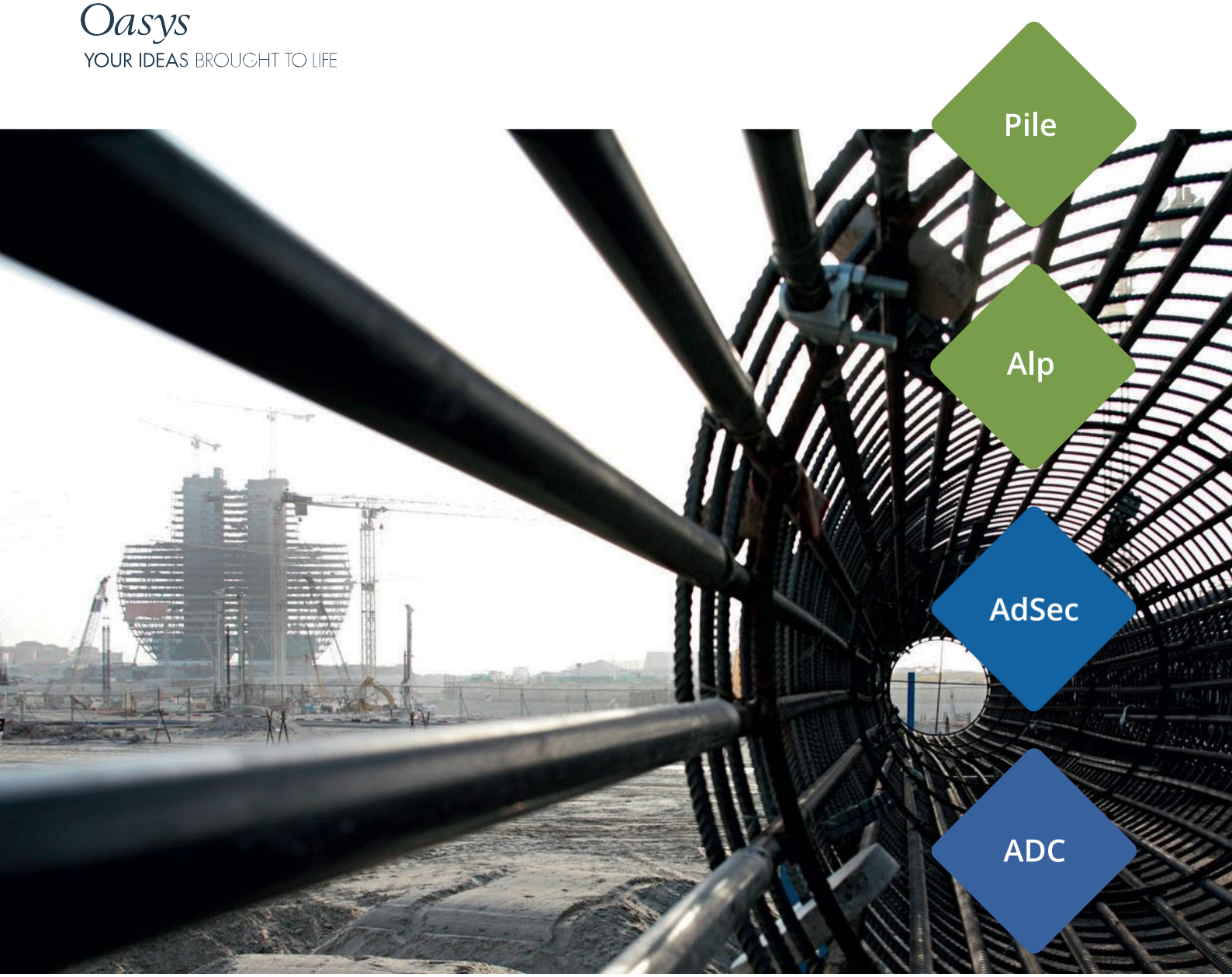
Oasys
YOUR IDEAS BROUGHT TO LIFE



Piling Analysis Tutorial Manual

OASYS

Contents

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 for dates and program content.

Section 1

Step-by-Step Tutorial 1
Oasys Pile – Pile Capacity and Settlement Analysis

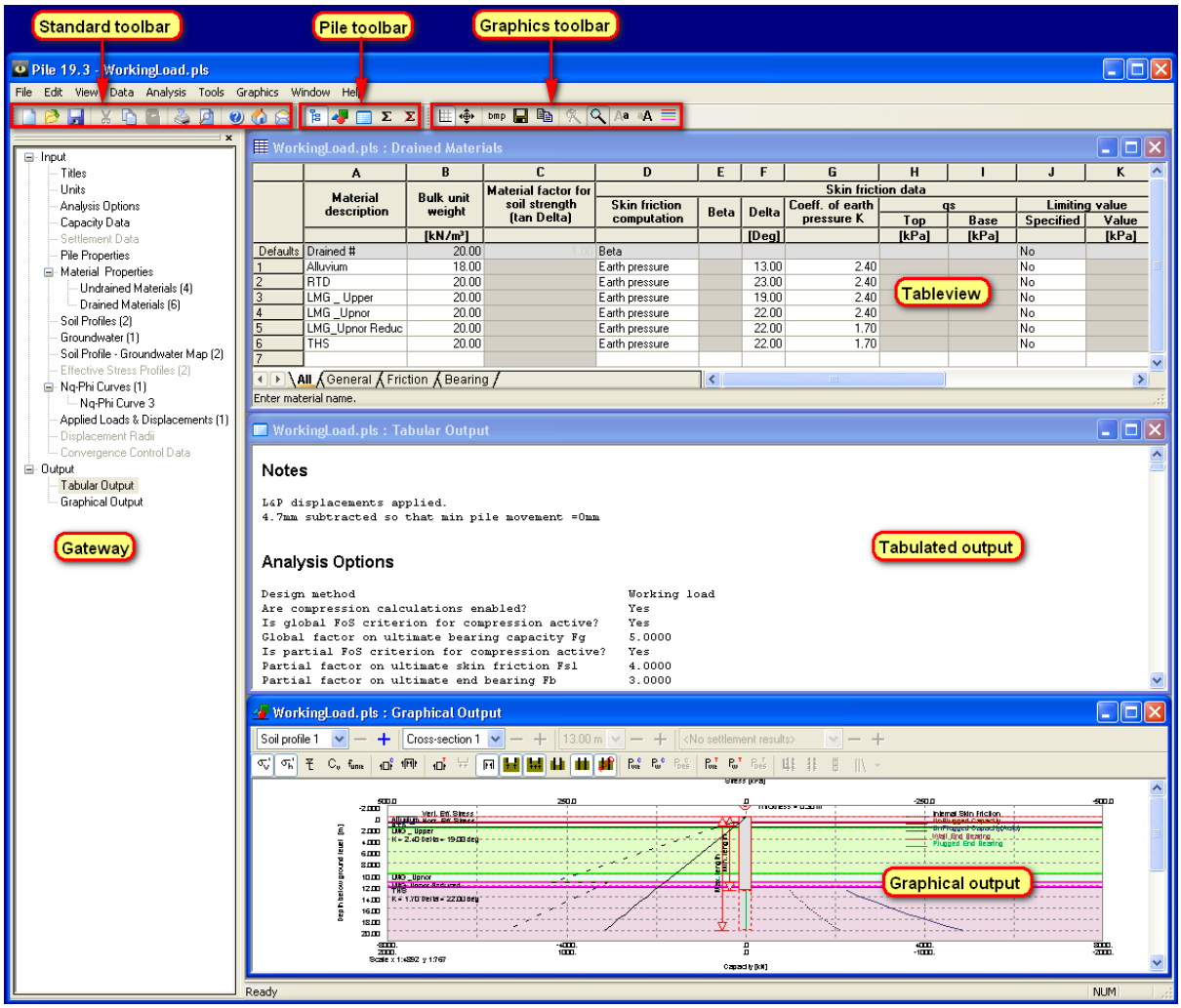
Objectives

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.

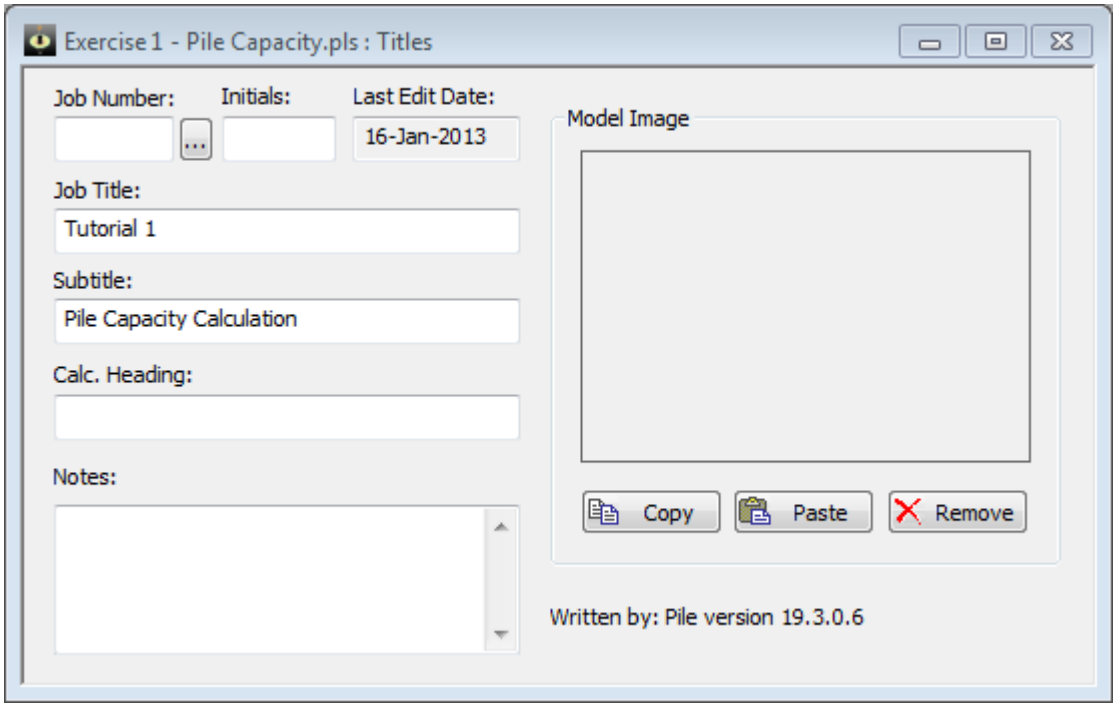


Hint: When using the Gateway, click '+' to expand an item. Double click on the item to open the corresponding window

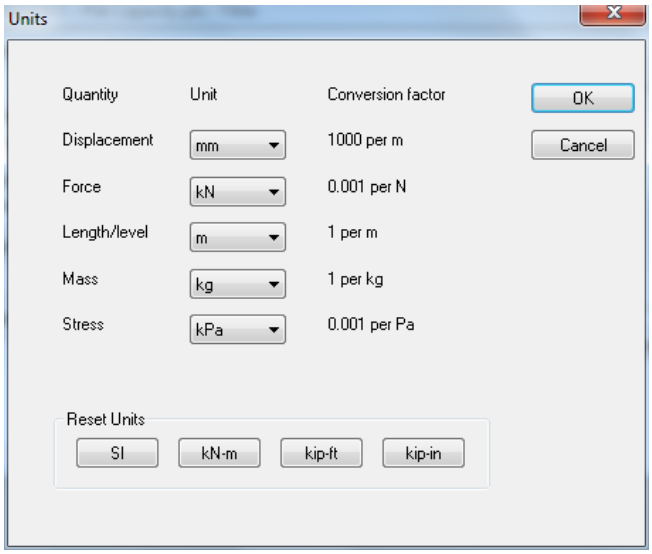
1.2 Pile Capacity Analysis

1.2.1 Creating the input

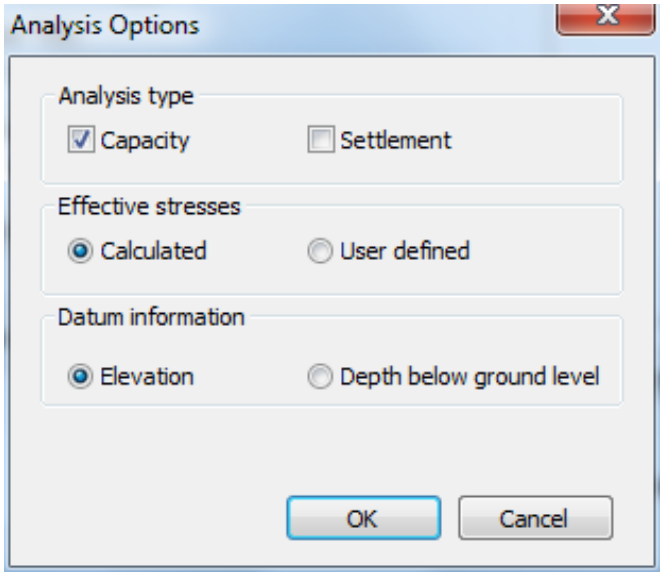
- Once opened, create a new file by clicking the '***New File'*** icon  on the top left of the program or clicking ***Ctrl + N***
- Fill the ***Titles and Units*** dialog box:



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

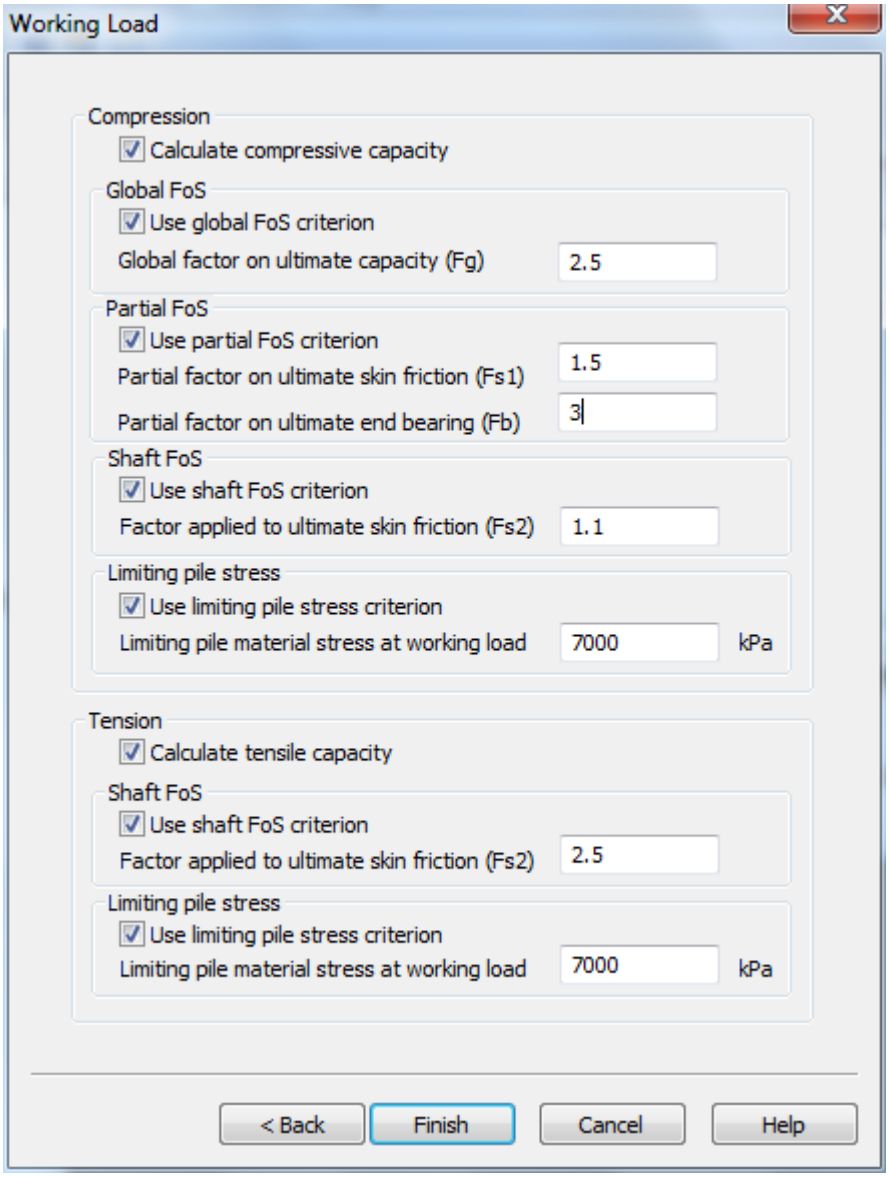


- Double click on *Analysis* ***Options*** in the Gateway.
- Select ***Capacity*** and the following options:



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

- Choose the following options:



- 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^3)	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^2)	Unlimited
Nq - specified or computed?	Specified

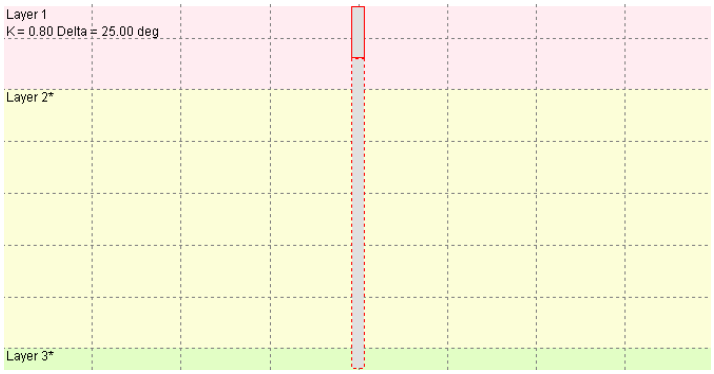
- Specify Nq for Material #1 as 50.

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

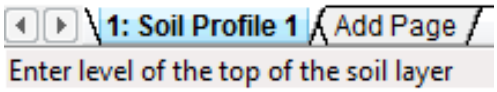
- See the help file for instructions on how to set the limiting value of end bearing to 'unlimited'.

