



# Piling Analysis Tutorial Manual

OASYS

# Contents

Introduction .....	1
Section 1 .....	2
Step-by-Step Tutorial 1 Oasys Pile – Pile Capacity and Settlement Analysis .....	2
1.1 Components of the User Interface.....	3
1.2 Pile Capacity Analysis .....	4
1.2.1 Creating the input.....	4
1.2.2 Analysis and Outputs .....	10
1.2.2.1 Tabular Outputs .....	10
1.2.2.2 Exporting Tabular Results .....	12
1.2.2.3 Graphical Outputs.....	12
1.2.2.4 Printing Graphical Outputs .....	15
1.3 Pile Settlement Analysis.....	16
1.3.1 Creating the input.....	16
1.3.2 Analysis and Outputs .....	18
1.3.3 Tabular Outputs.....	18
1.3.4 Graphical Outputs .....	19
Section 2 .....	21
Step by Step Tutorial 2 Oasys Alp – Lateral Pile Capacity .....	21
2.1 Components of the User Interface.....	22
2.2 Creating the input.....	23
2.3 Analysis and Outputs .....	28
2.3.1 Tabular Outputs.....	28
2.3.2 Graphical Outputs .....	29
Section 3 .....	31
Step by Step Tutorial 3 Oasys AdSec –Section Analysis .....	31
3.1 Input .....	32
3.1.1 Creating the Section .....	32
3.1.2 Applying the Load Cases.....	37
3.2 Analysis and Outputs .....	39
3.2.1 Load Moment Curve .....	39
3.2.2 SLS Loads.....	40
3.2.3 ULS Loads .....	41

Section 4 ..... 42

    Step by step Tutorial 4 Oasys ADC –Section Analysis ..... 42

    Objectives ..... 42

    4.1    Input ..... 43

    4.2    Analysis and Results ..... 50

        4.2.1    Tabular Outputs..... 54

        4.2.2    Plotting Charts ..... 55

Appendix ..... 56

    EC7 – UK SAMPLE CALCULATION IN PILE ..... 56

# Introduction

Oasys Pile, Alp, AdSec and ADC are programs that analyse single piles. The input wizards enable the quick generation of complex staged construction sequences. The outputs are designed to help the user examine relevant results, as well as visually communicate these results to clients. The calculation itself is based on verified and robust numerical methods and the simplicity of calculation enables new users to set up and run problems within hours of completion of this tutorial.

The various tutorial examples deal with a range of practical pile analysis but this Tutorial Manual is intended to familiarise the user with the programs. The examples should therefore not be used as a basis for practical projects.

Users are expected to have an understanding of soil mechanics and geotechnical theory, and should be able to work in a Windows environment.

It is important to realise that Pile, Alp, AdSec and ADC advanced programs and the user must be fully aware of the various methods of analysis, requirements and limitations discussed in the User Manuals before use. The Tutorial Manual will not provide theoretical background information on the analysis method, nor does it explain the details of various methods of analysis available in the programs. These details can be found in the User Manuals. This also contains detailed information on the available program features.

Short courses are also regularly organised and should you be interested in more hands-on experience you can contact [oasys@arup.com](mailto:oasys@arup.com) for dates and program content.

# Section 1

## Step-by-Step Tutorial 1

### Oasys Pile – Pile Capacity and Settlement Analysis

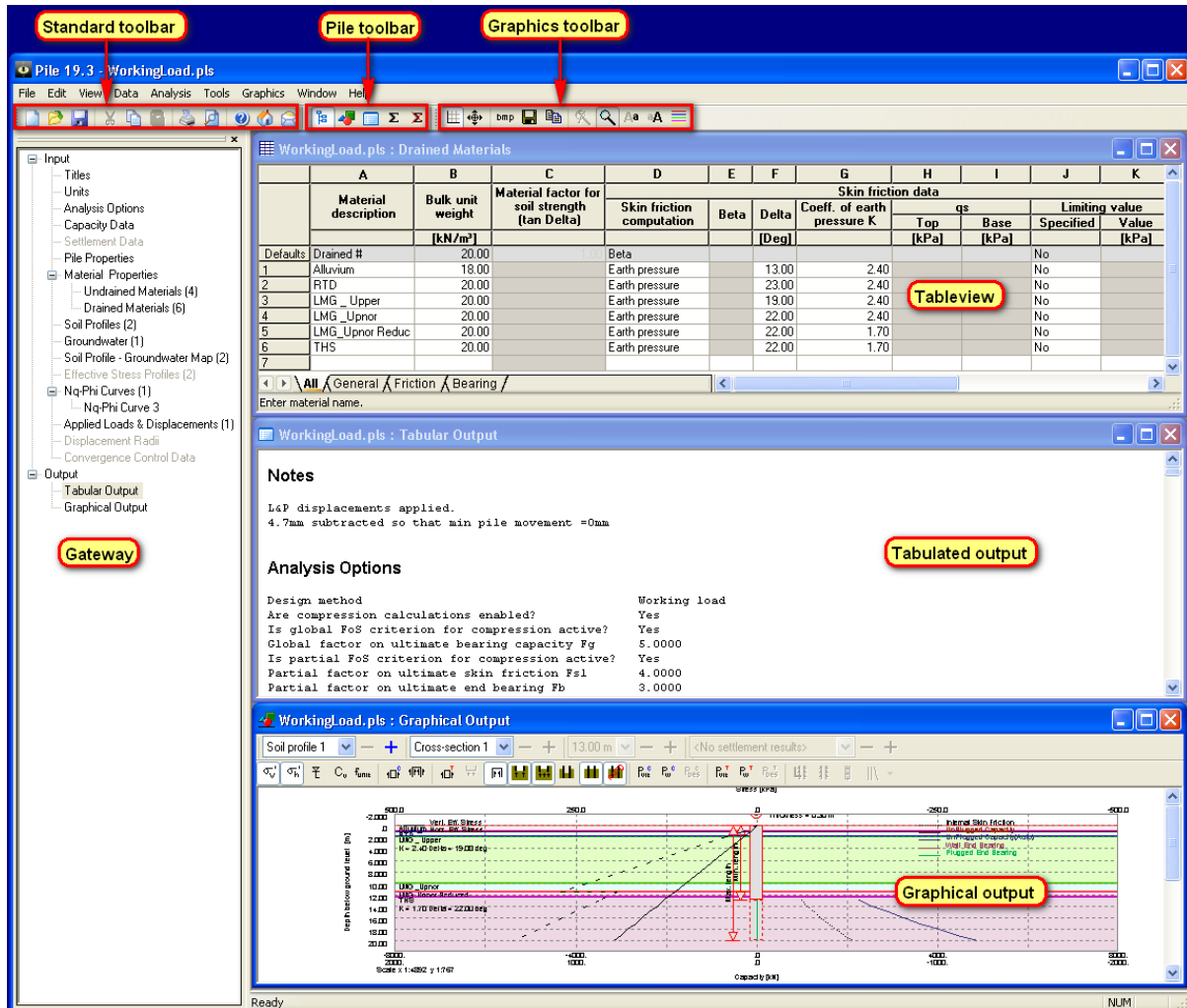
#### Objectives

By the end of the session the user should be able to:

- Navigate the Pile Interface
- Input Capacity and Settlement Data
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs
- Export tabular outputs for further analysis

## 1.1 Components of the User Interface


The principal components of Pile's user interface are the Gateway, Table Views, Graphical Output, Tabular Output, toolbars, menus and input dialogs. These are illustrated below.

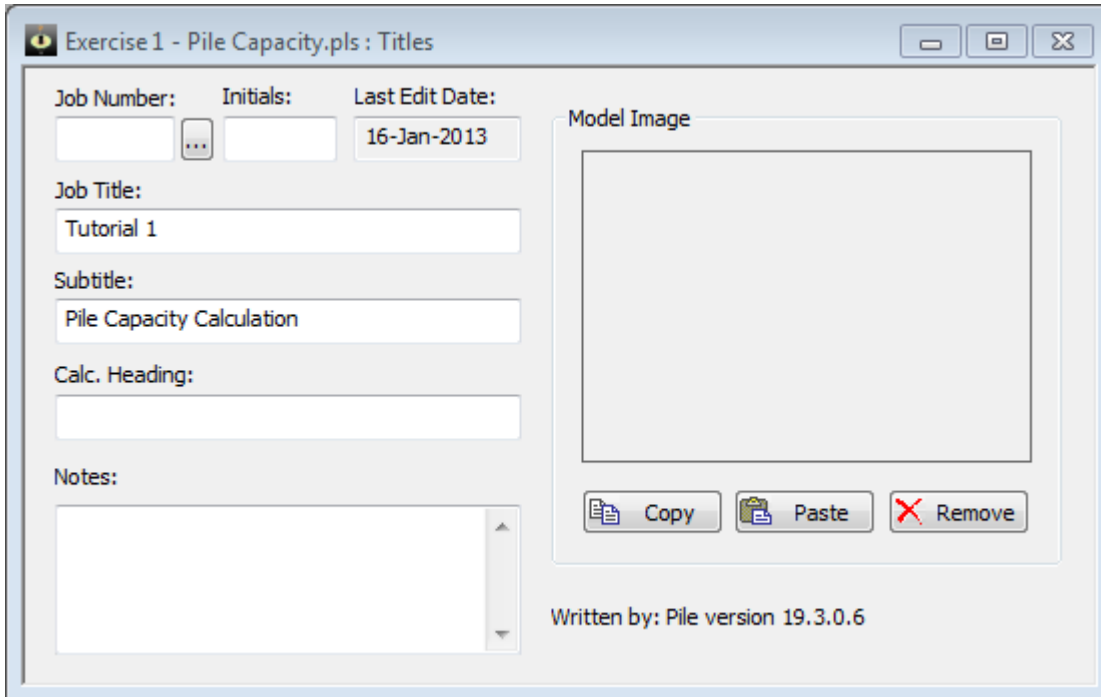


**Hint:** When using the Gateway, click '+' to expand an item. Double click on the item to open the corresponding window

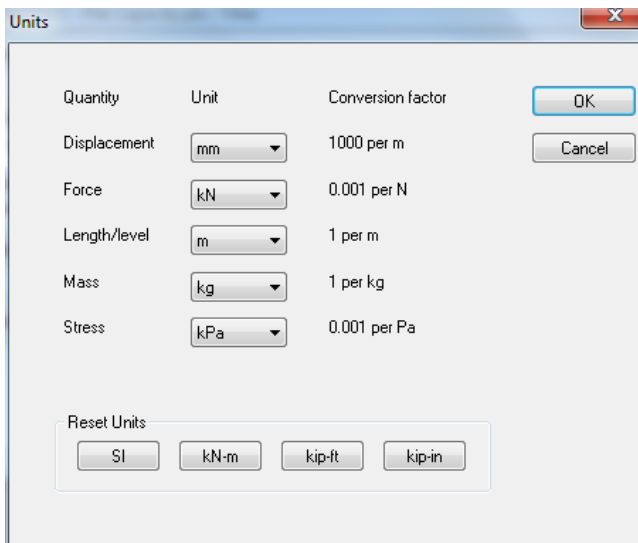
## 1.2 Pile Capacity Analysis

### 1.2.1 Creating the input

- Once opened, create a new file by clicking the '**New File**' icon  on the top left of the program or clicking **Ctrl + N**
- Fill the **Titles and Units** dialog box:



**Hint:** Should the user need to change the Units, they should do so at this point by double clicking on **Units** in the Gateway

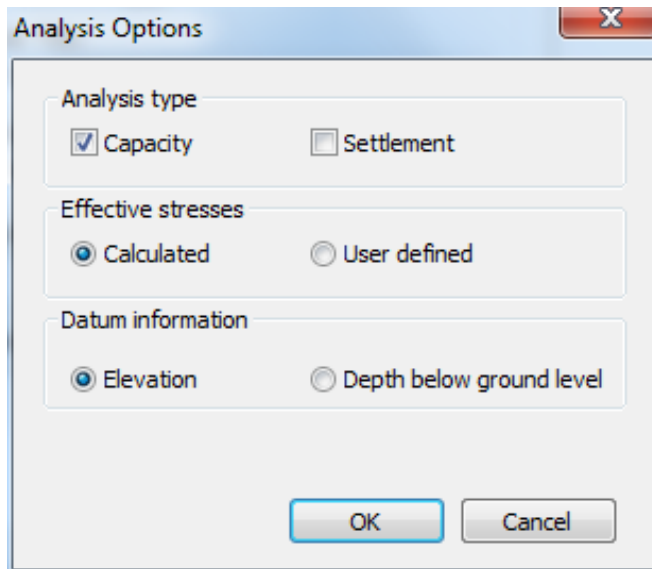


Quantity	Unit	Conversion factor
Displacement	mm	1000 per m
Force	kN	0.001 per N
Length/level	m	1 per m
Mass	kg	1 per kg
Stress	kPa	0.001 per Pa

Reset Units

SI kN-m kip-ft kip-in

- Double click on *Analysis Options* in the Gateway.
- Select **Capacity** and the following options:



- Double click on **Capacity Data** in the Gateway
- Select the **Working Load Calculation Method** and click **OK**



- Choose the following options:

**Working Load**

**Compression**

☒ Calculate compressive capacity

**Global FoS**

☒ Use global FoS criterion

Global factor on ultimate capacity ( $F_g$ )

**Partial FoS**

☒ Use partial FoS criterion

Partial factor on ultimate skin friction ( $F_{s1}$ )

Partial factor on ultimate end bearing ( $F_b$ )

**Shaft FoS**

☒ Use shaft FoS criterion

Factor applied to ultimate skin friction ( $F_{s2}$ )

**Limiting pile stress**

☒ Use limiting pile stress criterion

Limiting pile material stress at working load  kPa

**Tension**

☒ Calculate tensile capacity

**Shaft FoS**

☒ Use shaft FoS criterion

Factor applied to ultimate skin friction ( $F_{s2}$ )

**Limiting pile stress**

☒ Use limiting pile stress criterion

Limiting pile material stress at working load  kPa

< Back Finish Cancel Help

- Double click on **Pile Properties** in the Gateway  
 Select a Solid Pile without under-ream,  $E = 20\,000\,000$  kPa  
 Select the inputs to calculate the Pile Capacity between 5m and 35m below ground level, calculating the values every metre  
 Input Pile Diameters of 600mm and 700mm.

**Hint:** Move onto the next line in the input table to store the previous values inputted

- Double click on **Material Properties** in the Gateway
- Define materials which make up the soil profile around the pile.

Material #1	
General Properties	
Description	Layer 1
Type	Effective Stress
Bulk unit weight (kN/m <sup>3</sup> )	20
Contributes to negative skin friction?	No
Effective Stress Properties	
Skin friction computed by	Earth pressure method
Angle of friction, delta (deg)	25
Horizontal effective stress profile is :	Calculated
Coefficient of earth pressure, K	0.8
Limiting value of skin friction per unit area (kN/m <sup>2</sup> )	Unlimited
Nq - specified or computed?	Specified

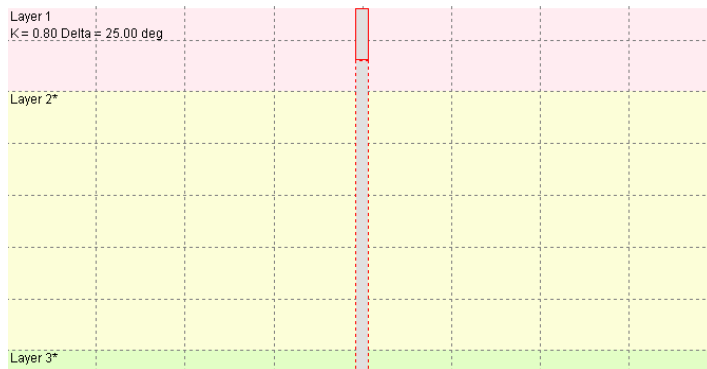
- Specify Nq for Material #1 as 50.

Material #2	
General Properties	
Description	Layer 2
Type	Total Stress
Bulk unit weight (kN/m <sup>3</sup> )	20
Contributes to negative skin friction?	No
Total Stress Properties	
Cu at Top of material (kN/m <sup>2</sup> )	60
Cu at Bottom of material (kN/m <sup>2</sup> )	260
Method of computation of $\alpha$	User-specified
$\alpha$ value	0.45
Limiting value of skin friction per unit area (kN/m <sup>2</sup> )	200
Method of computation of $N_c$	User-specified
$N_c$ value	9
Limiting value of end bearing stress (kN/m <sup>2</sup> )	Unlimited

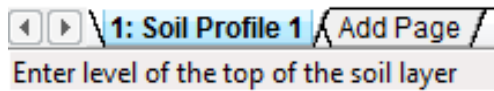
- See the help file for instructions on how to set the limiting value of end bearing to 'unlimited'.

Material #3	
General Properties	
Description	Layer 3
Type	Total Stress
Bulk unit weight (kN/m <sup>3</sup> )	20
Contributes to negative skin friction?	No
Total Stress Properties	
Cu at Top of material (kN/m <sup>2</sup> )	260
Cu at Bottom of material (kN/m <sup>2</sup> )	260
Method of computation of $\alpha$	User-specified
$\alpha$ value	0.45
Limiting value of skin friction per unit area (kN/m <sup>2</sup> )	200
Method of computation of $N_c$	User-specified
$N_c$ value	9
Limiting value of end bearing stress (kN/m <sup>2</sup> )	Unlimited

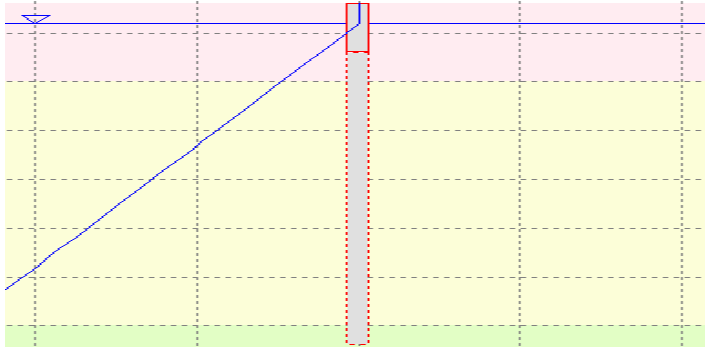
- Double click on *Soil Profiles* in the Gateway
- Define layers constituting the soil profile around the pile:
  - top of Layer 1 at 8mOD
  - top of Layer 2 at 0mOD
  - top of Layer 3 at -25mOD



**Note:** The tab at the bottom of the table shown below allows the user to enter more than one soil profile, should they wish to compare the impact of different soil properties or stratigraphy boundaries



- Double click on **Groundwater** in the Gateway




- To define a hydrostatic pore-water distribution, just define the level of the phreatic surface.
- For this example, input the two phreatic surfaces for the soil profile to compare the impact of dewatering around the pile:
  - Groundwater Profile 1: 6 mOD
  - Groundwater Profile 2 : 0 mOD

**Hint:** Use the tab at the bottom of the table to input two different groundwater profiles

- Double click on **Soil Profile - Groundwater Map** in the Gateway.
- Assign the inputted soil profile to Groundwater Profile 1.

