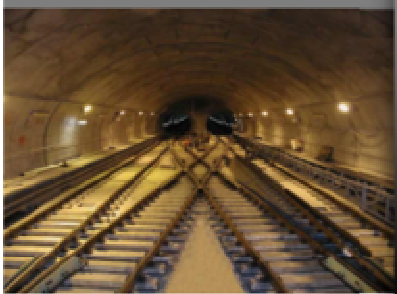
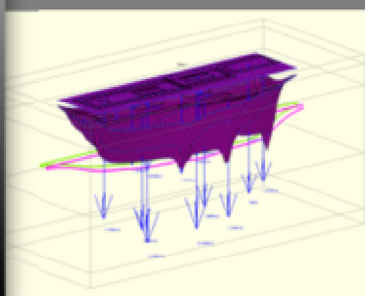
Oasys

The software house of Arup

PDisp Tutorial Manual





Oasys Pdisp

Printed: February 2013

Contents

1. Introduction p1
2. Pdisp Analysis Methodology p2
3. Pdisp Program Features p6
4. Components of the Pdisp Interface p7
5. Loaded Soil Tutorial p8
6. Graphical Input Tutorial p31

1. Introduction

Pdisp (Pressure Induced Displacement Analysis) calculates the displacements (and stresses if required) within a linear elastic or non-linear soil mass, arising from uniform normal or tangential pressure, applied to rectangular and circular loaded planes. The program is ideal for predicting the displacements that may arise due to the action of several loads in a soil mass.

The outputs enable the user to study the settlements and stresses within the soils. The calculation itself is based on verified and robust numerical methods and the simplicity of design enables new users to set up and run an analysis easily

The tutorial examples deal with practical loading applications but this Tutorial Manual is intended to familiarise the user with Pdisp. The examples should therefore not be used as a basis for practical projects.

Users are expected to have a basic understanding of soil mechanics and geotechnical theory, and should be able to work in a Windows environment. The tutorial lessons are also available in the examples folder and can be used to check your results.

It is important to realise that Pdisp is an advanced program analysing a complex problem and the user must be fully aware of the various methods of analysis, requirements and limitations discussed in the User Manual before use. The Tutorial Manual will not provide theoretical background information on the methods, nor does it explain the details of various methods of analysis available in the program. These details can be found in the User Manual for Pdisp. 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.

2. Pdisp Analysis Methodology

Two methods of analysis are available in Pdisp:

- Boussinesq
- Mindlin

It is important that the user is familiar with both methods and their suitability to different types of analysis. The selection of analysis method also influences inputs such as the number of displacement points and the final outputs the user obtains from their analysis.

Consequently, before using the program, read Sections 2.1 to 2.3 of the user manual to enable to you answer the questions below.

Space has been left for you to write your answers for future reference.

2.1 Boussinesq Analysis

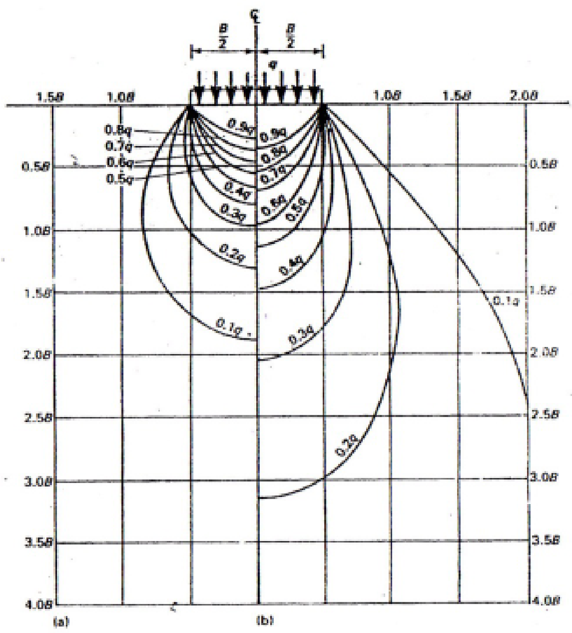


Fig. (6) : Pressure bulbs for vertical stress
(a) Circular foundation (b) Strip foundation

1. Stresses and strains are calculated in Pdisp using Boussinesq equations. How are displacements derived from these values?

2. What impact does decreasing the displacement intervals in a soil-strata have on the accuracy of the analysis? Why are displacement intervals important for the Boussinesq method?