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

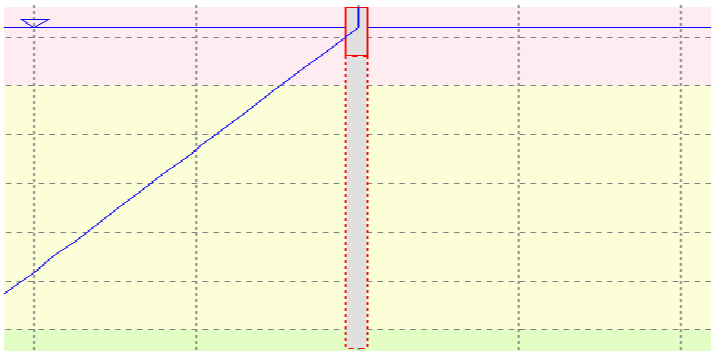
- Double click on *Soil Profiles* in the Gateway
- Define layers constituting the soil profile around the pile:
    - top of Layer 1 at 8mOD
    - top of Layer 2 at 0mOD
    - top of Layer 3 at -25mOD



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

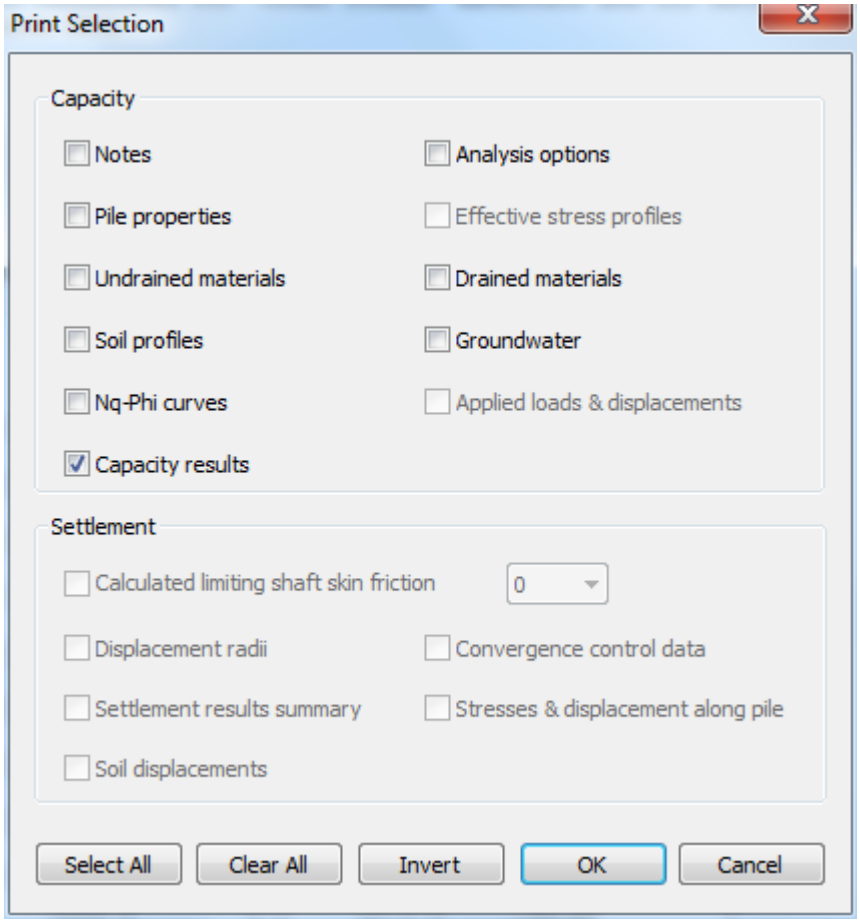
1.2.2 Analysis and Outputs

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

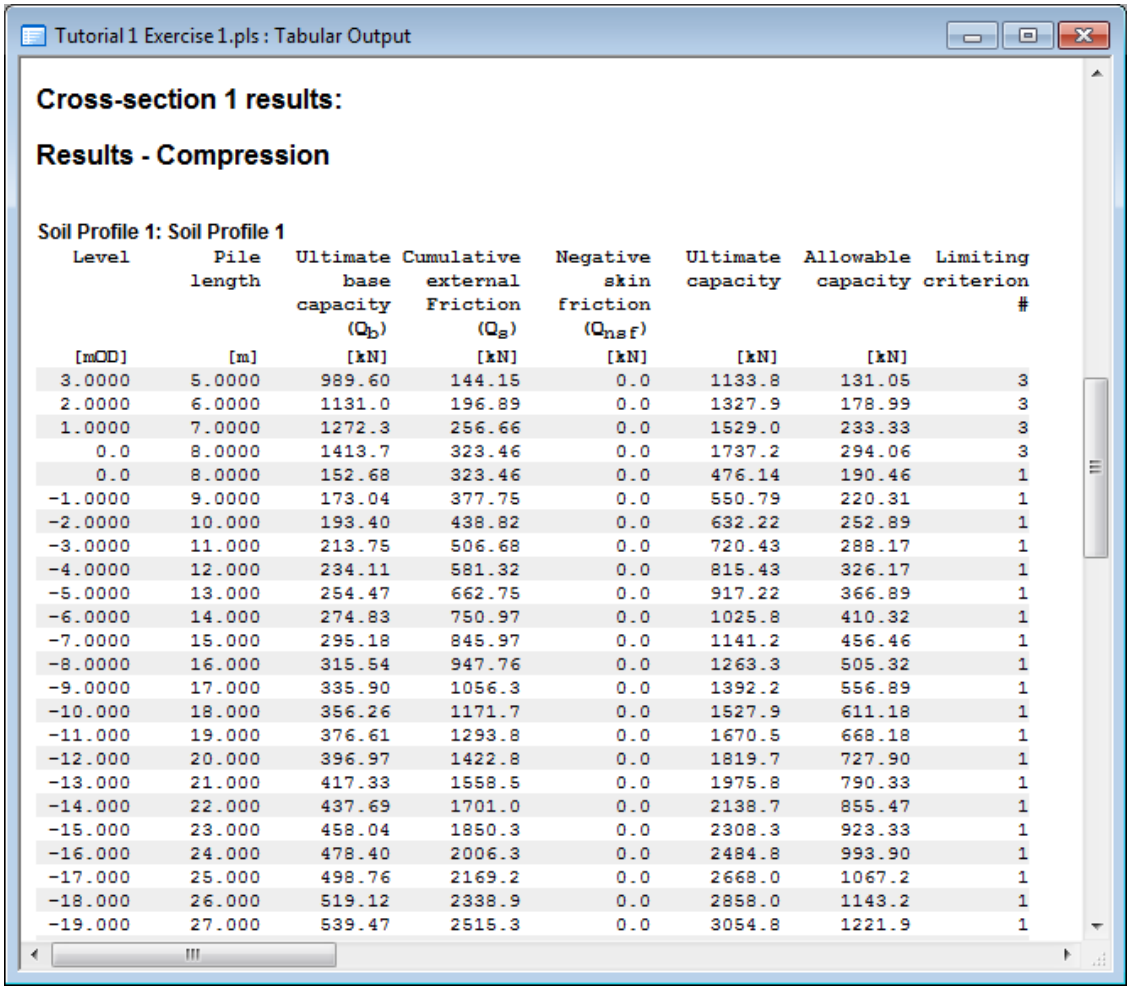
1.2.2.1 Tabular Outputs

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

- Select the outputs only by ticking Capacity Results:



- The 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

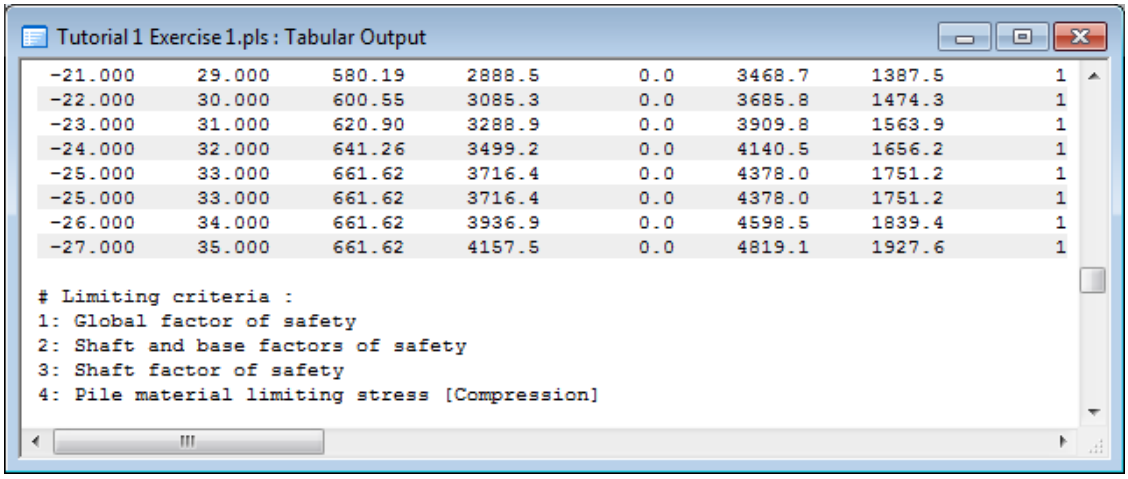
Cross-section 1 results:

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:



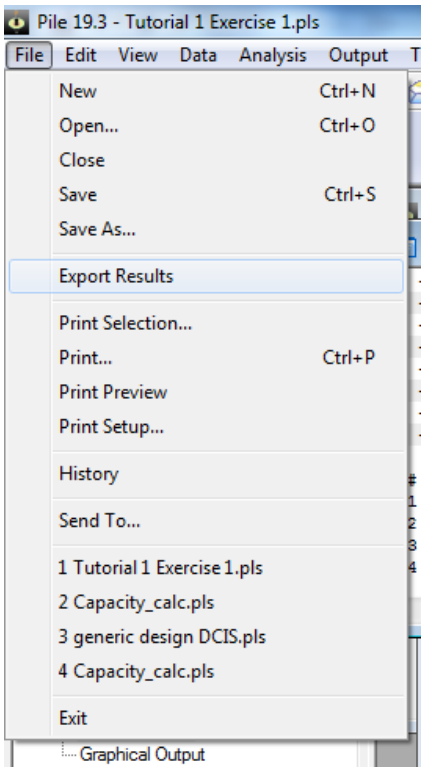
Tutorial 1 Exercise 1.pls : Tabular Output

-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.

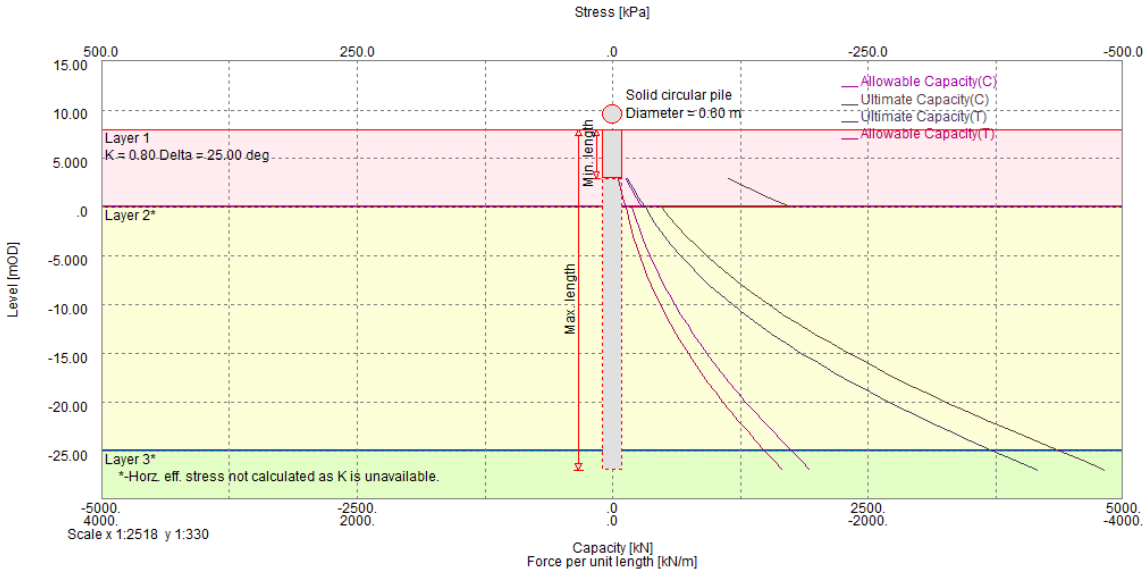
Ultimate load tension - toggles the ultimate load tension plot.

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.

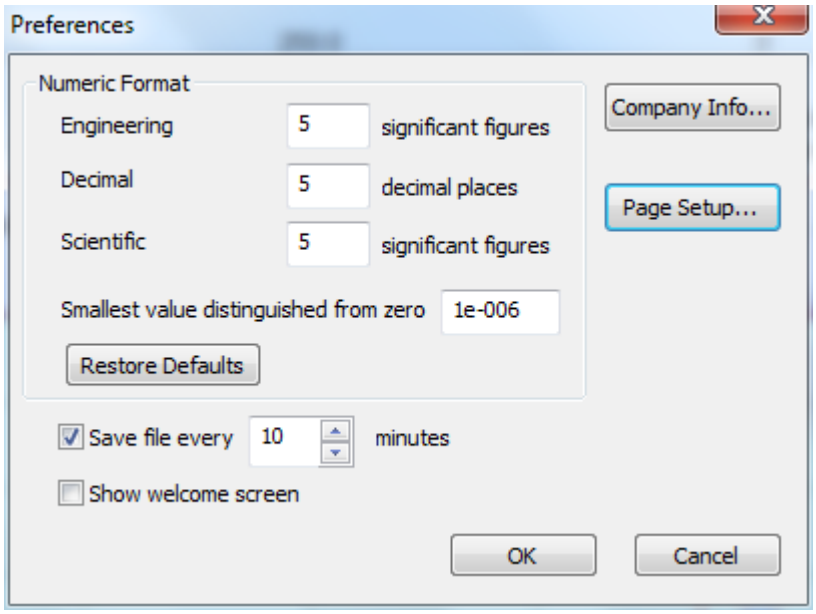
- The graphical output is shown below:



- 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



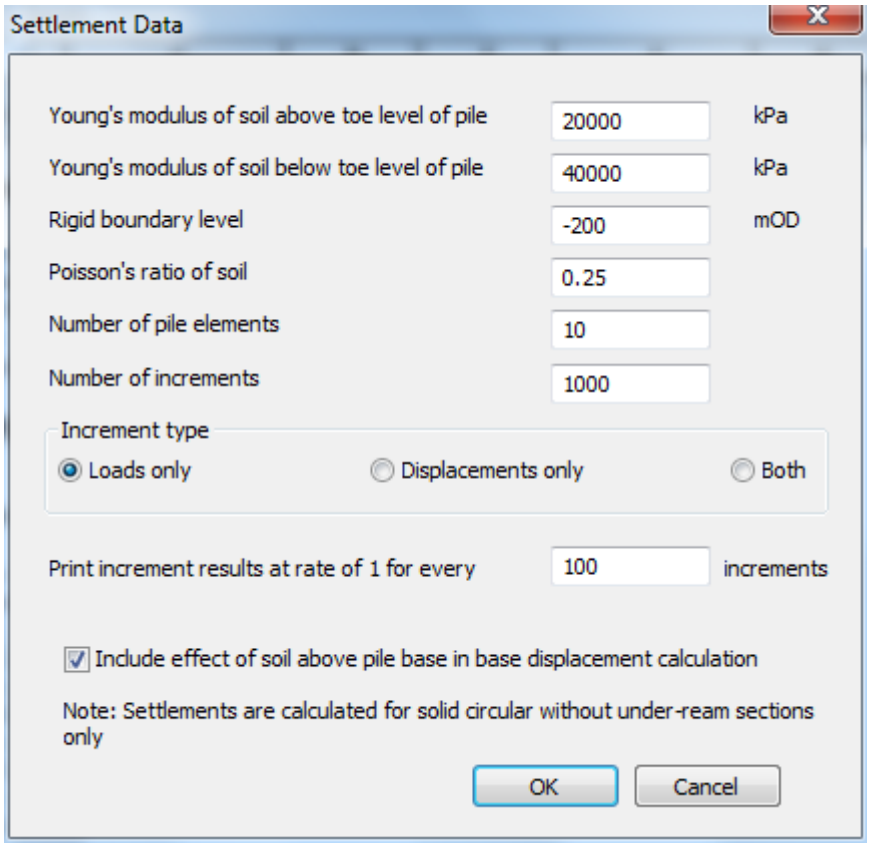
- Select appropriate options for the print output, including company information and logos.
- Select ***File > Print Preview*** in the menu bar to check the output for printing

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:



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 Σ 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.

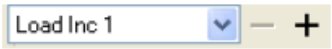

Pile Stress - toggles the pile stress plot.


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

Settlement results can be viewed for a selected soil profile 

and cross-section 

and selected pile length 

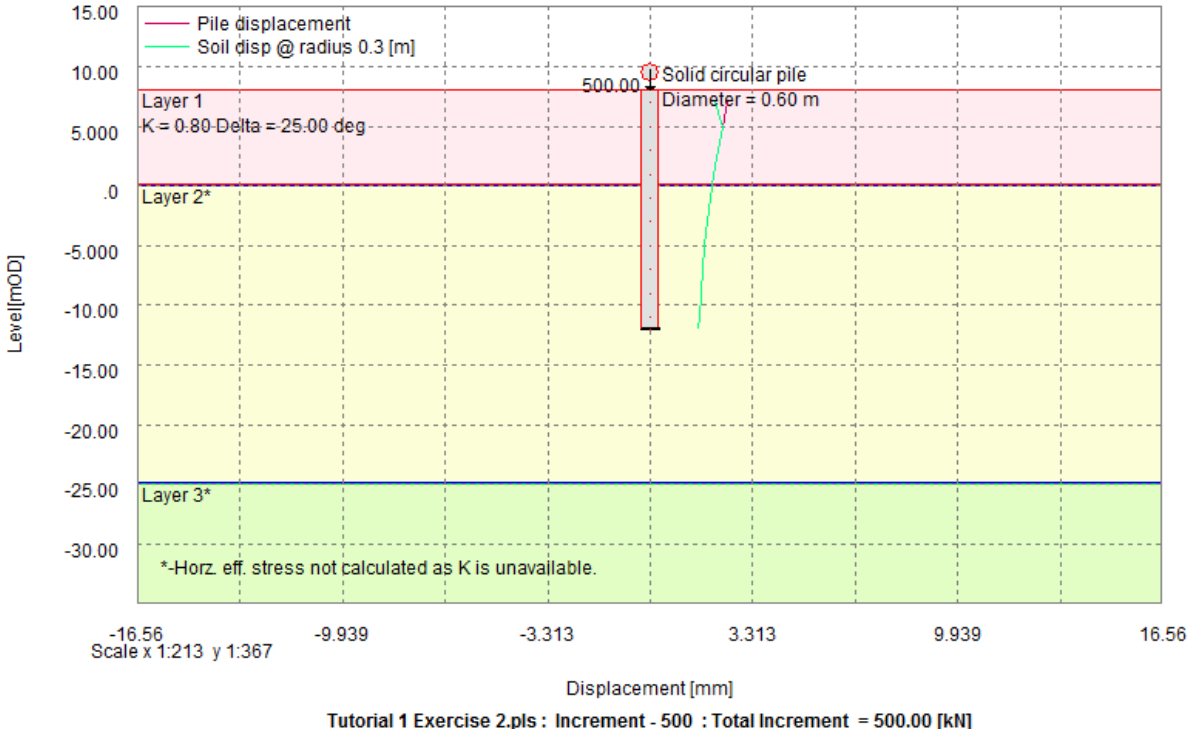
and selected load/applied displacement increments 

(if load increments exist) .

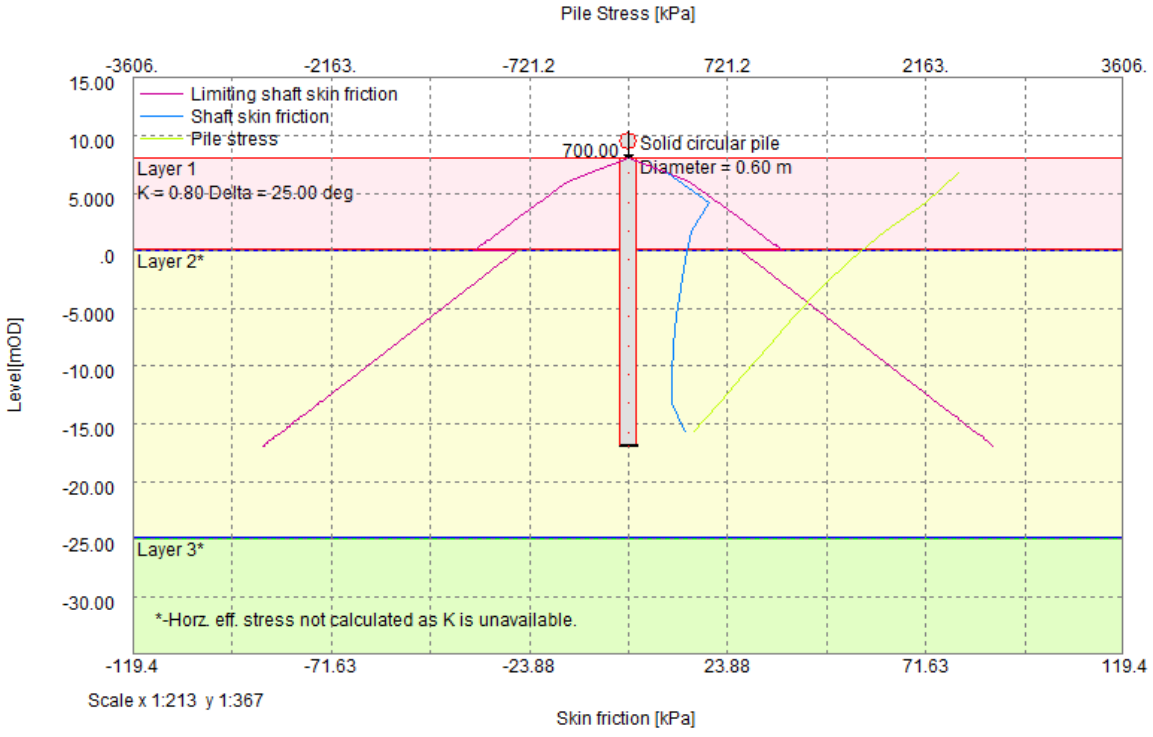
1 Pile displacement and Soil displacement at 0.3m for a 20m long pile with 500kN load acting

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:



Tutorial 1 Exercise 2.pls : Increment - 500 : Total Increment = 500.00 [kN]



Tutorial 1 Exercise 2.pls : Increment - 700 : Total Increment = 700.00 [kN]