**Note:** This feature allows the user to analyse the impact of different groundwater levels on the soil stratigraphy in one file. Once the analysis has been run for one profile, the file can be reanalysed to compare results

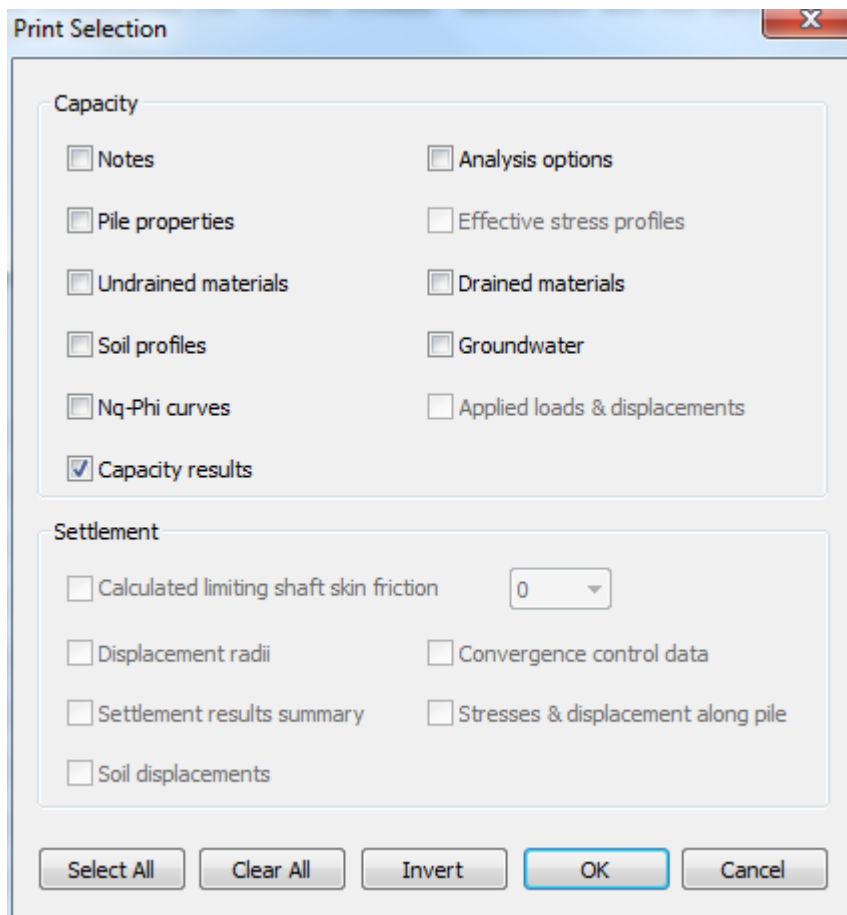
## 1.2.2 Analysis and Outputs

- Prior to analysis, check all inputs and save the file
- Click on the  analyse button to carry out the analysis
- The solution progress window will automatically appear and the analysis will run.

### 1.2.2.1 Tabular Outputs

The program will give the user the option to view tabular inputs and outputs.

- Select the outputs only by ticking Capacity Results:



The image shows a 'Print Selection' dialog box with two main sections: 'Capacity' and 'Settlement'. The 'Capacity' section contains several checkboxes, with 'Capacity results' being the only one checked. The 'Settlement' section contains several checkboxes, all of which are unchecked. At the bottom of the dialog are five buttons: 'Select All', 'Clear All', 'Invert', 'OK', and 'Cancel'.

Capacity	
<input type="checkbox"/> Notes	<input type="checkbox"/> Analysis options
<input type="checkbox"/> Pile properties	<input type="checkbox"/> Effective stress profiles
<input type="checkbox"/> Undrained materials	<input type="checkbox"/> Drained materials
<input type="checkbox"/> Soil profiles	<input type="checkbox"/> Groundwater
<input type="checkbox"/> Nq-Phi curves	<input type="checkbox"/> Applied loads & displacements
<input checked="" type="checkbox"/> Capacity results	

Settlement	
<input type="checkbox"/> Calculated limiting shaft skin friction	0 ▼
<input type="checkbox"/> Displacement radii	<input type="checkbox"/> Convergence control data
<input type="checkbox"/> Settlement results summary	<input type="checkbox"/> Stresses & displacement along pile
<input type="checkbox"/> Soil displacements	

Select All   Clear All   Invert   OK   Cancel

- The tabular outputs enable the user to look at stress profiles and specific values for the pile as shown below:

**Tutorial 1 Exercise 1.pls : Tabular Output**

**Cross-section 1 results:**

**Results - Compression**

**Soil Profile 1: Soil Profile 1**

Level	Pile length	Ultimate base capacity ( $Q_b$ )	Cumulative external friction ( $Q_s$ )	Negative skin friction ( $Q_{nsf}$ )	Ultimate capacity	Allowable capacity	Limiting criterion #
[mOD]	[m]	[kN]	[kN]	[kN]	[kN]	[kN]	
3.0000	5.0000	989.60	144.15	0.0	1133.8	131.05	3
2.0000	6.0000	1131.0	196.89	0.0	1327.9	178.99	3
1.0000	7.0000	1272.3	256.66	0.0	1529.0	233.33	3
0.0	8.0000	1413.7	323.46	0.0	1737.2	294.06	3
0.0	8.0000	152.68	323.46	0.0	476.14	190.46	1
-1.0000	9.0000	173.04	377.75	0.0	550.79	220.31	1
-2.0000	10.000	193.40	438.82	0.0	632.22	252.89	1
-3.0000	11.000	213.75	506.68	0.0	720.43	288.17	1
-4.0000	12.000	234.11	581.32	0.0	815.43	326.17	1
-5.0000	13.000	254.47	662.75	0.0	917.22	366.89	1
-6.0000	14.000	274.83	750.97	0.0	1025.8	410.32	1
-7.0000	15.000	295.18	845.97	0.0	1141.2	456.46	1
-8.0000	16.000	315.54	947.76	0.0	1263.3	505.32	1
-9.0000	17.000	335.90	1056.3	0.0	1392.2	556.89	1
-10.000	18.000	356.26	1171.7	0.0	1527.9	611.18	1
-11.000	19.000	376.61	1293.8	0.0	1670.5	668.18	1
-12.000	20.000	396.97	1422.8	0.0	1819.7	727.90	1
-13.000	21.000	417.33	1558.5	0.0	1975.8	790.33	1
-14.000	22.000	437.69	1701.0	0.0	2138.7	855.47	1
-15.000	23.000	458.04	1850.3	0.0	2308.3	923.33	1
-16.000	24.000	478.40	2006.3	0.0	2484.8	993.90	1
-17.000	25.000	498.76	2169.2	0.0	2668.0	1067.2	1
-18.000	26.000	519.12	2338.9	0.0	2858.0	1143.2	1
-19.000	27.000	539.47	2515.3	0.0	3054.8	1221.9	1

- The user can determine which limiting criterion applies to different lengths of pile by using the key provided in the Tabular Output:

**Tutorial 1 Exercise 1.pls : Tabular Output**

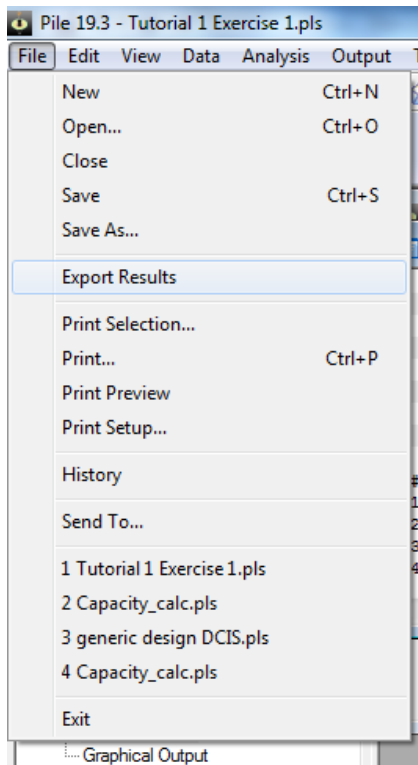
-21.000	29.000	580.19	2888.5	0.0	3468.7	1387.5	1
-22.000	30.000	600.55	3085.3	0.0	3685.8	1474.3	1
-23.000	31.000	620.90	3288.9	0.0	3909.8	1563.9	1
-24.000	32.000	641.26	3499.2	0.0	4140.5	1656.2	1
-25.000	33.000	661.62	3716.4	0.0	4378.0	1751.2	1
-25.000	33.000	661.62	3716.4	0.0	4378.0	1751.2	1
-26.000	34.000	661.62	3936.9	0.0	4598.5	1839.4	1
-27.000	35.000	661.62	4157.5	0.0	4819.1	1927.6	1

# Limiting criteria :

- 1: Global factor of safety
- 2: Shaft and base factors of safety
- 3: Shaft factor of safety
- 4: Pile material limiting stress [Compression]

### 1.2.2.2 Exporting Tabular Results

- Select **File** > **Export Results**



- Save the file as **Tutorial 1 Exercise 1.csv** to export the results to Excel

### 1.2.2.3 Graphical Outputs

- Double click on **Graphical Output** in the Output section of the Gateway
- Using the excerpt from the User Manual shown below, plot the ultimate and design capacities for the pile and determine the length of pile required to support a load of 1000kN.

## Graphical toolbar buttons



**Axis** - provides a reference grid behind the drawing.



**Set Scale** - this allows the user to toggle between the default 'best fit' scale, the closest available engineering scale. e.g. 1:200, 1:250, 1:500, 1:1000, 1:1250, 1:2500, or exact scaling. The same options are available via the View menu "Set exact scale" command.



**Save Metafile** - this save icon allows the image 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 word processors, spreadsheets and drawing packages.



**Zoom Facility** - 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.



**Smaller/Larger font** - allows adjustment of the font sizes on the graphical output view.



**Edit colours** - allows line and fill colours to be edited.



**Axis** - provides a reference grid behind the drawing.



**Save BMP** - allows the file to be saved in the format of bitmap.



**Copy** - allows to copy the graphical view to be copied to clip board.

## Capacity



**Vertical effective stress** - toggles the vertical effective stress plot.



**Horizontal effective stress** - toggles the horizontal effective stress plot.



**Pore water pressure** - toggles the pore water pressure plot.



**Undrained cohesion** - toggles the undrained cohesion plot.



**Unit shaft friction** - toggles the unit shaft friction plot.



**External skin friction compression** - toggles the external skin friction compression plot.



**Total skin friction compression** - toggles the total skin friction compression plot.



**Total skin friction tension** - toggles the total skin friction tension plot.



**End bearing capacity** - toggles the end bearing capacity plot.



**Internal skin friction** - toggles the internal skin friction plot.












**Wall end bearing** - toggles the wall end bearing plot.



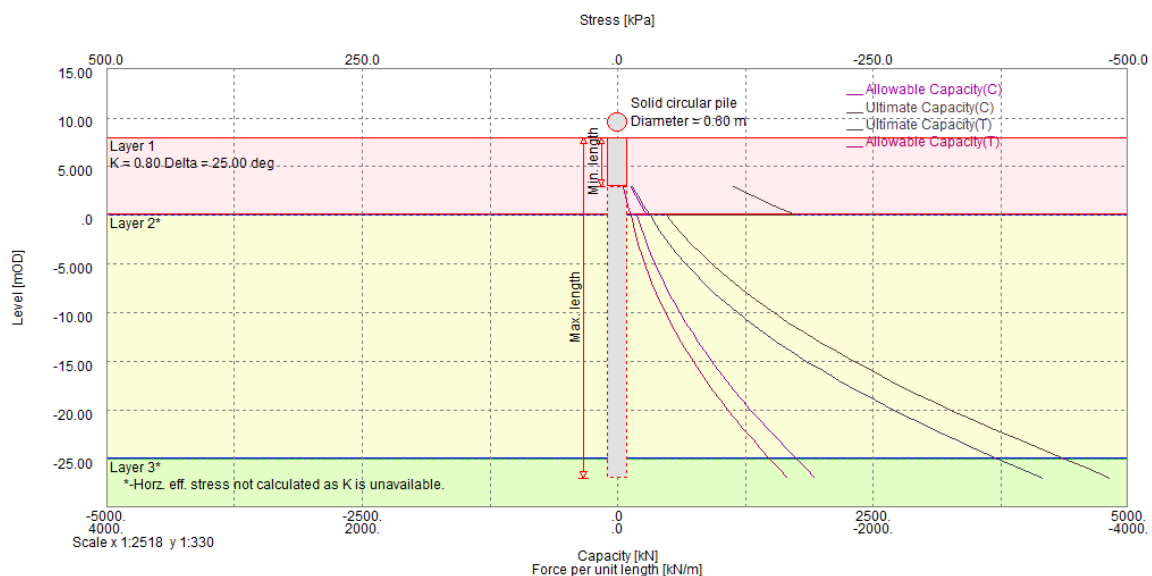
**Plugged end bearing** - toggles the plugged end bearing plot.



-  **Plugged capacity** - toggles the plugged capacity plot.
-  **Unplugged capacity** - toggles the unplugged capacity plot.
-  **Unplugged capacity - auto plugged** - toggles the unplugged capacity - auto plugged plot.
-  **Ultimate load compression** - toggles the ultimate load compression plot.
-  **Working load compression** - toggles the working load compression plot.
-  **Design load compression** - toggles the design load compression plot.
-  **Ultimate load tension** - toggles the ultimate load tension plot.
-  **Working load tension** - toggles the working load tension plot.
-  **Design load tension** - toggles the design load tension plot.

Capacity results can be viewed for a selected soil profile Soil Profile 1 — + and cross-section Cross-section 1 — +.

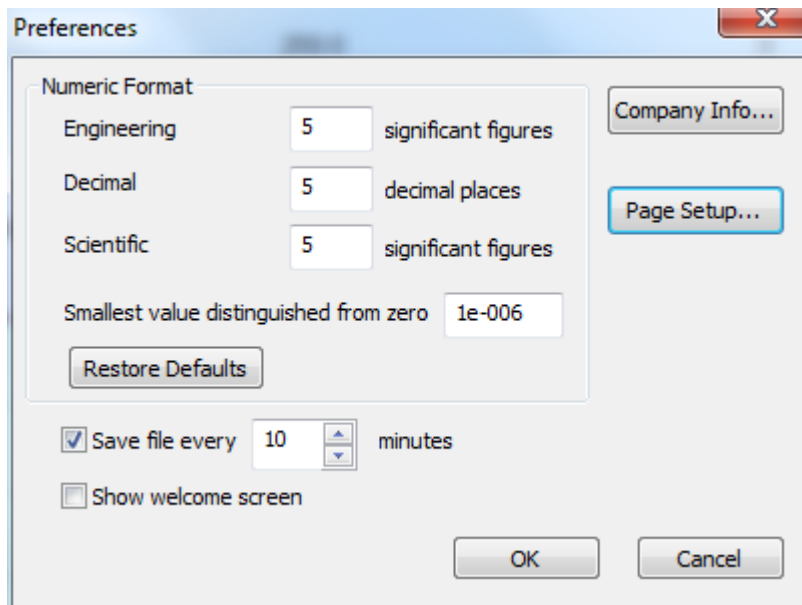
- The graphical output is shown below:



- Compare the results for a 600mm and 700mm diameter pile
- Re-run the analysis to view the impact of the other groundwater profile (phreatic surface at 0mOD) to the pile capacity

#### 1.2.2.4 Printing Graphical Outputs

- Go to **Tools > Preferences** in the **menu** bar




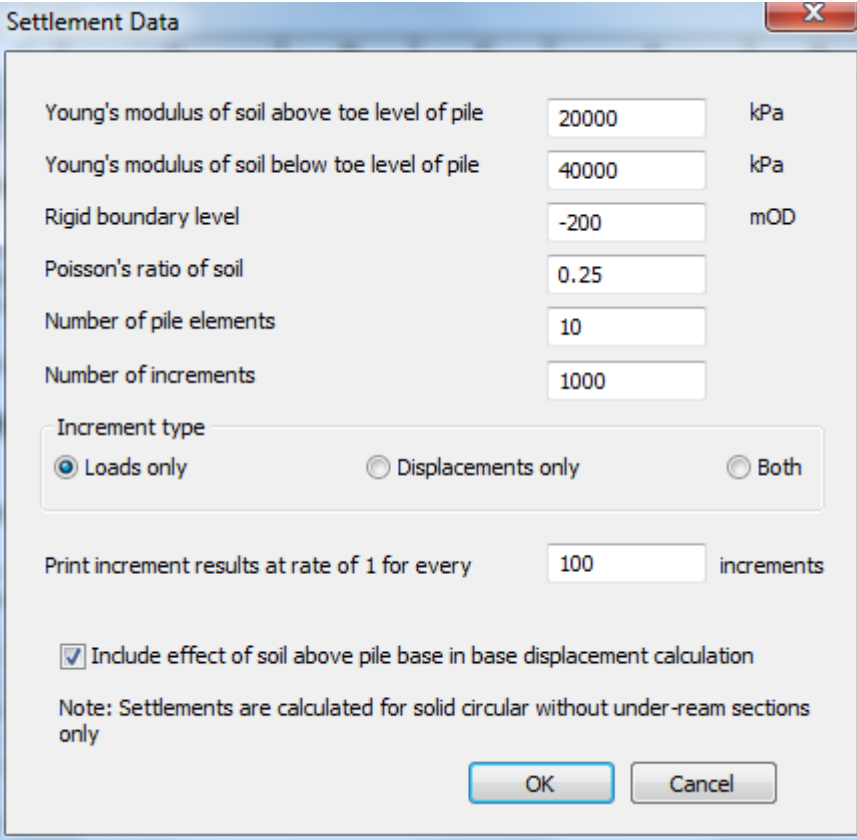
- Select appropriate options for the print output, including company information and logos.
- Select **File > Print Preview** in the menu bar to check the output for printing

**Note:** The user can also print the tabular output in this format

## 1.3 Pile Settlement Analysis

### 1.3.1 Creating the input

- Once opened, create a new file by clicking the '**New File**' icon  on the top left of the program or clicking **Ctrl + N**
- Fill the **Titles and Units** dialog box, outlining the Program is carrying out a Pile Settlement Analysis.
- Double click on *Settlement Data* in the Gateway.
- Input the following:



The image shows a screenshot of the 'Settlement Data' dialog box. It contains several input fields and options for configuring the settlement analysis. The fields are: Young's modulus of soil above toe level of pile (20000 kPa), Young's modulus of soil below toe level of pile (40000 kPa), Rigid boundary level (-200 mOD), Poisson's ratio of soil (0.25), Number of pile elements (10), and Number of increments (1000). There is a section for 'Increment type' with three radio buttons: 'Loads only' (selected), 'Displacements only', and 'Both'. Below this is a field for 'Print increment results at rate of 1 for every' (100) increments. A checkbox 'Include effect of soil above pile base in base displacement calculation' is checked. At the bottom, there is a note: 'Note: Settlements are calculated for solid circular without under-ream sections only'. The dialog has 'OK' and 'Cancel' buttons at the bottom right.

Young's modulus of soil above toe level of pile	20000	kPa
Young's modulus of soil below toe level of pile	40000	kPa
Rigid boundary level	-200	mOD
Poisson's ratio of soil	0.25	
Number of pile elements	10	
Number of increments	1000	

Increment type

☒ Loads only    ☐ Displacements only    ☐ Both

Print increment results at rate of 1 for every 100 increments

☒ Include effect of soil above pile base in base displacement calculation

Note: Settlements are calculated for solid circular without under-ream sections only

OK Cancel

**Note:** the excerpt from the manual outlines the definition for the different inputs and the user must read these before moving on.

Settlement data is enabled when settlement analysis is selected.

**Young's Modulus of soil above toe level of pile and Young's Modulus of soil below toe level of pile** are average value representing the soil stiffness above and below the pile toe respectively.

**Poisson's ratio** is the average value from the different soil layers around the pile.

Include effect of soil above pile base in base displacement calculation.

For calculation of stiffness at the base node user can include/exclude effect of soil above pile base.

**Depth of rigid boundary** – the pile is divided into the number of elements and Pile Stiffness is calculated for each element.

