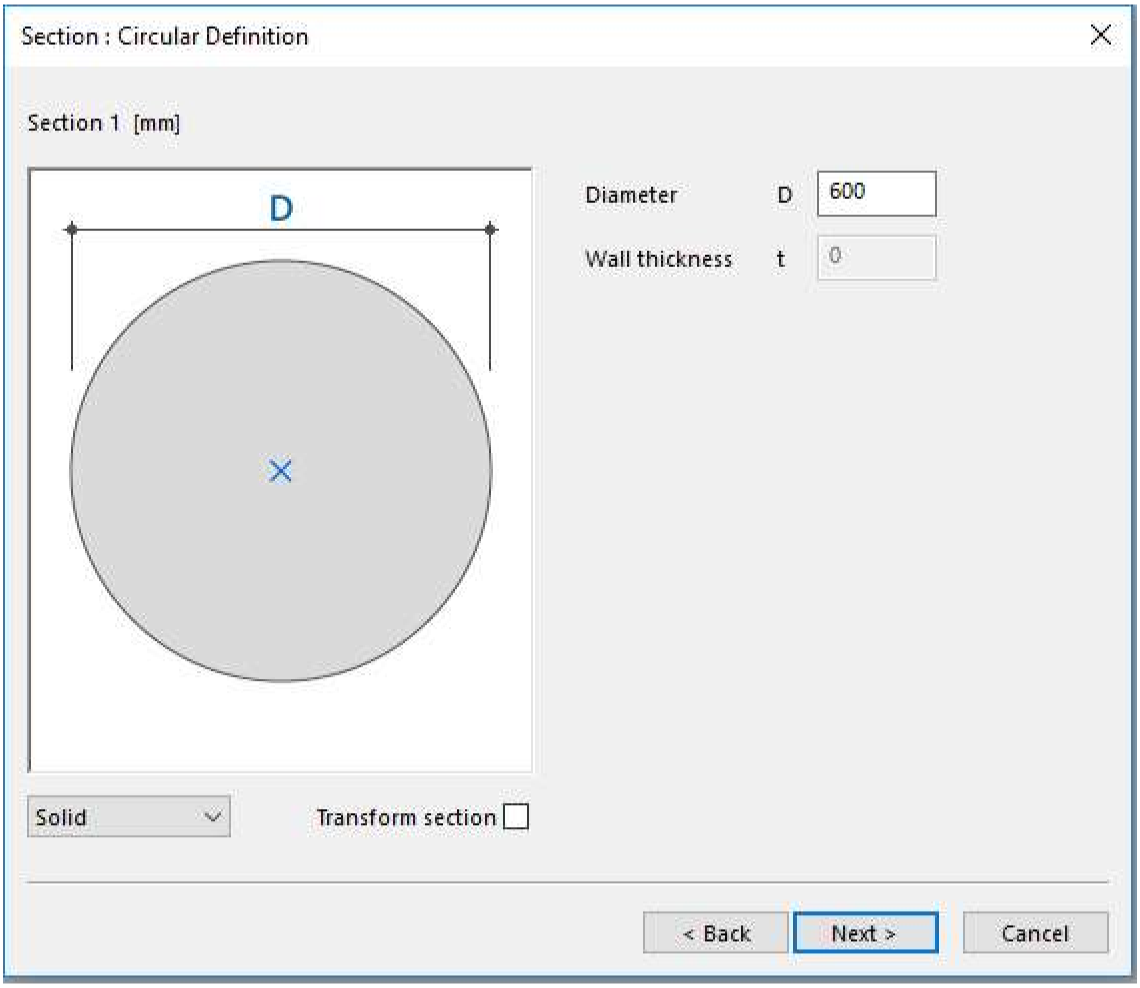
Material Type
The drop down box can be used to select either a concrete or steel pile.

Material
The specific class of material is selected using the 'Material' drop down list. For concrete the available classes is affected by the design code selected in the general data dialog.

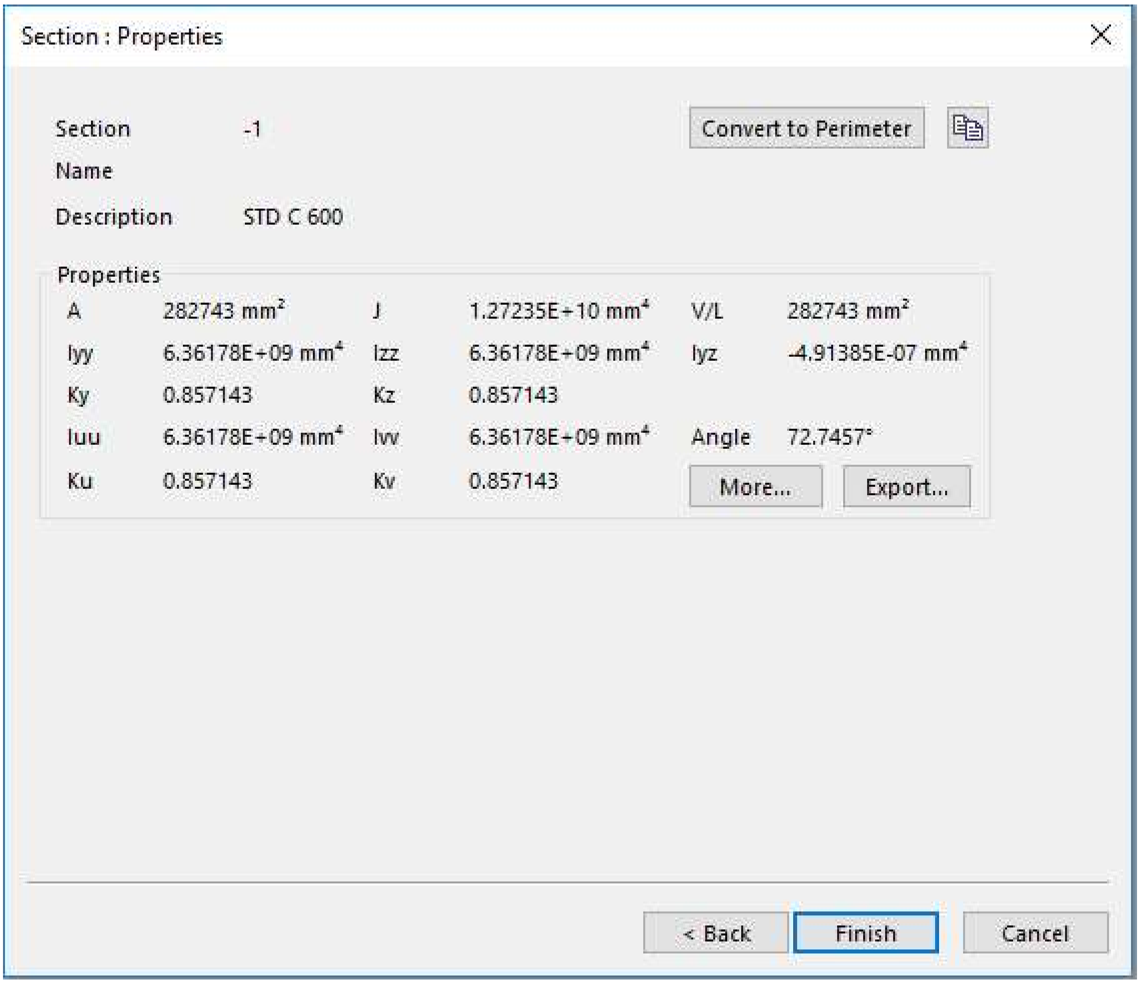
Profile
Clicking on the 'Profile' button opens up a series of pages that can be used to define the shape of the pile. Alternatively, where the user is familiar with the codes, they may wish to specify section profiles directly by typing the shape code directly into the text box adjacent to the 'Profile' button.



The 'Definition' page allows the user to select the general pile shape by clicking on the appropriate radio button, then clicking 'Next'.



The shape section definition page allows the user to input the dimensions of the section. The units and solid/hollow section options are selected using the drop down boxes in the top left of the screen. The dimensions are then typed into the relevant boxes. If the transform section option is checked this will allow the user to rotate the section, e.g. for consideration of loads that are non perpendicular to the principle axes.



The final page of the profile wizard shows the parameters calculated for the section. Additional options available include:

- Convert to Perimeter - this stores and defines the shape as a series co-ordinates that mark the external boundary. This affects how the shape and described in the main page and consequently how it can be modified subsequently by the user.
- More - this shows further section properties calculated by the wizard.
- Export - this option creates a table listing the section properties as a .dxf file.

Following the creation of the section profile the main window of the wizard is amended to show the section profile and include the section description.

The text in the section description can be amended to edit the section profile. The format for codes is listed in the shape codes section. Where the user is familiar with the codes, they may wish to specify section profiles directly by typing in the shape code.

Component Attributes

Reinforcement

Clicking on the 'Reinforcement' button opens up the main reinforcement dialog, that can be used to add reinforcement bars to a concrete section.

Properties

This button opens the properties dialog box. This lists the properties calculated for the section profile input by the user.