Section 2

Step by Step Tutorial 2
Oasys Alp – Lateral Pile Capacity

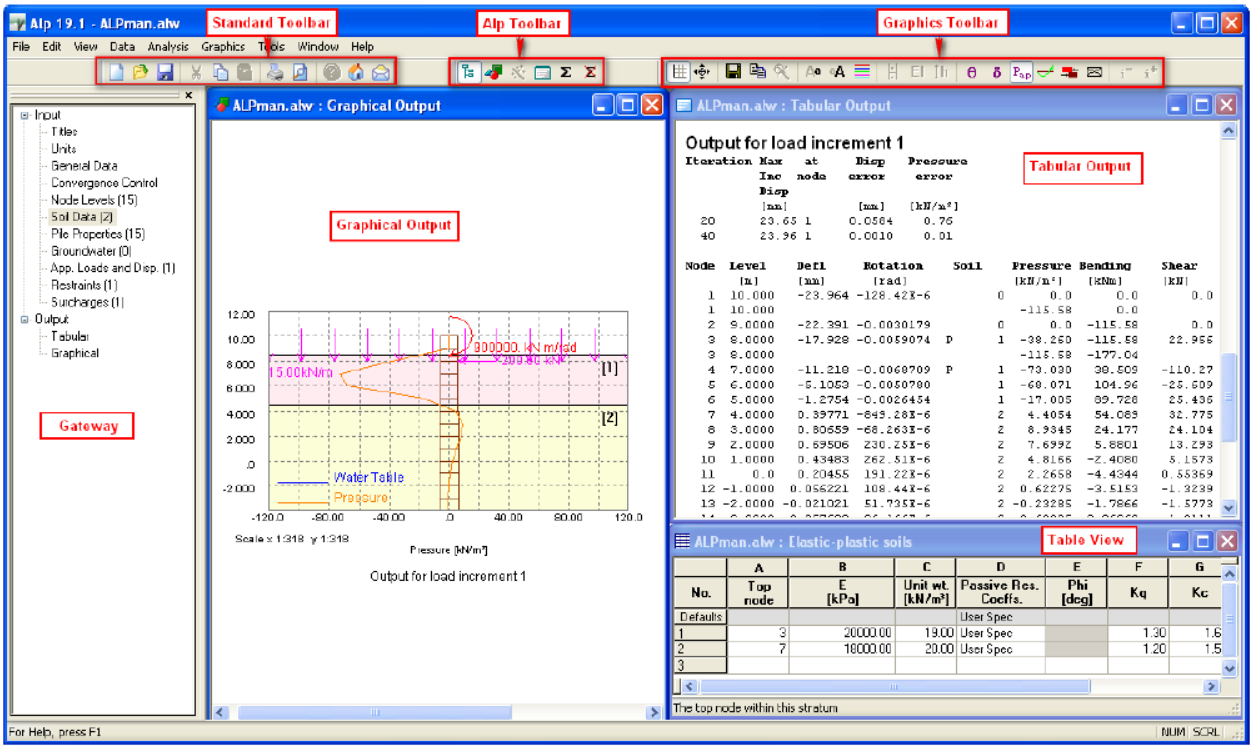
Objectives

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

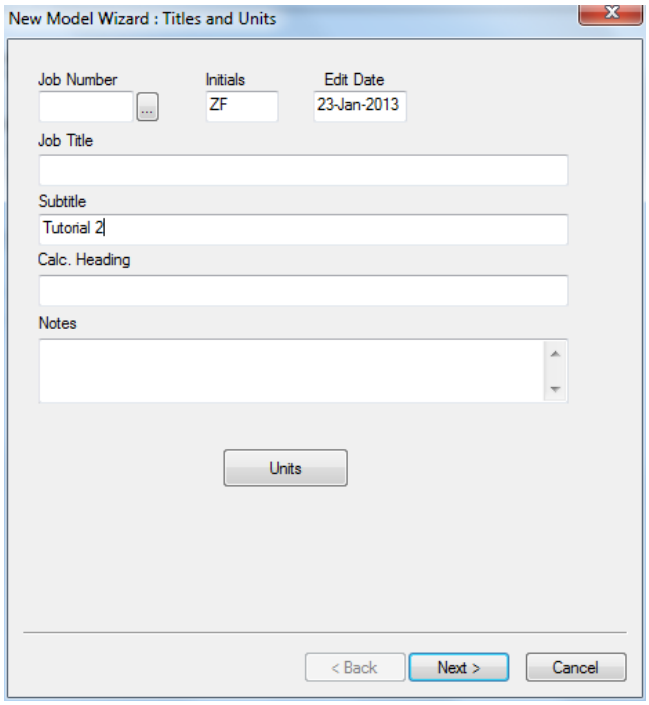
2.1 Components of the User Interface

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.

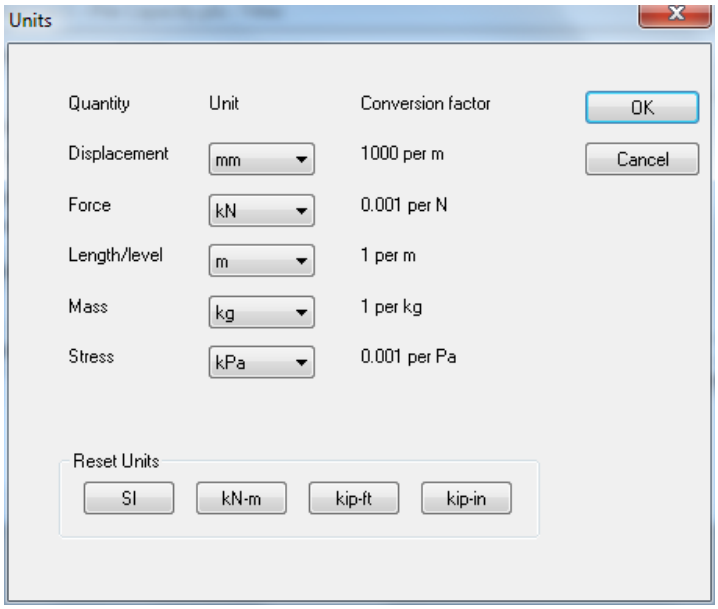


2.2 Creating the input

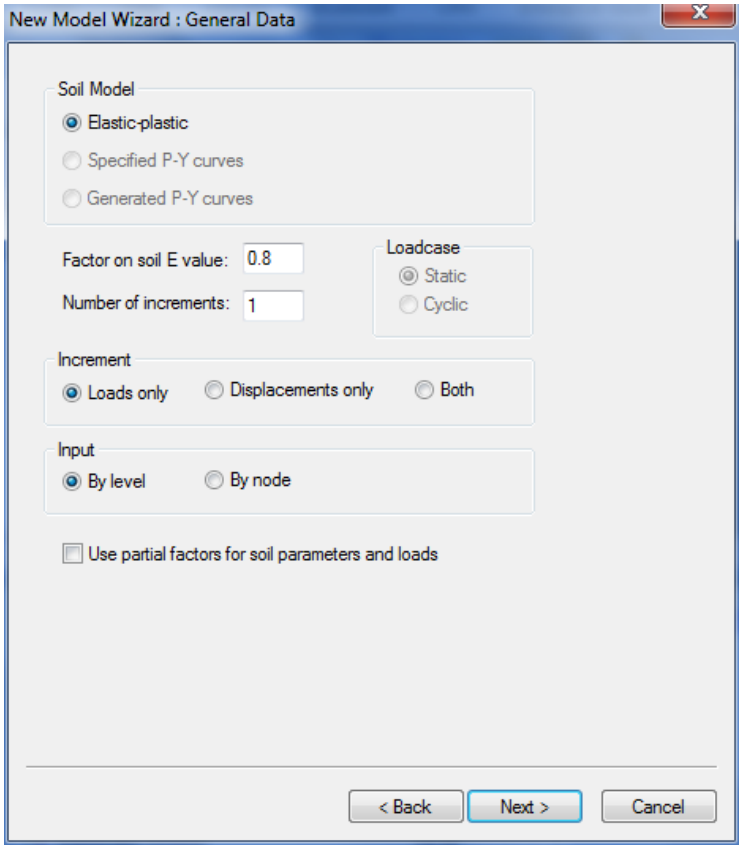
- Once opened, create a new file by clicking the '***New File'*** icon  on the top left of the program or clicking ***Ctrl + N***
- Fill the **Titles and Units** dialog box:



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.

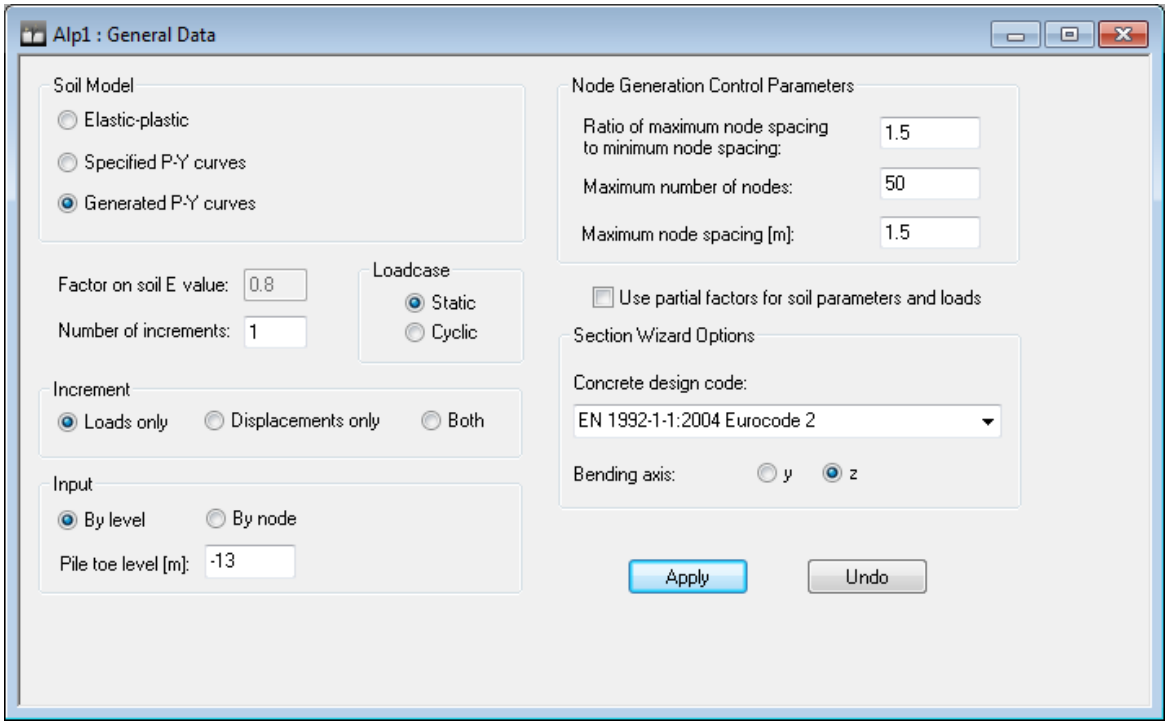


- Select the default options for the **General Data** box.



- (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 kNm2
    - 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



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

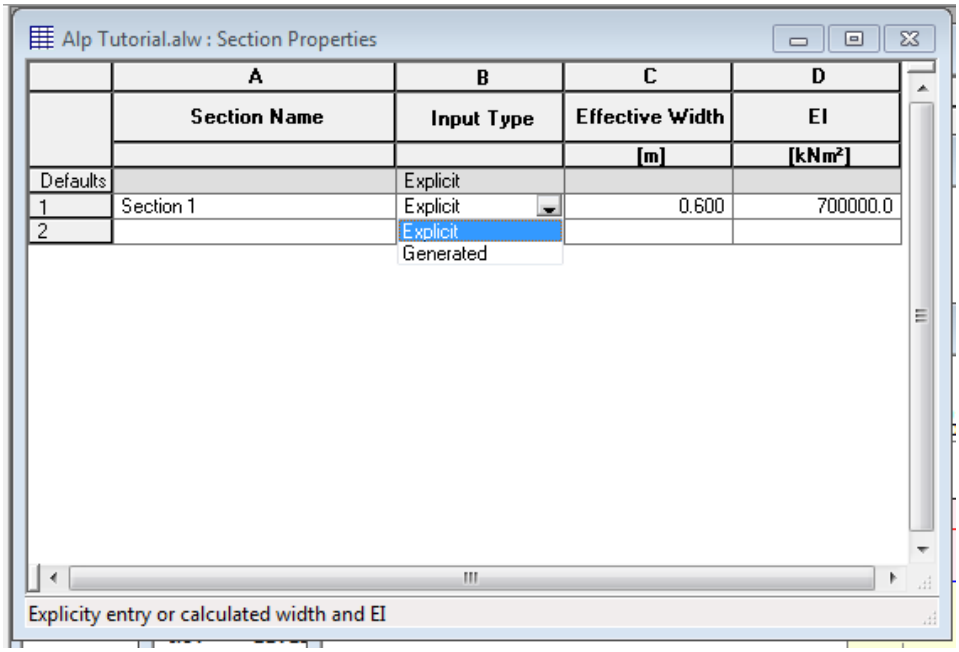
Alp Tutorial.alw : Generated P-Y curves

No.	A Top Level [m]	B Type	C Unit wt. [kN/m³]	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									

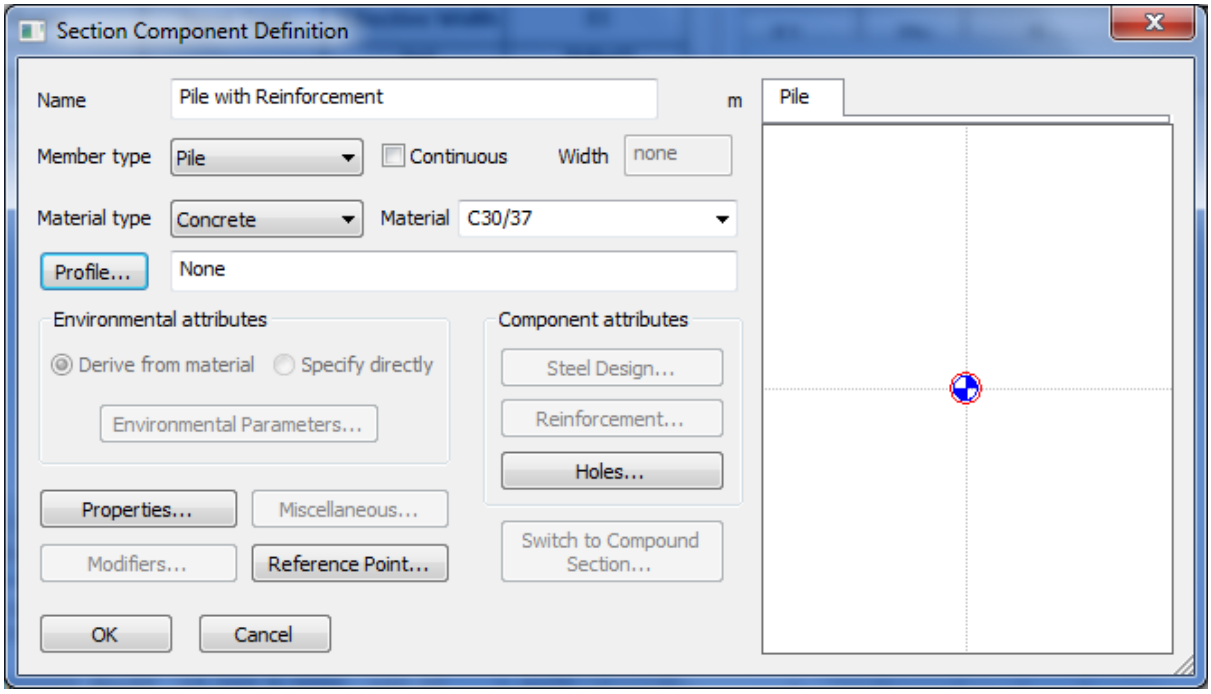
  - Soft Clay 0mOD
  - Stiff Clay -3mOD
  - Sand (API 18) -8mOD

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

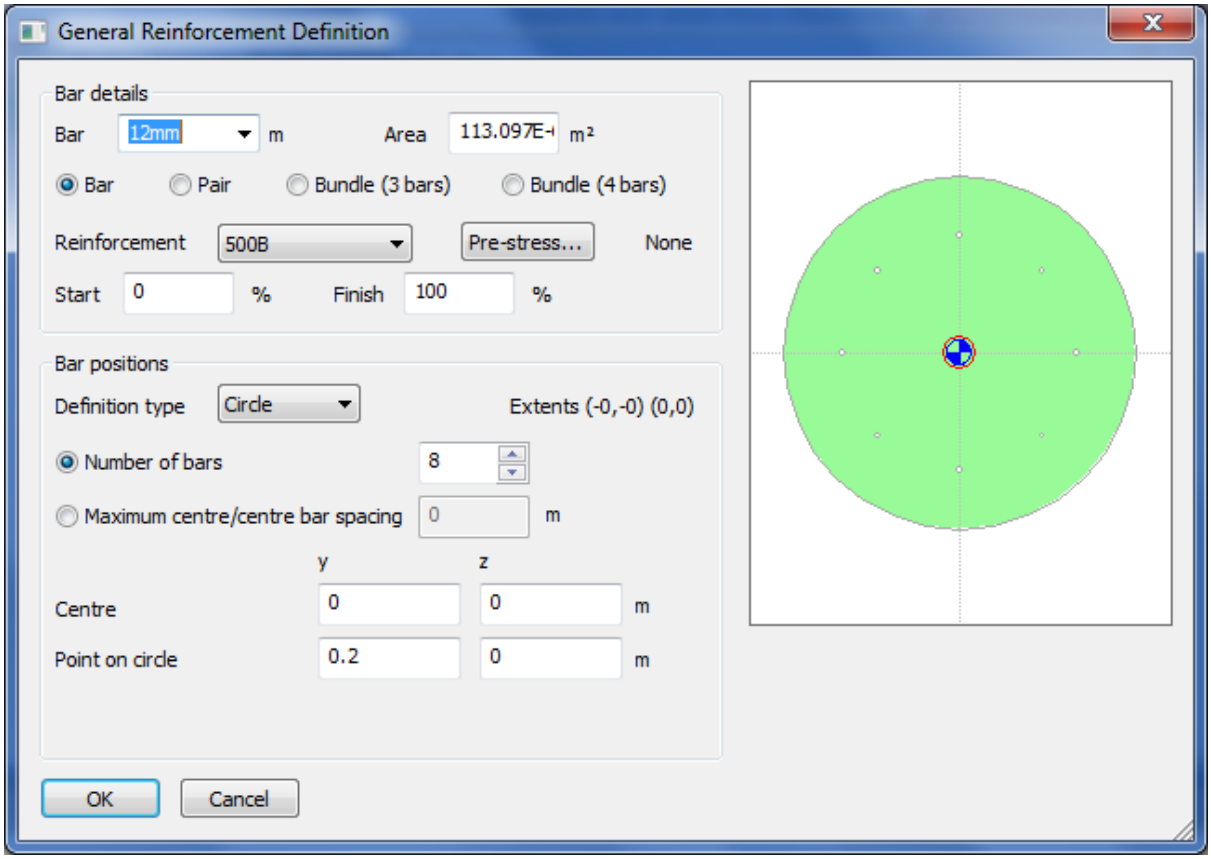


- The Wizard will appear. Name the section ***Pile with Reinforcement***.



- 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:



- 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 Σ 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.

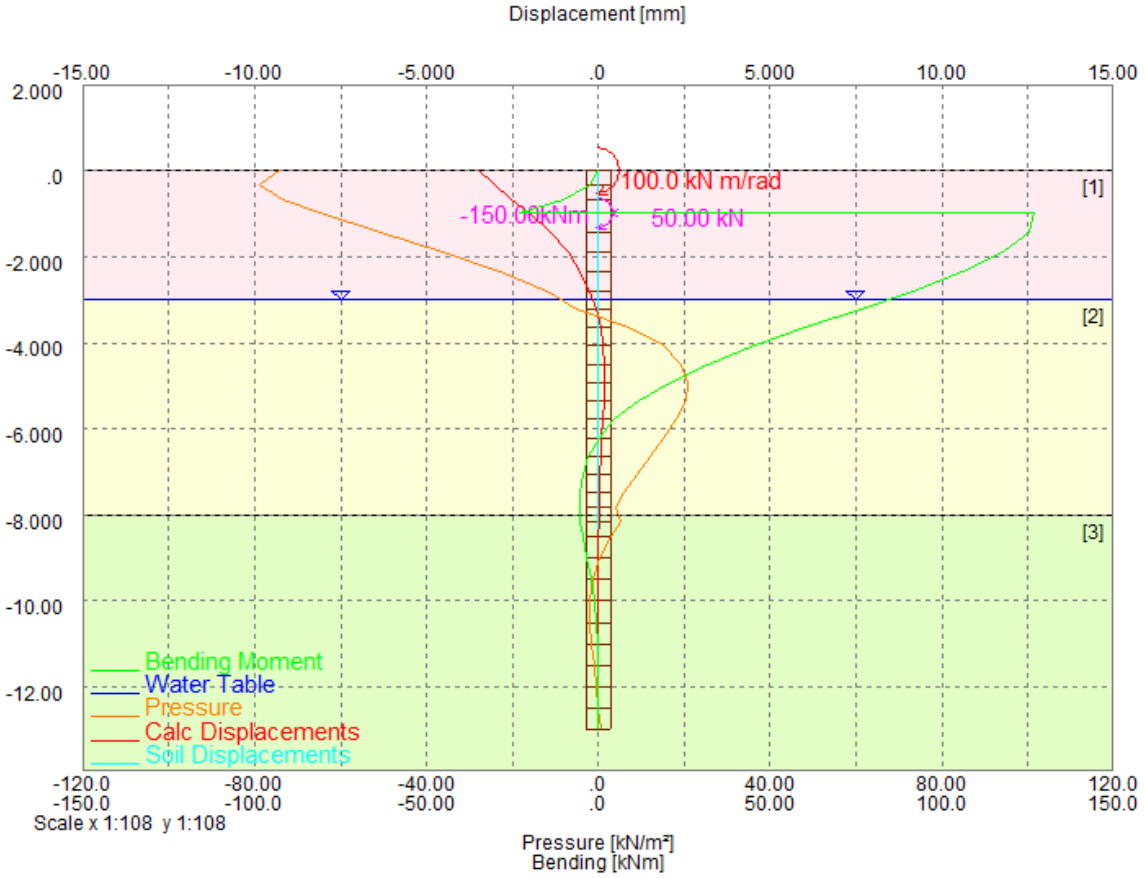
- View the tabular output.