**Number of Increments** – the load is applied in a number of equal increments, and

**Increment type** – i.e. whether load alone is incremented, or applied displacement alone is incremented, or both of them are incremented.

Increasing the increments helps to reduce any incompatibilities between relative displacements at the pile-soil interface, and the mobilized skin friction.

The user may also specify the rate at which the results from various increments need to be printed, i.e. one in every 10 increments etc. Irrespective of the frequency specified, the program always prints the last increment.

- Enter the same data for the following as the previous example:
  - Pile Properties
  - Material Properties
  - Soil Profile
  - Groundwater Data

**Note:** You can copy and paste between tables in different pile analysis files.


- Double click on *Applied loads and displacements* in the Gateway Menu  
Input a 1500kN load

**Question:** At what loads will the pile stresses be presented in the output?

- Double click on *Displacement* Radii in the Gateway Menu
- Input displacement radii at:
  - 0.3m
  - 1m
  - 2m

**Question:** Why has a radii of 0.3m been chosen?

### 1.3.2 Analysis and Outputs

- Prior to analysis, check all inputs and save the file
- Click on the analyse button  to carry out the analysis
- The solution progress window will automatically appear and the analysis will run.

### 1.3.3 Tabular Outputs

The program will give the user the option to view tabular inputs and outputs.

- Select the Settlement outputs only

### 1.3.4 Graphical Outputs

A number of different options are available. Use the definitions below to produce two graphs outlined.

#### Settlement



**Limiting Shaft Skin Friction** - toggles the limiting shaft skin friction plot.



**Axis** - provides a reference grid behind the drawing.



**Shaft Skin Friction** - toggles the shaft skin friction plot.



**Pile Stress** - toggles the pile stress plot.

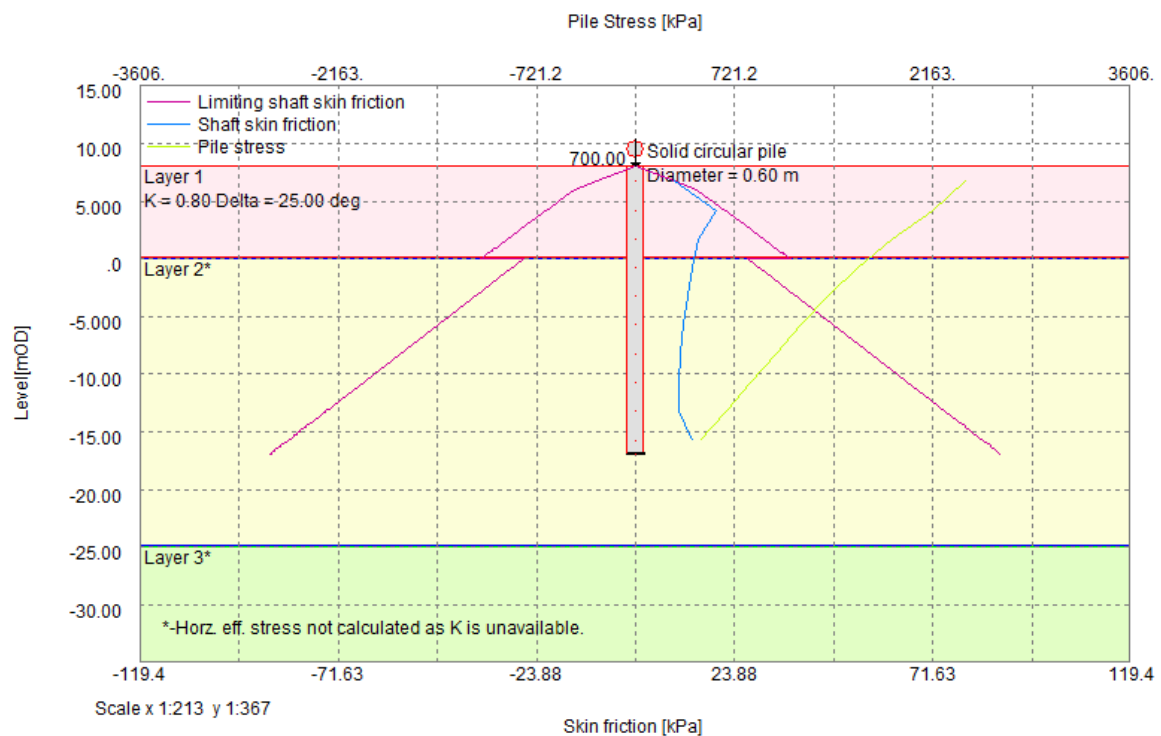
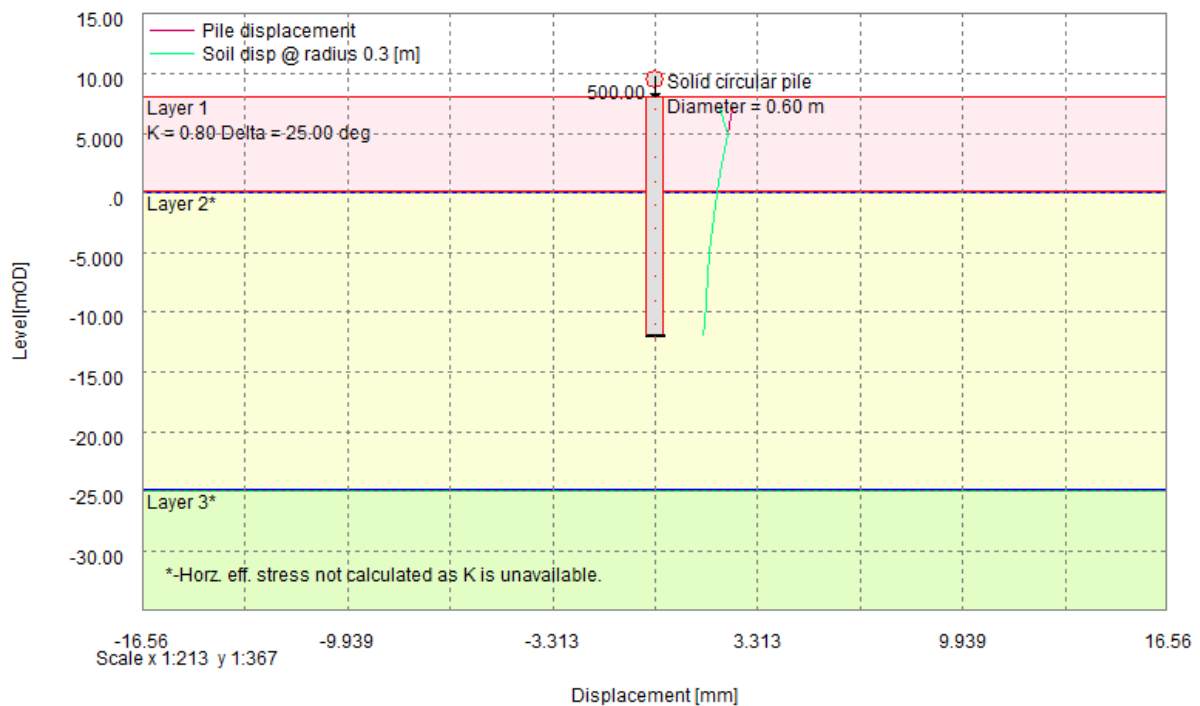


**Pile/Soil Displacement** - toggles the displacements for pile or soil.

Settlement results can be viewed for a selected soil profile Soil Profile 1 — + and cross-section Cross-section 1 — + and selected pile length 10.00 m — + and selected load/applied displacement increments Load Inc 1 — + (if load increments exist).

- 1 Pile displacement and Soil displacement at 0.3m for a 20m long pile with 500kN load acting
- 2 Limiting shaft skin friction, shaft skin friction and pile stress for a 25m long pile with 700kN load acting

The expected outputs are shown below:



# Section 2

## Step by Step Tutorial 2 Oasys Alp – Lateral Pile Capacity

### Objectives

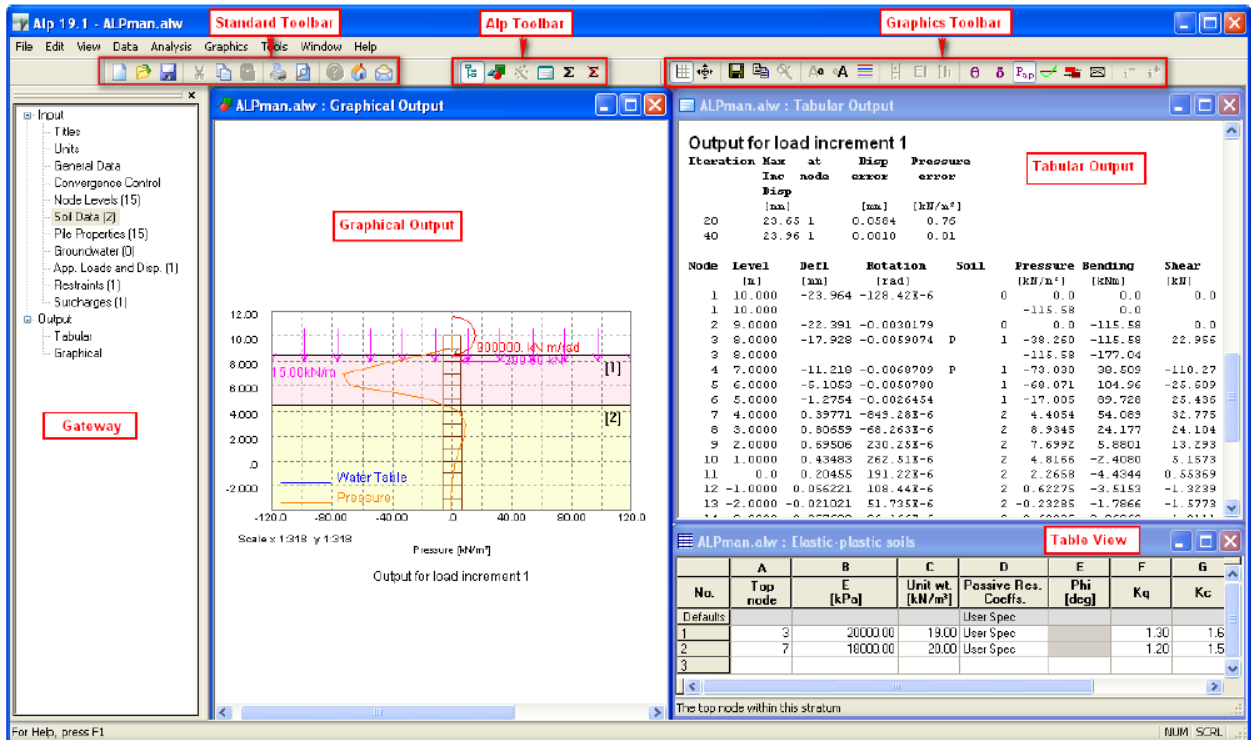
By the end of the session the user should be able to:

- Navigate the Alp Interface
- Use the Alp Wizard and Gateway
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs
- Export tabular outputs for further analysis




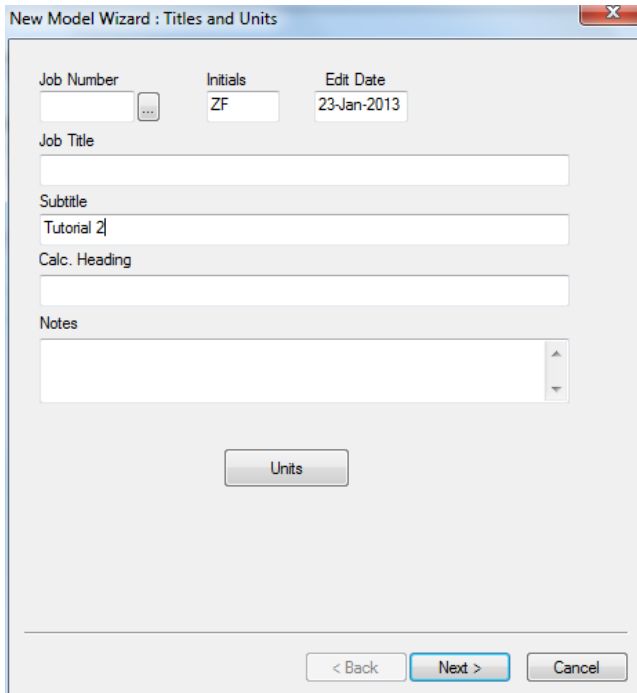
## 2.1 Components of the User Interface

The principal components of Alp's user interface are the Gateway, Table Views, Graphical Output, Tabular Output, toolbars, menus and input dialogs. These are illustrated below.



## 2.2 Creating the input

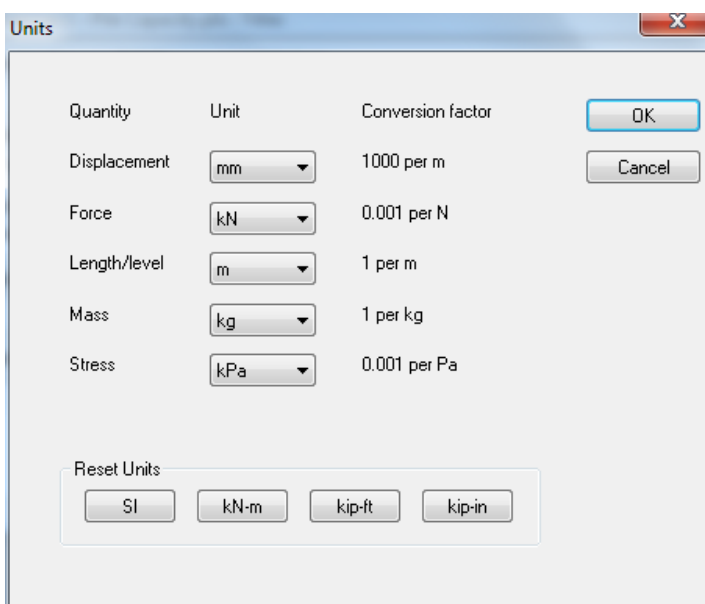
- Once opened, create a new file by clicking the '**New File**' icon  on the top left of the program or clicking **Ctrl + N**
- Fill the **Titles and Units** dialog box:



The 'New Model Wizard : Titles and Units' dialog box contains the following fields and controls:

- Job Number:** A text input field with a 'New' button next to it.
- Initials:** A text input field containing 'ZF'.
- Edit Date:** A date input field containing '23-Jan-2013'.
- Job Title:** A large text input field.
- Subtitle:** A text input field containing 'Tutorial 2'.
- Calc. Heading:** A text input field.
- Notes:** A large text area with a scroll bar.
- Units:** A button located below the Notes field.
- Navigation:** '< Back', 'Next >', and 'Cancel' buttons at the bottom right.

**Hint:** Should the user need to change the Units, they should do so at this point or by double clicking on *Units* in the Gateway.



The 'Units' dialog box displays a table of units and conversion factors, with 'OK' and 'Cancel' buttons on the right.

Quantity	Unit	Conversion factor
Displacement	mm	1000 per m
Force	kN	0.001 per N
Length/level	m	1 per m
Mass	kg	1 per kg
Stress	kPa	0.001 per Pa

Below the table is a 'Reset Units' section with four buttons: SI, kN-m, kip-ft, and kip-in.

- Select the default options for the **General Data** box.

New Model Wizard : General Data

Soil Model

☒ Elastic-plastic

☐ Specified P-Y curves

☐ Generated P-Y curves

Factor on soil E value: 0.8

Number of increments: 1

Loadcase

☒ Static

☐ Cyclic

Increment

☒ Loads only ☐ Displacements only ☐ Both

Input

☒ By level ☐ By node

☐ Use partial factors for soil parameters and loads

< Back Next > Cancel

- (Note: The user will have the opportunity change the Soil Model if required on leaving the wizard via the Gateway menu)
- In the **Pile and Soil Data** box, input the parameters for the following, keeping the rest of the options for Soil Properties and Node Generation Control Parameters as default:
  - Soil top 0mOD
  - Pile top 0mOD
  - Pile bottom -13mOD
  - Pile diameter 0.6m
  - Pile EI 70 000 kNm<sup>2</sup>
  - Force 50kN at -1mOD
  - Moment -150kN at -1mOD
  - Groundwater -2mOD

- Double click on *General Data* in the Gateway.
- Select the Generated P-Y curve option

**Alp1 : General Data**

**Soil Model**

☐ Elastic-plastic

☐ Specified P-Y curves

☒ Generated P-Y curves

Factor on soil E value:

Number of increments:

**Loadcase**

☒ Static

☐ Cyclic

**Increment**

☒ Loads only ☐ Displacements only ☐ Both

**Input**

☒ By level ☐ By node

Pile toe level [m]:

**Node Generation Control Parameters**

Ratio of maximum node spacing to minimum node spacing:

Maximum number of nodes:

Maximum node spacing [m]:

☐ Use partial factors for soil parameters and loads

**Section Wizard Options**

Concrete design code:

Bending axis: ☐ y ☒ z

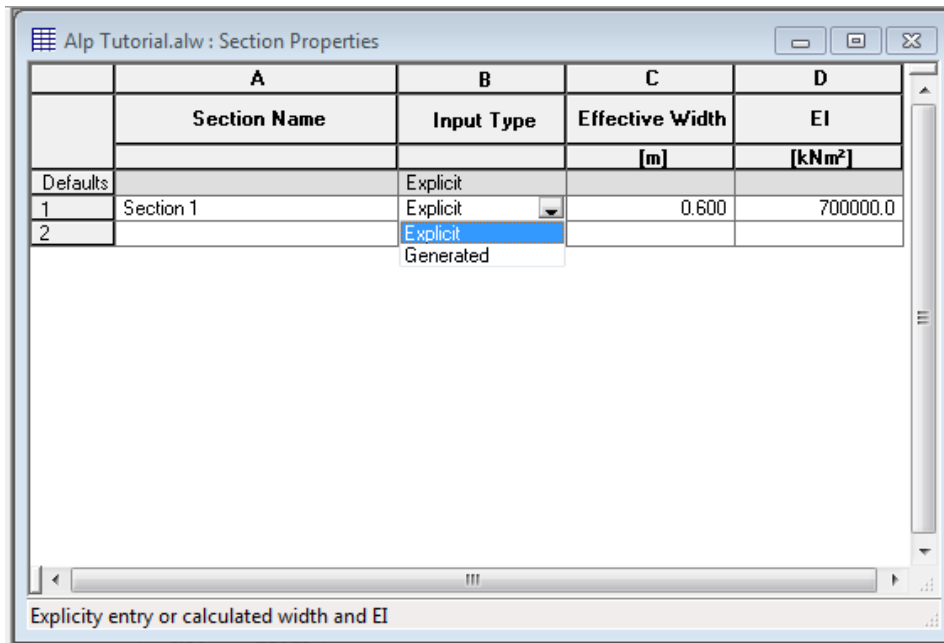
- Double click on **Soil Data** in the Gateway menu
- Enter the properties outlined below for the following stratigraphy:

Alp Tutorial.alw : Generated P-Y curves									
No.	A Top Level [m]	B Type	C Unit wt. [kN/m <sup>3</sup> ]	D E50	E Cu (top) [kPa]	F dCu/dz [kPa/m]	G K0	H K1 [kPa/m]	I Phi [deg]
Defaults		Soft Clay							
1	0.00	Soft Clay	17.00	0.0100	80.00	1.50			
2	-3.00	Stiff Clay	19.00	0.0050	100.00	1.50			
3	-8.00	Sand (API 21)	18.00						35.00
4									

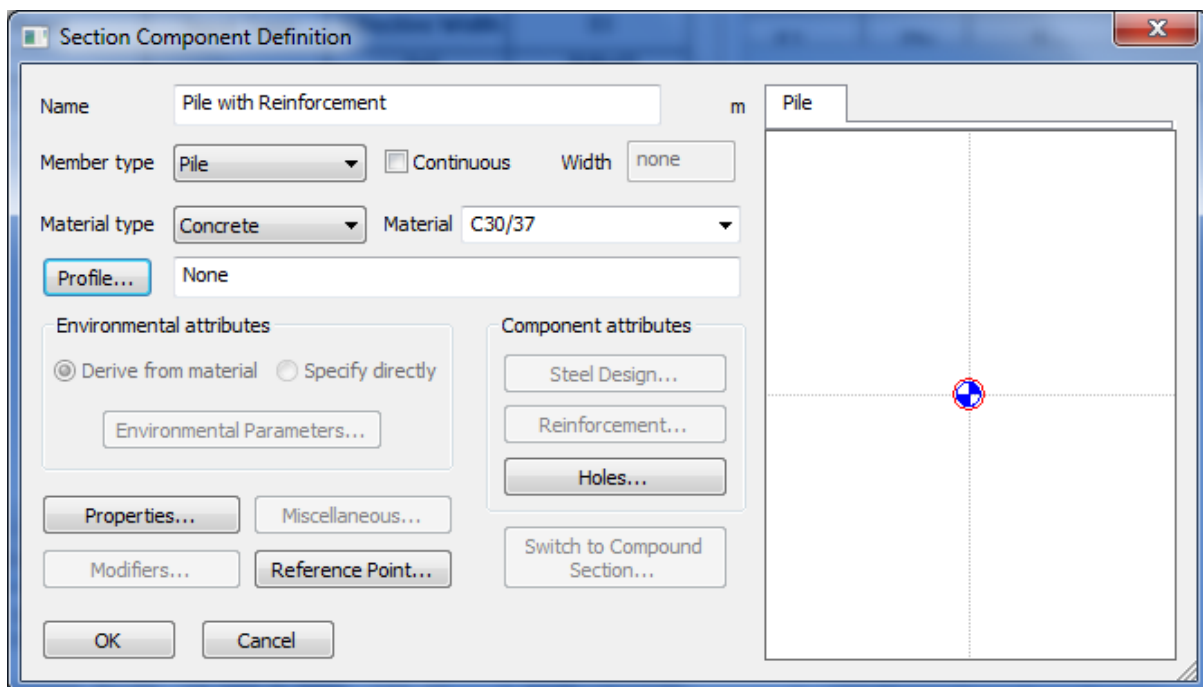
- Soft Clay      0mOD
- Stiff Clay      -3mOD
- Sand (API 18)   -8mOD

Use the User Manual to determine the E50 values-which Section have they been derived from and what do the curves look like?