3. To calculate settlements and stresses below a small loaded area carrying a significant load accurately, what is the best vertical spacing of displacement levels close the load?

4. If you apply a larger maximum allowable ratio between values of E at adjacent displacement levels, how does this affect the accuracy of the Boussinesq analysis?

5. For loads below the ground level in which there is a sudden change in stress, how would you improve the accuracy of the Boussinesq solution?

6. When selecting Global Poisson's ratio for materials with very different elasticity, should you choose the higher or lower Poisson's ratio and why?

2.2 Mindlin Analysis

1. When starting a new analysis, would you use the Legacy Mindlin Method? Why?

2. a. How would you model displacements above a load where there is a significant difference in stiffness between the soils below and above the load?

   b. Which heave correction factor should be used for a new analysis?

2.3 General

1. Which analysis should be used if you are interested in horizontal displacements and why?

2. Which analysis should be used if you need to calculate stresses and displacements in the soil?

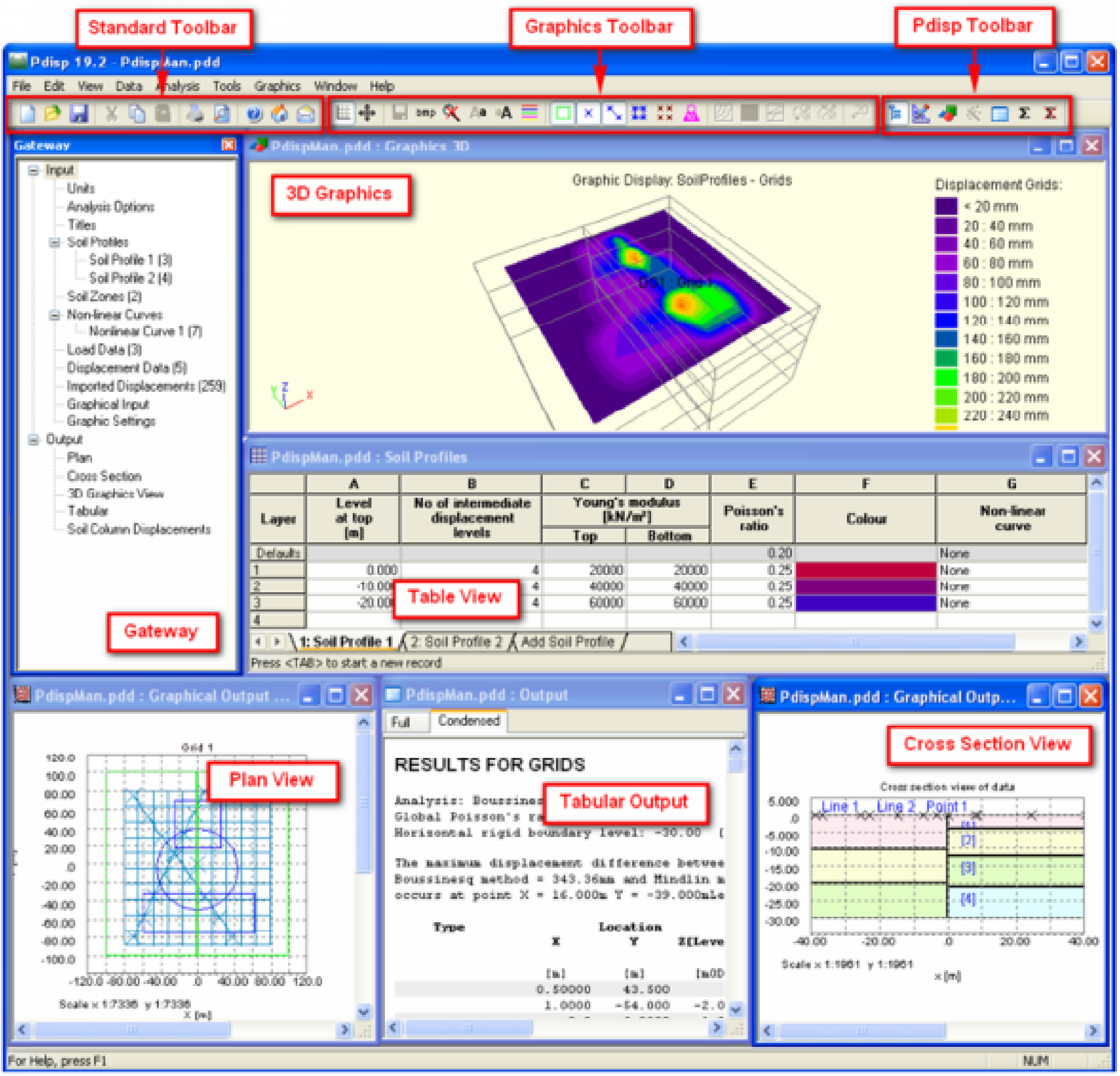
3. Which analysis is best if you are interested in accurately representing settlements above a load?