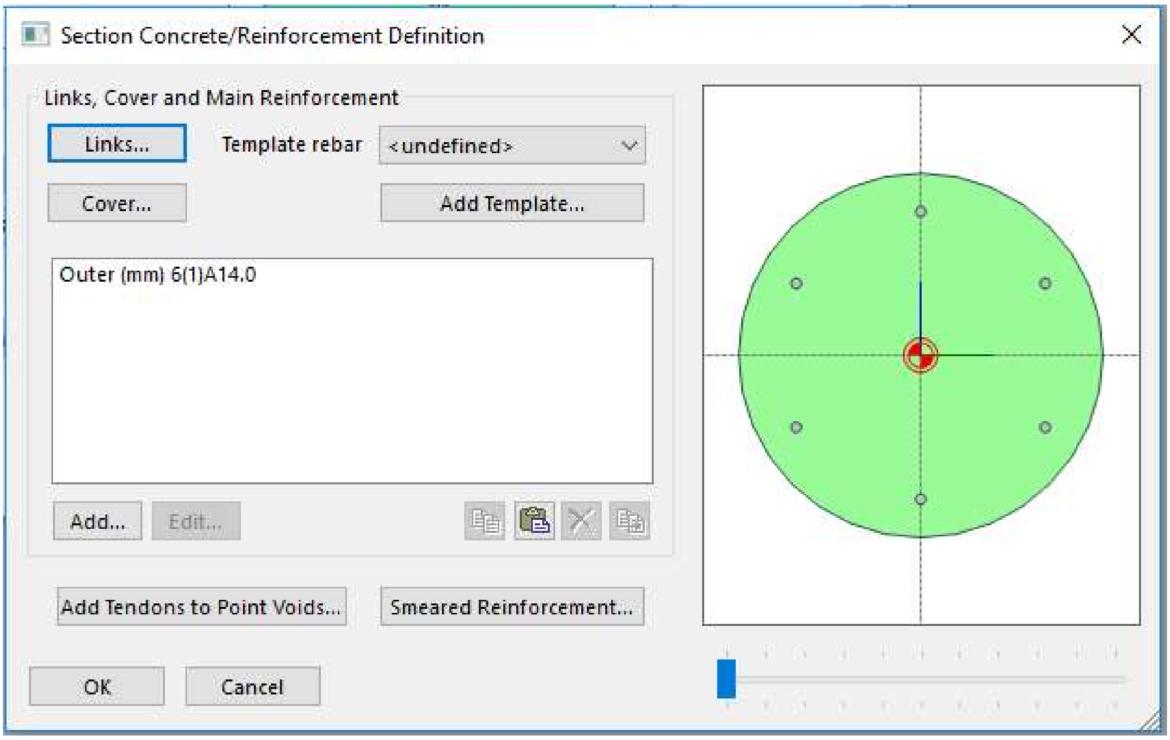
5.3.8.2.1.1 Shape Codes

Shape codes can be used to explicitly define a section shape. Codes for typical pile shapes are listed below.

Shape	Code							
	Text	Number 1	Number 2	Number 3	Number 4	Number 5	Number 6	Number 7
Rectangle	STD R	Depth	Width					
Hollow Rectangle	STD RHS	Depth	Width	Thickness (D walls)	Thickness (W walls)			
Circular	STD C	Diameter						
Hollow Circular	STD CHS	Diameter	Thickness					
I Section	STD I	Depth	Width	Thickness (web)	Thickness (flange)			
General I Section	STD GI	Depth	Width (flange top)	Width (flange bottom)	Thickness (web)	Thickness (flange top)	Thickness (flange bottom)	
Tapered I Section	STD TI	Depth	Width (flange top)	Width (flange bottom)	Thickness (web top)	Thickness (web bottom)	Thickness (flange top)	Thickness (flange bottom)
Secant Pile Wall	STD SPW	Pile Diameter	Pile Centre Spacing	Number of Piles				
Secant Pile Section	STD SP	Pile Diameter	Pile Centre Spacing	Number of Piles				

5.3.8.2.1.2 Reinforcement

Reinforcement is added to the generated section through the reinforcement dialog.



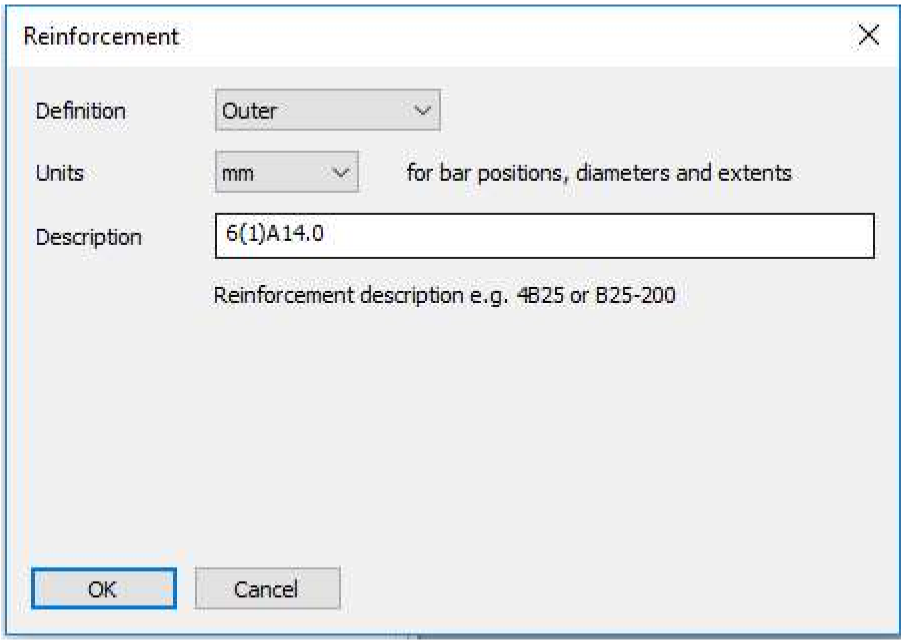
Reinforcement Settings

The 'Links', 'Cover' and 'Smeared Reinforcement' buttons, open up dialog boxes that allow the user to set the default values for these options.

Adding / Editing Reinforcement

To create a new reinforcement click on the 'Add...' button or right click within the list box and select 'Add' from the drop-down menu. To edit an existing reinforcement entry, select the required entry in the list box then either click on the 'Edit...' button, or right click on the selected section and select 'Modify' from the drop-down menu.

Either option will open the reinforcement description dialog. Enter the required description using the guidelines in the Reinforcement Syntax page. For example, "6(1)A 14.0" represents 6 groups of one bar per group, reinforcement grade A, diameter 14mm.



Deleting Reinforcement

To delete reinforcement, select the required reinforcement entry from the list and either click on the delete icon , or right click and select delete from the drop down menu.

Bar group type

This specifies the type of group. There are several options:

- Group types that position bars parametrically, based on the section geometry
  Top, Bottom, Side, Left, Right, Perimeter and Inner perimeter
- Group types that position bars independently of the section geometry
  Point, Line, Circle and Arc

Units

The default units for all dimensions is mm. Other units can be specified here.

Bar group position

Where the bar group type is independent of section geometry, bar position inputs are required

Group type	**Position inputs required**
Point	coordinates of the bar
Line	start and end coordinates of the line
Circle	centre, radius and angle of first bar (relative to section's y axis)
Arc	centre, radius, start angle and sweep angle

Bar group definition

The definition of the bars is a text string that describes the both the layout of the bars and the bar's material.

Simple bar groups

The simplest bar groups can be defined as a number of bars or bars at a given pitch

Intention	Definition string	Explanation
Group defined by number of bars	4B16	4No grade 'B' bars, diameter 16mm
Group defined by pitch	B16-200	16mm diameter grade 'B' bars at 200mm centres

Note, the bar grade is either the material grade tag (defined in the reinforcement grade property) or the grade name.
For example, in Eurocode, 500B reinforcement can be referred to as "B" (e.g. 4B16) or "500B" (e.g. 4"500B"16)

Multiple layers of bars

Groups can contain multiple layers of reinforcement by adding '+' between two (or more) simple groups

Intention	Definition string	Explanation
Two layers by number of bars	4B20 + 2B16	One layer with 4No 20mm bars, second layer with 2No 16mm bars
Group defined by pitch	B16-200+ B10-400	One layer with 16mm bars at 200mm centres and a second layer with 10mm bars at 400mm centres

Groups can also contain layers of mixed number/pitch layers.

Bundled bars

Where bundled bars are required, the bar diameter can be prefixed with the number of bars per bundle

Intention	Definition string	Explanation
Bundled bars by number	4(2)B16	4No bundles, each containing 2No 16mm bars
Bundled bars by pitch	(2)B10-150	Bundles of 2No 10mm bars at 150mm centres

Multiple bar definitions in the same layer

Where a single layer requires multiple bar diameters there are two options

Intention	Definition string	Explanation
Multiple diameters	2B32 2B16 2B32	6No bars in a single layer. The input order is maintained.
Alternately placed bars	B20-200 B16-200 AP	16mm bars alternately placed with 20mm bars at 200mm centres. The pitch must be equal.

Bar group extents

Where the bar group is being defined in the context of a 1D member, extents can be used to define the position of the group along the length of the member. This can be done using absolute or relative positioning.

Intention	Definition string	Explanation
Group to exist along whole member	0%:100%	Bar group starts at 0% and extends to 100% of the member. This is the default if not specified.
Group to exist in the middle half of the member	25%:75%	Bar group starts at 25% and extends to 75% of the member length.
Group to start 900mm from the member start and end at the member end	900:100%	Absolute numbers refer to absolute dimensions. Percentages are relative to the member length.
Group to start at the member start and end 900mm from the member end	0:-900	Negative numbers are relative to the member end (whereas positive numbers relative to the member start).

5.3.9 Pile Properties/Node Levels

The pile properties and node level pages are considered together as they present the same input options and data. When using the input by node setting the table is identical for both. When using the input by level setting, the pile properties table is used to input the level for the top of pile, base of pile and any change in section and the corresponding section type. In this setting the node levels table is only accessible once nodes have been generated for the pile and shows the node levels for user information only - these levels are not editable.