- Double click on **Restraints** in the Gateway menu
- Input a rotational restraint at 0mOD with a stiffness of 100 kN m/radian
- Double click on **Sections** in the Gateway menu
- Select **Generated** in the **Input Type** column



- The Wizard will appear. Name the section **Pile with Reinforcement**.



- Click on the **Profile** button in the Wizard and choose a Circular section with 600mm diameter.
- Click on the **Reinforcement** button.

- The Section Concrete/Reinforcement Definition Box will appear. Click on the **Add** button.
- In the General Reinforcement box, select the following options for the reinforcement bars:

**General Reinforcement Definition**

Bar details

Bar: 12mm m Area: 113.097E-4 m<sup>2</sup>

☒ Bar ☐ Pair ☐ Bundle (3 bars) ☐ Bundle (4 bars)

Reinforcement: 500B Pre-stress... None

Start: 0 % Finish: 100 %

Bar positions

Definition type: Circle Extents: (-0,-0) (0,0)

☒ Number of bars: 8

☐ Maximum centre/centre bar spacing: 0 m


Centre: y: 0 z: 0 m

Point on circle: y: 0.2 z: 0 m

OK Cancel

- Now click on **OK** and exit the Section Wizard.

## 2.3 Analysis and Outputs

- Prior to analysis, check all inputs and save the file
- Click on the analyse button  to carry out the analysis
- The solution progress window will automatically appear and the analysis will run.

### 2.3.1 Tabular Outputs

The program will give the user the option to view tabular inputs and outputs.

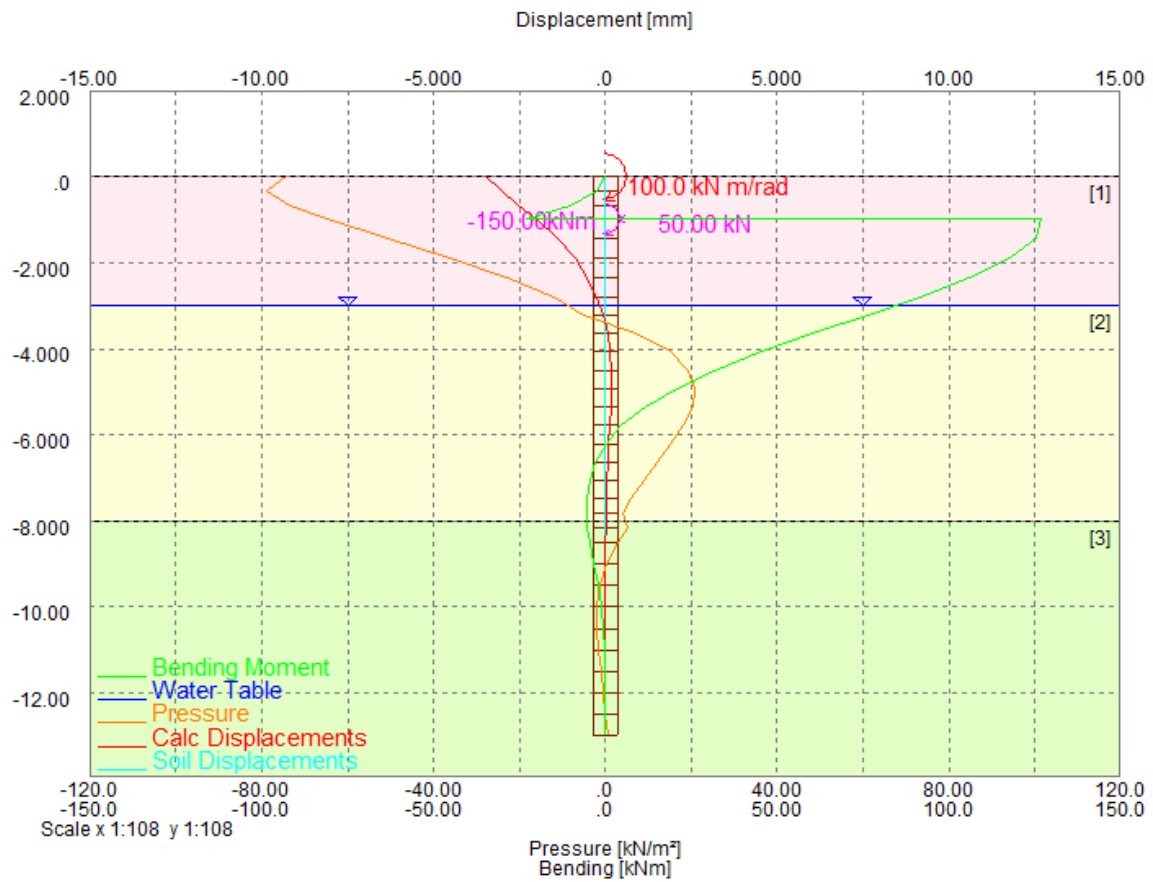
- View the tabular output.

**Question:** What is load increment?

- Export the tabular output

## 2.3.2 Graphical Outputs

- Double click on **Graphical Output** in the Gateway



- Use the following icons to determine the maximum bending moment and effective length of the pile:



deflection



rotation



pressure



bending moment



shear



- Zoom into a section of the pile between -8mOD and -10mOD and unzoom

**Hint:** To zoom, right left click and draw a box. To unzoom, use the unzoom icon

- Check your results with the tabular results

**Question:** Go back to *General Data* and increase the number of increments. How does it change your calculation?

# Section 3

## Step by Step Tutorial 3 Oasys AdSec –Section Analysis

### Objectives

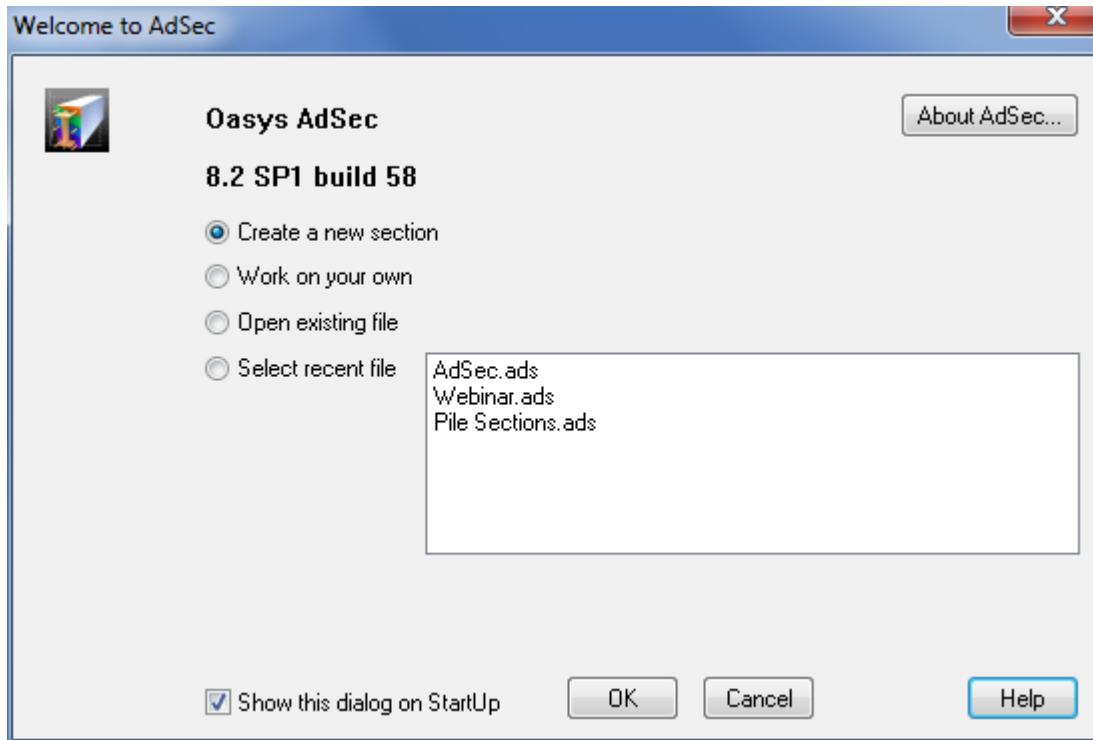
By the end of the session the user should be able to:

- Navigate the AdSec Interface
- Use the Alp Wizard and Gateway
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs and plot load cases
- Understand the difference between Serviceability and Ultimate Limit State Calculations

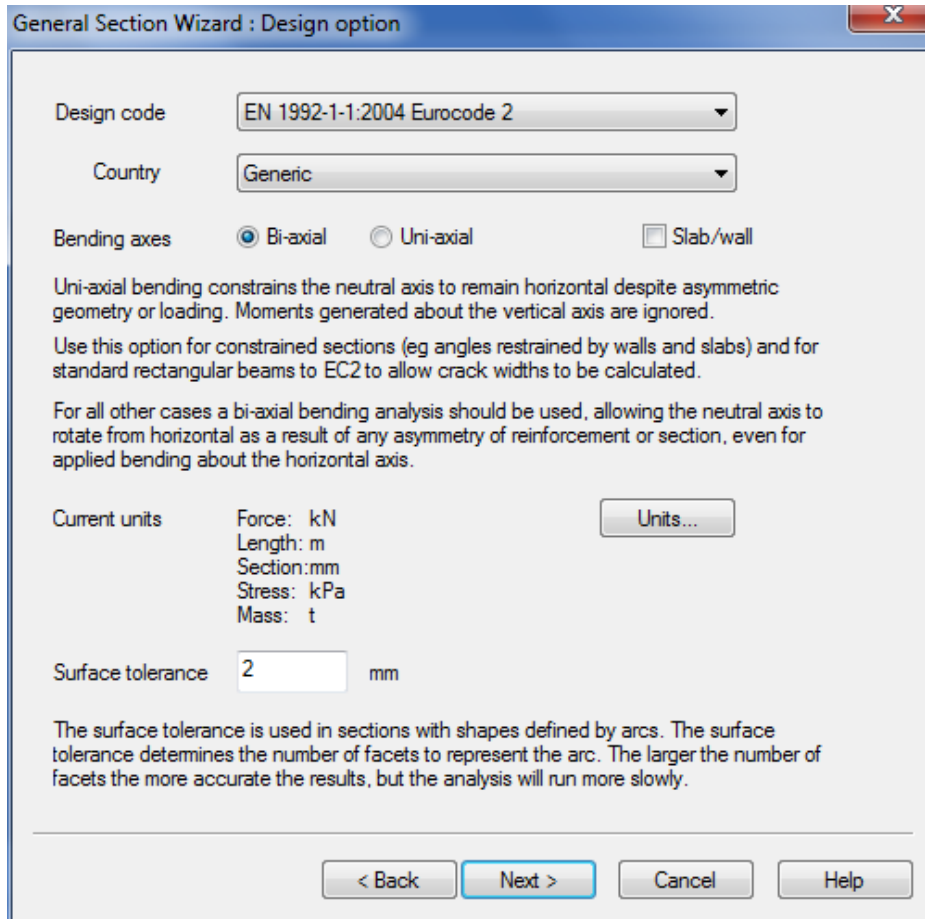
## 3.1 Input

### 3.1.1 Creating the Section

- Once opened, the program start up screen will give a number of options.
- Choose to **Create a New Section**.



- Complete the **Titles and Units** section of the Wizard.
- In the Design Option section, choose **Eurocode 2**



The dialog box is titled "General Section Wizard : Design option". It contains the following fields and options:

- Design code:** A dropdown menu showing "EN 1992-1-1:2004 Eurocode 2".
- Country:** A dropdown menu showing "Generic".
- Bending axes:** Two radio buttons, "Bi-axial" (selected) and "Uni-axial".
- Slab/wall:** A checkbox that is currently unchecked.
- Text description:**

Uni-axial bending constrains the neutral axis to remain horizontal despite asymmetric geometry or loading. Moments generated about the vertical axis are ignored.

Use this option for constrained sections (eg angles restrained by walls and slabs) and for standard rectangular beams to EC2 to allow crack widths to be calculated.

For all other cases a bi-axial bending analysis should be used, allowing the neutral axis to rotate from horizontal as a result of any asymmetry of reinforcement or section, even for applied bending about the horizontal axis.
- Current units:** A list showing "Force: kN", "Length: m", "Section: mm", "Stress: kPa", and "Mass: t". To the right is a "Units..." button.
- Surface tolerance:** A text input field containing the number "2", followed by the unit "mm".
- Text description:**

The surface tolerance is used in sections with shapes defined by arcs. The surface tolerance determines the number of facets to represent the arc. The larger the number of facets the more accurate the results, but the analysis will run more slowly.
- Navigation buttons:** At the bottom are four buttons: "< Back", "Next >" (highlighted in blue), "Cancel", and "Help".

**Note:** The user can click on the Units button if you wish to change from SI Units

- In the **Definition** section of the Wizard, name the section appropriately and choose an Aggregate Size of 20mm.

General Section Wizard : Definition

Section

Name: Reinforced Pile

Material Type: Concrete

Material Grade: C30/37

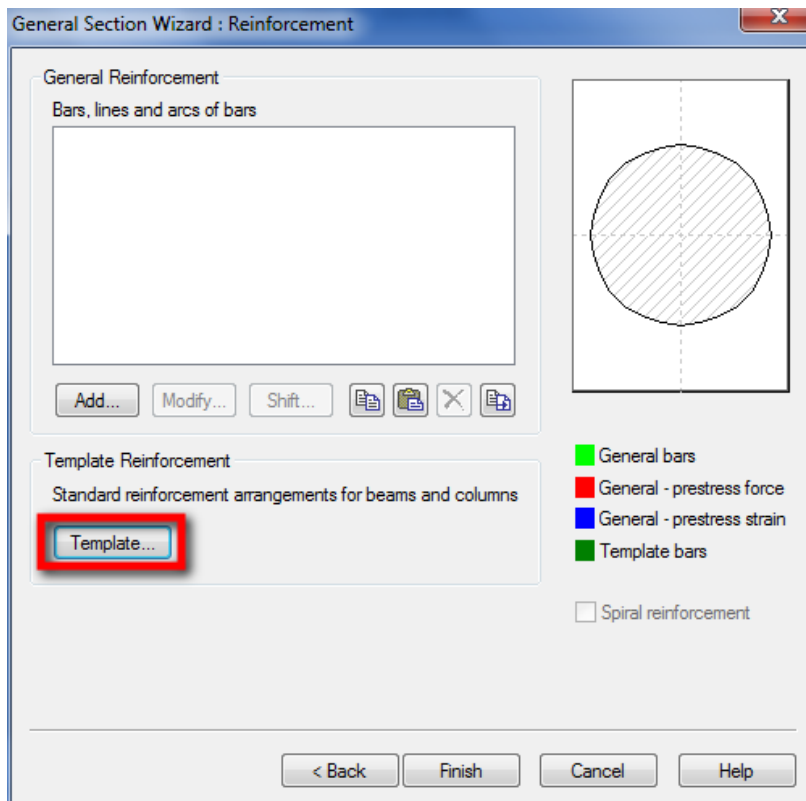
**Section...** STD C 600.  
eg STD R 400 300

The 'Section...' button opens the 'Section Wizard' to define the new section shape and dimensions.

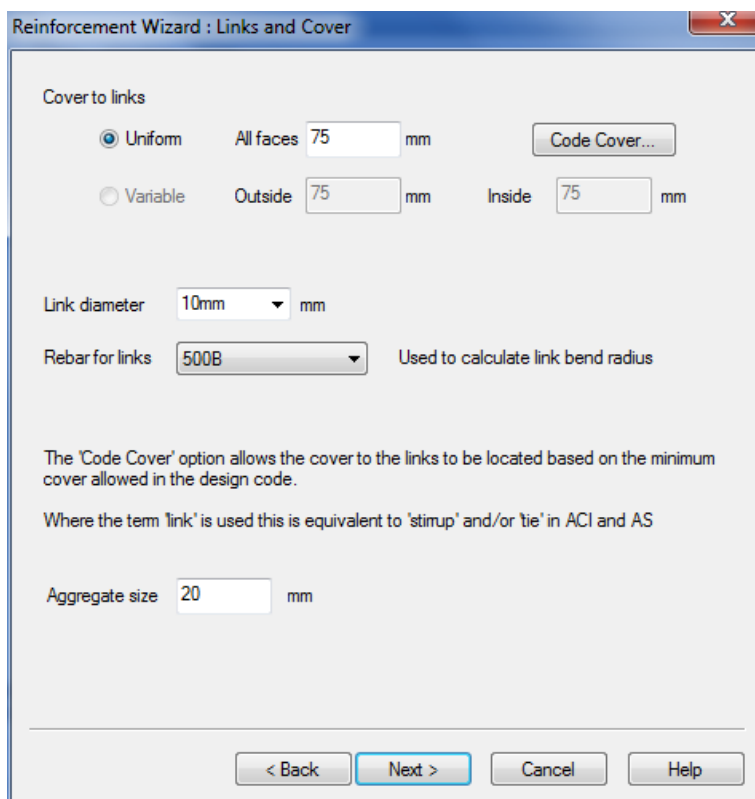
Aggregate size: 20 mm

< Back Next > Cancel Help

- Click on the **Section** button to open the Section Wizard.
- Choose a circular section with 600mm diameter.
- In the **Reinforcement** section of the wizard, click on the **Template** button.