3. Pdisp Program Features

The main features of **Pdisp** are summarised below.

- **Displacements** are calculated using a profile of Young's modulus with depth. The modulus can be constant or vary linearly with depth for each soil layer. Non-linear stress/strain curves may also be specified.
- **Displacements and stresses** can be calculated at several levels and at any location in plan. The location of the displacement/stress points is defined by means of two-dimensional orthogonal grids, by lines or by points. The results from the grids can be displayed graphically as **contours**.
- **Loaded areas** can be defined as rectangles or circles. Pressures are applied as vertical and/ or horizontal uniformly distributed loads to rectangular and/or circular loaded areas. Areas can be superimposed and placed at any level. All loads are completely flexible i.e. no allowance is made for stiffness of the structures.
- The base of the model is defined by the specification of an equivalent **rigid boundary.**
- The **ground** is modelled using a series of vertical **soil profiles** each consisting of a number of horizontal soil layers. The plan distribution of the soil profiles is specified in rectangular areas known as **soil zones**. These can be superimposed allowing the development of complicated ground models.

4. Components of the Pdisp Interface



5. Loaded Soil Tutorial

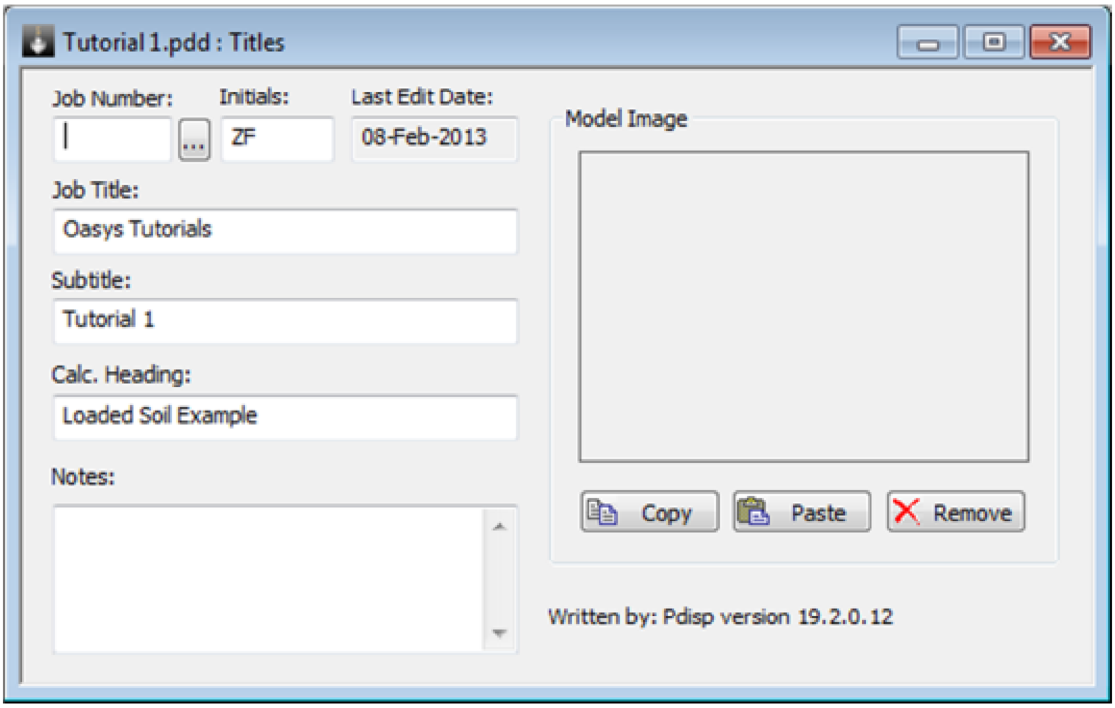
By the end of the session the user should be able to:

- Navigate the Pdisp Interface
- Input load data
- Input displacement data
- Input soil data, including non-linear curves
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs
- Export tabular outputs for further analysis
- Export contours for dxf drawings