The pile is modeled as a series of elastic beam elements. The details of the which are given in Pile Stiffness. The required input data is the pile section (which gives the pile diameter and bending stiffness (EI)) and the level of nodes (which are generated automatically if the input by level setting is used). The diameter and bending stiffness can be varied along the length of the pile, at the positions of nodes.

The values of diameter and flexural (bending) stiffness (EI) are assumed constant between nodes, i.e. they change **at** the nodes. For generating the P-Y curve coordinates at a node, the diameter specified **below the node** is used in the calculations.

As well as defining changes in the pile properties, nodes are also required at:

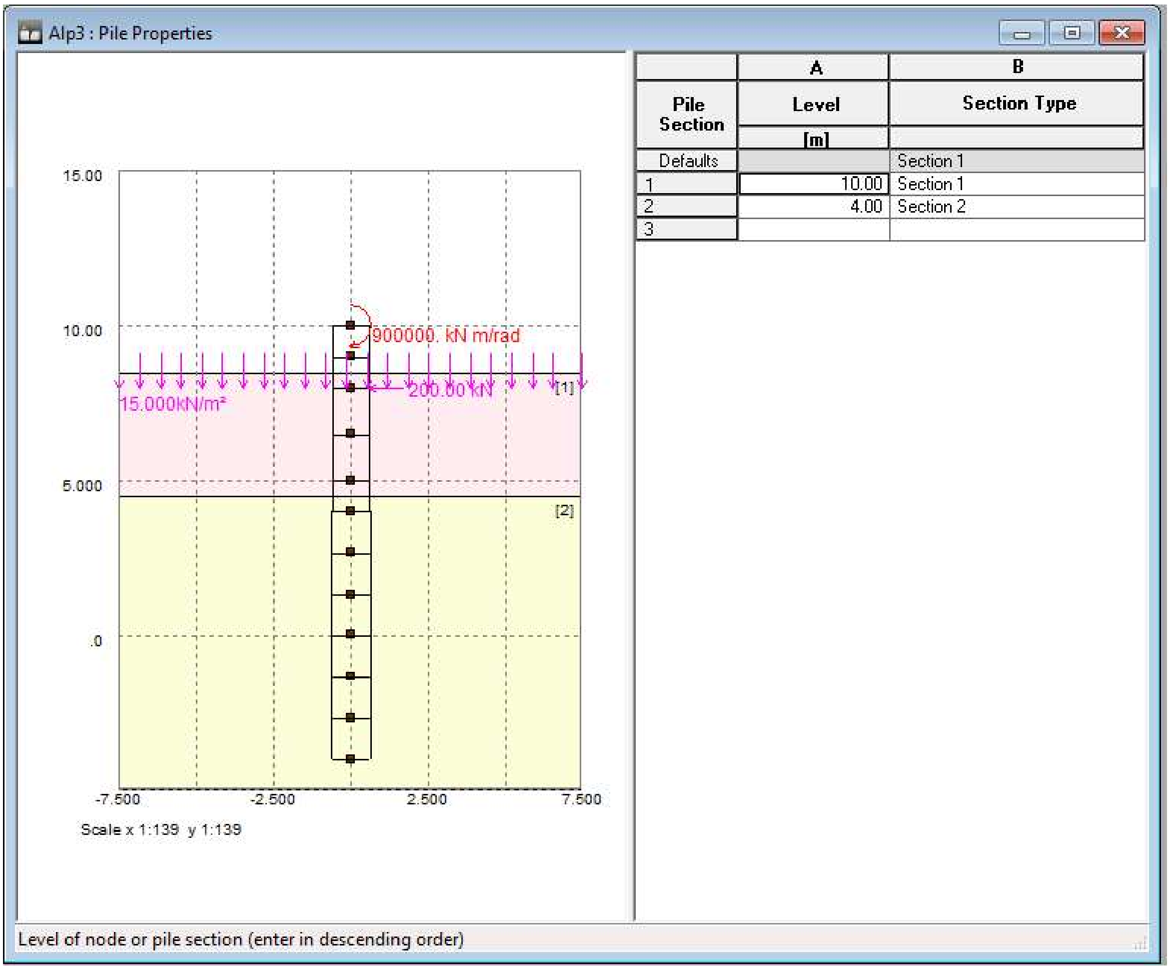
- restraint levels
- load, moment and displacement levels
- soil stratum boundaries, where these boundaries occur midway between nodes. **Note!** As a special case the ground surface may coincide with the top node.

The calculated pressures, deflections, bending moments and shear forces are output only at node positions. Extra nodes are therefore required in order to model the flexibility of the pile and to obtain a reasonably complete representation of the forces, pressures and bending moments.

Data can be entered directly using the tables in both input by level and input by node settings.

Level Based Input

When inputting data by level, it is only necessary to define the top of the pile, the pile toe level (via the General Data dialog box) and each point within the pile at which the section changes. The level at the top of pile and at each change in section must be typed into the level column in descending order, and the corresponding section type selected from the drop-down box in the section type column. This is done through the pile properties option in the gateway.



To delete a pile section, left click on the entry you wish to delete in the pile section column. Then right click anywhere on that row and select 'Delete'.

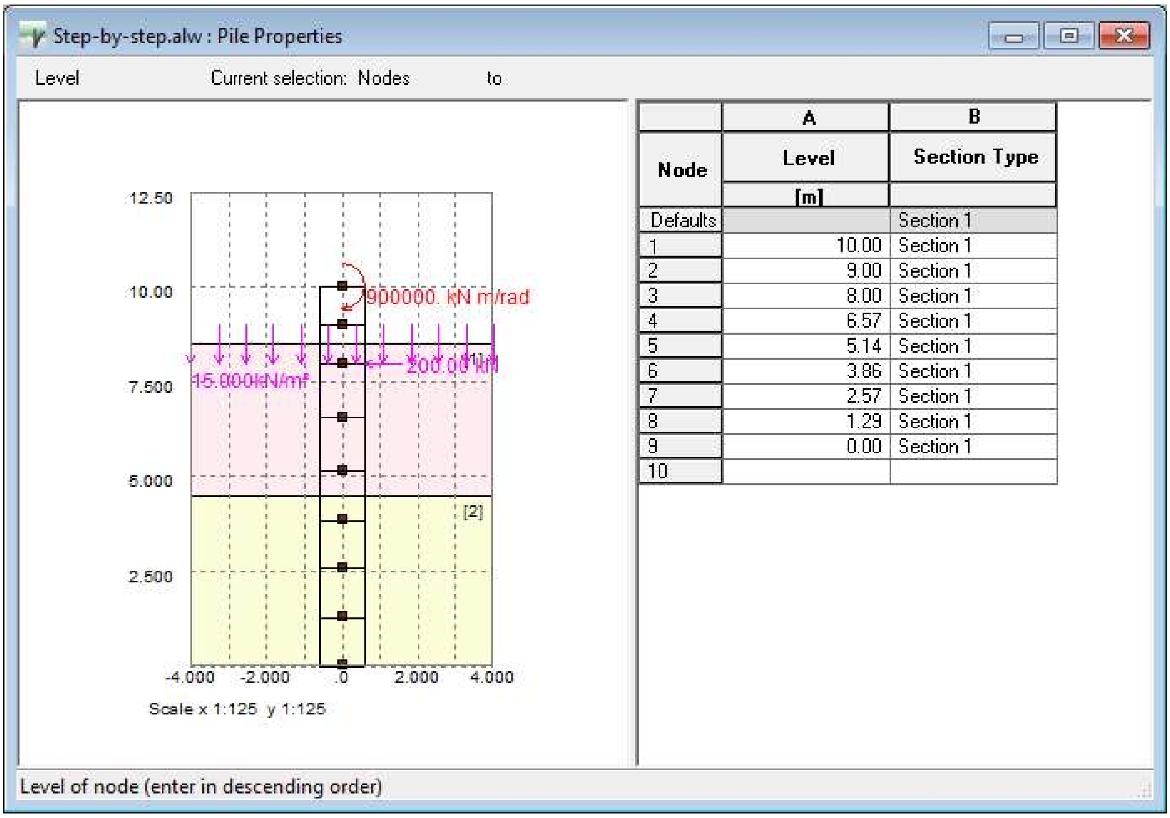
To view the node levels click on the node level option in the gateway. This table will show the node levels generated by Alp, however, these levels cannot be modified by the user and the table can only be accessed once nodes have been generated.

Node Based Input

When inputting data by node, each node is specified explicitly with the level and section type entered into the table. The level is entered in the level column and the section selected from the drop-down box in the section type column. Nodes must be entered in descending order, with the last entry representing the pile toe level.

A larger number of nodes will provide a more numerically accurate solution, but for most problems 20 to 30 should suffice.

Note! To avoid numerical instability, it is recommended that the spacing between **any two pairs of nodes should not differ in length by more than a factor of two**.



To delete a node, left click on the entry you wish to delete in the node column. Then right click anywhere on that row and select 'Delete'.

5.3.10 Groundwater

A single groundwater profile is entered to surround the pile. This can be hydrostatic or piezometric. Groundwater can be entered in tabular form by selecting 'Groundwater' from the Data menu or the Gateway.

ALPman.alw : Groundwater data

	A	B	C
No.	Level [m]	Pressure [kN/m²]	Unit wt of water [kN/m3]
Defaults			
1	7.00	0.00	10.00
2	5.00	20.00	
3	3.00	42.00	
4			

The data is entered in tabular form. The points should be entered in order of increasing depth and the first (or highest) point must specify zero water pressure.
This first line of the table allows a single value for the unit weight of water to be added. On subsequent lines, levels and pressures can be entered to create a piezometric profile. Interpolation between the points is linear and the water profile beneath the lowest point is assumed to be hydrostatic.