- In the **Template Definition** box, choose Column Arrangement.
- In the **Links and Cover** box, choose the following options:



- Finally, in the **Circular/Elliptical Reinforcement** section of the wizard, choose the following options:

The screenshot shows a software dialog box titled "Reinforcement Wizard : Circular/Elliptical Reinforcement". It is divided into two main sections: "Outside Reinforcement" and "Inside Reinforcement".

**Outside Reinforcement:**

- Bars per ring:** A numeric input field with the value "8".
- Diameter:** A dropdown menu showing "40mm" followed by "mm".
- Number of rings:** A numeric input field with the value "1".
- Clear distance between rings:** A numeric input field with the value "25" followed by "mm".
- Rebar:** A dropdown menu showing "500B".

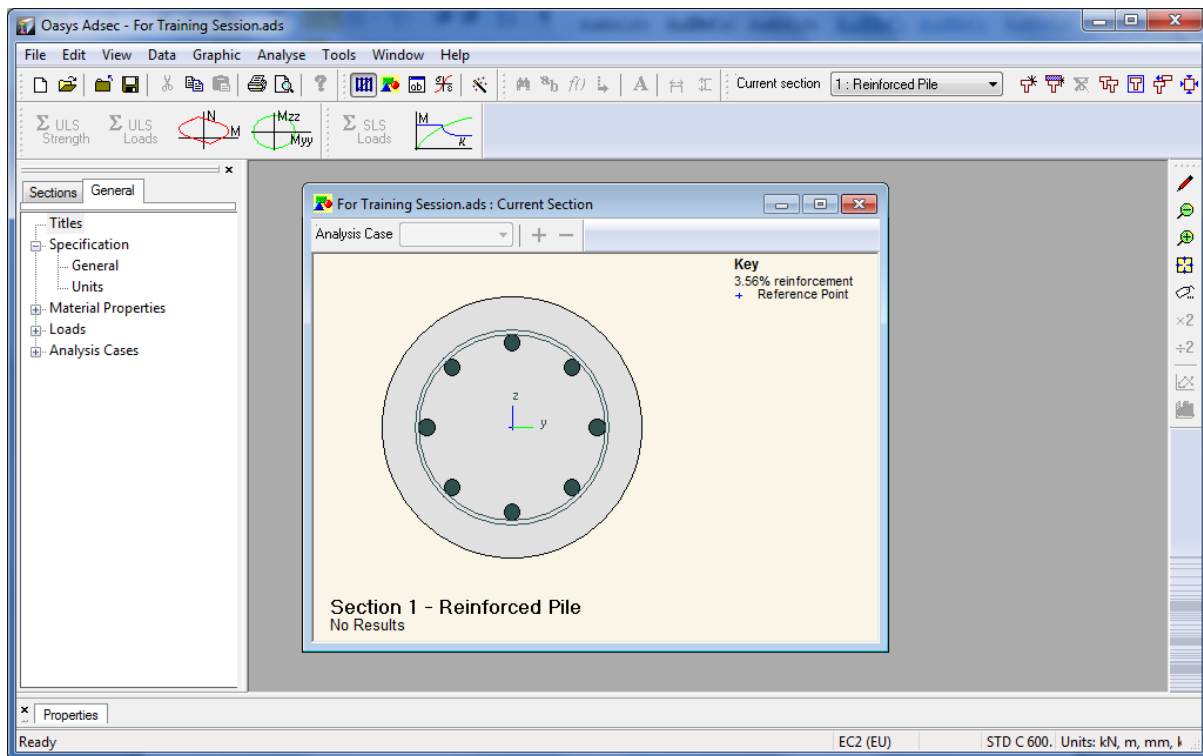
**Inside Reinforcement:**

- Include:** An unchecked checkbox.
- Bars per ring:** A numeric input field with the value "0".
- Diameter:** A dropdown menu (empty) followed by "mm".
- Number of rings:** A numeric input field with the value "0".
- Clear distance between rings:** A numeric input field with the value "25" followed by "mm".
- Rebar:** A dropdown menu showing "500B".

At the bottom of the dialog are four buttons: "< Back", "Finish" (highlighted in blue), "Cancel", and "Help".

### 3.1.2 Applying the Load Cases

- Once the User exits the Wizard, they can view the Section and the Gateway:



**Note:** Using the Material Properties section of the Gateway, the User can amend standard properties and apply these to their section if the options offered are not sufficient)

- Expand Loads in the **General** tab of the Gateway.
- Double click on Loading and input the following loads:

For Training Session.ads : Loads					
Load	A Load Case	B Load Type	C Force N [kN]	D Moment	
				Myy [kNm]	Mzz [kNm]
Defaults	1	Section Force	0	0	0
1	1	Section Force	3000	100	0
2	2	Section Force	1700	650	0
3					

Define the loads (or strains) applied to the section (or component)



- Double click on **Analysis Cases** in the Gateway and input the following cases:

#### Ultimate Limit State Case

	A	B	C
Analysis Case	Name	Description	Prestress Factor
Defaults	ULS Case #	L#	1
1	ULS Case 1	1.4L1	1
2	ULS Case 2	1.4L2	1
3			



#### Servicability Limit State Case

	A	B	C	D	E	F	G
Analysis Case	Name	Analysis Type	Load Description	Prestress Factor	Creep Coeff.	Crack Width Strain [mm]	Duration Factor
Defaults	SLS Case #	Long	L#	1	2	Equation 7.9	Normal
1	SLS Case 1	Long	L1	1	2	Equation 7.9	Normal
2	SLS Case 2	Long	L2	1	2	Equation 7.9	Normal

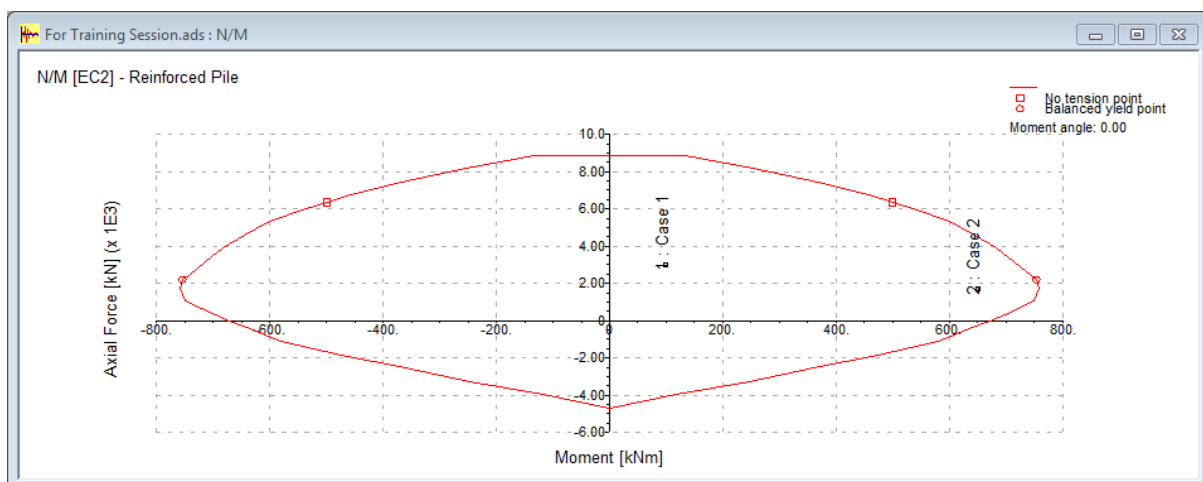
## 3.2 Analysis and Outputs

- Prior to analysis, check all inputs and save the file


### 3.2.1 Load Moment Curve

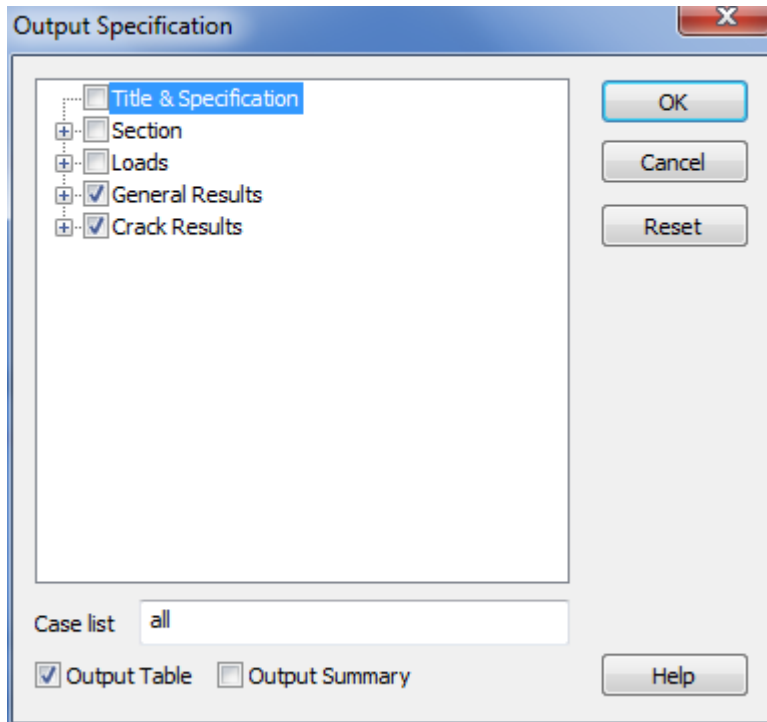
- Click on the N-M Chart Icon 
- Plot the N-M Chart
- Click on the *Additional chart Points* icon 
- Use the Load Point button to plot the specified loads on the chart

**Note:** Which load case is likely to cause the most cracking and why? Stop to discuss with the trainer



### 3.2.2 SLS Loads


- Click on the SLS Loads icon. 
- Select all load cases and continue
- Choose to view General Results and Crack Results




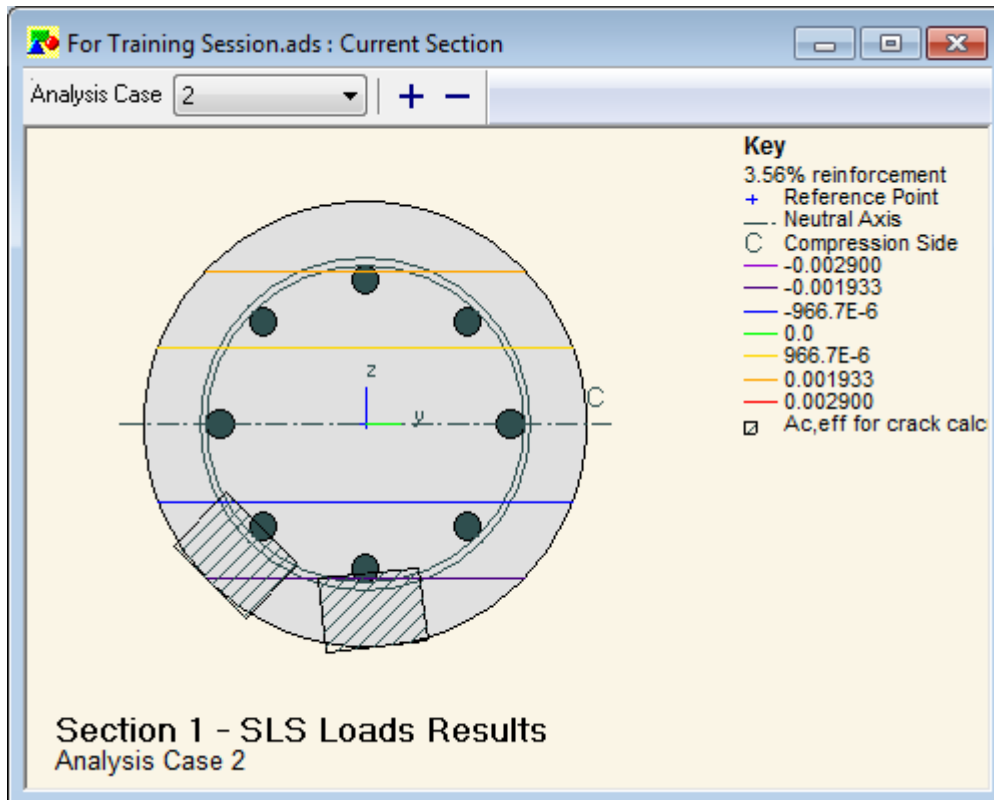
**Note:** Look carefully at the results. Have the results converged successfully? Stop and discuss with the trainer

- View Material and Reinforcement strains and stresses.
- View crack widths.

**Question:** Do the results correspond with the predictions made from the N-M chart?

- Click on **Section View** to view the section. 
- Choose Analysis Case 2 in the Section View window. The user will be able to view where cracking is most likely to occur.

- Click on the Label icon  to show strains on the section for SLS Load Case 2



### 3.2.3 ULS Loads

- Use the ULS Load and ULS Strength icons.
- How do the Stresses/Strains compare between the two results sets and why? Stop to discuss with your trainer.

# Section 4

## Step by step Tutorial 4 Oasys ADC –Section Analysis

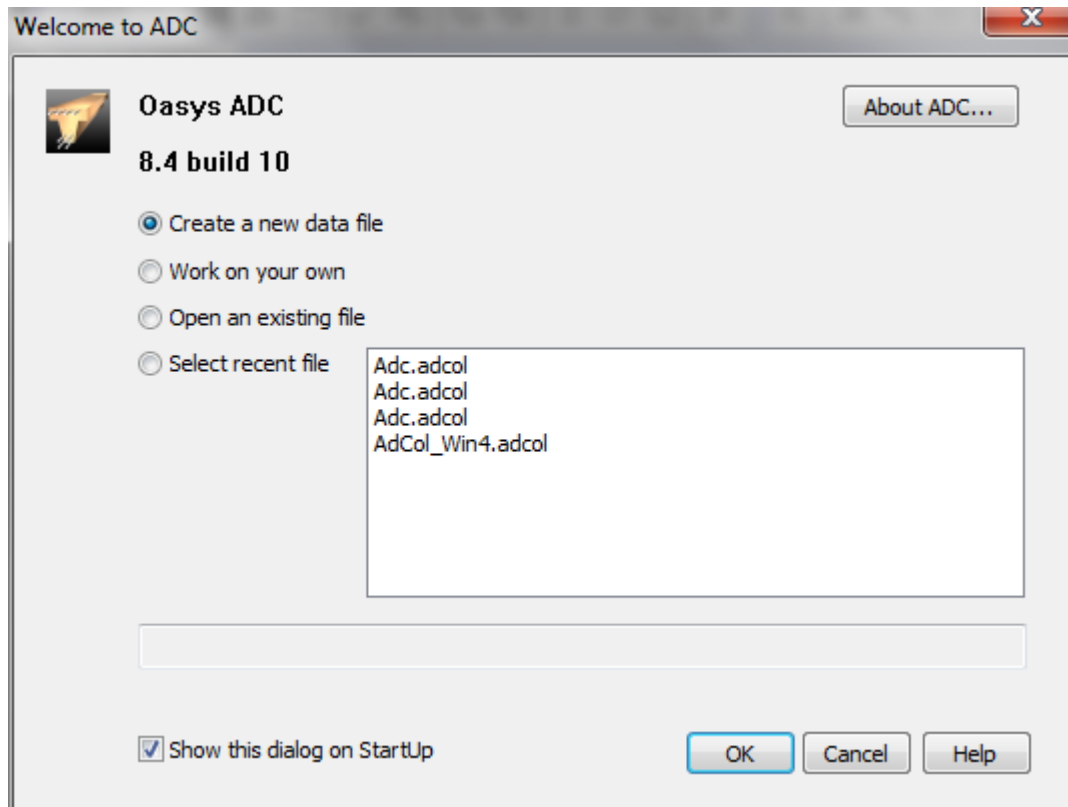
### Objectives

By the end of the session the user should be able to:

- Navigate the ADC Interface
- Use the ADC Wizard and Gateway
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs

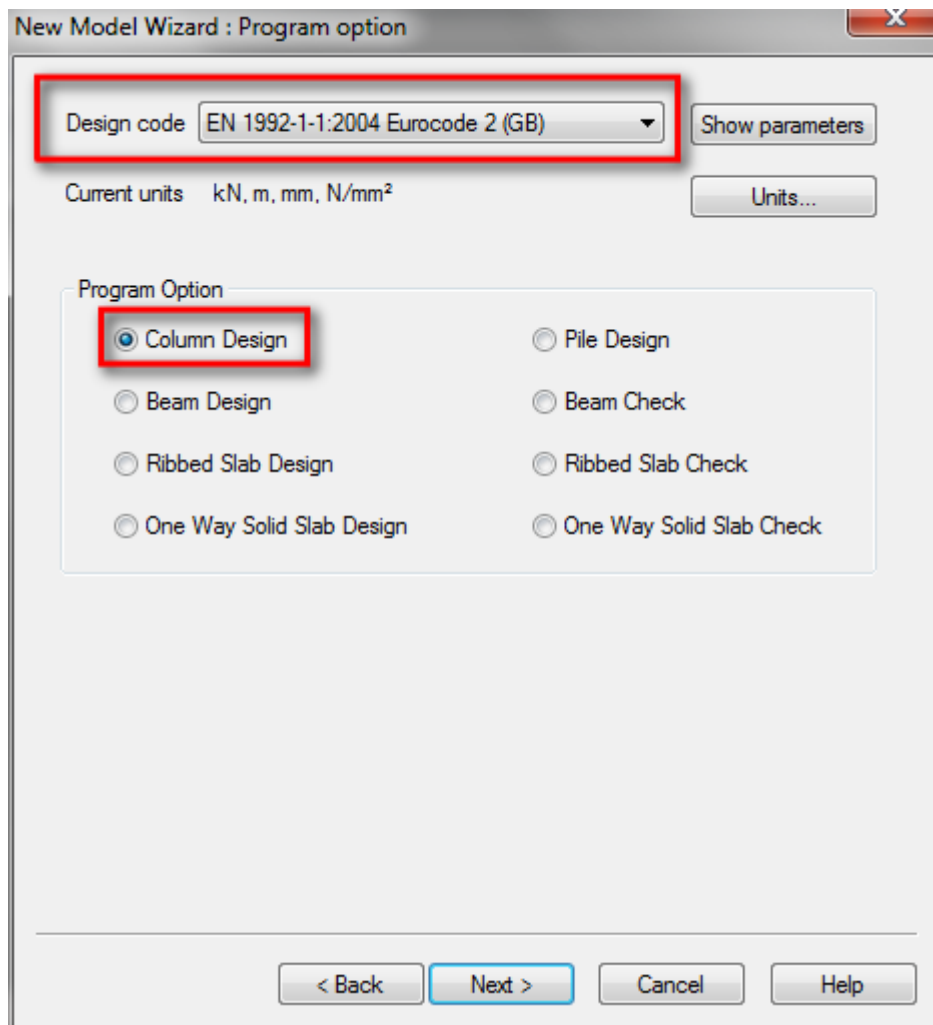
## 4.1 Input

- Once opened, the program start up screen will give a number of options.  
Choose to *Create a New Section*.



- Complete the ***Titles and Units*** section of the Wizard.

- In the **Program Option** section, choose Column Design and the Eurocode 2 Design Code



**Note:** The user can click on the Units button if you wish to change from SI Units

- In the **Definition** section of the Wizard, name the section appropriately and choose an Aggregate Size of 20mm.