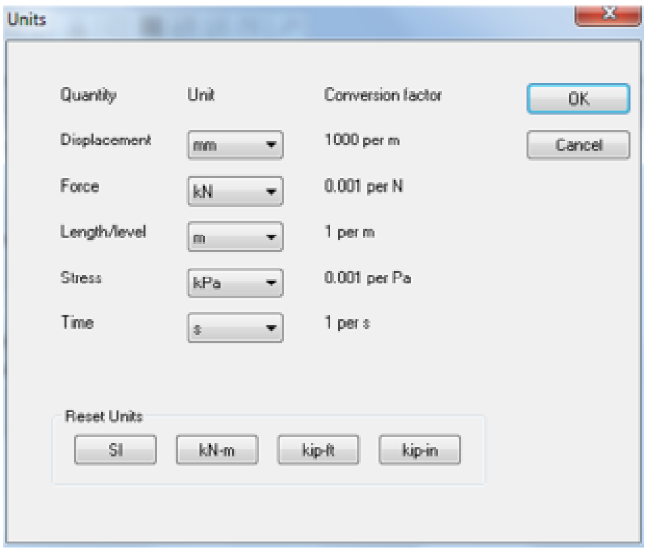
5.1 Creating the Input

Once opened, create a new file by clicking the 'New File' icon on the top left of the program or clicking Ctrl + N

1. Double click on *Titles* in the Gateway and fill in the dialog box:

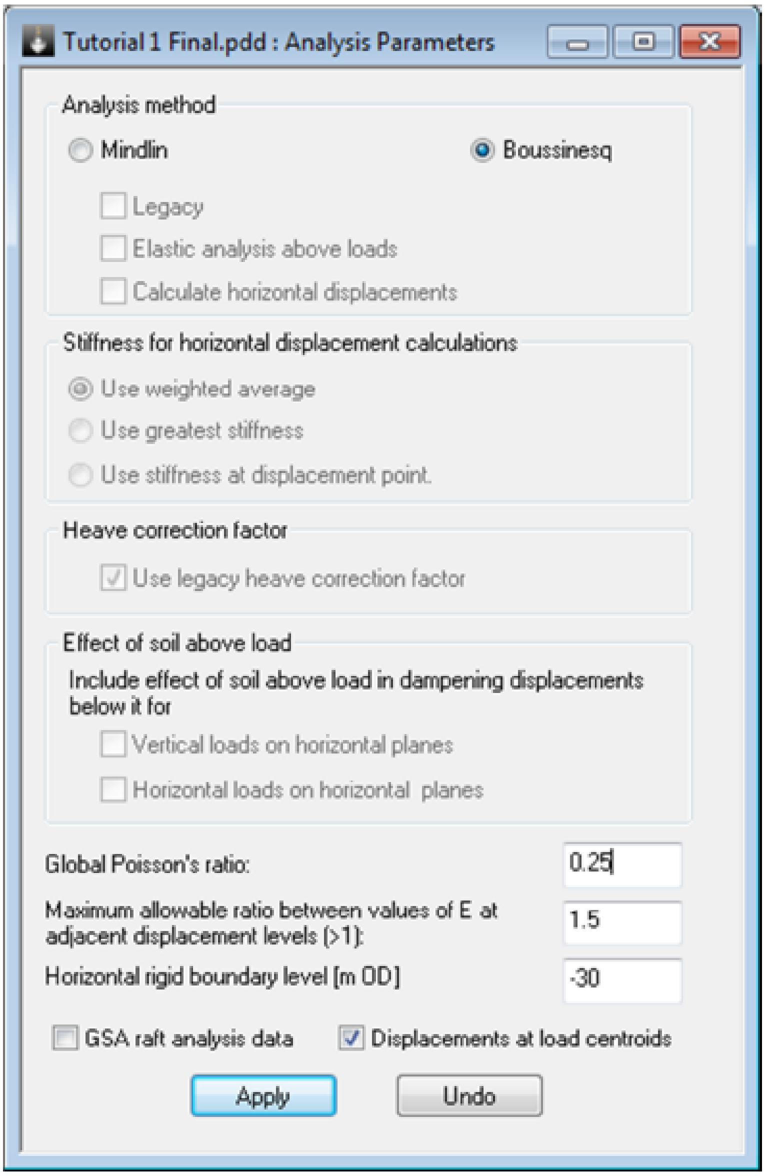


(Hint: Should the user need to change the Units, they should do so at this point by double clicking on *Units* in the Gateway)