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

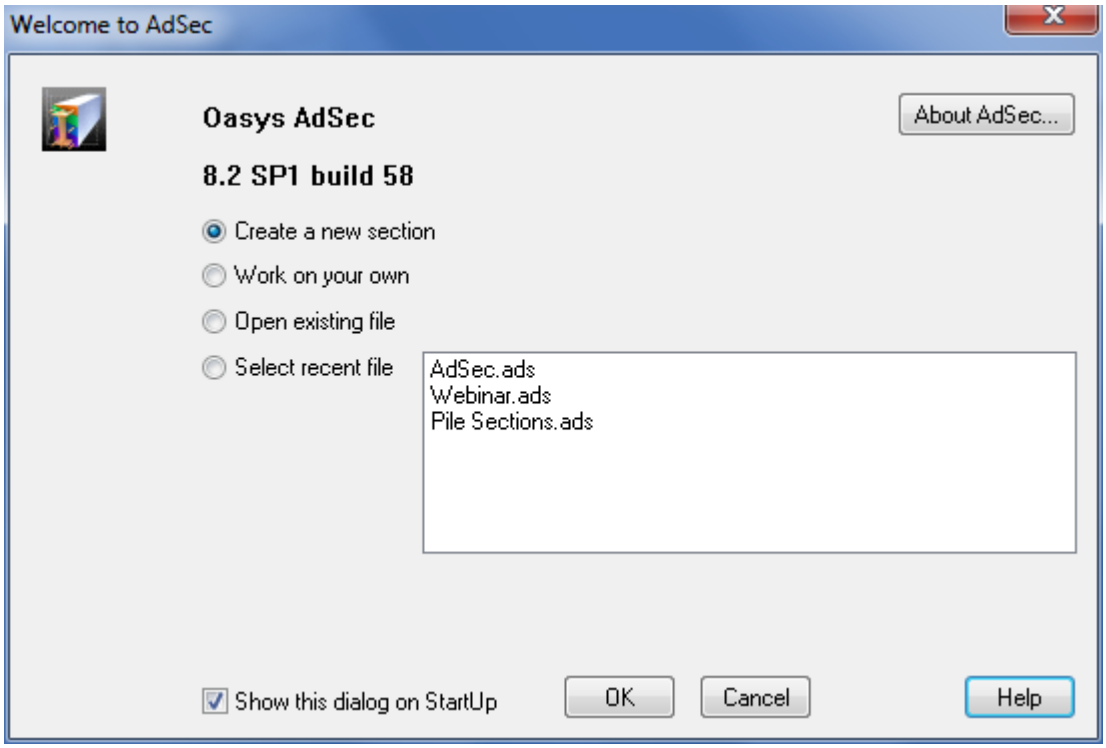
By the end of the session the user should be able to:

- 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

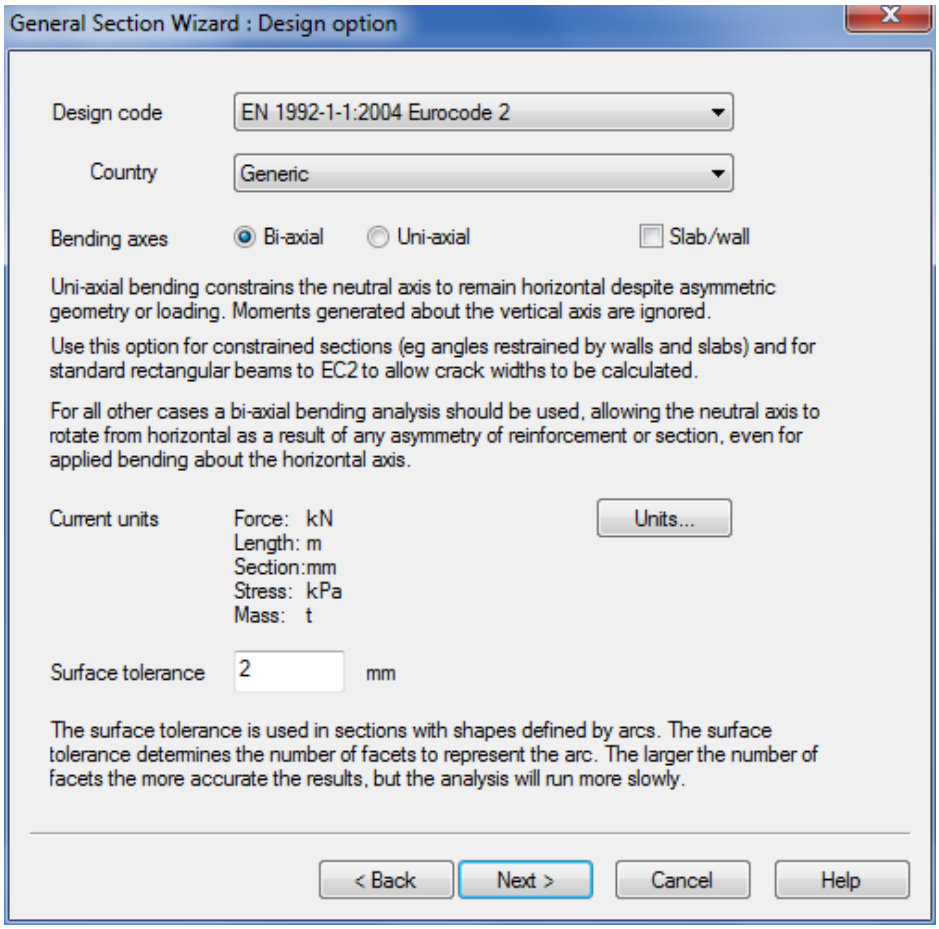
3.1 Input

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***.

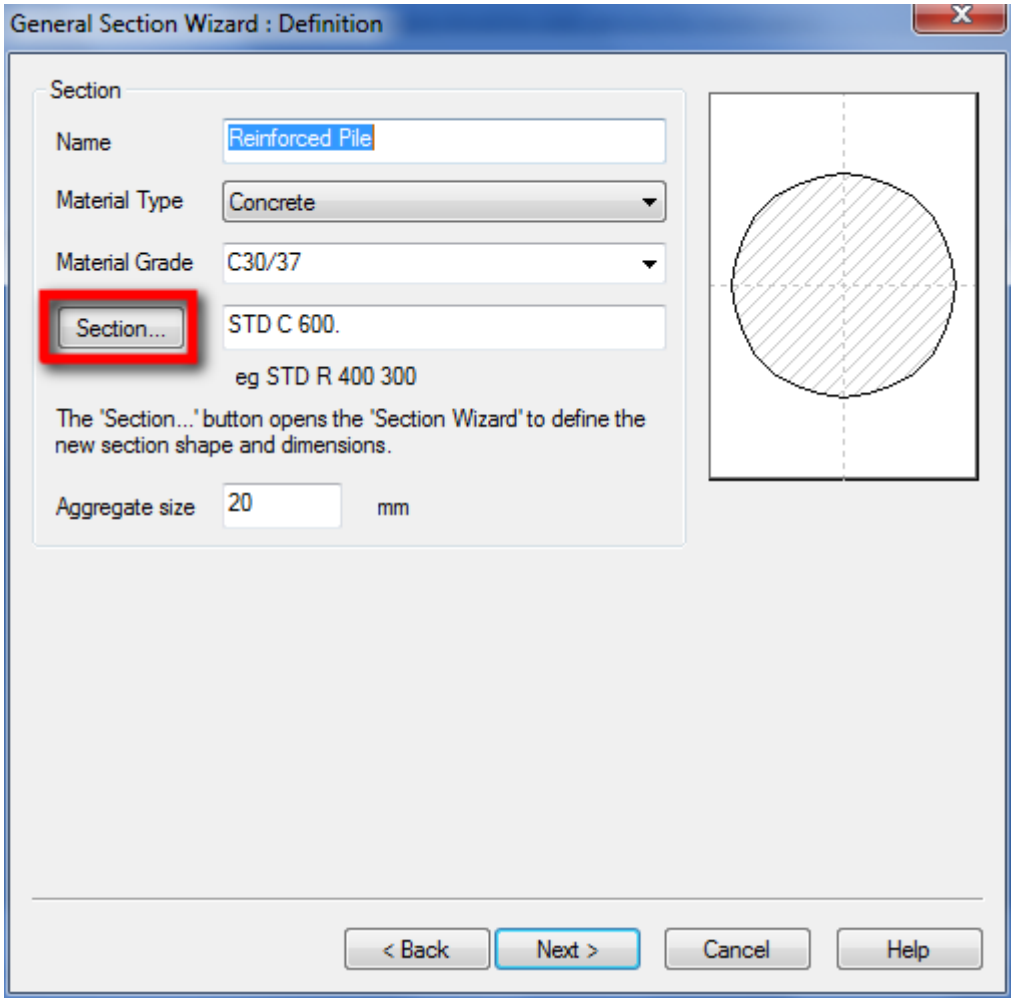


- Complete the ***Titles and Units*** section of the Wizard.
- In the Design Option section, choose ***Eurocode 2***

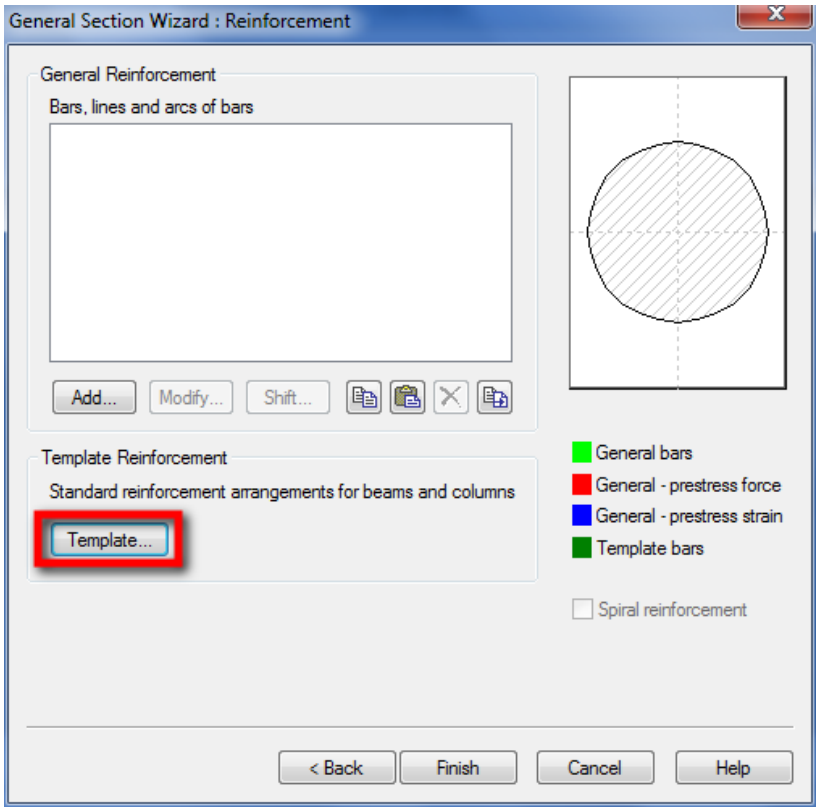


Note: The user can click on the Units button if you wish to change from SI Units

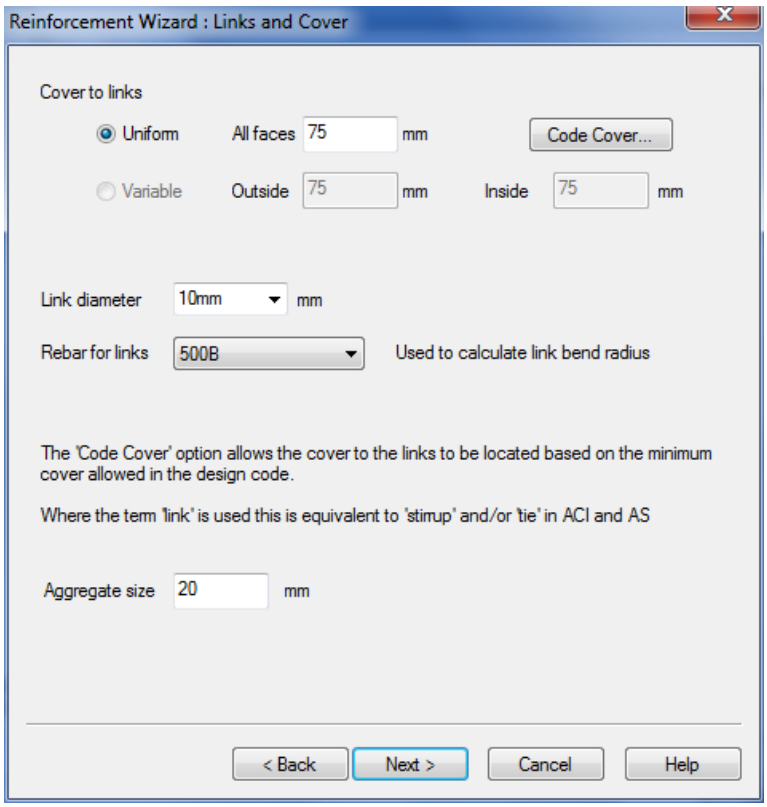
- In the ***Definition*** section of the Wizard, name the section appropriately and choose an Aggregate Size of 20mm.



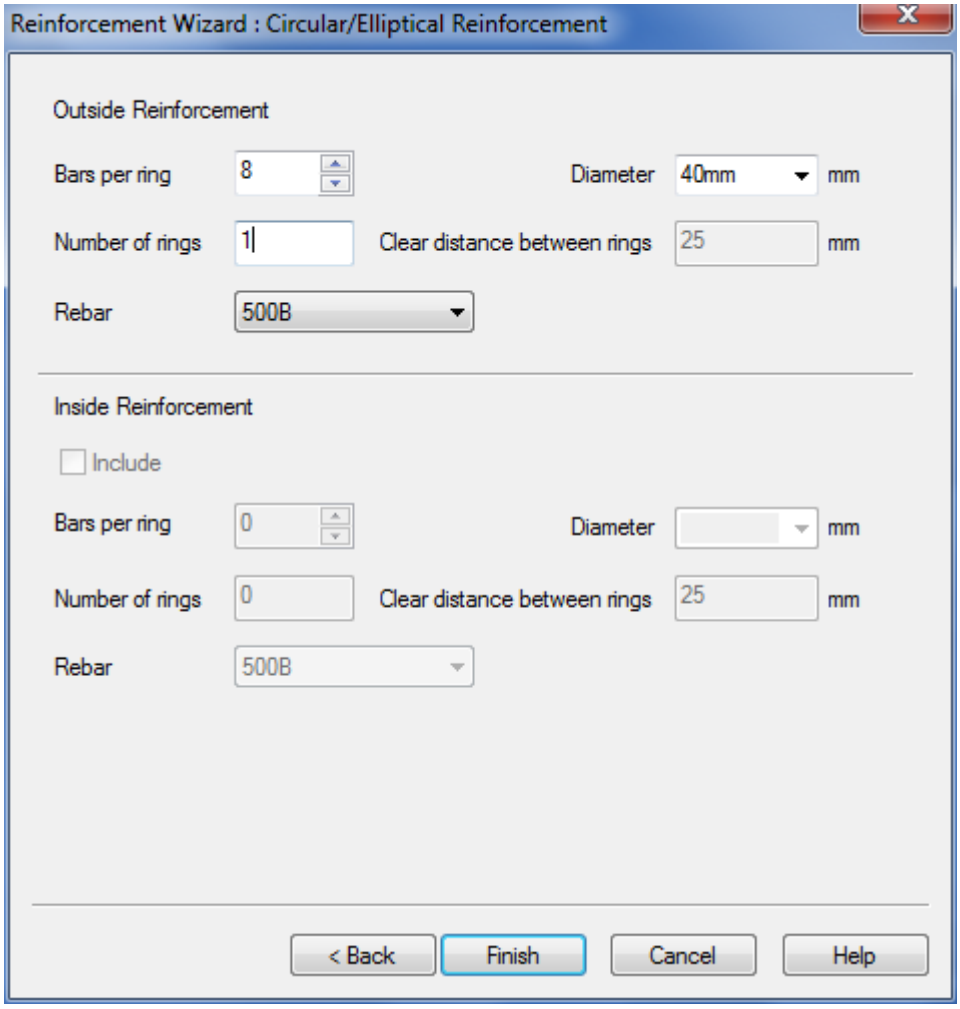
- Click on the **Section** button to open the Section Wizard.
- Choose a circular section with 600mm diameter.
- In the ***Reinforcement*** section of the wizard, click on the ***Template*** button.