New Model Wizard : Section and Materials

Column Section

**Section...** STD C 600.

e.g. STD R 400 300

The 'Section...' button opens the 'Section Wizard' to define the new section shape and dimensions.

Materials

Concrete

Column C20/25

Reinforcement

Main 500B

Link 500B

< Back Next > Cancel Help

Click on the **Section** button to open the Section Wizard.

Choose a circular section with 600mm diameter.

**Note:** Click on the Concrete and Reinforcement options to see what is offered by the program



- In the **Slenderness Details** box, choose the following options for a 9m pile:

New Model Wizard : Slenderness details


Slenderness details


Clear height between restraints (y direction) 9 m


Clear height between restraints (z direction) 9 m


y ☐ ☐ ☐ ☐ ☒


z ☐ ☐ ☐ ☐ ☒

  
 $l_0 = l$

  
 $l_0 = 2l$

  
 $l_0 = 0.7l$

  
 $l_0 = l/2$

  
 $l_0 = l$

☐ Column is compression member in regular frame (to specify relative flexibilities)

< Back Next > Cancel Help

Apply the following Load Case (the moments are obtained from the Alp Output):

Axial Force = 1000kN

Top Moment (yy) = 200kNm

Bottom Moment (yy) = 0kNm

**New Model Wizard : Load details**

**Ultimate Load**

Axial force  kN Sign Conventions

**Moments**

	yy	zz
Top	<input type="text" value="200"/> kNm	<input type="text" value="0"/> kNm
Bottom	<input type="text" value="0"/> kNm	<input type="text" value="0"/> kNm

**Effective Creep Ratios**

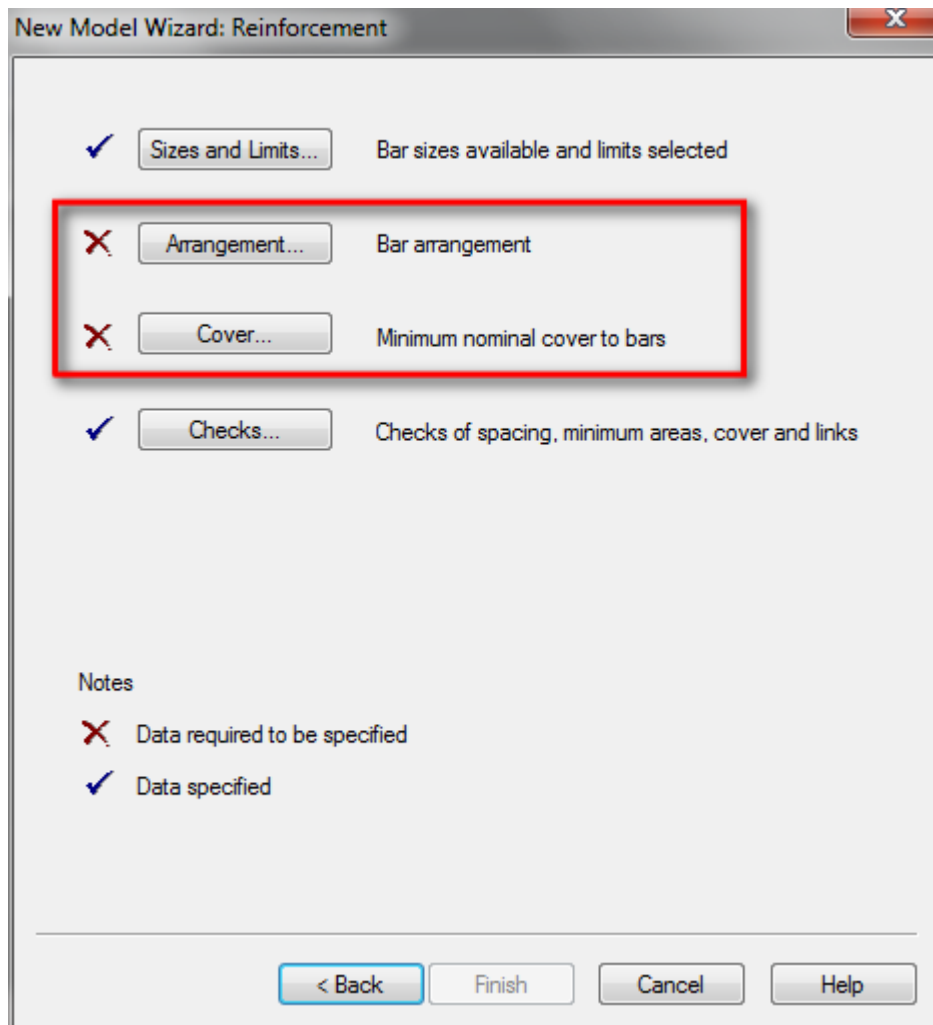
Moment ratio y	<input type="text" value="0.65"/>	Creep coefficient	<input type="text" value="2"/>
Moment ratio z	<input type="text" value="0.65"/>		

**Notes**

1 Additional loads and analysis cases may be input after completion of the Wizard.  
2 Click the 'Sign Conventions' button for help on load sign conventions in AdCol.

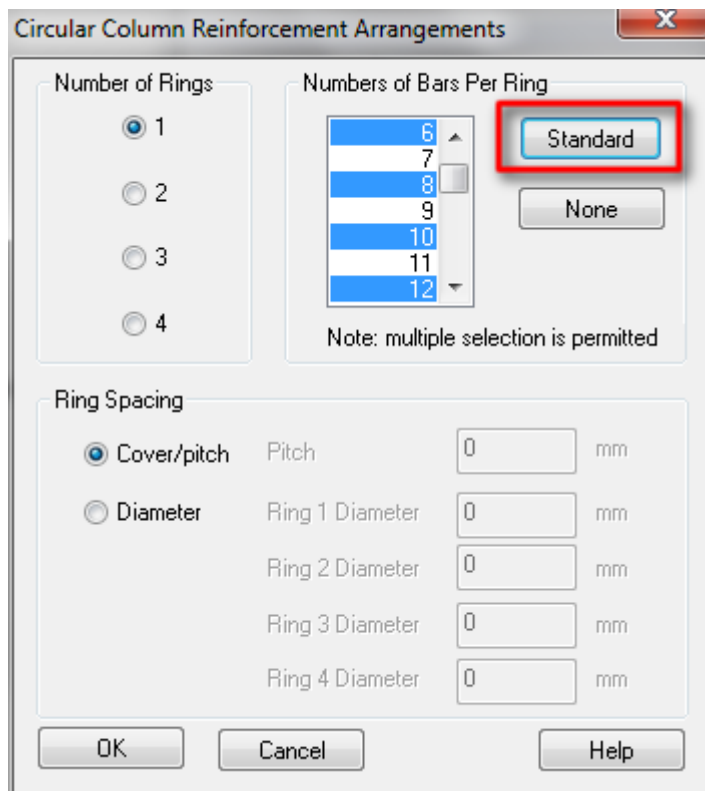
< Back Next > Cancel Help

Finally, in the **Reinforcement** section, the Arrangement and Cover need to be specified:



- Click on the **Arrangement** box.

Click on the **Standard** box to select the number of bars per ring.



The dialog box is titled "Circular Column Reinforcement Arrangements". It contains two main sections: "Number of Rings" and "Numbers of Bars Per Ring".

**Number of Rings:** This section has four radio buttons labeled 1, 2, 3, and 4. The radio button for "1" is selected.

**Numbers of Bars Per Ring:** This section features a list box containing the numbers 6, 7, 8, 9, 10, 11, and 12. The number "6" is highlighted. To the right of the list box are two buttons: "Standard" (which is highlighted with a red rectangle) and "None". Below the list box, a note states: "Note: multiple selection is permitted".

**Ring Spacing:** This section has two radio buttons: "Cover/pitch" (selected) and "Diameter".

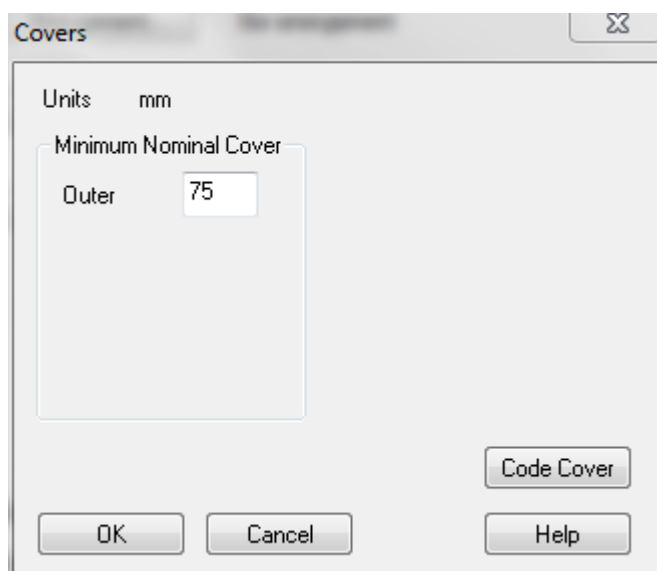
**Under "Cover/pitch":** There is a "Pitch" label followed by a text input field containing "0" and the unit "mm".

**Under "Diameter":** There are four labels: "Ring 1 Diameter", "Ring 2 Diameter", "Ring 3 Diameter", and "Ring 4 Diameter". Each label is followed by a text input field containing "0" and the unit "mm".

At the bottom of the dialog box are three buttons: "OK", "Cancel", and "Help".

- Click on the **Cover** box.

Input the cover of 75mm.



The dialog box is titled "Covers". It contains the following elements:

**Units:** A label "Units" followed by the text "mm".

**Minimum Nominal Cover:** A section containing a label "Outer" followed by a text input field containing the value "75".

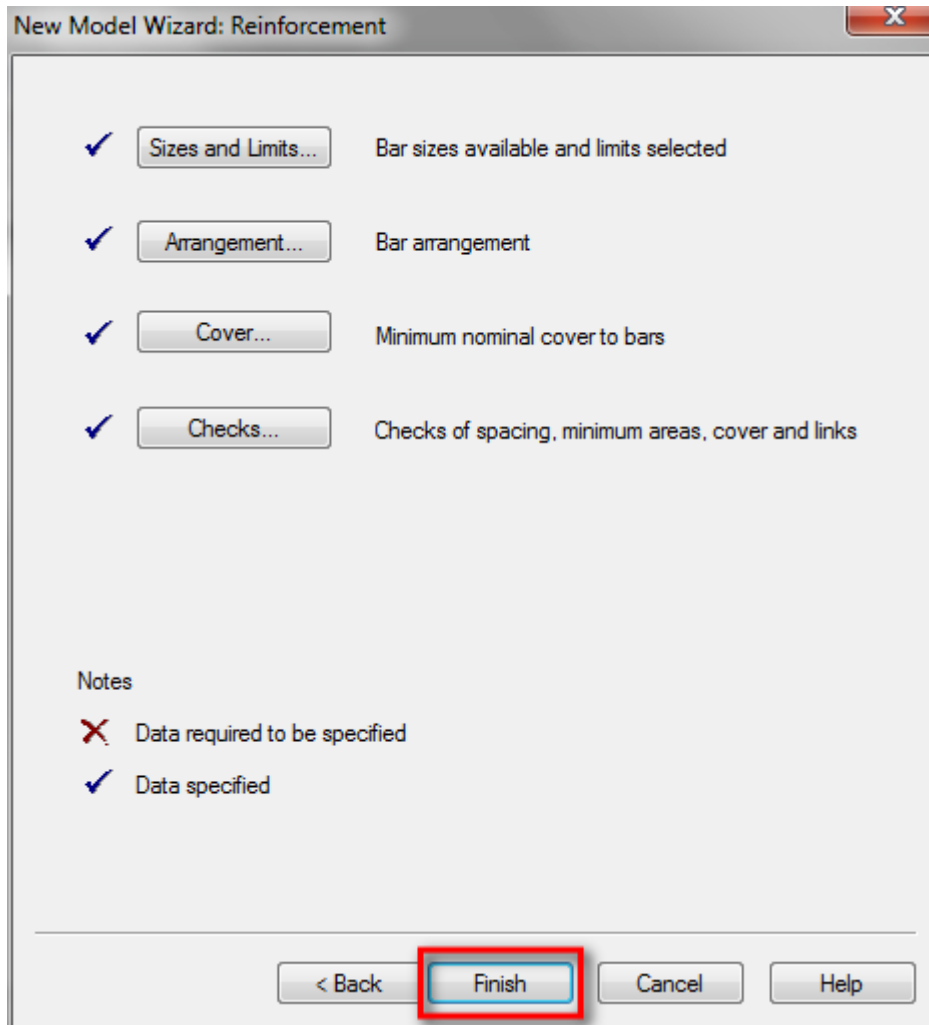
At the bottom right of the dialog box is a button labeled "Code Cover".

At the bottom of the dialog box are three buttons: "OK", "Cancel", and "Help".

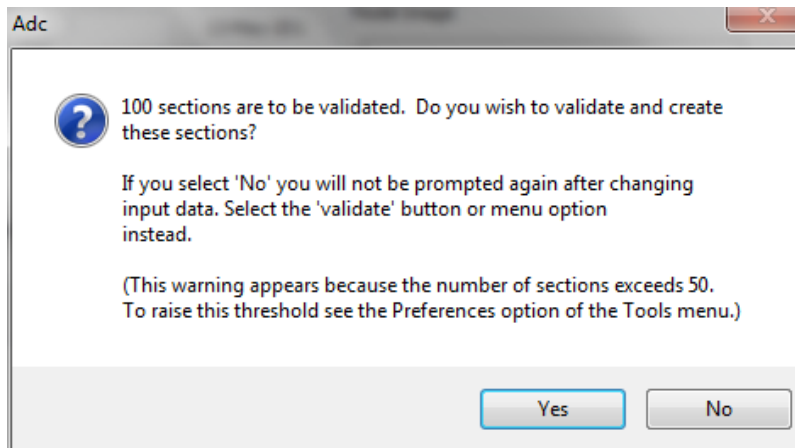
**Note:** Click the Code Cover box to see what the requirement is of the code you have specified

## 4.2 Analysis and Results

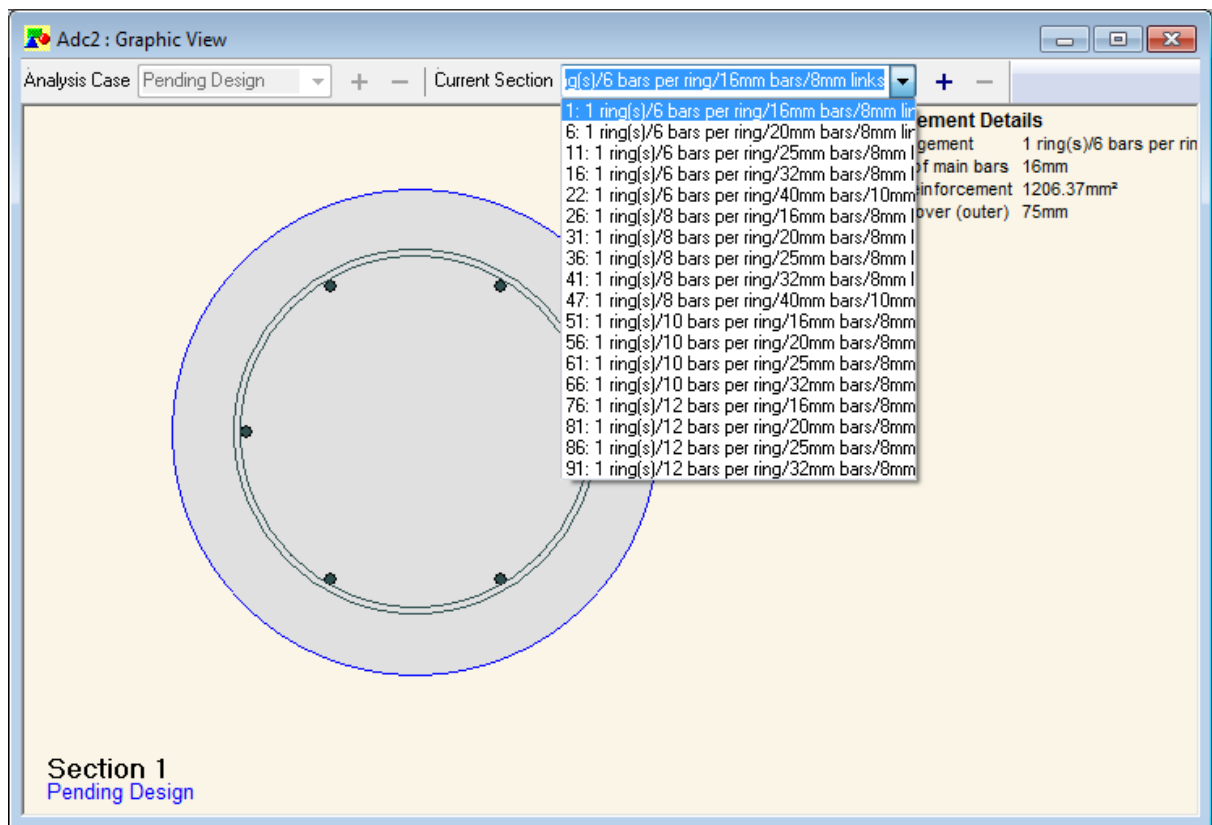
- Now that all the Data is specified in the Reinforcement section, click on the **Finish** button




- The following box will appear, checking if you are happy with the number of sections. Click **OK** to proceed with the analysis.



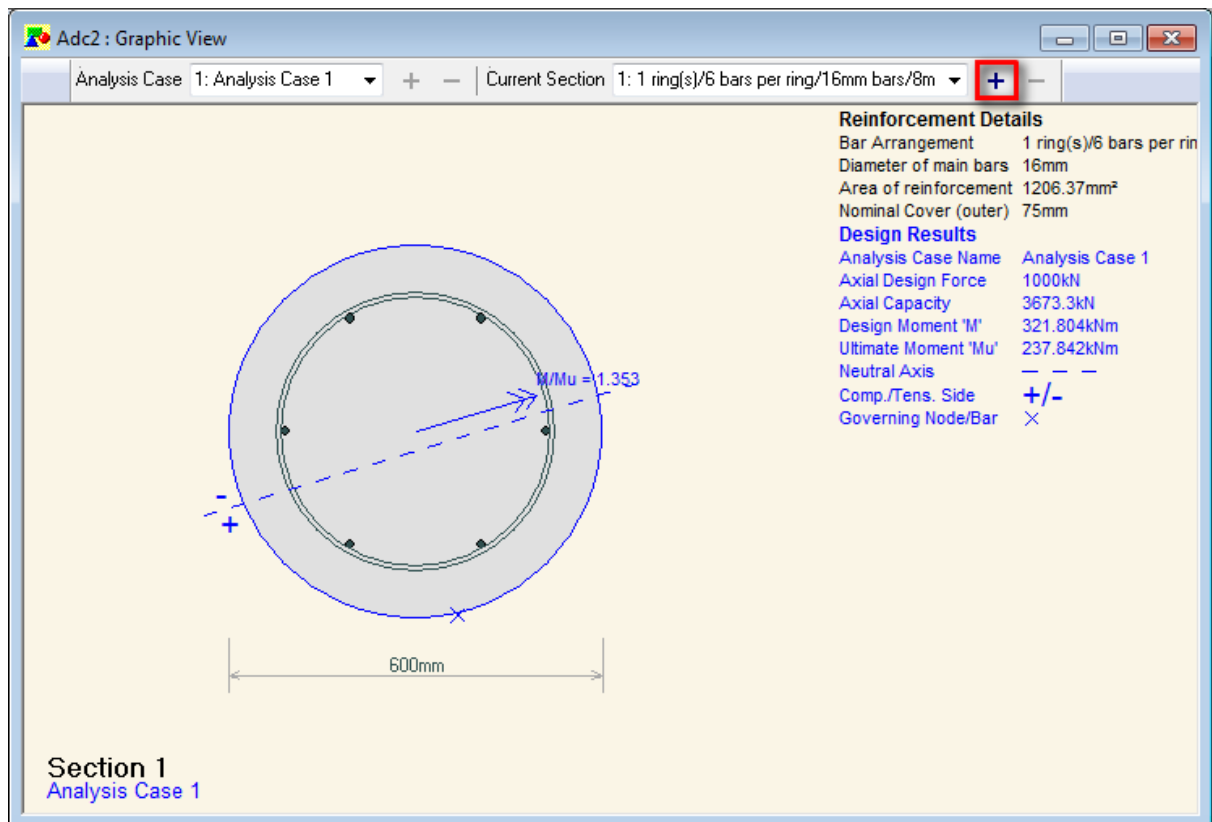
- The **Graphic View** will appear, showing the acceptable reinforcement:



- Click on the *Design* icon  to run the analysis for the load case specified


- The following Graphical View will appear.