If only one data point is entered, the program will also assume a **hydrostatic groundwater**

distribution.

For **hydrostatic** distributions the water pressure (u) is calculated from

$u = z_w \gamma_w$

where

z_w = depth below water table level

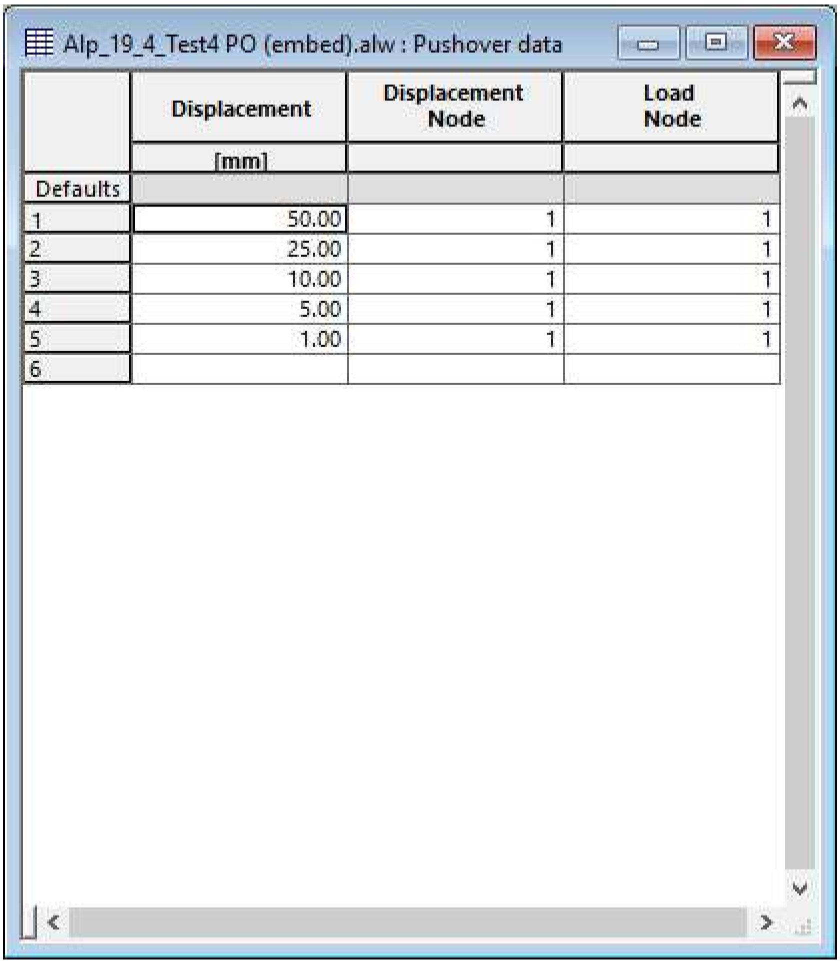
γ_w = specified unit weight of water

Thus a partial hydrostatic condition can be modelled by specifying a value of γ_w less than 10kN/m³.

For **piezometric profiles** the level and pressure at each known point must be entered. If more than one data point is entered, the program will assume that the points represent piezometers, and the ground water pressure will be interpolated vertically between the specified points. Below the lowest point, groundwater pressure will be assumed to extend hydrostatically.

5.3.11 Pushover Data

Where the user has specified for a pushover analysis in the General Data dialog, the points for the pushover curve can be specified in the pushover data table. The pushover data table can be accessed by double clicking on 'Pushover data' in the gateway, or by selecting pushover in the Data menu.



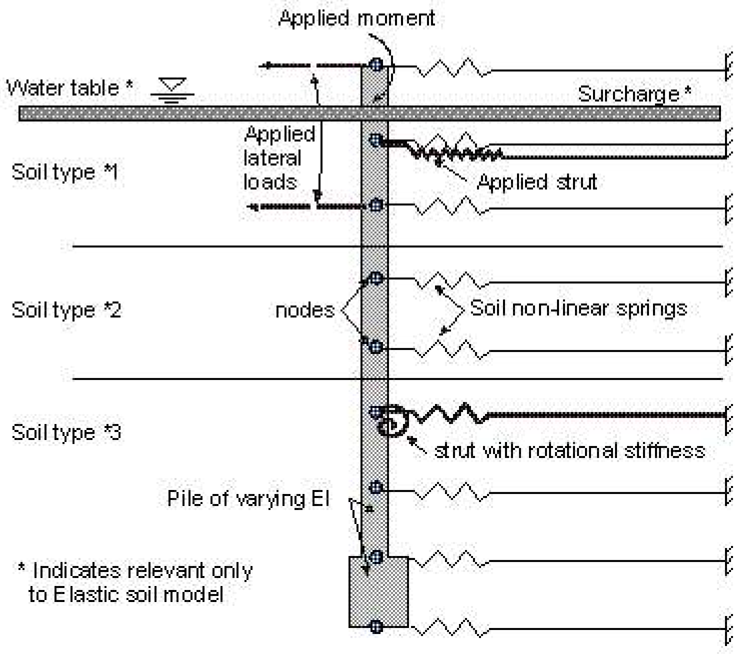
For each point the user can specify either the required load or displacement in the first column. And the level or node at which the displacement is calculated and the load applied in the second and third columns. Note that rotations are also calculated at the same node or level as the displacements.

5.3.12 Applied Loads and Soil Displacements

Lateral loads can be placed on the pile at nodes by specifying a load and moment. The sign convention for the loading is:

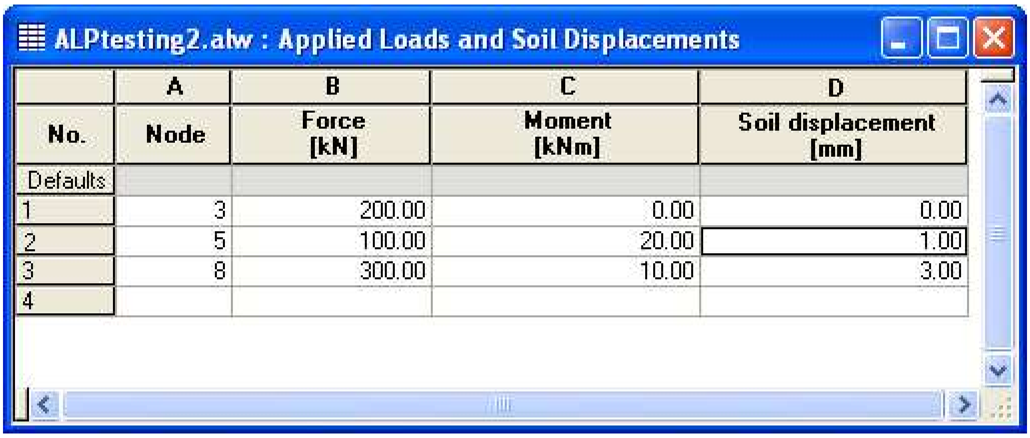
- Load - Positive to the left.
- Moment - Positive clockwise.

Note! The sign convention has been changed from that in the DOS program prior to Version 2.3. Old **Alp** data files will be converted automatically by the program to the new sign convention.



The applied loads shown above act in the positive direction.

Lateral soil movements, which would occur if no pile was present, can be specified at any node. As stated in Applied Lateral Loads and Displacements, linear interpolation is used between specified nodes and constant values of soil displacement are assessed outside of the node range specified. Loads and Displacements can be entered in tabular form by selecting 'Applied Loads and Displacements' from the Data menu or the Gateway.



ALPtesting2.alw : Applied Loads and Soil Displacements

	A	B	C	D
No.	Node	Force [kN]	Moment [kNm]	Soil displacement [mm]
Defaults				
1	3	200.00	0.00	0.00
2	5	100.00	20.00	1.00
3	8	300.00	10.00	3.00
4				

Node no	The node number at which it acts
Force	The applied lateral force, **positive values indicate a force to the left**
Moment	The applied moment, positive values indicate a **clockwise** applied moment
Soil Displacement	Lateral soil movements that would occur if there were no pile present, **positive values indicate soil movement to the RIGHT** .

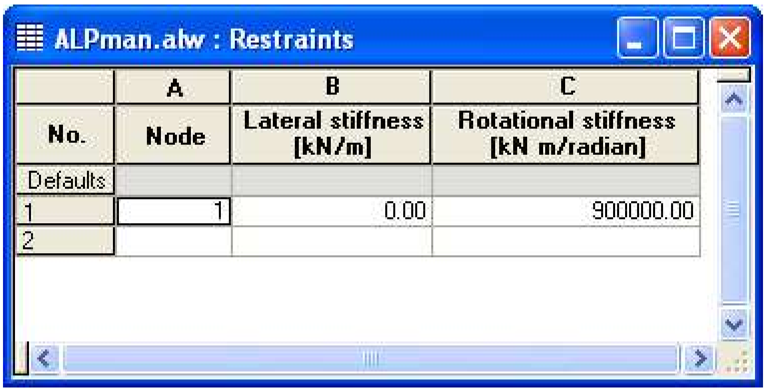
Increments

The user must specify the number of increments of applied load or displacement and then, if the number of increments is greater than one, whether the increments are applied to the loads, displacements or both. This is done via the General data dialog box, see General Data.

5.3.13 Restraints

Restraints can be applied to any node and given a combination of lateral and rotational stiffness. Restraints can be entered in tabular form by selecting 'Restraints' from the Data menu or the Gateway.