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

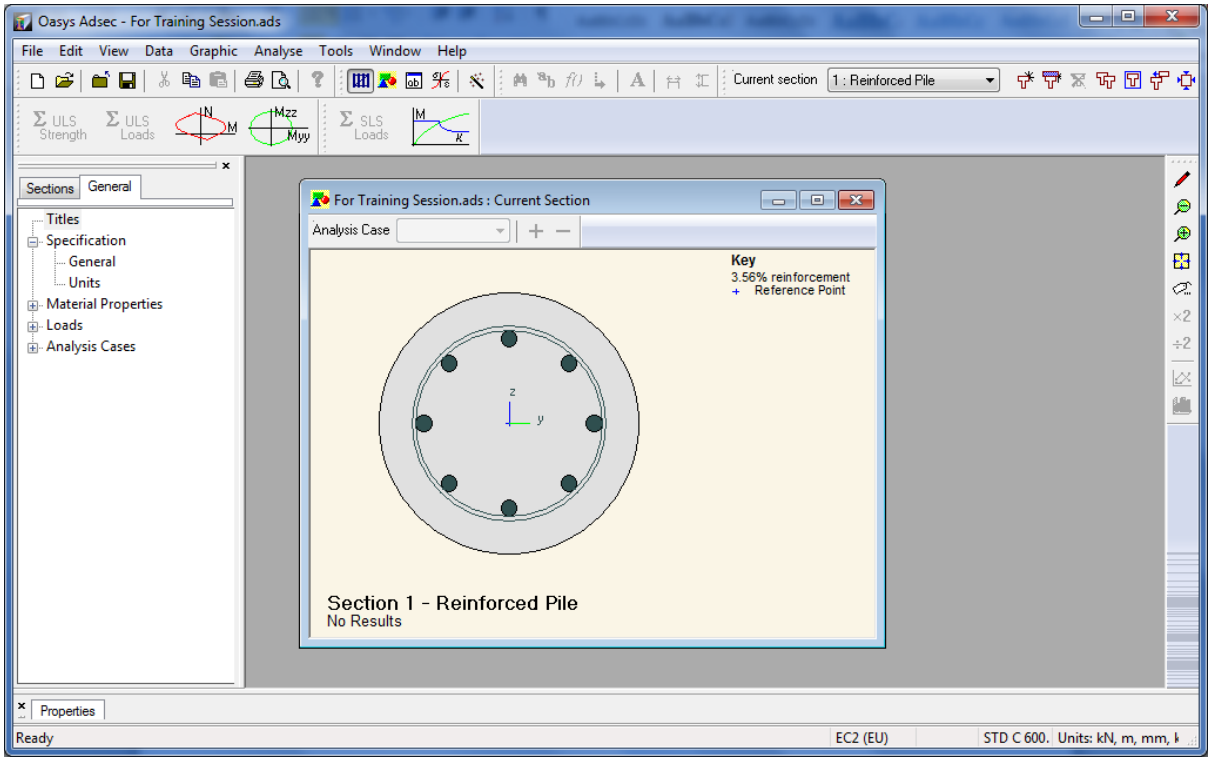


- Finally, in the ***Circular/Elliptical Reinforcement*** section of the wizard, choose the following options:



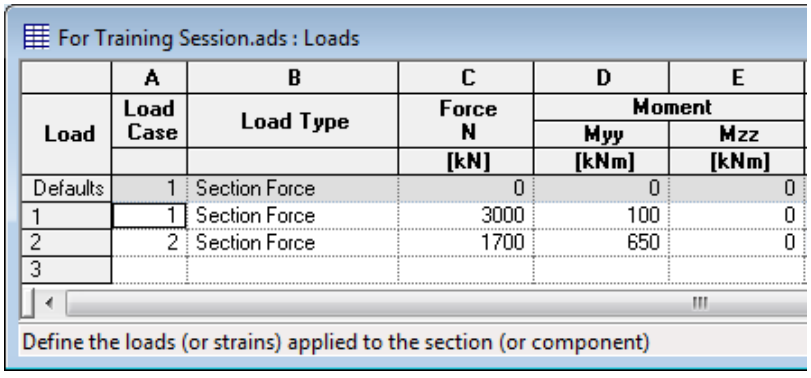
3.1.2 Applying the Load Cases

- 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:



	A	B	C	D	E
Load	Load Case	Load Type	Force N	Moment Myy	Moment Mzz
			[kN]	[kNm]	[kNm]
Defaults	1	Section Force	0	0	0
1	1	Section Force	3000	100	0
2	2	Section Force	1700	650	0
3					

- 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	Duration Factor
						[mm]	
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

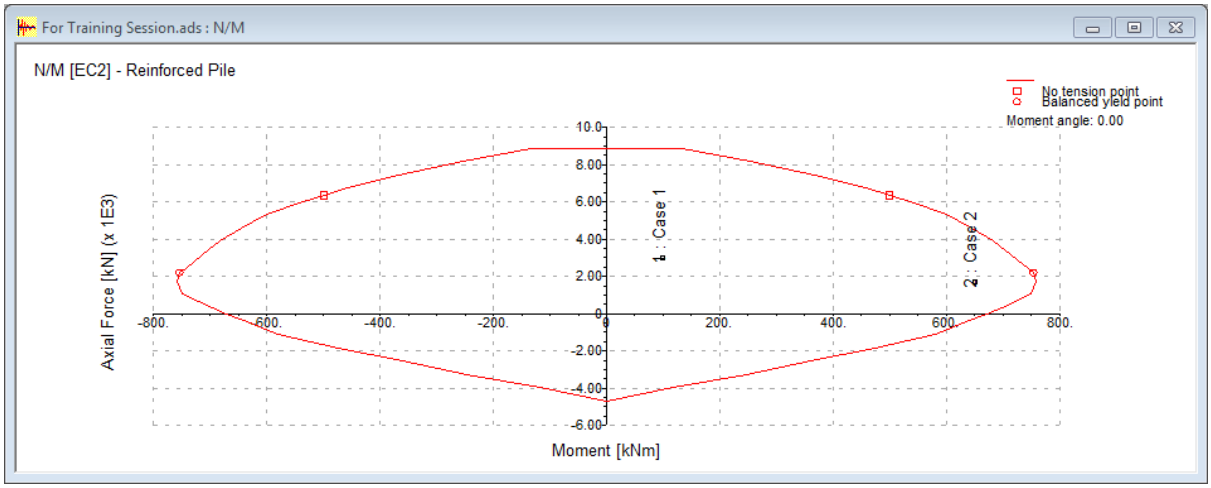
- Prior to analysis, check all inputs and save the file

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

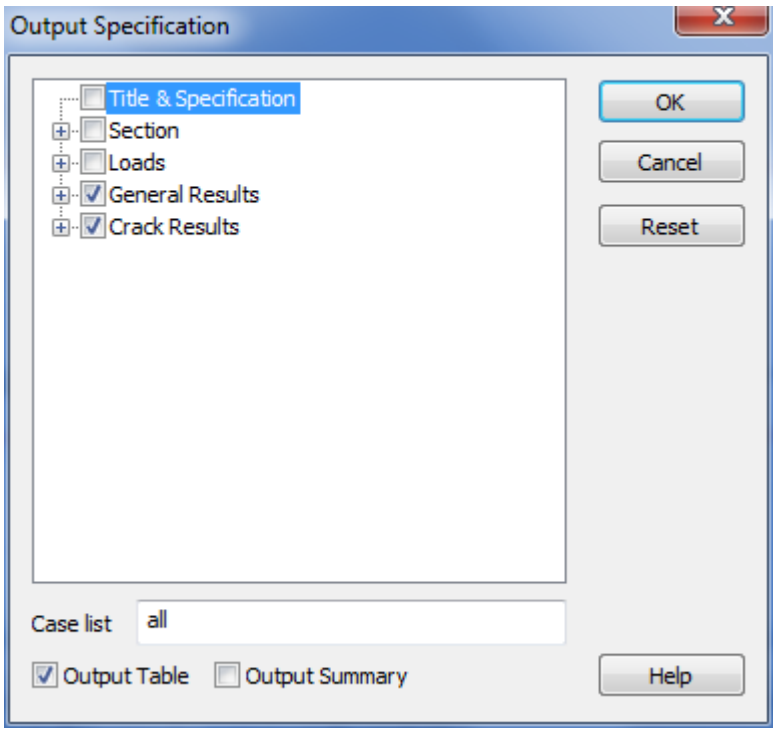
Note: Which load case is likely to cause the most cracking and why? Stop to discuss with the trainer



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



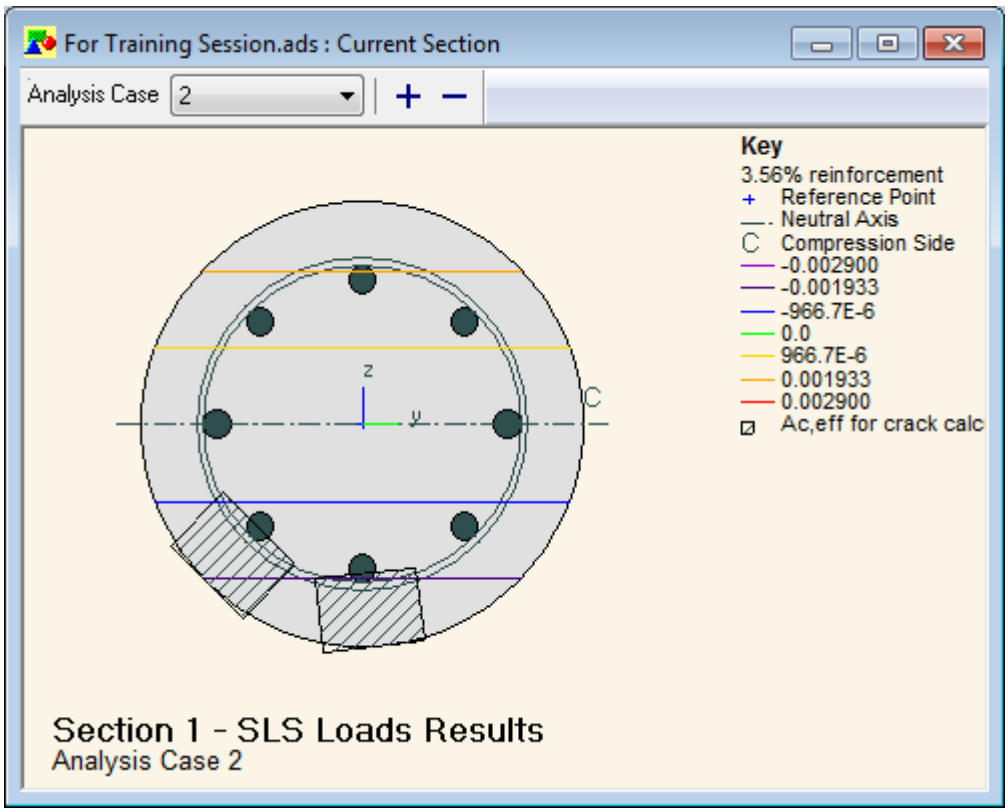
Note: Look carefully at the results. Have the results converged successfully? Stop and discuss with the trainer

- 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

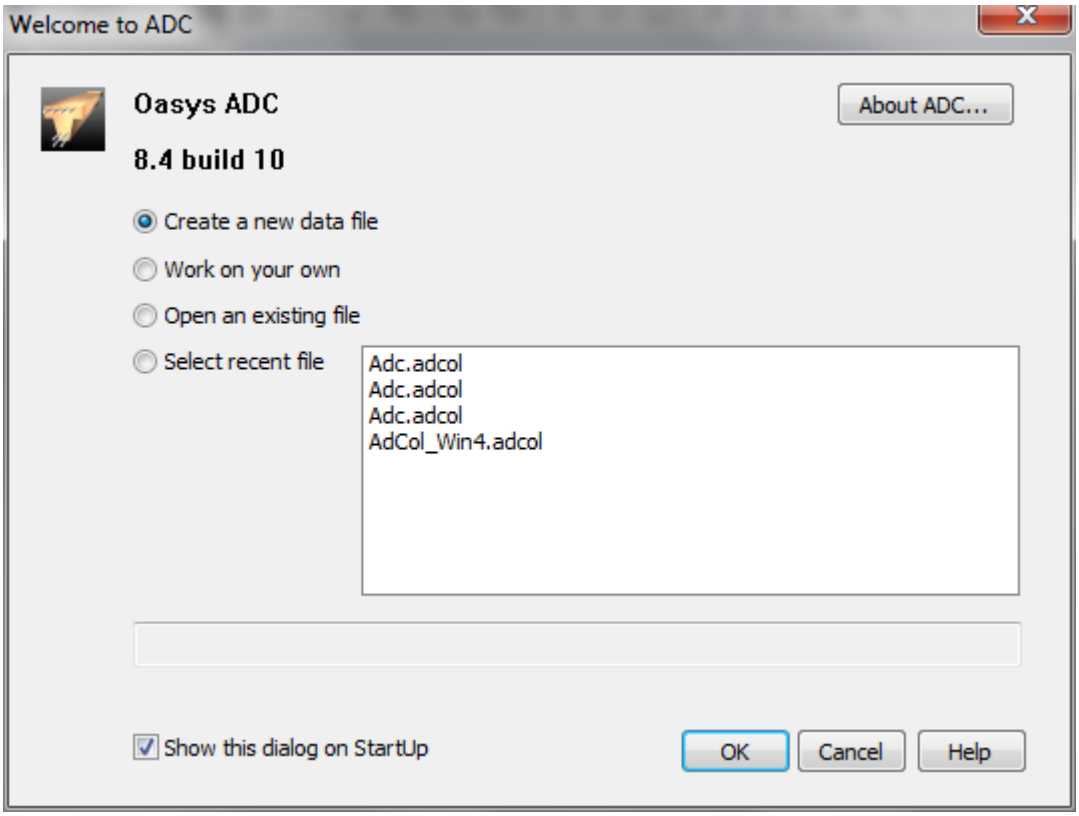
By the end of the session the user should be able to:

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

4.1 Input

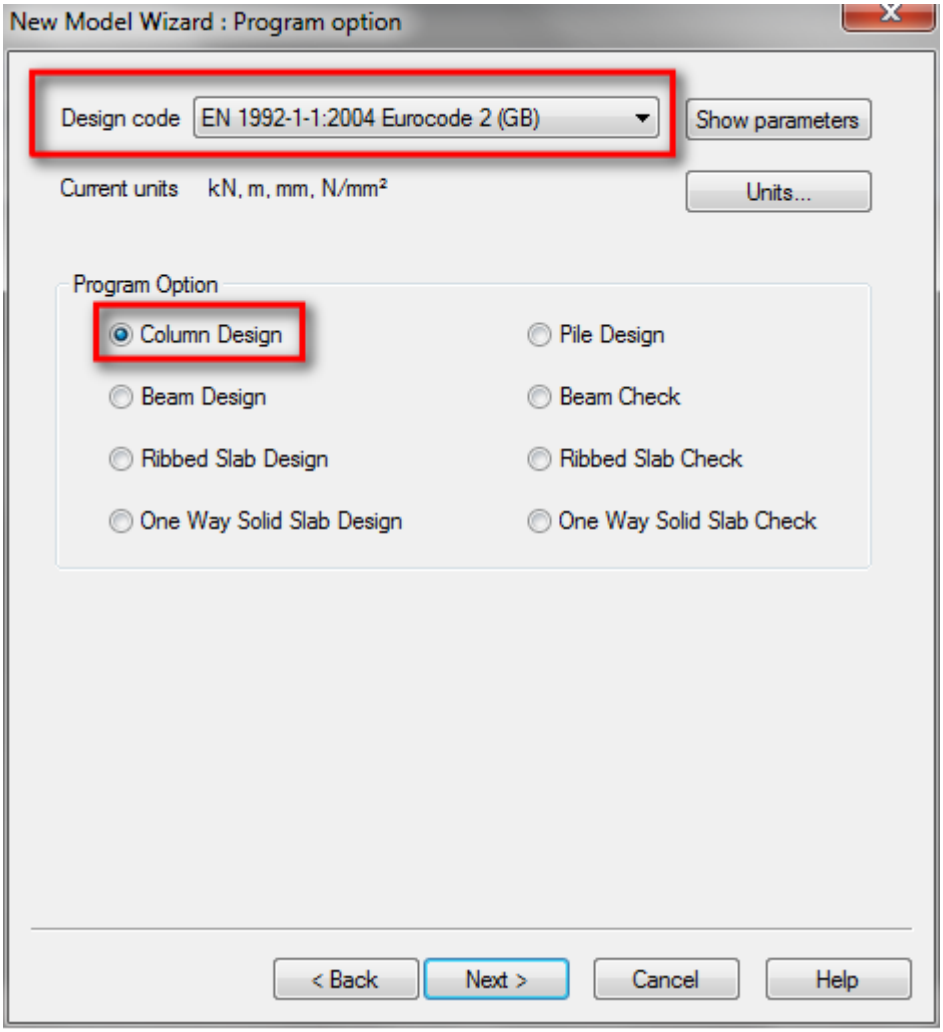
- Once opened, the program start up screen will give a number of options.

  Choose to *Create a New Section*.



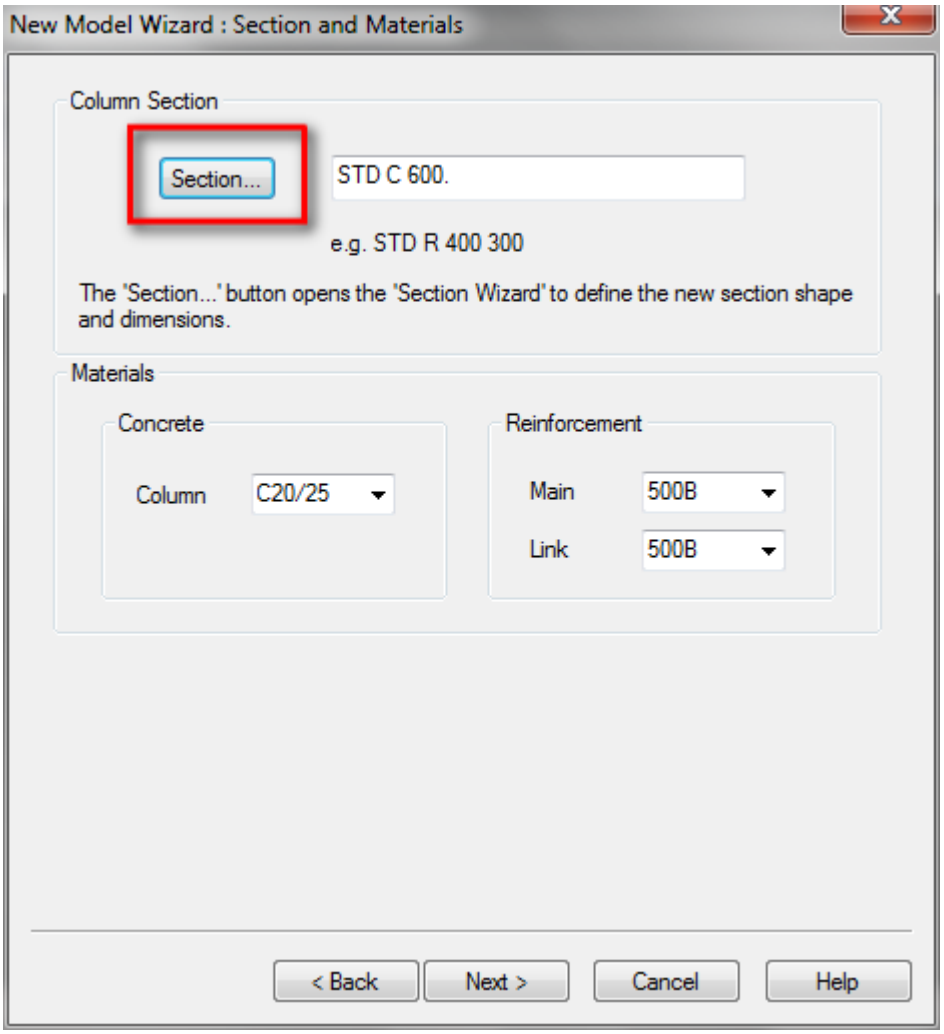
- Complete the ***Titles and Units*** section of the Wizard.

- 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.

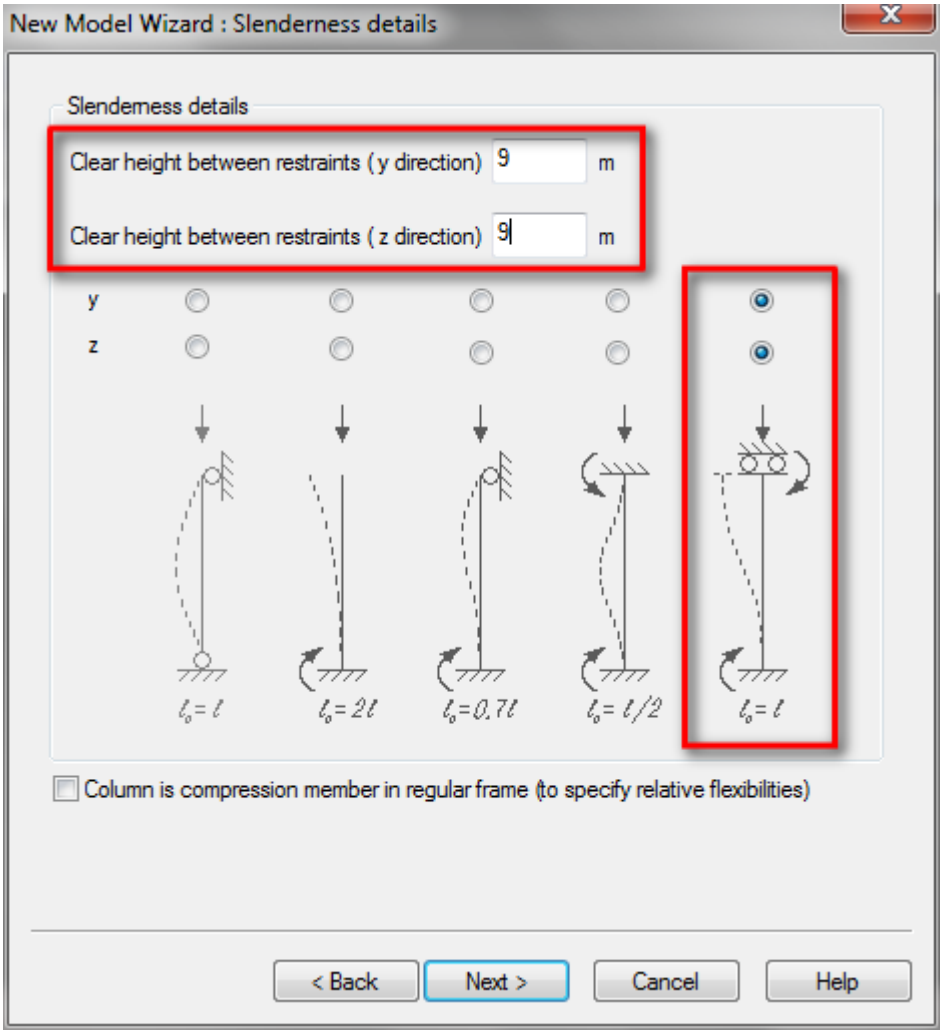


Click on the **Section** button to open the Section Wizard.