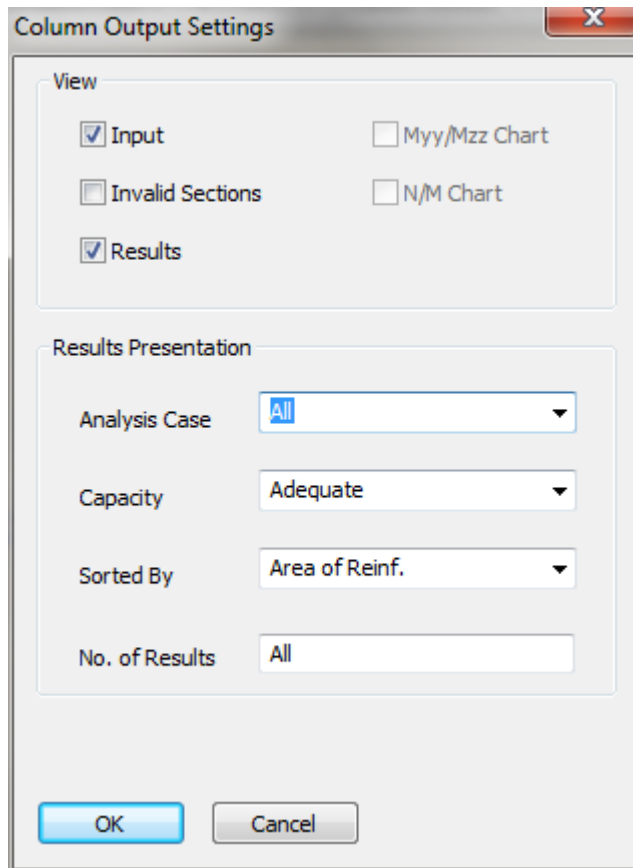
Toggle through the different cases by pressing the + button to see the results.







### 4.2.1 Tabular Outputs

- Click on the **Output View** icon 
- Select the following options in the **Column Output Settings** window to view the sections with adequate capacity for the load case specified in order of area of reinforcement:



- View the Tabular Output results  
Scroll to the Column Summary Results to view the section with the lowest reinforcement area with adequate capacity

## 4.2.2 Plotting Charts

- Click on the *N/M Chart* Icon The icon shows a red circle with a diagonal line through it, and the letters 'N' and 'M' are positioned on either side of the line.
- Choose the appropriate section to plot and view the results, noting the load case within the envelope
- Click on the *Myy/Mzz Chart* Icon The icon shows a green circle with a diagonal line through it, and the letters 'M' and 'M' are positioned on either side of the line.
- Choose the appropriate section to plot and view the results, noting the load case within the envelope

# Appendix

## EC7 – UK SAMPLE CALCULATION IN PILE

The following example has been based on the design example given in “Design of pile foundations following Eurocode 7” by R. Frank. The final analysis file is also available in the Sample Files provided with the program.

### **Pile Geometry:**

Solid circular pile with diameter 600 mm. The pile is “driven”.

### **Soil Data:**

Sand layer with following properties:

Density = 21 kN/m<sup>3</sup>

Angle of internal friction = 35 degrees

SPT N = 25

Based on SPT, following values are assumed:

End bearing stress,  $q_b = 1370$  kPa

Skin friction stress,  $q_s = 70$  kPa

These values are assumed to be uniform throughout the layer.

### **Groundwater Data:**

GWT is located at a depth of 2 m below the ground level.

### **Load Data:**

Characteristic permanent load  $G_k = 1200$  kN

Characteristic variable load  $Q_k = 200$  kN

**Design Code:**

EC7 (United Kingdom)

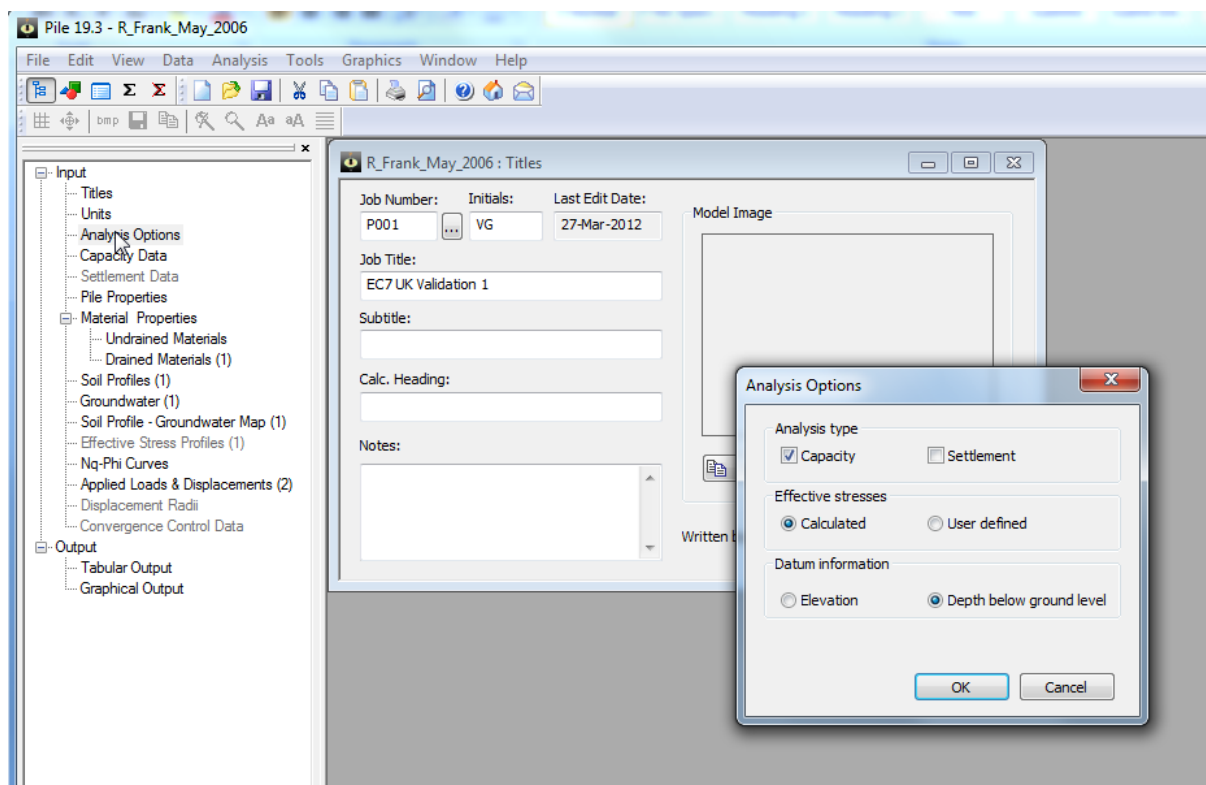
Following data need to be entered in the Pile data file:

- 1 Analysis Options
- 2 Capacity Data (for selecting design code)
- 3 Pile Properties
- 4 Materials
- 5 Soil Profiles
- 6 Groundwater
- 7 Soil -profile Groundwater Map
- 8 Applied Loads & Displacements

Each of the preceding items are explained in detail in the following sections:

**Analysis Options:**

This and other data modules can be invoked by clicking the relevant item in the Gateway:



**Analysis type:**

In this example, only capacity calculations are carried out. Hence, only “Capacity” check box is checked.

**Effective Stresses:**

The program will be calculating the effective stresses, as opposed to user specifying the same. Hence, “Calculated” radio button is selected.

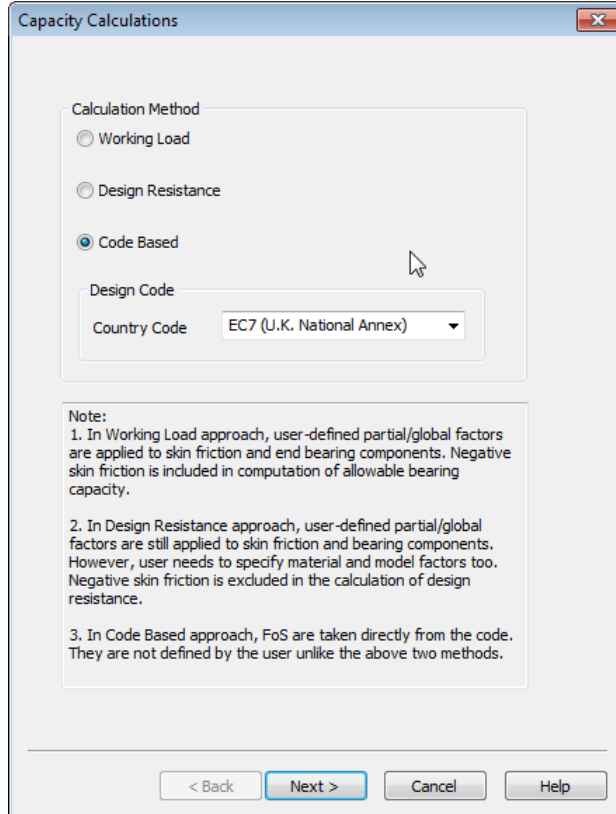
**Datum information:**

The data corresponding to soil layers, groundwater etc. is based on depth. Hence, “depth below ground level” is selected.

## Capacity Data:

Selecting this item in the Gateway brings up a wizard. To enable EC7 calculations, “Code based” option should be selected in the first page.

Then, “EC7 (U.K. National Annex)” should be selected in the “Country Code” drop down box.



The screenshot shows a window titled "Capacity Calculations" with a close button in the top right corner. Inside the window, there is a section labeled "Calculation Method" with three radio button options: "Working Load", "Design Resistance", and "Code Based". The "Code Based" option is selected. Below these options, there is a "Design Code" label and a "Country Code" dropdown menu. The dropdown menu is open, showing "EC7 (U.K. National Annex)" as the selected option. At the bottom of the window, there are four buttons: "< Back", "Next >", "Cancel", and "Help".

Capacity Calculations

Calculation Method

☐ Working Load

☐ Design Resistance

☒ Code Based

Design Code

Country Code EC7 (U.K. National Annex)

Note:

1. In Working Load approach, user-defined partial/global factors are applied to skin friction and end bearing components. Negative skin friction is included in computation of allowable bearing capacity.

2. In Design Resistance approach, user-defined partial/global factors are still applied to skin friction and bearing components. However, user needs to specify material and model factors too. Negative skin friction is excluded in the calculation of design resistance.

3. In Code Based approach, FoS are taken directly from the code. They are not defined by the user unlike the above two methods.

< Back Next > Cancel Help

Then, the following page is shown on clicking “Next”

The screenshot shows a software window titled "Eurocode 7 (U.K.)". It contains several sections for configuring design parameters:

- Design Approach:** Three radio buttons are present: "DA1(C1 + C2)" (selected), "DA2", and "DA3".
- Pile Type:** Three radio buttons are present: "Driven" (selected), "Bored", and "CFA".
- Model factor:** A text input field containing the value "1.4".
- Partial Factors On Negative Skin Friction:** Two text input fields: "Set A1 partial factor" and "Set A2 partial factor", both containing the value "1".
- Checkboxes:**
  - ☐ Serviceability is verified by load tests (preliminary/working) carried out on more than 1% of constructed piles to loads not less than 1.5 times the representative load for which they are designed.
  - ☐ Resistance is verified by a maintained load test taken to the calculated, unfactored, ultimate resistance.
  - ☐ Print detailed output of capacities from all combinations.
- Buttons:** At the bottom, there are four buttons: "< Back", "Finish" (highlighted in blue), "Cancel", and "Help".

It can be seen that only DA1 design approach is allowed, as is the case with U.K. national annex.

The pile type should be specified as “driven” as per the data given for this example.

The “Model factor” is read only. This is set to 1.4 by default.

The pile type should be specified as “driven” as per the data given for this example. The “Model factor” is read only. This is set to 1.4 by default.

However, if the user specifies that the resistance is verified by maintained load test taken to calculated, unfactored ultimate resistance by clicking the relevant check box, then the model factor is set to 1.2.

The user also needs to verify whether explicit verification of SLS is carried out, by checking the relevant check box. This selection influences the R4 resistance factors as per Table A.NA.6 in U.K. national annex.

The A1 and A2 partial factors on negative skin friction would be used for calculating the factored load due to negative skin friction, which is treated as an unfavourable action. The users may refer to A.3.1. section in UK national annex for guidance on these factors. These values can be ignored if there is no negative skin friction in the model.

Clicking on the last check box in this page allows the user to see the results from both the combinations of DA1.

Clicking on “Finish” closes the wizard and saves the code specific data.

**Pile Properties:**

The pile geometry needs to be specified in the wizard invoked by selecting the relevant item in the Gateway:

Pile Properties

Pile cross-section: Solid Circular

Young's modulus: 2e+007 kPa

Reduction factor for internal skin friction: 0.9

Under-reams (Solid only)

☐ With under-ream ☒ Without under-ream

Note: Settlements are calculated for solid circular without under-ream sections

< Back Next > Cancel Help

For the given model, “Solid circular” and “Without under-ream” options need to be selected.

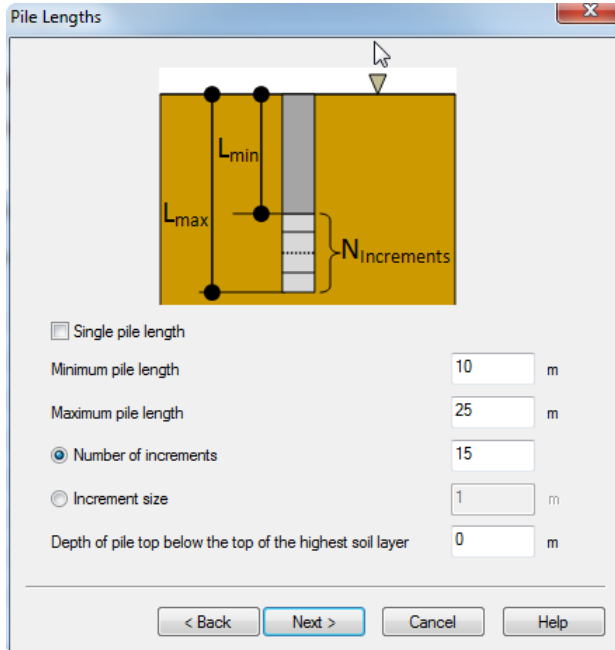
The “Young’s modulus” is not relevant to capacity calculations. Hence, leave the default value as it is.

Once this data is entered, the user can specify the range of pile lengths for which capacity calculations need to be carried out.

For this model, the range of pile lengths from 10m to 25m is suggested with an increment of 1m.

Also, since basement piles are not being modelled, “Depth of pile top below top of highest soil layer” needs to be set to zero as shown.



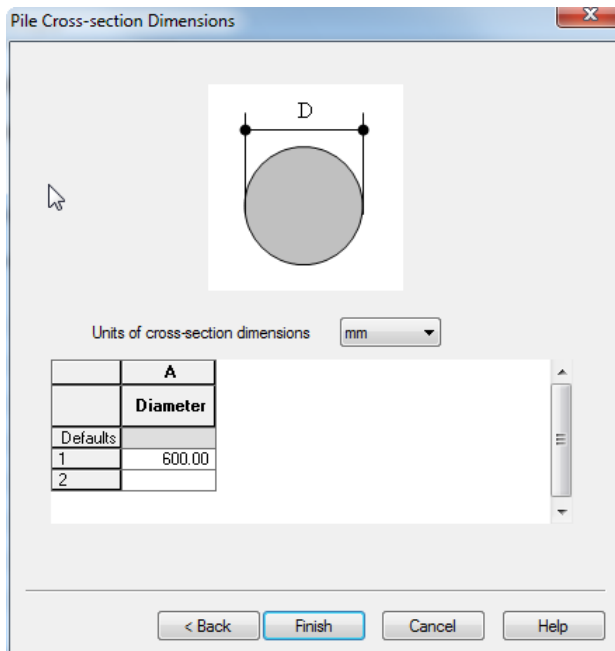


The **Pile Lengths** dialog box contains a schematic diagram of a pile in soil. The pile is shown as a vertical grey bar with a downward arrow at the top. The soil is represented by a yellow area. The pile length is divided into segments:  $L_{min}$  at the top,  $L_{max}$  at the bottom, and a central section divided into  $N_{increments}$  equal parts. Below the diagram, there are input fields for:
 

- ☐ Single pile length
- Minimum pile length: 10 m
- Maximum pile length: 25 m
- ☒ Number of increments: 15
- ☐ Increment size: 1 m
- Depth of pile top below the top of the highest soil layer: 0 m

 At the bottom are buttons for < Back, Next >, Cancel, and Help.

Next, the dimensions of a range of piles needs to be specified.



The **Pile Cross-section Dimensions** dialog box shows a schematic of a circular pile cross-section with diameter  $D$ . Below the diagram, the units are set to mm. A table lists the dimensions for two different pile types (1 and 2), with a default diameter of 600.00 mm for type 1.

	A
	Diameter
Defaults	
1	600.00
2	

At the bottom are buttons for < Back, Finish, Cancel, and Help.

In this model, we are interested in the capacity of only 600 mm diameter pile. Hence, only one entry is given.

Clicking on “Finish” closes the wizard and saves the “Pile Properties” data.

## Material Properties – Drained materials:

In this model, there is only one drained material - Sand. The following data for the same needs to be entered in the “Undrained Materials” table by invoking the same from the Gateway.

As the material tables are wide, it is suggested to enter the data directly in the “General”, “Friction” and “End Bearing” worksheets, instead of using the “All” worksheet.

R\_Frank\_May\_2006 : Drained Materials

	A	B	C
	Material description	Bulk unit weight [kN/m <sup>3</sup> ]	Material factor for soil strength (tan Delta)
Defaults:	Drained #	20.00	0.00
1	Sand	21.00	
2			

Enter material name.