Note that the specification of a high rotational stiffness at Node 1 means that the pile is effectively 'fixed' at the top.

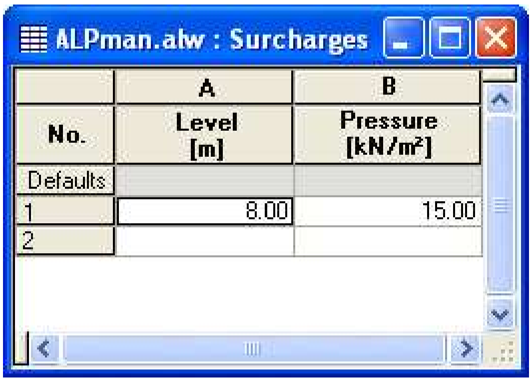


ALPman.alw : Restraints

	A	B	C
No.	Node	Lateral stiffness [kN/m]	Rotational stiffness [kN m/radian]
Defaults			
1	1	0.00	900000.00
2			

5.3.14 Surcharges

Vertical surcharges within or on the soil are not confined to node locations and can be specified at any level. Surcharges can be entered in tabular form by selecting 'Surcharges' from the Data menu or the Gateway.



A positive pressure acts vertically downwards.

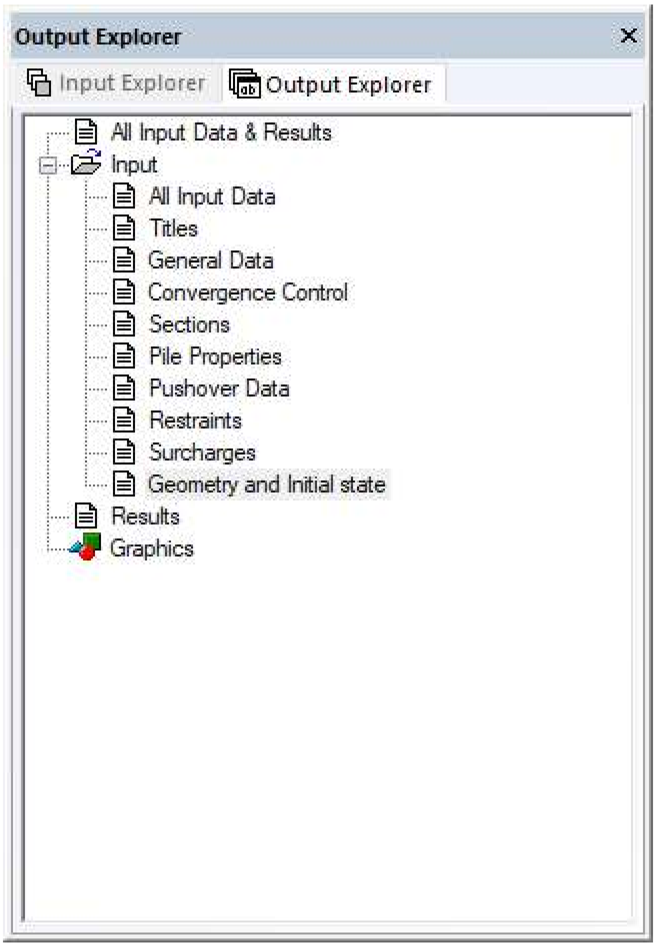
Note! This option has no effect for specified P-Y curves.

Part VI

6 Output

6.1 Output Explorer

The Output Explorer is accessible from the 'View' menu, or via the 'Explorers' button  on the Alp Toolbar. It allows access to all output: the input data and tables of results for inclusion in reports; the Graphical Output; or Results Charts.

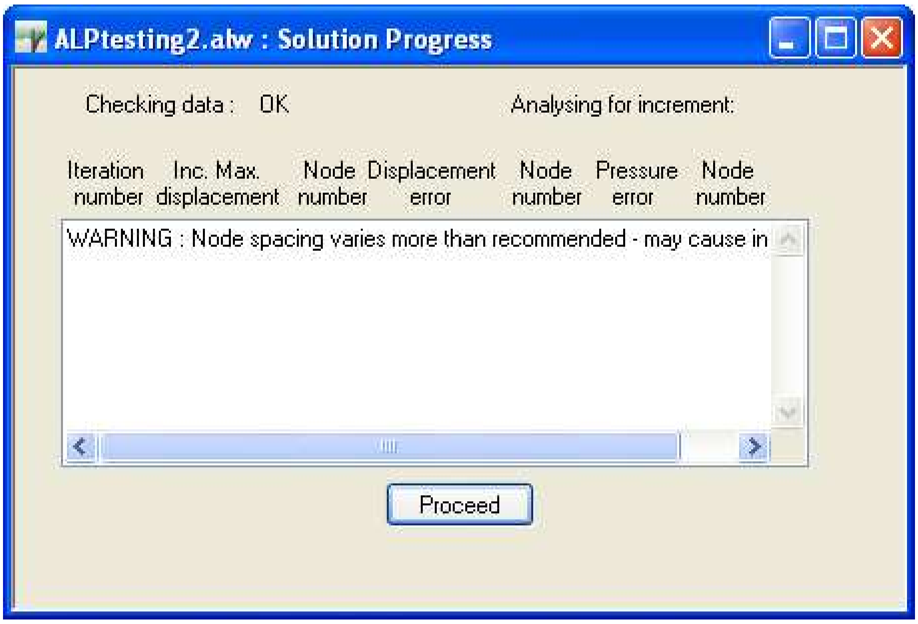


A context menu, to expand or collapse individual or all items, is available by right-clicking in the window.

6.2 Analysis and Data Checking

Results can be obtained by selection of the Analysis menu.

Prior to analysis the program carries out a data check. Warnings are provided should data errors be located.



If no errors are found then the calculation can proceed. Select Proceed.

Note: The option to Examine results becomes available once the calculations have been completed.

6.3 Text Output

Text Output is available in the Text View. Input data or results may be viewed in text by selecting its category from the Output Explorer.

Right-clicking on the Text View opens a context menu with the following options:

- **Stripey Output** - toggles shading of alternate rows in the text output, to improve readability of tables
- **Fully populate fields** - toggles whether blank fields are to be shown where data is repeated from one row to the next, or whether that data is to be repeated, to allow more effective data sorting if exported to a spreadsheet
- **Grid View** - opens the Grid View or rows and columns, for easy copying to a spreadsheet
- **Export...** - opens a dialog to allow the text output to be exported to a CSV, HTML, RTF or TXT file

The **Stripey Output, Fully populate fields** and **Grid View** options are also available via "Output" from the program menu when the Text View is active.

The **Export...** option is also available via "File | Export" from the program menu when the Text View is active.

The font size of the Text Output can be controlled for printing, and so to size tables to fit better to page widths, via the font size control on the Alp toolbar.

Typical Text View of Input Data:

Alp_Elastic-plastic.alw : Tabular Output

Geometry and Initial state

Node	Level [cm]	Soil	EI [Ncm2]	Effective Width [cm]	Water Pressure [N/cm^2]	Soil Disp [mm]
1	1000.0	0	0.0	120.00	0.0	0.0
2	900.00	0	0.0	120.00	0.0	0.0
3	800.00	1	0.0	120.00	0.0	0.0
4	700.00	1	0.0	120.00	0.0	0.0
5	600.00	1	0.0	120.00	0.0	0.0
6	500.00	1	0.0	120.00	0.0	0.0
7	400.00	2	0.0	130.00	0.0	0.0
8	300.00	2	0.0	130.00	0.0	0.0
9	200.00	2	0.0	130.00	0.0	0.0
10	100.00	2	0.0	130.00	0.0	0.0
11	0.0	2	0.0	130.00	0.0	0.0
12	-100.00	2	0.0	130.00	0.0	0.0
13	-200.00	2	0.0	130.00	0.0	0.0
14	-300.00	2	0.0	130.00	0.0	0.0
15	-400.00	2	0.0	130.00	0.0	0.0

Typical Text View of Results:

Alp_Elastic-plastic.alw : Tabular Output

Output for load increment 1

Iteration	Max Inc Disp [mm]	at node	Disp error [mm]	Pressure error [N/cm²]
20	23.65	1	0.0584	0.76
40	23.96	1	0.0010	0.01

Node	Level [cm]	Defl [mm]	Rotation [rad]	Soil	Pressure [N/cm²]	Bending [Ncm]	Shear [N]	
1	1000.0	-23.964	-128.42E-6	0	0.0	0.0	0.0	
1	1000.0					-11.558E+6	0.0	
2	900.00	-22.391	-0.0030179	0	0.0	-11.558E+6	0.0	
3	800.00	-17.928	-0.0059073	1	-3.8260	-11.558E+6	22956.	P
3	800.00					-11.558E+6	-177040.	
4	700.00	-11.218	-0.0068708	1	-7.3030	3.8510E+6	-110270.	P
5	600.00	-5.1053	-0.0050780	1	-6.8070	10.496E+6	-25610.	
6	500.00	-1.2754	-0.0026454	1	-1.7005	8.9730E+6	25435.	
7	400.00	0.39770	-849.27E-6	2	0.44053	5.4092E+6	32775.	
8	300.00	0.80658	-68.262E-6	2	0.89345	2.4181E+6	24104.	
9	200.00	0.69506	230.25E-6	2	0.76991	588440.	13292.	
10	100.00	0.43483	262.51E-6	2	0.48166	-240310.	5156.7	
11	0.0	0.20455	191.22E-6	2	0.22658	-442900.	553.13	
12	-100.00	0.056220	108.44E-6	2	0.062275	-350930.	-1324.4	
13	-200.00	-0.021021	51.735E-6	2	-0.023285	-178010.	-1577.9	
14	-300.00	-0.057629	26.166E-6	2	-0.063835	-35359.	-1011.6	
15	-400.00	-0.079957	20.409E-6	2	-0.088567	24307.	-20.972	

- The letter "P" next to a result indicates that the effective earth pressure is greater than 0.99 times the pa

EXTREME values so far:-

Deflections Min [mm]	Deflections Max [mm]	Rotations Min [rad]	Rotations Max [rad]	Moments Min [Ncm]	Moments Max [Ncm]	Shears Min [N]	Shears Max [N]
-23.964	0.80658	-0.0068708	262.51E-6	-11.558E+6	10.496E+6	-177040.	32775.