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:

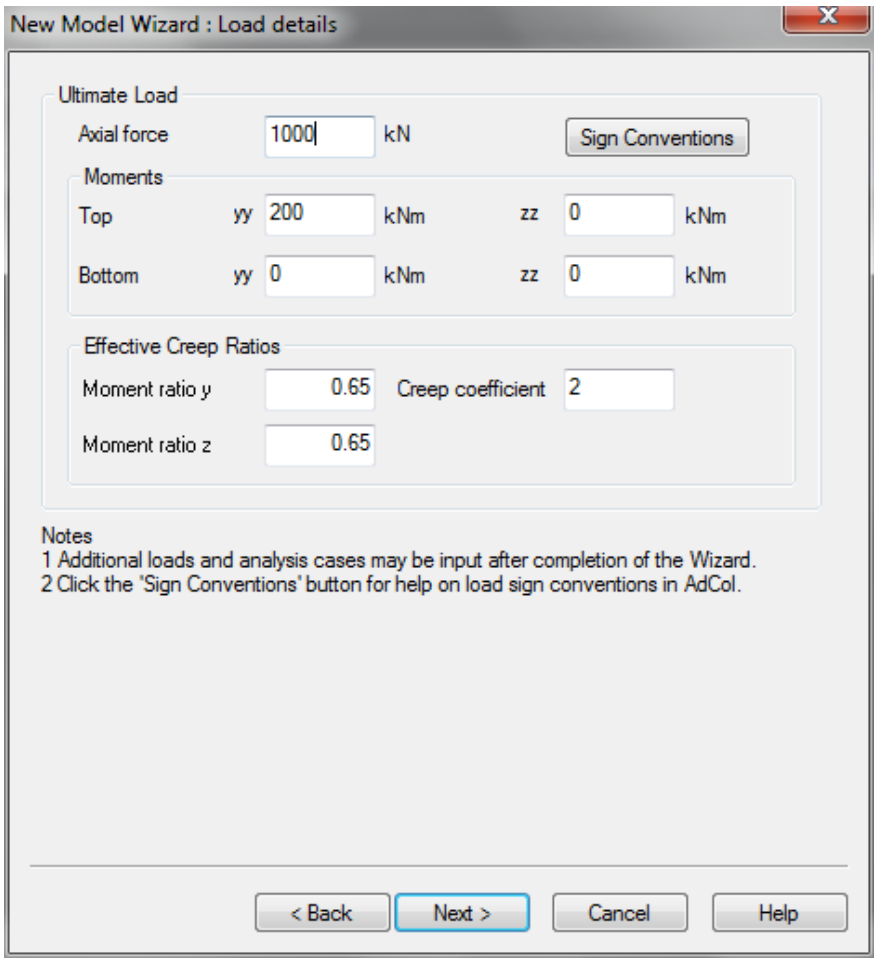


Apply the following Load Case (the moments are obtained from the Alp Output):

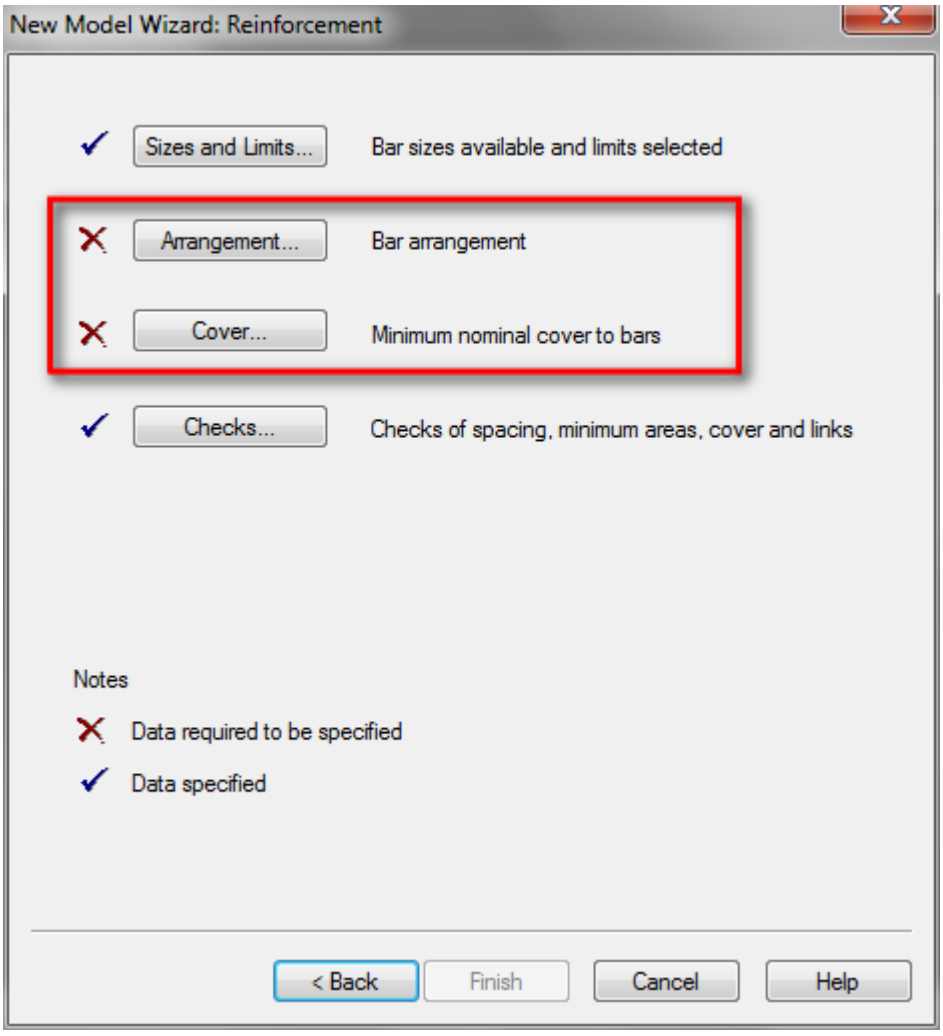
Axial Force = 1000kN

Top Moment (yy) = 200kNm

Bottom Moment (yy) = 0kNm

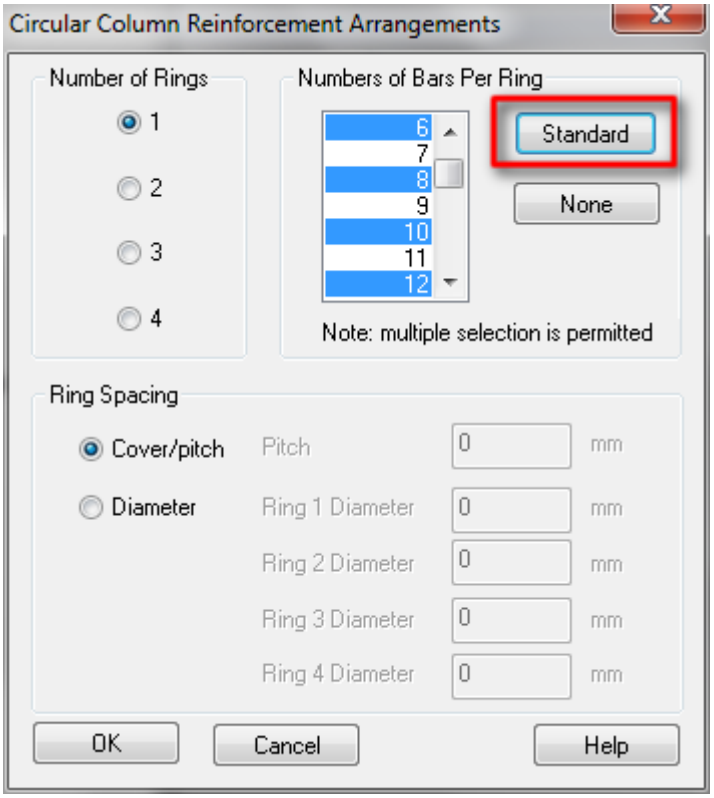


Finally, in the ***Reinforcement*** section, the Arrangement and Cover need to be specified:



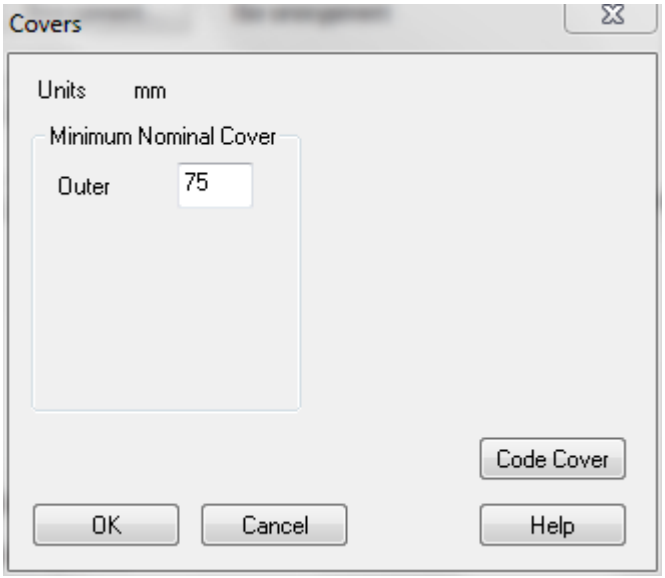
- Click on the ***Arrangement*** box.

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



- Click on the ***Cover*** box.

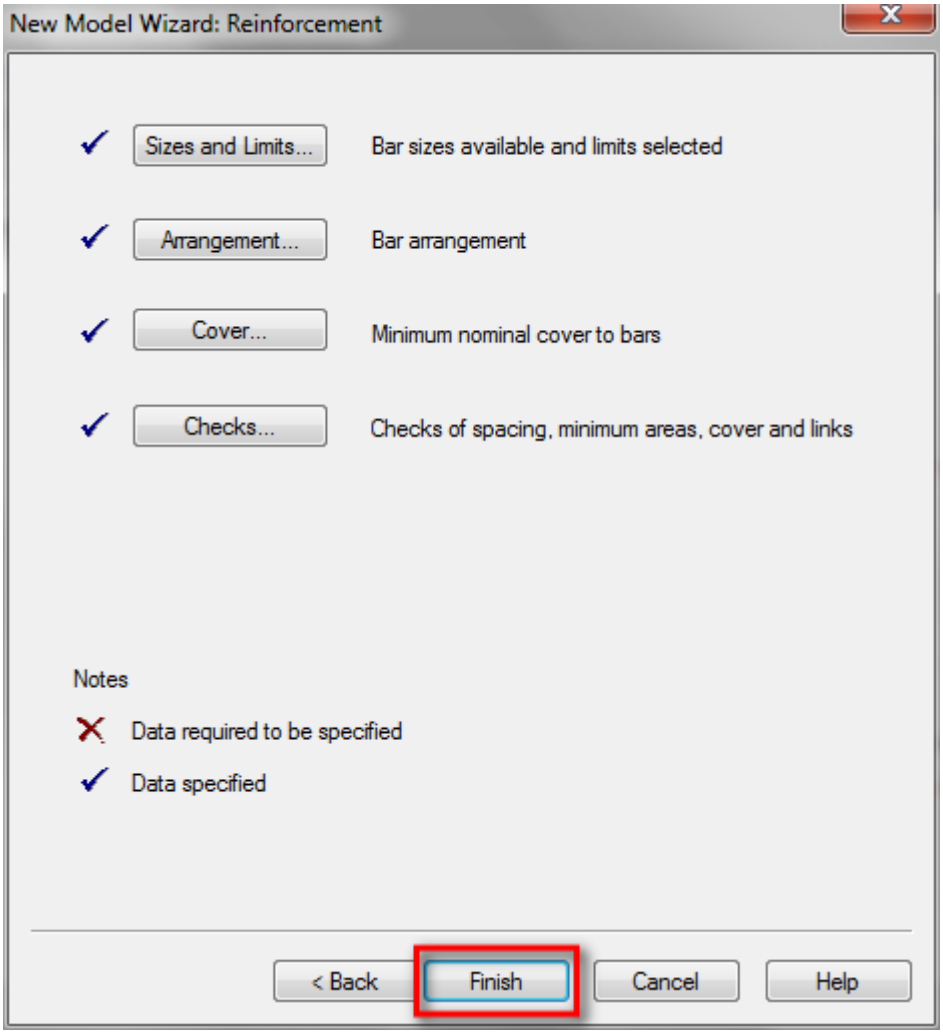
  Input the cover of 75mm.



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

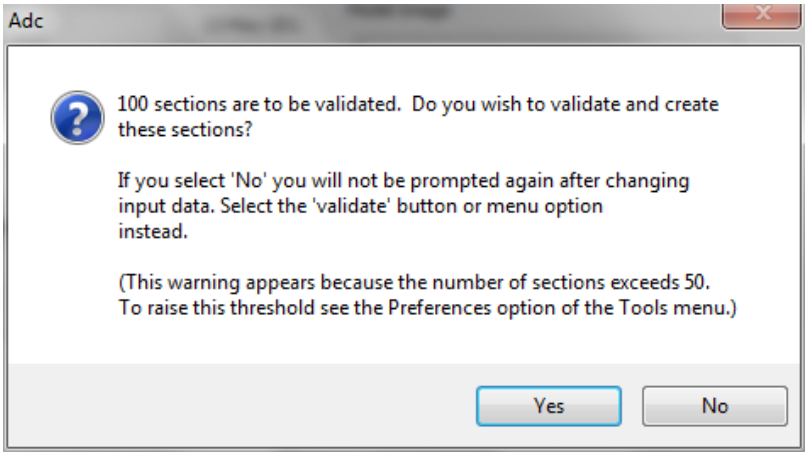
4.2 Analysis and Results

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

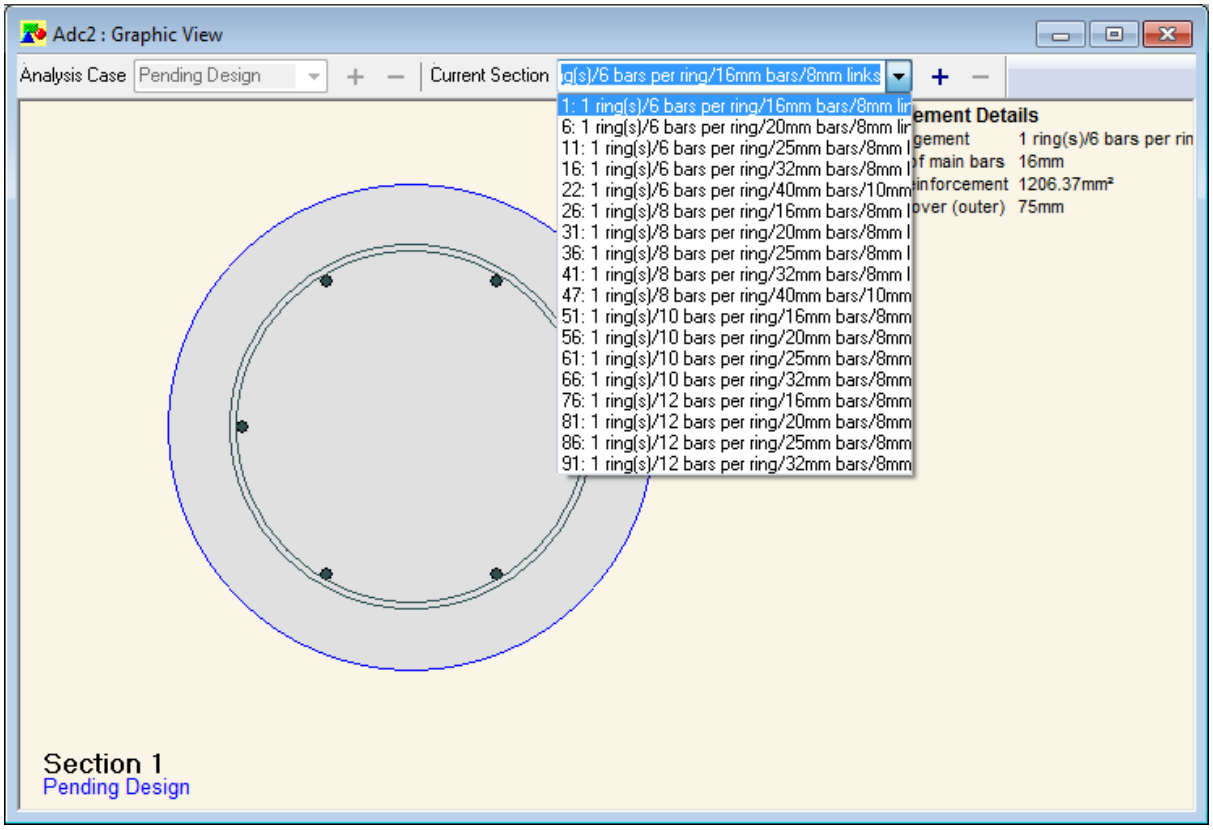


- The following box will appear, checking if you are happy with the number of sections.

  Click **OK** to proceed with the analysis.



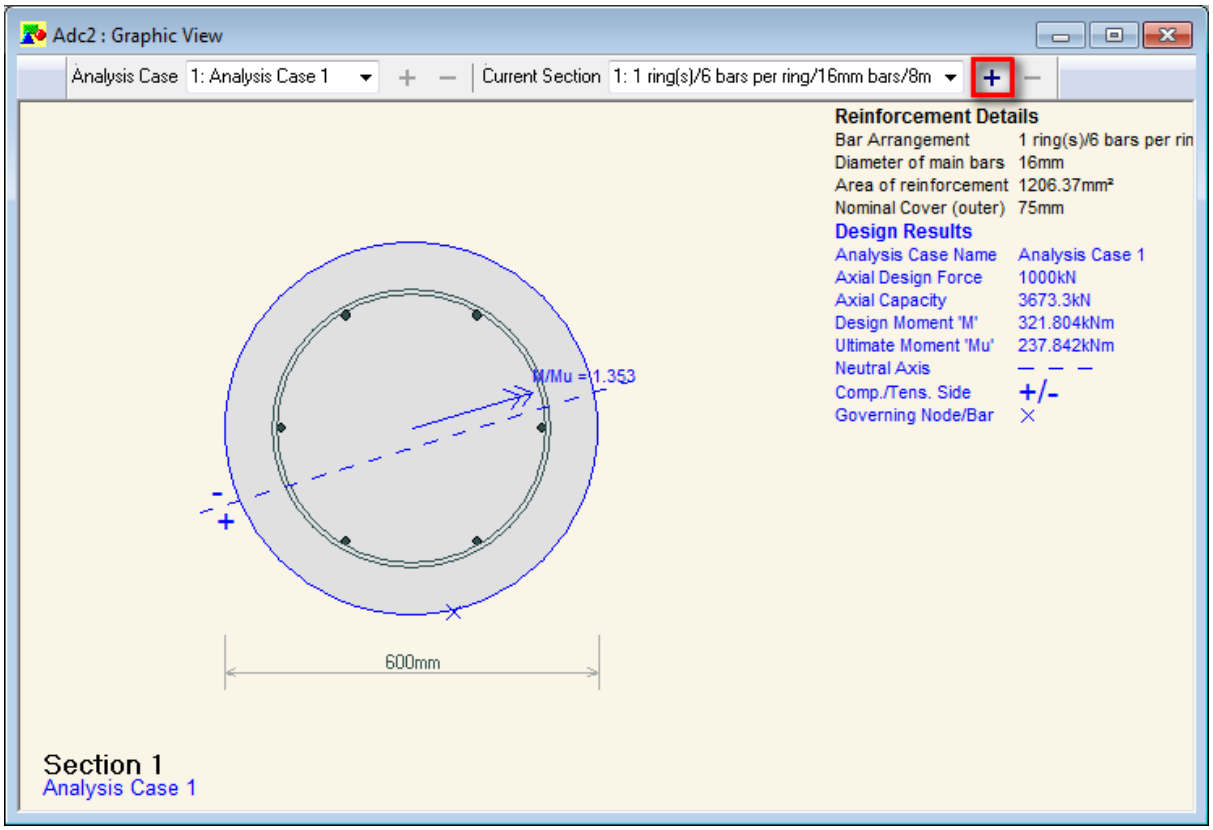
- The ***Graphic View*** will appear, showing the acceptable reinforcement:



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

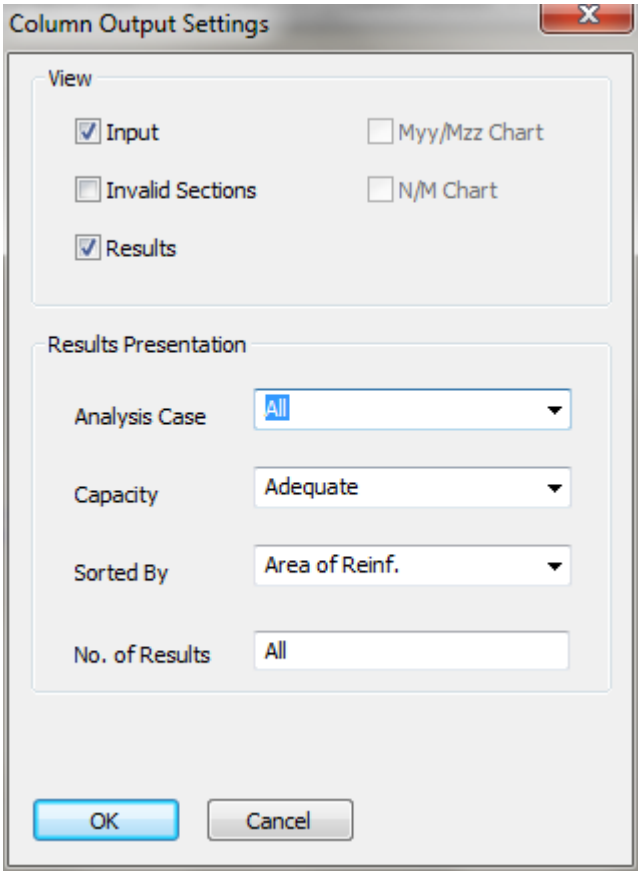
- The following Graphical View will appear.