R\_Frank\_May\_2006 : Drained Materials

	A	B	C	D	E	F	G	H	I	J	K
	Material description	Skin friction computation	Beta	Delta [Deg]	Coeff. of earth pressure K	Skin friction data		Limiting value		Qs material factors	
						Top	Base	Specified	Value	M1	M2
						[kPa]	[kPa]		[kPa]		
Defaults:	Drained #	Beta						No		1.00	1.00
1	Sand	qs specified				70.00	70.00			1.00	1.00
2											

Enter material name.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	
	Material description	End bearing computation	Nq	Phi'	PhiD	Phicv'	Ir	Nq material factors		End bearing data		Limiting value		Nq-Phi curve		Qb material factors	
				[Deg]	[Deg]	[Deg]		M1	M2	Top	Base	Specified	Value			M1	M2
										[kPa]	[kPa]						
Defaults	Drained #	Nq specified						1.00	1.00			No		Berezantzev Ak Bk Curves		1.00	1.00
1	Sand	qb specified								1370.00	1370.00					1.00	1.00
2																	
3																	

In the "Friction" and "Bearing" worksheets, the user needs to select the "qs -specified" and "qb-specified" options, and enter the relevant values respectively as shown.

The fields M1 and M2 correspond to the material factor sets. In EC7 (U.K.), all the M1 factors are always 1.0. Hence the fields are not editable.

The field M2 may need to be explicitly entered by the user only in some circumstances. For example, DA1 C2 uses M2 factors to calculate the unfavourable Geo actions – which translates to negative skin friction for this program. However, the code only specifies M2 factors for

- Angle of shearing resistance,
- Effective cohesion,
- Undrained shear strength, and
- Unconfined strength.

The user need not explicitly enter the material factors if the skin friction and end-bearing values are solely computed on the basis of the above 4 parameters. However, if the user directly specifies qs, Nc, Nq or qb, then the program does not know the theory/formula which has been used in computing the same. In these cases, the program uses the material factors specified by the user in the tables above, to derive the "material factored" unfavourable geotechnical actions.

For this example, however, as there is no negative skin friction involved, these factors can be left at the default value of 1.0.

**Soil Profiles :**

The user then has to define the soil profiles. Each soil profile is a collection of material layers.

The user can specify multiple soil profiles, particularly if he is using the “Model Pile” procedure”.

However, for EC7 (U.K.) national annex, the program only uses the “Alternative procedure”. Hence, the user would be entering only one soil profile “S1” as shown:

	A	B	C
Layer	Depth below ground level	Material	Contribute to negative skin friction
	[m]		
Defaults	0.00	Sand	No
1	0.00	Sand	No
2			

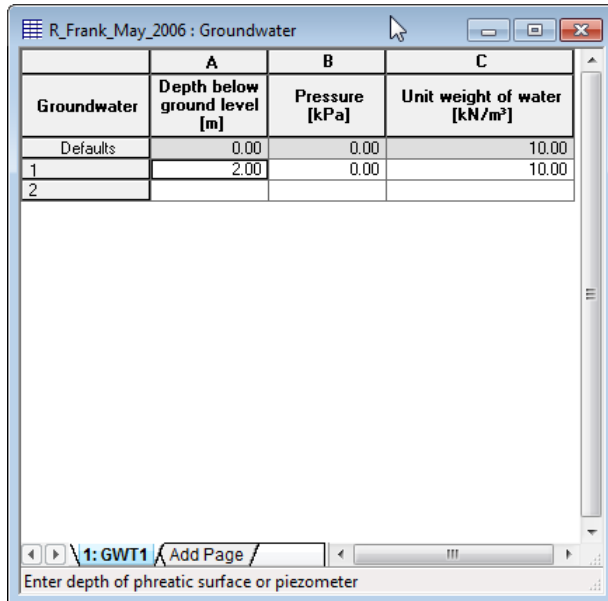
1: S1 Add Page /

Enter depth below ground level

The user can enter multiple soil profiles for the “Alternative approach” also. In this case, the program calculates design resistance of pile in each of these soil profiles independently.

**Groundwater:**

The user can specify the groundwater tables in the model in these worksheets.



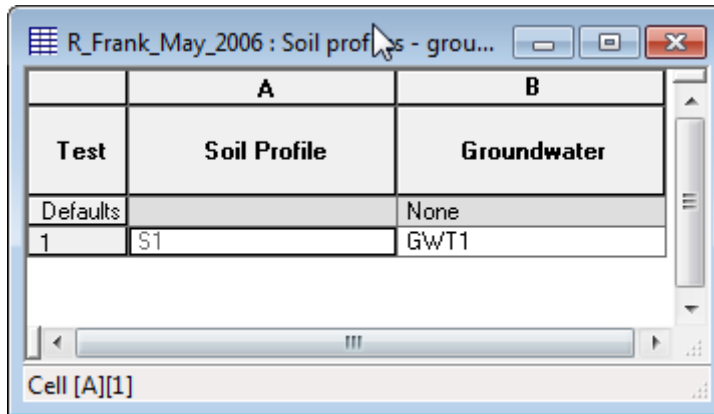
	A	B	C
Groundwater	Depth below ground level [m]	Pressure [kPa]	Unit weight of water [kN/m³]
Defaults	0.00	0.00	10.00
1	2.00	0.00	10.00
2			

In this particular example, there is only one soil profile and one ground water table. The same is entered as shown.

If the user wants to model perched groundwater in a soil profile, he can use multiple data points to model the pore-pressure in the soil profile. In this case, the pore pressure distribution will not be hydrostatic.

**Soil –profile Groundwater Map:**

This table is really used to associate a particular groundwater table to a particular soil profile.



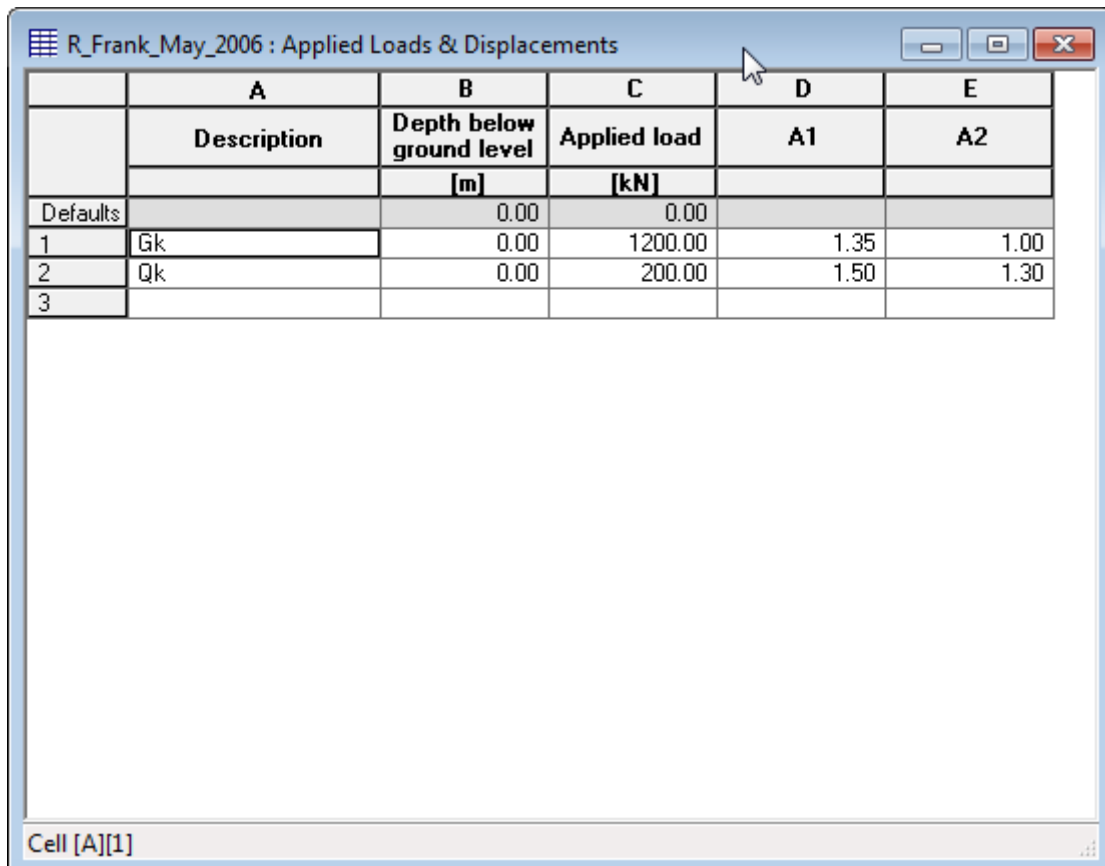
	A	B
Test	Soil Profile	Groundwater
Defaults		None
1	S1	GW1

**IMPORTANT:** By default, no groundwater table is associated to the soil profiles created by the user. Hence, it is important to explicitly enter the data in this table when groundwater is present in a soil profile.

**Applied Loads & Displacements:**

For the capacity part, the “Displacements” are not relevant.

The following load data needs to be entered as shown:



	A	B	C	D	E
	Description	Depth below ground level	Applied load	A1	A2
		[m]	[kN]		
Defaults		0.00	0.00		
1	Gk	0.00	1200.00	1.35	1.00
2	Qk	0.00	200.00	1.50	1.30
3					

Fields A1 and A2 refer to the action factor sets in EC7 (U.K.) N.A.

For this model example, it is assumed that for

Action factor set A1:

Dead load factor = 1.35

Live load factor = 1.50

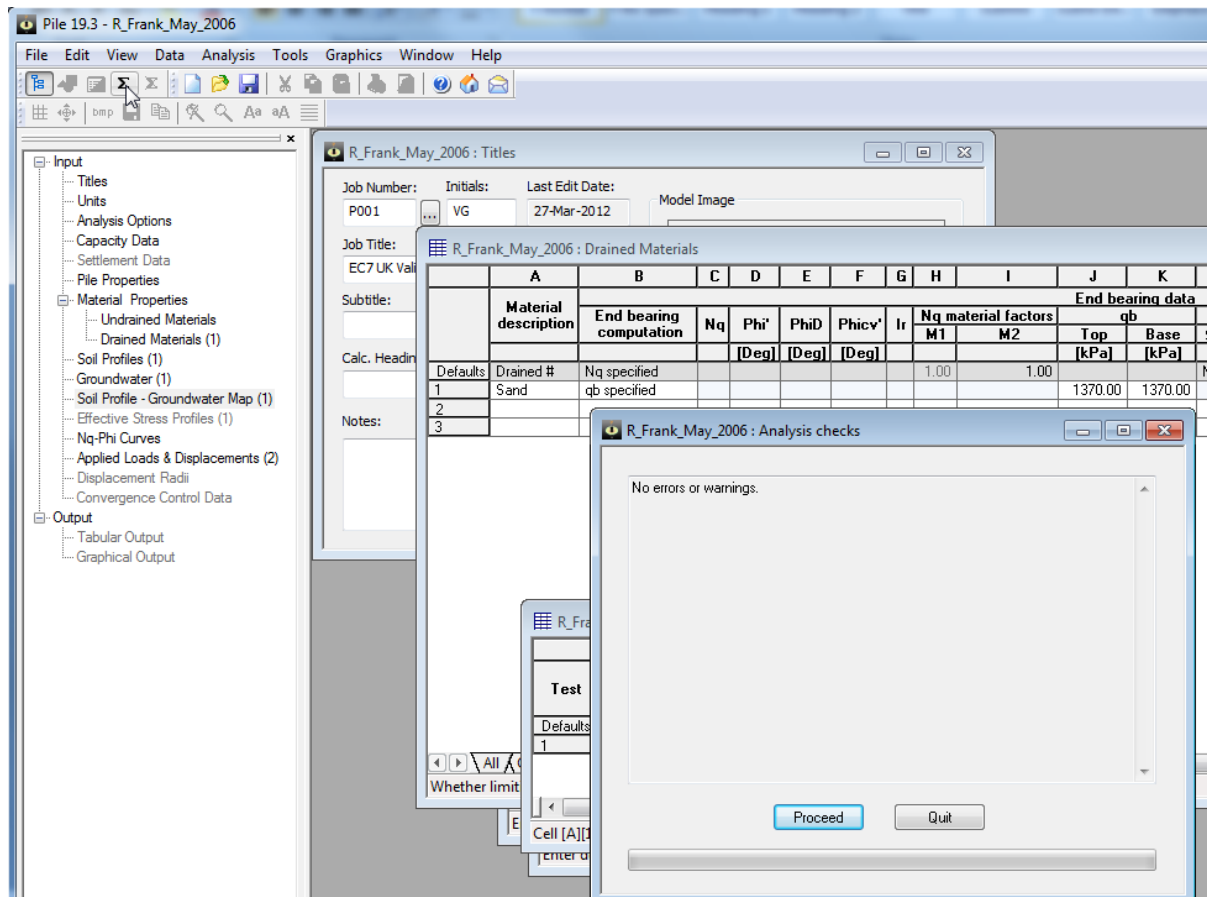
Action factor set A2:

Dead load factor = 1.00

Live load factor = 1.30

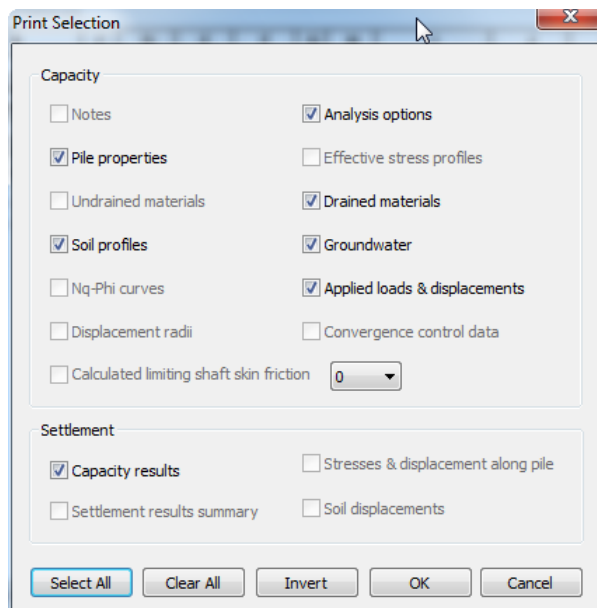
## Analysis & Results:

Once the data is input as shown above, the user can proceed for analysis by clicking the “Analyse” button as shown:



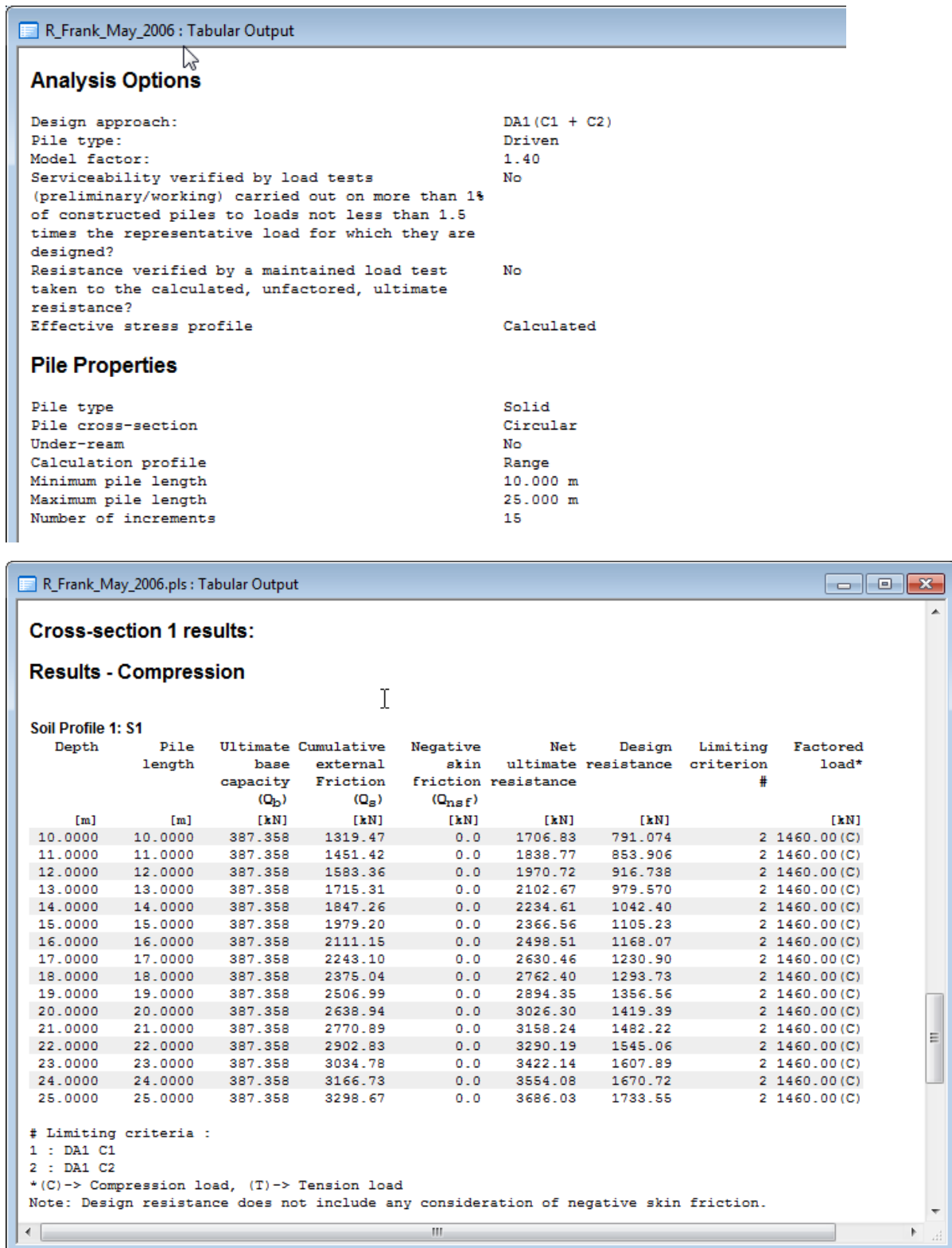
This brings up the Pre-Analysis check dialog giving any warnings or errors in the model. If there are no errors, the user can run the analysis by clicking the “Proceed” button.

After completion of analysis, the following print selection dialog appears:





The user can select the data of interest to be shown/printed in the tabular output, which comes up after clicking the “OK” button above:



The program prints the factored load and design resistance information at the pile lengths specified by the user.

**Validation:**

A short validation is given below for the pile length of 25 m:

Unit end bearing stress = 1370 kPa

Area of the pile base =  $(\pi/4) * (0.6) * (0.6) = 0.283 \text{ m}^2$

Ultimate end bearing capacity =  $1370 * 0.283 = 387.36 \text{ kN}$

Unit skin friction = 70 kPa

Length of pile = 25 m

Ultimate skin friction capacity =  $\pi * 0.6 * 70 * 25 = 3298.67 \text{ kN}$

DA1 C1

"A1 + M1 + R1"

Action set A1:

Factored load =  $1.35 * 1200 + 1.5 * 200 = 1920 \text{ kPa}$

Partial factor combination:

Design resistance =  $((\text{Ultimate shaft resistance}/\text{shaft factor}) + (\text{Ultimate base resistance}/\text{base factor}))/\text{model factor}$   
 $= (3298.67/1.0 + 387.6/1.0)/1.4 = 2632.88 \text{ kPa}$

DA1 C2

"A2 + M1 + R4"

Action set A1:

Factored load =  $1.00 * 1200 + 1.3 * 200 = \mathbf{1460 \text{ kPa}}$

Note: R4 factor set is based on "No explicit SLS verification"

Partial factor combination:

Design resistance (Partial Factors) =  $((\text{Ultimate shaft resistance}/\text{shaft factor}) + (\text{Ultimate base resistance}/\text{base factor}))/\text{model factor}$   
 $= (3298.67/1.5 + 387.6/1.7)/1.4 = \mathbf{1733.65 \text{ kPa}}$

Design capacity is the minimum from all the 4 cases above.

Design capacity is the minimum from all the 2 cases above.

**Hence, the design capacity is 1733.65 kN, and the governing combination is DA1 C2. The corresponding factored load is 1460 kN.**