RESTRAINT FORCES

No.	Node	Lateral force [N]	Moment [Ncm]
1	1	0.0	115580.

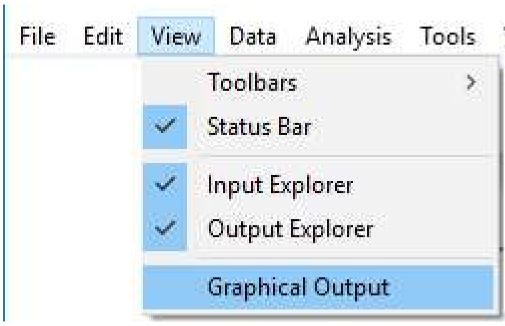
The Text Output provides a listing of displacements, bending moments and shear forces at each node. An indication is given in the output to show if the passive soil pressure limit is exceeded. The letter "P" next to a result indicates that the effective earth pressure is greater than 0.99 times the passive limit, but within the convergence pressure limit.

The maximum and minimum deflections, bending moments and shear forces are reported with a summary of the convergence errors.

Where a pushover analysis has been undertaken summary results are included for each point including the load applied, the deflection and rotation at the indicated node, and the maximum bending moment in each section of the pile.

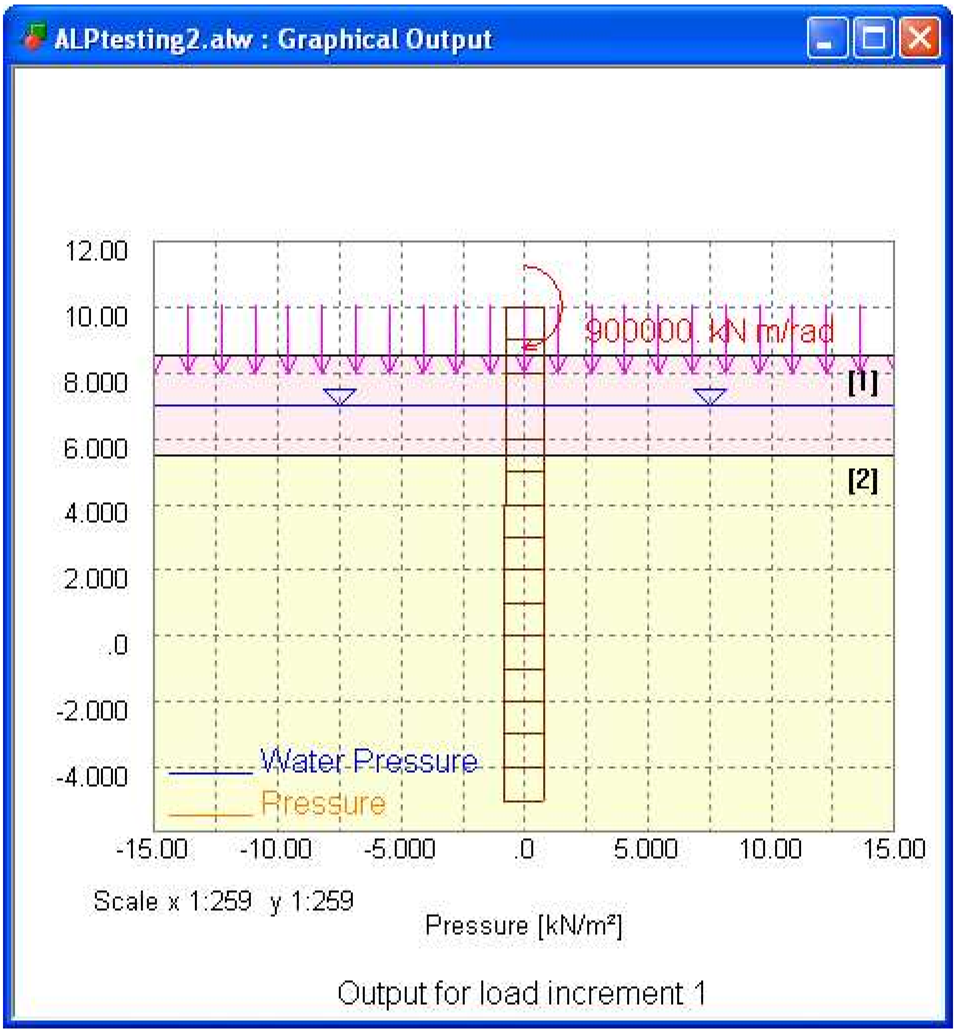
6.4 Graphical Output

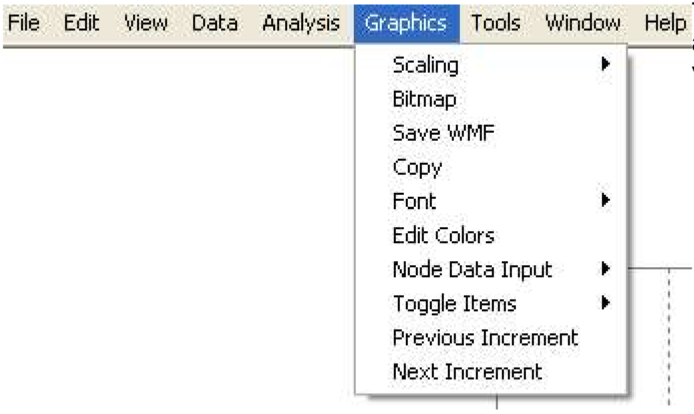
To obtain a plot of data and results, select 'Graphical Output' from the 'View' menu, the Gateway or the Alp Toolbar.



For information on the use of the Toolbar and Status bar functions please see the Index list.

Input data is always shown, and individual result types can be switched on or off as required using the buttons on the graphics toolbar.

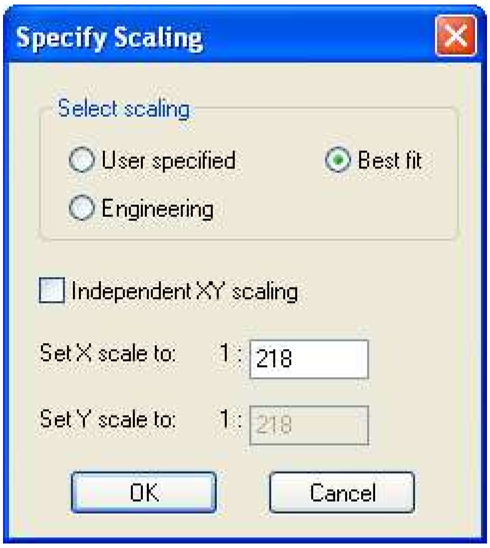




The Graphics menu will be available when Graphical Output view is activated.

6.4.1 Set Exact Scale

Selection of Set Exact Scale icon  allows you to set any required scale for the graphics. This is done using the following data entry screen,



which allows best fit, specified or engineering scales. The X and Y scales can also be changed to give different values and distort the drawing

6.5 Charts

Results charts are most easily accessible from the Output Explorer.

The following charts are viewable in the program providing the appropriate data and results are available.

Pushover Data

- Force vs Displacement
- Max. Moment vs (Force or Displacement)
- Rotation vs (Force or Displacement)
- (Displacement or Force) vs Max. Moment

Notes

[1] Force or Displacement in the pushover charts depends on the option given by the user for Pushover analysis in the General Data dialog.

6.6 Toolbars

Toolbars provide a short cut to the more commonly used commands. Toolbars except can be docked (attached to the application frame) or floating (free to be positioned by the user). The toolbars can be switched on and off as required from the “View | Toolbars” menu command.

6.6.1 Standard Toolbar

The Standard Toolbar provides access to the following common Windows functions along with some that are specific to the program.

New — create a new model
Open — open an existing file
Save — save the model to file

Cut — cut the data and place on clipboard
Copy — copy the data and place on the clipboard
Paste — paste the data from the clipboard into the model

Print — print the current view
Print Preview — preview the current view

About — opens the program's About Dialog e.g. to show version information
Alp Home — opens the programs home page on the internet
Email — opens an email to the Oasys support team

 - Undo any change in the table view
 - Redo any change in the table view

6.6.2 Alp Toolbar

The Alp Toolbar provides access to the following functions.

 - open or close the Explorers

- open the Graphical Output
- specify a custom font scale to size text in the Plan View, or when printing the Text Output (dependent on which of those views is active)
- select from a list of font scales to size text as above
- open the context sensitive wizard
- perform an analysis
- delete the results

6.6.3 Graphics Toolbar

The following buttons are available on the graphical output toolbar:.

Axis : Provides a reference grid behind the drawing.

Engineering Scale : This opens the Specify Scaling dialog box which allows best fit, specified or engineering scales. The X and Y scales can also be changed to give different values and distort the drawing.