  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 
- Choose the appropriate section to plot and view the results, noting the load case within the envelope
- Click on the *Myy/Mzz Chart* Icon 
- 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/m3

Angle of internal friction = 35 degrees

SPT N = 25

Based on SPT, following values are assumed:

End bearing stress, qb = 1370 kPa

Skin friction stress, qs = 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 Gk = 1200 kN

Characteristic variable load Qk = 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

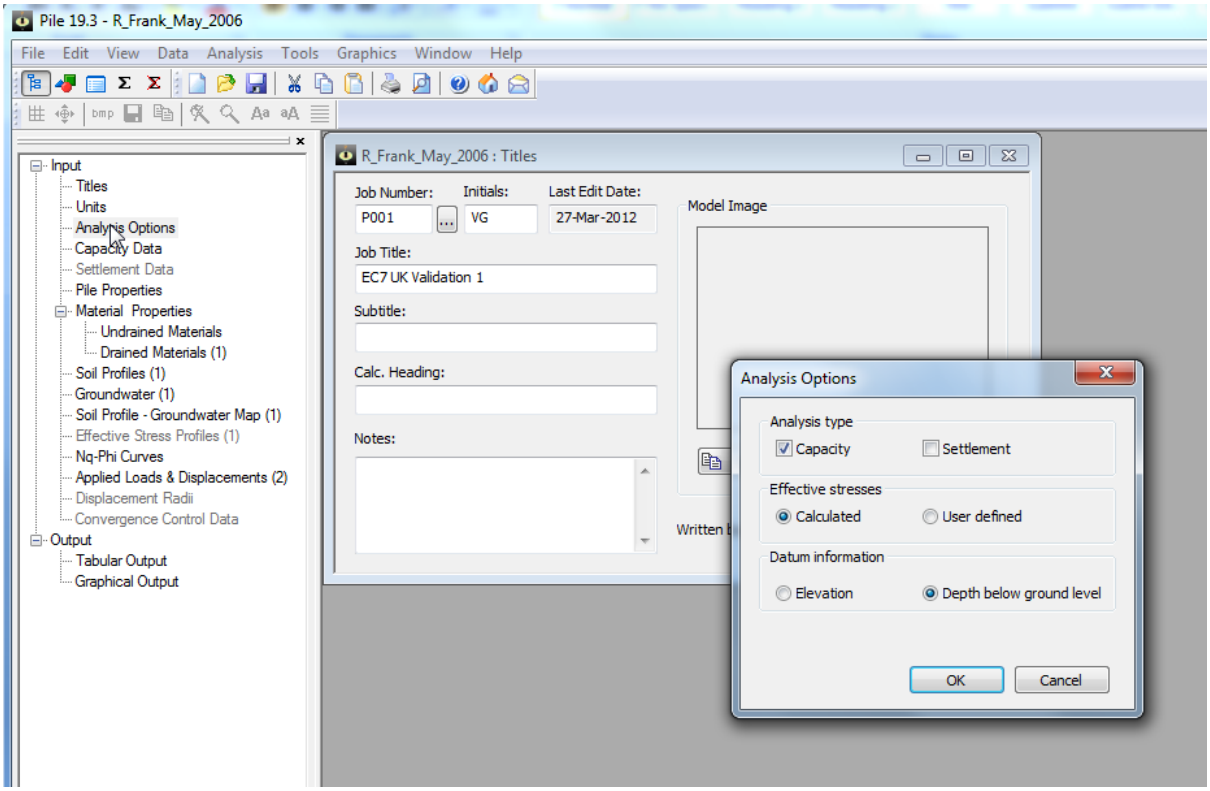
7 Soil –profile Groundwater Map

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.

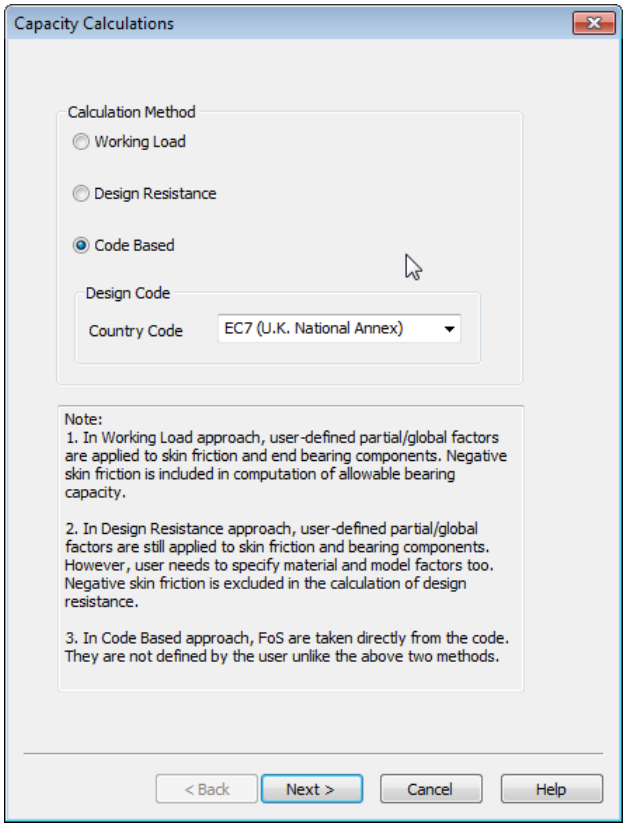
Datum information:

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

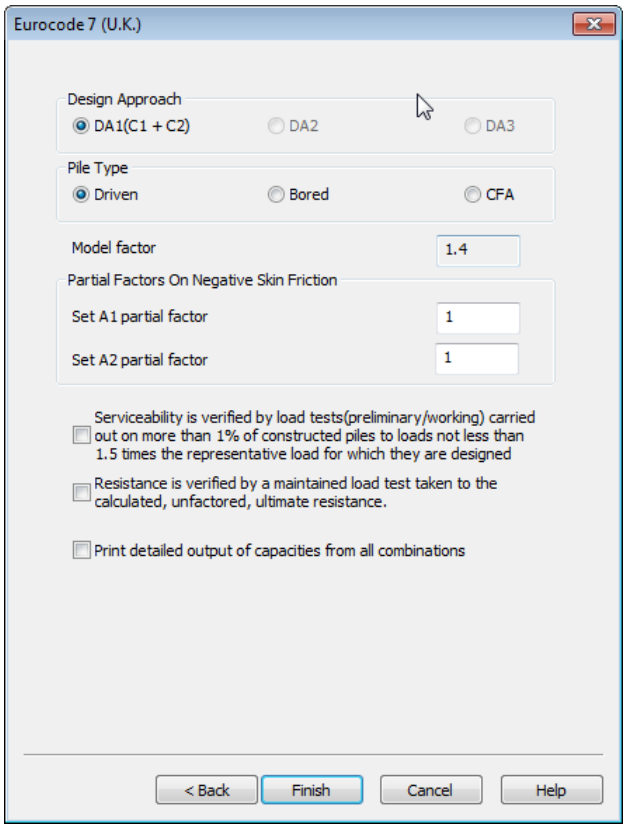
Capacity Data:

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.



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



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.

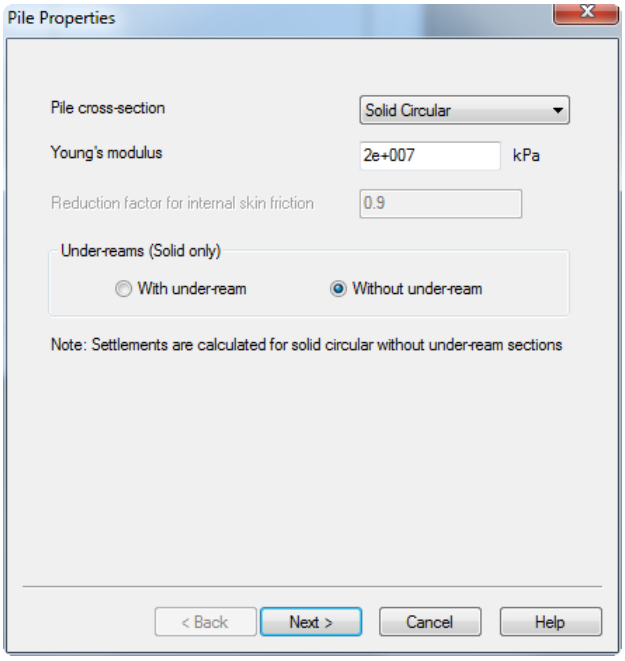
The A1 and A2 partial factors on negative skin friction would be used for calculating the factored load due to negative skin friction, which is treated as an unfavourable action. The users may refer to A.3.1. section in UK national annex for guidance on these factors. These values can be ignored if there 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:



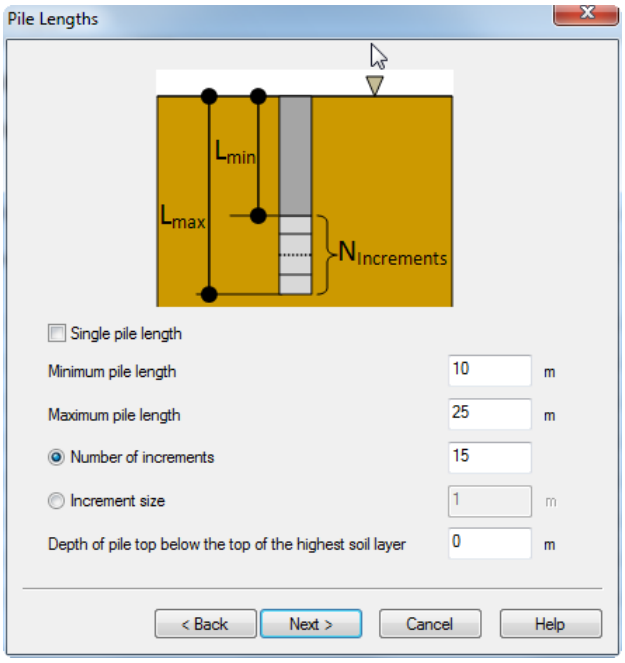
For the given model, “Solid circular” and “Without under-ream” options need to be selected.

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

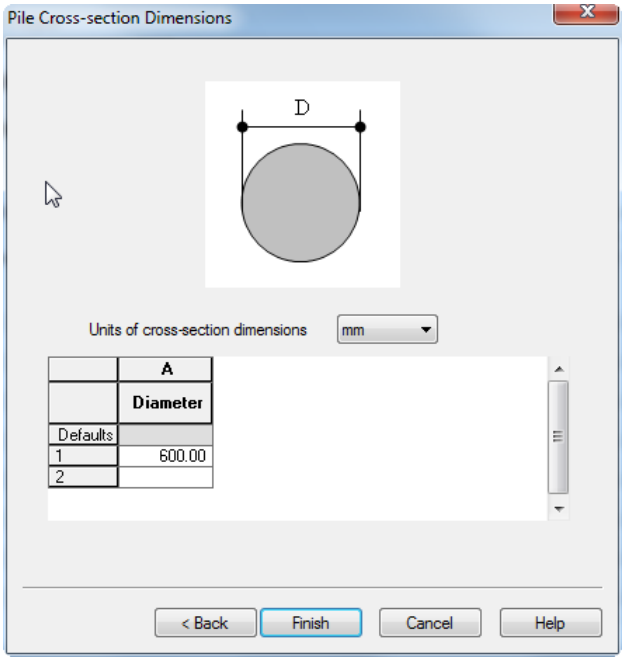
Once this data is entered, the user can specify the range of pile lengths for which capacity calculations need to be carried out.

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.



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



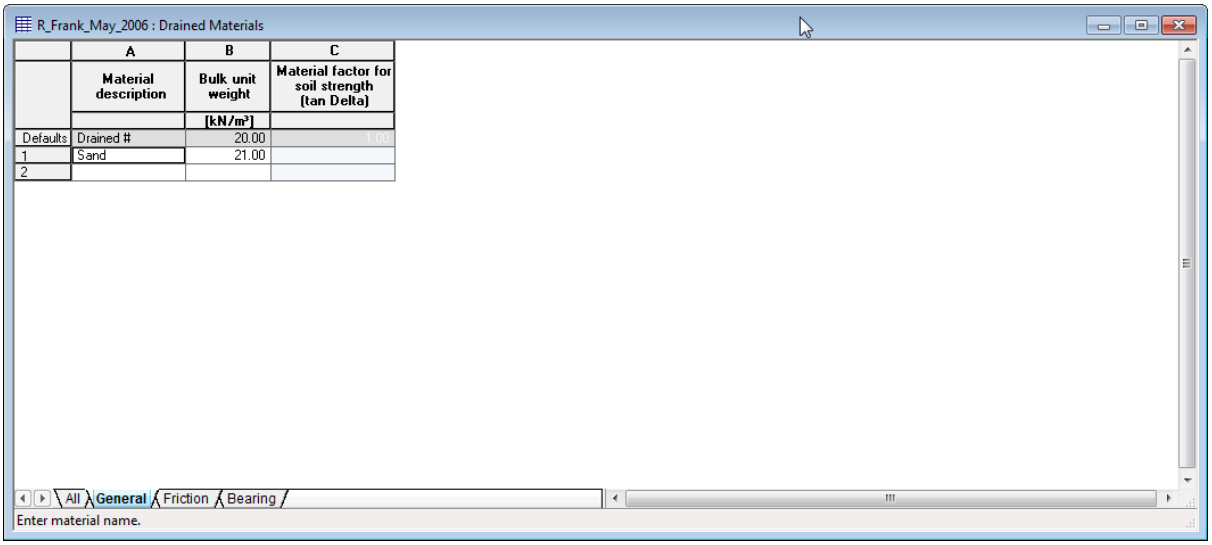
In this model, we are interested in the capacity of only 600 mm diameter pile. Hence, only one entry is given.

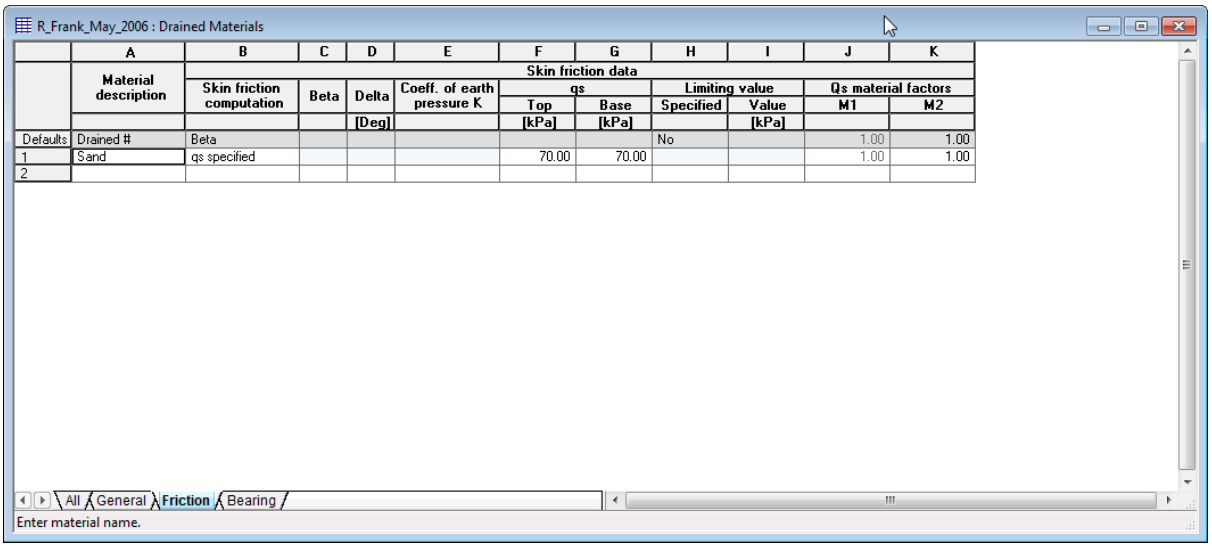
Clicking on “Finish” closes the wizard and saves the “Pile Properties” data.

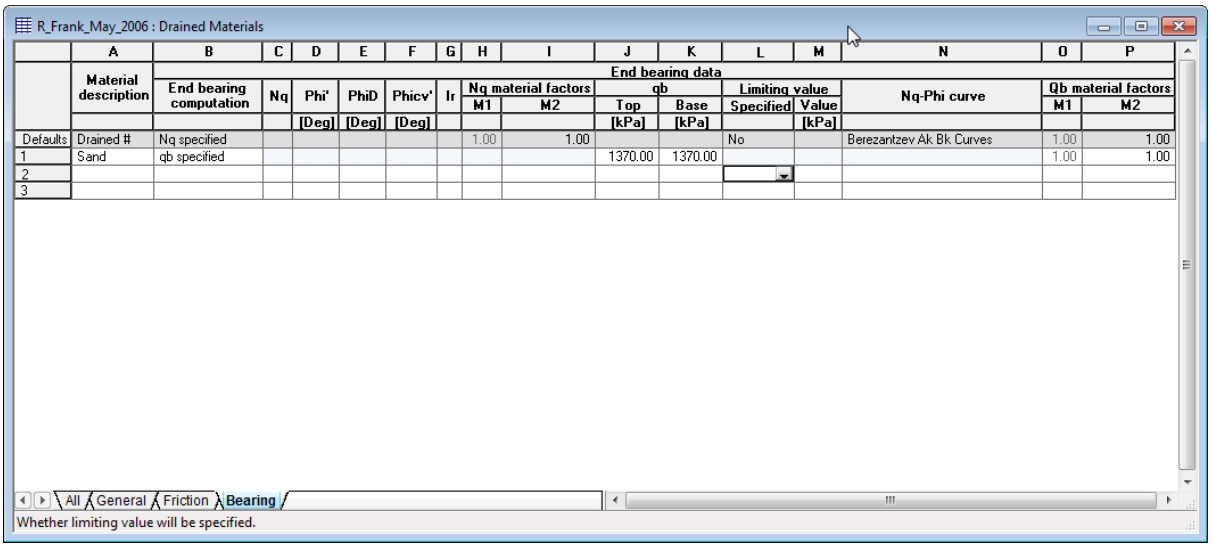
Material Properties – Drained materials:

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.







	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
		End bearing data														
	Material description	End bearing computation	Nq	Phi'	PhiD	Phicv'	Ir	Nq material factors		qb		Limiting value		Nq-Phi curve	Qb material factors	
				[Deg]	[Deg]	[Deg]		M1	M2	Top [kPa]	Base [kPa]	Specified	Value [kPa]		M1	M2
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.

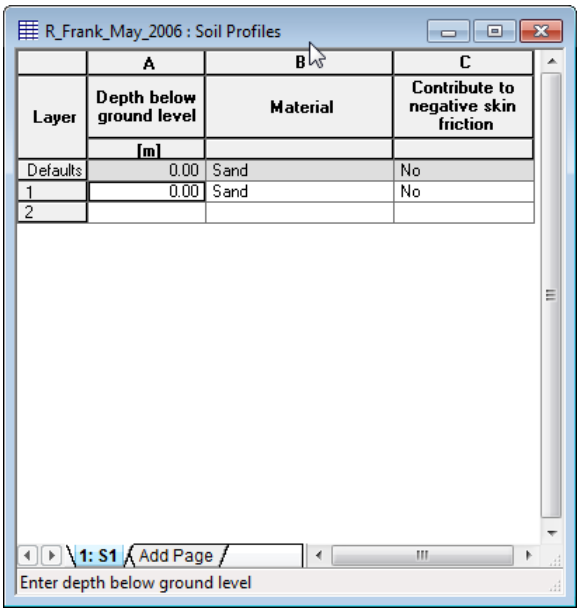
For this example, however, as there is no negative skin friction involved, these factors can be left at the default value of 1.0.