Save Metafile : this save icon allows the file to be saved in the format of a Windows Metafile. This retains the viewed scale. The metafile can be imported into other programs such as a word processor, spreadsheets and drawing packages.

Copy : This icon allows the view to be copied to the clipboard in the form of a Windows Metafile.

Restore Zoom : The user can select an area to 'zoom in' to by using the mouse to click on a point on the drawing and then dragging the box outwards to select the area to be viewed. The program will automatically scale the new view. The original area can be restored by clicking on the 'restore zoom' icon as shown here.

Deflection of the pile.

Rotation of the pile.

Soil Pressure down the passive side of the pile.

Bending Moments within the pile.

Shear Forces within the pile.

Pressure envelope - Whereas other graphical results are for single increments, the pressure envelope provides the envelope of results for all increments in the calculation.

Previous increment : This button will be available when more than one increment is selected in General Data.

Next increment : This button will be available when more than one increment is selected in General Data.

Part VII

7 Programming Interface

Programming interface is provided by COM Automation in Alp

COM Automation allows commands to be issued from a separate process via a set of VBA or C++ instructions. The separate process could be a separate program or, indeed, a VBA script in a spreadsheet.

7.1 COM Automation

COM Automation allows other programs to access Alp by creating an instance of Alp class "alplib.AlpAuto" and calling the exported functions of this class. The available COM export functions are listed below.

Note that function names are case sensitive.

Examples of COM use from Excel (VBA) and Python are installed in the Samples folder with the program. These are simple examples to show basic use and are provided as an illustration only.

More:
COM Export Functions

7.1.1 COM Export Functions

The Alp COM export functions are listed below:

The use of many of these functions is demonstrated in sample Excel (XLSX) files that are installed in the Samples folder in the Alp program folder. Unless otherwise stated, the functions usually return a "short" (integer) which is zero on successful execution, and -1 on failure. Where a function argument (in the brackets following the function name) has a '*' prefix, it indicates that it is a value modified by the function providing an output.

Note this has changed significantly from the COM functions in Alp 19.2 and 19.3.

File manipulation

NewFile (string filename)	Open a new model. The file suffix can be .alw or .json. If it is not included, it will default to a binary .alw file.
Open (string filename)	Open an existing file. The file suffix should be specified as either .alw or .json.
Save()	Save the data to the default file (i.e. overwriting the file that was opened or last saved)
SaveAs (string filename)	Save the data to*.alw file. ***filename*** – the name of the file to be saved, including path and extension.
ReadTextFile (string filename)	Sets all data in the current document to be equal to that set by a text file. A template and example '.txt' file are included with the installation demonstrating the required format for any files to be imported. ***filename*** – the name of the '.txt' file to be imported, including path and extension.
Close()	Close the current file.

Program Interface

Show()	Show the running instance of Alp created by the Automation client
UpdateVIews()	Refreshes all the Alp views currently displayed

Data

SetJobNumber(string jobnum) **SetJobTitle(string jobtitle)** **SetInitials(string initials)** **SetSubtitle(string subtitle)** **SetCalcHeader(string header)** **SetNotes(string notes)**	))))))These functions all set the title item to the input string
SetSoilModel(short imodel)	Sets whether elastic-plastic, generated or specified P-Y model is to be used (parameter should be 0, 1 or 2 respectively)
SetGlobalSoilEFactor(double dFactor)	Sets the global soil E factor
SetIncType(short itype)	Sets the increment type - 0 loads only, 1 displacements only, 2 both
SetNumIncs(short ninc)	Sets the number of analysis increments
SetAnalysisType(short itype)	Sets analysis type - 0 for standard or 1 for pushover analysis
GetNumIncs(short* ninc)	Gets the number of analysis increments
ClearNodes()	Clears all nodes from the data
SetToeLevel(double dToe)	Sets the toe level of the pile. This will regenerate nodes if required. The function fails if the input type is by node rather than by level.
GetToeLevel(double* dToe)	Returns the toe level of the pile
SetMaxIterations(short max) **SetDispTol(double dTol)** **SetPressTol(double Tol)** **SetDampingCoeff(double dDamp)** **SetMaxIncDisp(double dDisp)**	))))) These functions all set the control parameters for the numerical analysis
GetNumNodes(short* numnodes)	Gets the number of nodes in the current file. NOTE: node numbers are referenced as a 1-based list, so the top node is node 1 in other functions which set or get node-based properties.
GetNodeLevel(short inode, double* dLevel)	Gets the level of the node identified by sIndex
InsertNode(double	Inserts a node into the data at level dLev with pile section iSect. If the

dLev, short iSect)	input mode is by level, this function sets it to be by node.
SetNodeSection(short inode, short iSect)	Sets the pile section at node "inode" to section no. iSect
DeleteNode(short inode)	Deletes node "inode"
GetNodeEffWidth(short inode, double* dWidth)	Gets the pile diameter/effective width at the node identified by sIndex
GetNodeEI(short inode, double* dEI)	Gets the EI value of the pile at the node identified by sIndex
GetNumLoadDisps(short* nLoads)	Gets the number of applied loads/displacements
AddNodeLoadDisp(short inode, double dForce, double dMom, double dAppDisp)	Adds an applied load and/or displacement at node inode, with load/moment/displacement values in the dForce/dMom/dAppDisp parameters.
SetNodeLoadDisp(short inode, double dForce, double dMom, double dAppDisp)	As above but updates an existing record if one exists at this node.
SetNodeLoadAndType(short inode, double dForce, double dMom, double dAppDisp, short isFav, short isLive)	Adds a node load at "inode", allowing setting of whether the load is favourable/unfavourable, live/dead by a 1 or 2 respectively in the isFav/isLive parameters.
GetNumSections(short* numSect)	Gets the number of pile sections specified in the data. Section references are a 1-based list so section 1 is the first in the data.
SetSection(short iSect, string sName, short iType, double dEff, double dEI)	Sets an explicit pile section with the given parameters. This either creates a new section or updates an already existing section.
DeleteSection(short iSect)	Deletes a pile section from the model
ClearSections()	Erases all pile sections from the model
GetNumSoils(short* numsoils)	Gets the number of soil layers considered in the model, note this is the number of soil layers, not the number of materials. Soil layer references are 1-based.
GetSoilLevel(short sIndex, double* dLevel)	Gets the top level of the soil layer specified by sIndex, with the indexing going from 1 for the top soil layer and increasing with depth.
SetElasPlasSoil(short iTopNode, double dEval, double dUnitWt, double dCoh, double dCohGrad)	Creates a new elastic-plastic soil or edits an existing one, if iTopNode is already the top node of an existing soil. The soil parameters are set to the input values for the double parameters of this function.
SetEforSoil(short sIndex, double* dRefCoh)	Sets the E value for the specified soil
SetPhiforSoil(short	Sets the phi value for the specified soil

sIndex, double* dPhiSoil)	
SetRefCohesionforSoil(short sIndex, double* dRefCohesion)	Sets the cohesion value for the specified soil.
SetCohesionGradientforSoil(short sIndex, double* dCohGrad)	Sets the cohesion gradient for the specified soil
DeleteElasPlasSoil(short sIndex)	Deletes the elastic-plastic soil "sIndex" from the model
ClearElasPlasSoils()	Deletes all elastic-plastic soils from the model
AddNodePY(double dP1, double dY1, double dP2, double dY2, double dP3, double dY3, double dP4, double dY4, double dP5, double dY5, double dP6, double dY6)	This function adds a new specified PY point to the file, with dP1 corresponding to P1, dP2 corresponding to Y1, etc. As with the AddNode function the points must be added in descending order corresponding to the node levels.
SetNodePY(short iNode, [...same parameters as AddNodePY])	This function is similar to the AddNodePY function, however instead of adding a new PY curve it overwrites an existing curve for the node referenced by iNode. If no curve has been specified for the node this function will return a fail.
ClearSpecPY()	Deletes all specified PY curves from the currently selected file provided that the soils mode is set to specified PY, for other modes the function will return a fail.
SetKqforSoil(short sIndex, double* KqValue)	Sets Kq for the specified soil
SetKcforSoil(short sIndex, double* KcValue)	Sets Kc for the specified soil
GetNumWaterPoints()	Gets the number of water data points
SetWaterPoint(double dLev, double dPressure, double dUnitWt)	Sets a water data point at level dLev with the specified values for pore pressure and unit weight
GetPorePressureAtLevel(double dLEv, double* dPorePress)	Returns the pore pressure at the specified level. This is interpolated between water data points.
GetPorePressureAtNode(short inode, double* dPorePress)	Returns the pore pressure at the specified node. This is interpolated between water data points.
DeleteWaterPoint(short iref)	Deletes the water data point "iref" (1-based)
ClearWaterPoints()	Deletes all water data points
MaxDisp (double*	Gets the maximum displacement of the pile.