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”.

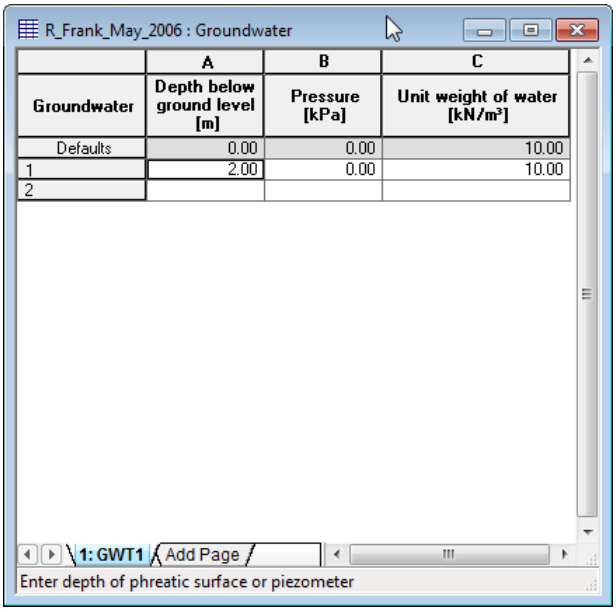
However, for EC7 (U.K.) national annex, the program only uses the “Alternative procedure”. Hence, the user would be entering only one soil profile “S1” as shown:



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.

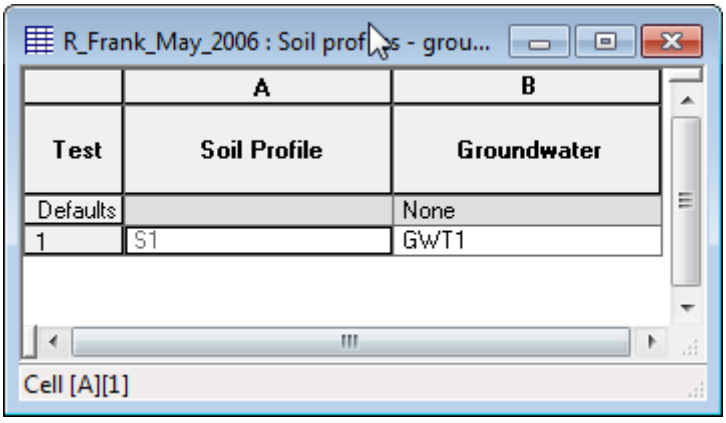


In this particular example, there is only one soil profile and one ground water table. The same is entered as shown.

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.

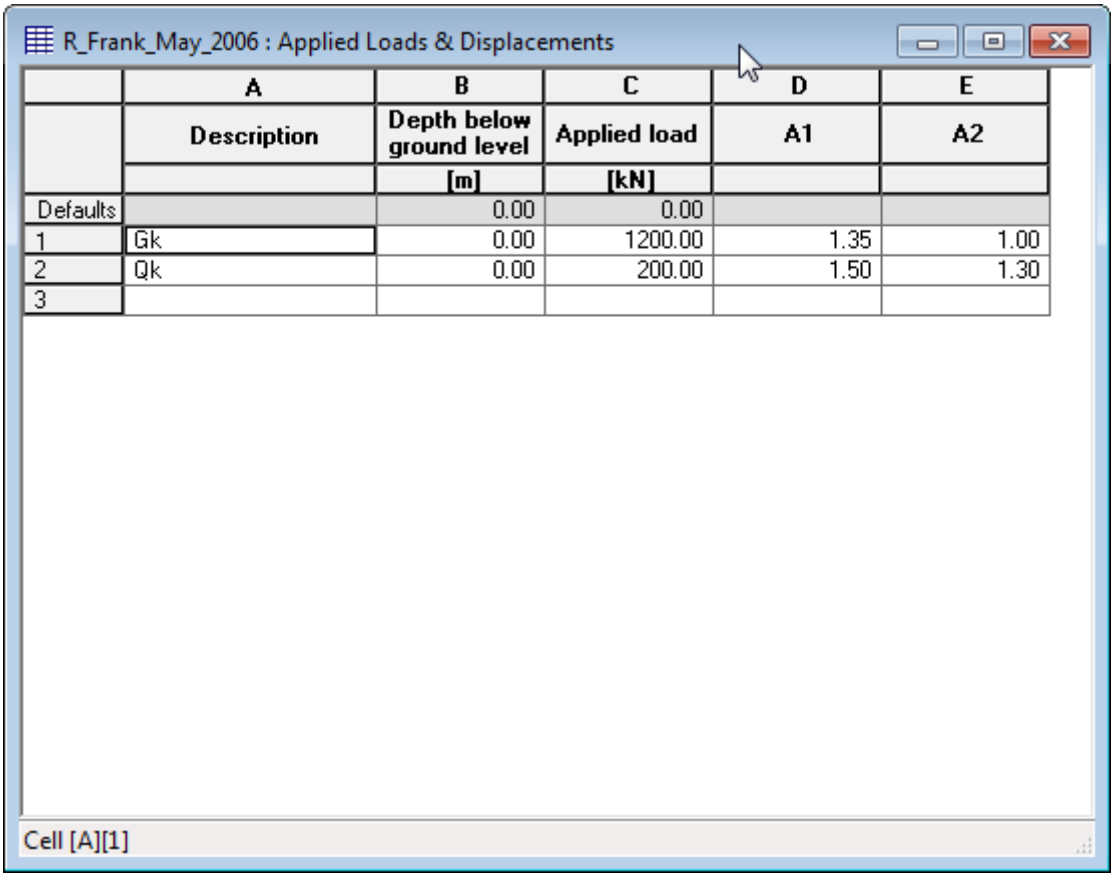


IMPORTANT: By default, no groundwater table is associated to the soil profiles created by the user. Hence, it is important to explicitly enter the data in this table when groundwater is present in a soil profile.

Applied Loads & Displacements:

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

The following load data needs to be entered as shown:



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

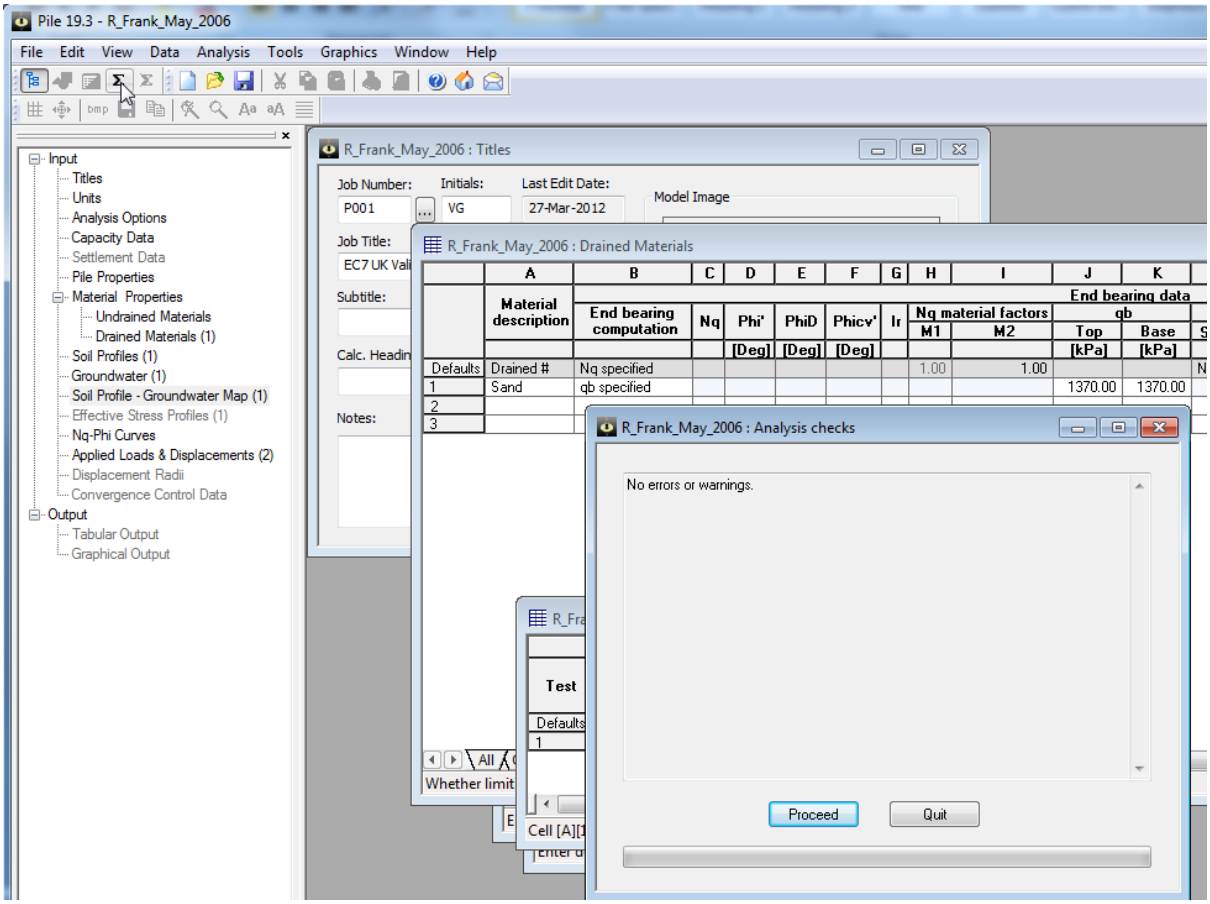
Action factor set A2:

Dead load factor = 1.00

Live load factor = 1.30

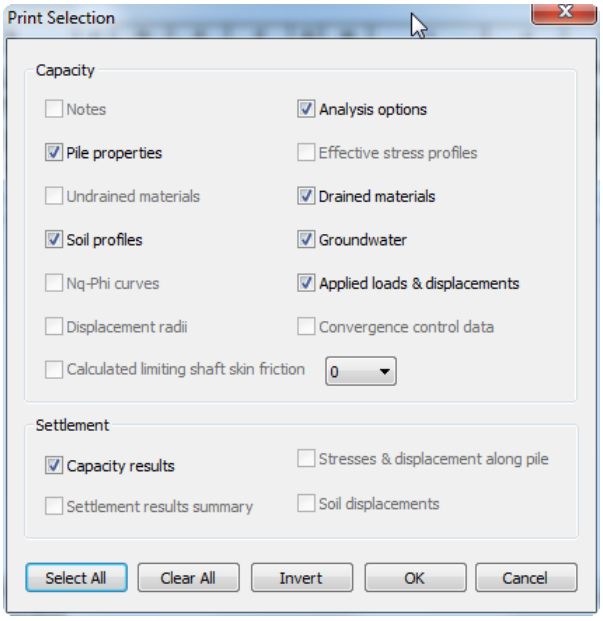
Analysis & Results:

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

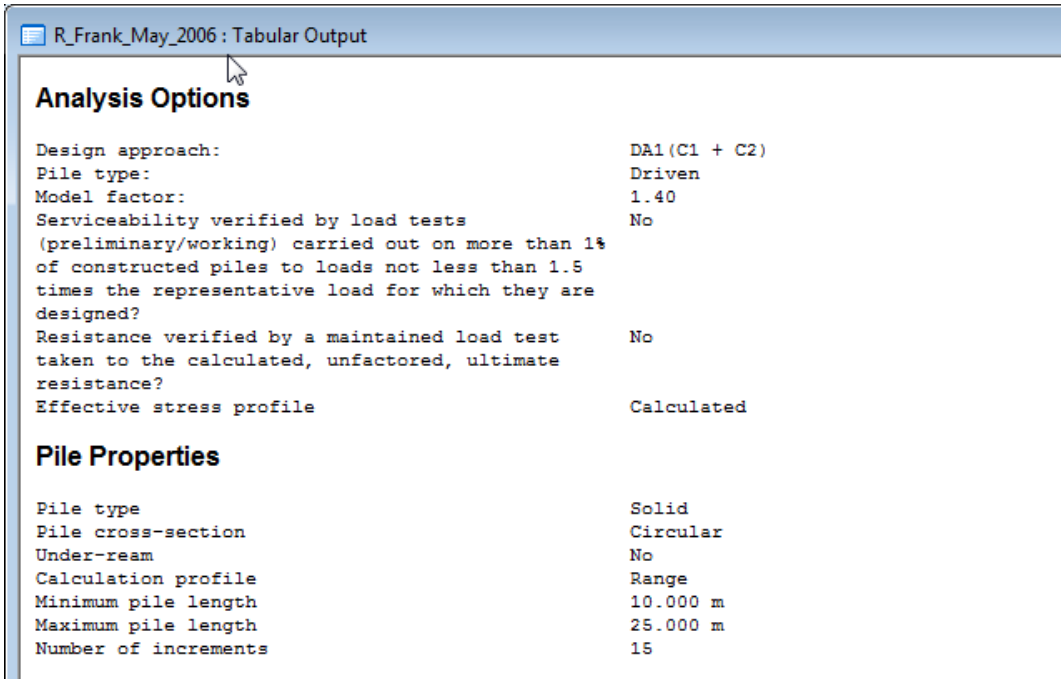


This brings up the Pre-Analysis check dialog giving any warnings or errors in the model. If there are no errors, the user can run the analysis by clicking the "Proceed" button.

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:



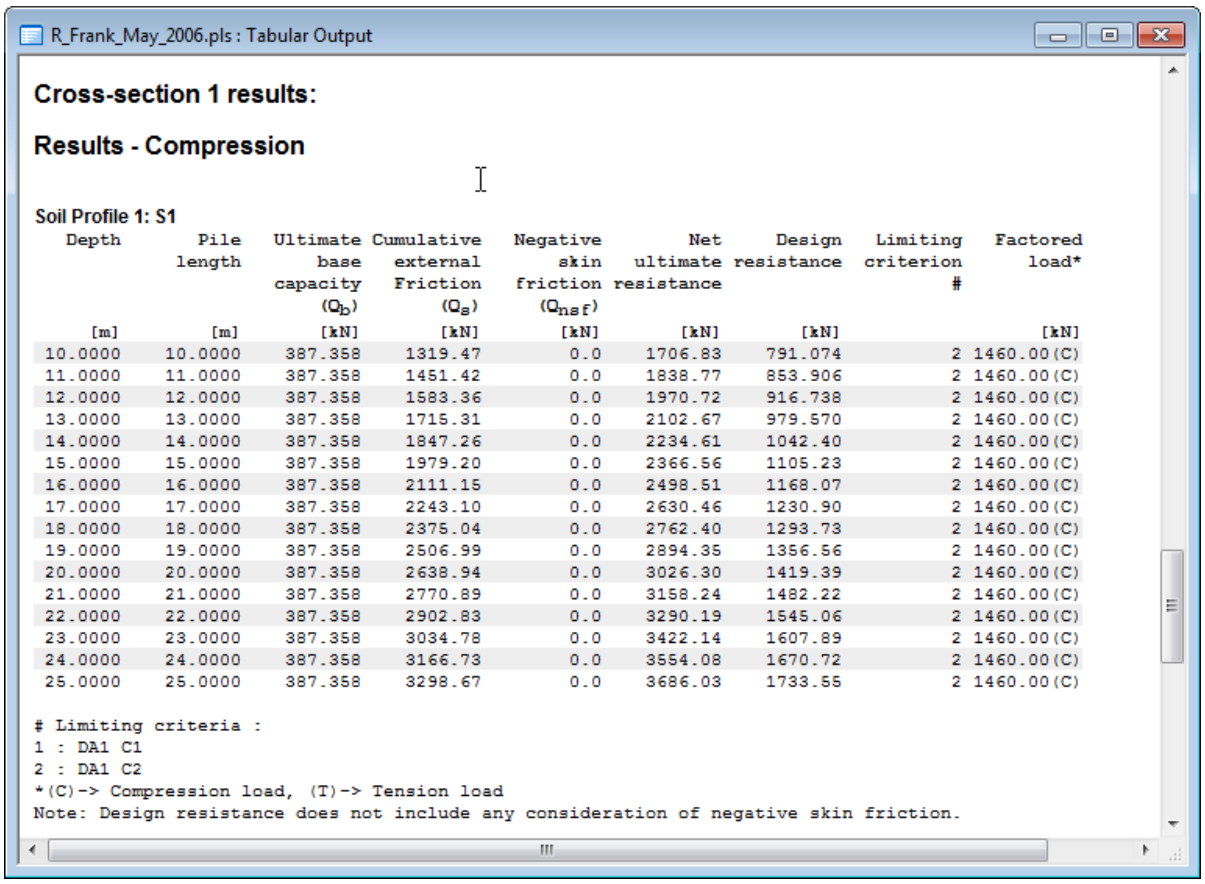
R_Frank_May_2006 : Tabular Output

Analysis Options

Design approach:	DA1(C1 + C2)
Pile type:	Driven
Model factor:	1.40
Serviceability 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?	No
Resistance verified by a maintained load test taken to the calculated, unfactored, ultimate resistance?	No
Effective stress profile	Calculated

Pile Properties

Pile type	Solid
Pile cross-section	Circular
Under-ream	No
Calculation profile	Range
Minimum pile length	10.000 m
Maximum pile length	25.000 m
Number of increments	15



R_Frank_May_2006.pls : Tabular Output

Cross-section 1 results:

Results - Compression

Soil Profile 1: S1

Depth	Pile length	Ultimate base capacity (Q_b)	Cumulative external Friction (Q_s)	Negative skin friction (Q_{nsf})	Net ultimate resistance	Design resistance	Limiting criterion #	Factored load*
[m]	[m]	[kN]	[kN]	[kN]	[kN]	[kN]		[kN]
10.0000	10.0000	387.358	1319.47	0.0	1706.83	791.074	2	1460.00(C)
11.0000	11.0000	387.358	1451.42	0.0	1838.77	853.906	2	1460.00(C)
12.0000	12.0000	387.358	1583.36	0.0	1970.72	916.738	2	1460.00(C)
13.0000	13.0000	387.358	1715.31	0.0	2102.67	979.570	2	1460.00(C)
14.0000	14.0000	387.358	1847.26	0.0	2234.61	1042.40	2	1460.00(C)
15.0000	15.0000	387.358	1979.20	0.0	2366.56	1105.23	2	1460.00(C)
16.0000	16.0000	387.358	2111.15	0.0	2498.51	1168.07	2	1460.00(C)
17.0000	17.0000	387.358	2243.10	0.0	2630.46	1230.90	2	1460.00(C)
18.0000	18.0000	387.358	2375.04	0.0	2762.40	1293.73	2	1460.00(C)
19.0000	19.0000	387.358	2506.99	0.0	2894.35	1356.56	2	1460.00(C)
20.0000	20.0000	387.358	2638.94	0.0	3026.30	1419.39	2	1460.00(C)
21.0000	21.0000	387.358	2770.89	0.0	3158.24	1482.22	2	1460.00(C)
22.0000	22.0000	387.358	2902.83	0.0	3290.19	1545.06	2	1460.00(C)
23.0000	23.0000	387.358	3034.78	0.0	3422.14	1607.89	2	1460.00(C)
24.0000	24.0000	387.358	3166.73	0.0	3554.08	1670.72	2	1460.00(C)
25.0000	25.0000	387.358	3298.67	0.0	3686.03	1733.55	2	1460.00(C)

Limiting criteria :
1 : DA1 C1
2 : DA1 C2
*(C)-> Compression load, (T)-> Tension load
Note: Design resistance does not include any consideration of negative skin friction.

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 m^2

Ultimate end bearing capacity = 1370*0.283 = 387.36 kN

Unit skin friction = 70kPa

Length of pile = 25 m

Ultimate skin friction capacity = PI*0.6*70*25 = 3298.67 kN

DA1 C1

"A1 + M1 + R1"

Action set A1:

Factored load = 1.35*1200 + 1.5*200 = 1920 kPa

Partial factor combination:

Design resistance = ((Ultimate shaft resistance/shaft factor) +

(Ultimate base resistance/base factor))/model factor

= (3298.67/1.0 + 387.6/1.0)/1.4 = 2632.88 kPa

DA1 C2

"A2 + M1 + R4"

Action set A1:

Factored load = 1.00*1200 + 1.3*200 = **1460 kPa**

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

Partial factor combination:

Design resistance (Partial Factors) = ((Ultimate shaft resistance/shaft factor) +

(Ultimate base resistance/base factor))/model factor

= (3298.67/1.5 + 387.6/1.7)/1.4 = **1733.65 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.