dMaxDisp)	
GetNodeDisp (short sIndex, double* dDisp)	Gets the pile displacement at the node identified by sIndex.
GetNodeRotation (short sIndex, double* dRot)	Gets the pile rotation at the node identified by sIndex.
GetNodeShear (short sIndex, BOOL Below, double* dShear)	Gets the pile shear at the node identified by sIndex. If the BOOL value is TRUE, the function returns the shear just below the node, otherwise it returns the shear just above the node.
GetNodeBM (short sIndex, double* dBM)	Gets the pile bending moment at the node identified by sIndex.
MaxBM(double* dMaxBending)	Gets the maximum **magnitude** of bending moment down the pile, i.e. -122kNm will be returned from this function if the maximum positive bending moment is less than 122kNm.
MaxShear(double* dMaxShear)	Gets the maximum **magnitude** of shear force down the pile, i.e. -95kN will be returned from this function if the maximum positive bending moment is less than 95kN.
ReadTextFile(string sPath)	Reads all model data from the specified text file.
Export(string sPath)	Exports the text output to various formats. This function may pop up message boxes.
ExportCSV(string sPath)	Exports the text output of results to a csv file. The file path and extension should be specified in the sPath variable.
ExportCSVResult(string sPath, short iType, short iInc)	Exports a csv table from the current file. The string sPAth needs to specify the full file path, name and extension. iType indicates which result to export and can have the following values - 0 for the bending moment profile with depth, 1 for the shear profile with depth and 2 the displacement profile with depth. iInc indicates the load increment for which the results are required.
ExportCSVPushOver(string sPath)	This function exports a csv table from the current file showing the pushover curve for the top node (i.e. the lateral load v displacement for the top node for the number of increments analysed). The string sPath needs to specify the full file path, name and extension.
ExportCSVPY(string sPath, short iNode)	Exports generated P-Y curves for node iNode to the file sPath.
short DeleteResults ()	Deletes the results in the current file.
short Analyse ()	Analyses the current data
PrintTabular(string bsPath, BOOL bNotes, BOOL bGen, BOOL bConv, BOOL bPartFacts, BOOL bSoils, BOOL bSects, BOOL bPileP, BOOL bLoads, BOOL bPush, BOOL bRest, BOOL	Prints the tabular output to a file "sPath". The BOOL parameters should be set to 1 when a particular data item is required in the output. The parameters are for Notes, General Data, Convergence Data, Partial Factors, Soil Data, Sections, ... , Applied Loads/Displacements, Pushover, Restraints, Surcharges, Geometry and Results respectively.

bSurch, BOOL bGeom, BOOL bResults)	
PrintGraphical(string sPath, BOOL bDisp, BOOL bRotation, BOOL bPressure, BOOL bBbending, BOOL bShear)	Prints a graphical view to a png file "sPath". The BOOL parameters switch on display of: displacement, rotation, pressure, bending moment and shear force respectively.

Part VIII

8 List of References

8.1 References

API, RP2A (1989). Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, 18th Edition, American Petroleum Institute.

API, RP2A (2000). Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms - Working Stress Design, 21st Edition, American Petroleum Institute.

Broms BB (1964b). Lateral Resistance of Piles in Cohesionless Soils, JSMFD, ASCE, Vol.90, SM3, pp123-156**.**

Broms B (1972). Stability of Flexible Structures (Piles and Pile Groups). General Report, Americal Proceedings 5th Euro Conf. SMFE, Madrid, Vol.2, pp 239-269.

Brinch Hansen J (1961). The Ultimate Resistance of Rigid Piles against Transversal Forces, Geoteknisk Institut. Bull. No.12, Copenhagen.

BSI (2009). Eurocode 7: Geotechnical design - Part 1: General rules, BS EN 1997-1:2004, British Standards Institute.

BSI (2009). UK National Annex to Eurocode 7: Geotechnical design - Part 1: General rules, NA to BS EN 1997-1:2004, British Standards Institute.

Georgiadis M (1983). Development of p-y curves for layered soils. In Geotechnical Practice in Offshore Engineering, pages 536 - 545. American Society of Civil Engineers, April 1983.

Matlock H (1970). Correlations for Design of Laterally Loaded Piles in Soft Clay, OTC 1204, Proceedings Offshore Technology Conference, Houston.

Pappin J W, Simpson B, Felton P J, and Raison C (1986). Numerical Analysis of Flexible Retaining Walls. Symposium on Computer Applications in Geotechnical Engineering. The Midland Geotechnical Society, April.

Poulos H G (1971). Behaviour of Laterally Loaded Piles:I-Single Piles. Proceedings of JSMFE, ASCE, Vol.97, 5, pp711-731.

Reese L C (1997). Analysis of Laterally loaded Piles in Weak rock, Journal of Geotechnical and Geoenvironmental engineering, Vol 123, no.11, 1997, pp1010-1017.

Reese L C and Matlock H (1956). Non-dimensional Solutions for Laterally Loaded Piles with Soil Modulus Assumed Proportional to Depth. Proceedings of the 8th Texas Conference on Soil Mechanics and Foundation Engineering, Special Publication 29, Bureau of Eng. Res., Univ. of Texas, Austin.

Reese L C, Cox W R, and Koop F D (1974). Analysis of Laterally-Loaded Piles in Sand, OTC 2080, Proceedings of Offshore Technology Conference, Houston.

Sullivan W R, Reese L C, and Fenske C W (1980). Unified Method for Analysis of Laterally Loaded Piles in Clay, Proceedings of Conference on Numerical Methods in Offshore Piling, ICE, London pp 135 – 146.

Terzaghi K (1955). Evaluation of the Coefficients of Subgrade Reaction, Geotechnique, Vol. 5, No.4, pp297-326.

Tomlinson M J (1981). Pile Design and Construction Practice, Second Edition, Viewpoint Publications.

Tomlinson MJ (1981). Foundation Design and Construction, Fifth Edition, Longman, pp445 (Fig 7.33)

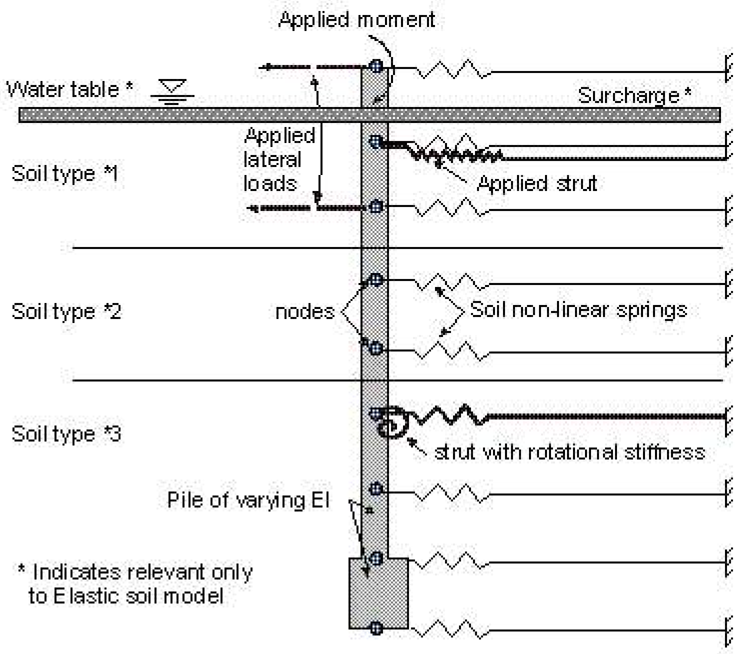
Turner J (2006). Rock-Socketed Shafts for Highway Structure Foundations, NCHRP Synthesis of Highway Practice, 2006, pp 58

Part IX

9 Alp

9.1 Brief Technical Description

Alp (Analysis of Laterally Loaded Piles) is a program that predicts the pressures, horizontal movements, shear forces and bending moments induced in a pile when subjected to lateral loads, bending moments and imposed soil displacements.



The pile is modelled as a series of elastic beam elements. The soil is modelled as a series of non-interactive, non-linear "Winkler type" springs. The soil load-deflection behaviour, can be modelled either assuming an elastic plastic behaviour, or by specifying or generating load-deflection (i.e. P-Y) data. Two stiffness matrices relating nodal forces to displacements are developed. One represents the pile in bending and the other represents the soil.

Index

- A -

ALP 106
Alp Toolbar 10
AlpToolbar 90
American Petroleum Institute 59
Analysis and Data Checking 84
API 59
Applied Lateral Loads and Displacements 45
Applied Loads and Soil Displacements 80
Assembling Data 50

- B -

Bending Moments 88
Brief Technical Description 106

- C -

COM interface
  COM automation 94
  COM functions 94
Components of the User Interface 10
Convergence Control Parameters 61
Copy 85

- D -

Data Input 57
Definition
  Reinforcement 74
Deflection 88

- E -

Elastic-Plastic data 63
Elastic-Plastic soil model 33
Excel 85

- G -

Gateway 10
General 32
General Data 59
General Program Description 8
Generated P-Y curves 36
Generated P-Y data 65
Graphical Output 10, 88
Graphics 91
Graphics menu 88
Graphics Toolbar 10, 91
Grid Output 85
Grid View 85
Groundwater 78

- I -

Input Explorer 56
Intranet link and e-mails 51

- M -

Moment Curvature 47

- N -

Next increment 88

- O -

Opening the program 50

- P -

P Modification Factors 47
Page Setup 85
Paste 85
Pile Properties 76
Pile stiffness 44
Pressure envelope 88
Previous increment 88
Program Features 8
programming interface
  programming interface 94

P-Y soil model 35

- R -

References 102
Reinforcement
    Definition 74
Restraints 45, 81
Results 85

- S -

Save Metafile 88
Set Exact Scale 89
Shear Forces 88
Soil Data 63
Soil Pressure 88
Specified P-Y curves 35
Specified P-Y data 64
Standard Toolbar 10, 90
Stripey Output 85
Surcharges 82

- T -

Table View 10
Tabular Output 10
Tabulated Output 85
Titles 55
Titles window - Bitmaps 55
Toolbar 10
Toolbars 90

- U -

Units and Preferences 57
User Interface 10

- V -

VBA interface
    COM automation 94
    COM functions 94
visual basic interface
    COM automation 94
    COM functions 94

- Z -

Zoom 88

Endnotes 2... (after index)

Back Cover