2. Double click on *Analysis Options* in the Gateway.

   Select the following:



(Note: The horizontal rigid boundary level sets the base of the analysis as where there is no settlement. This is often specified as the level of bedrock)

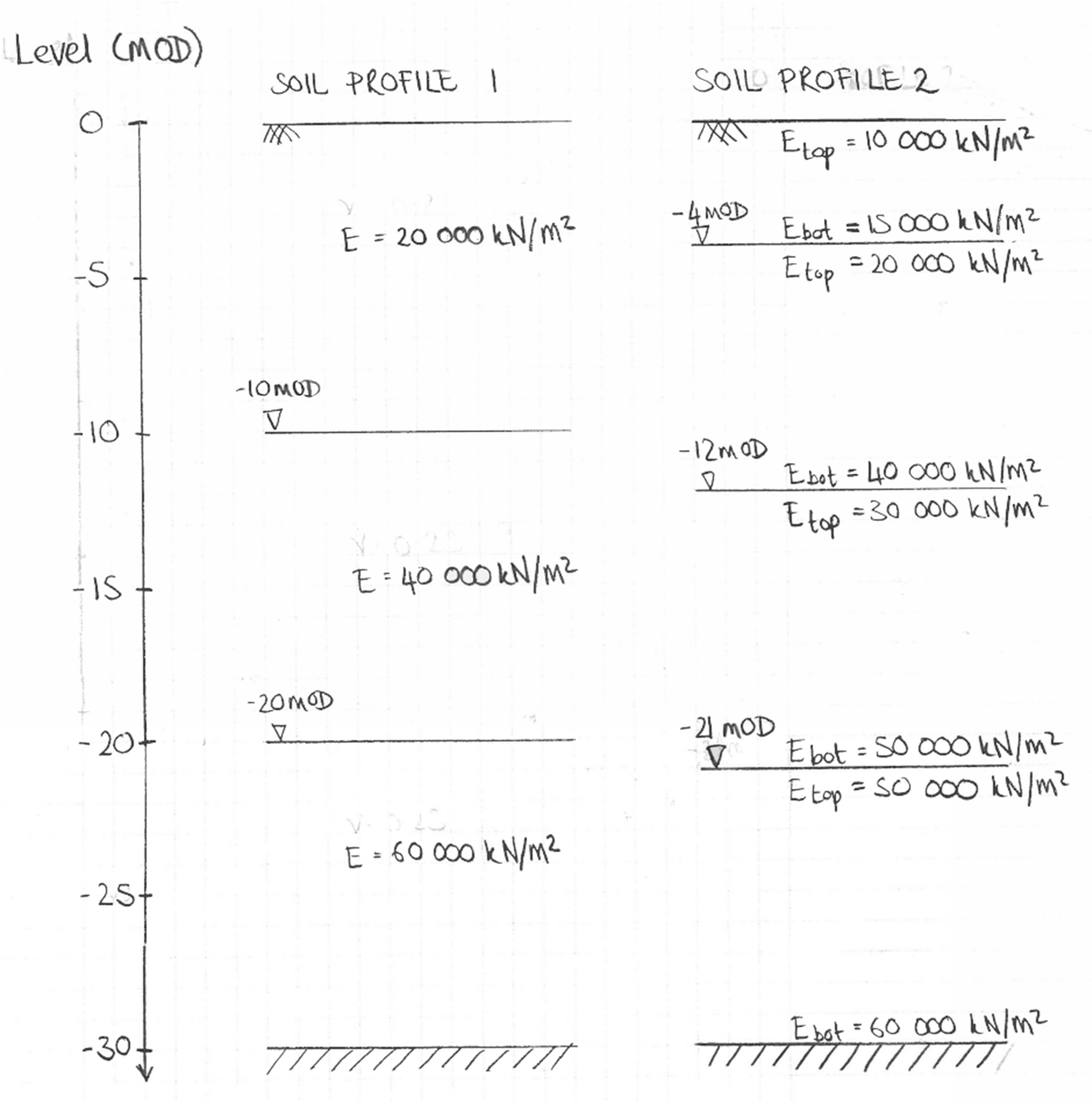
3. Double click on *Soil Profiles* in the Gateway

   Enter the data shown in the diagram on the following page for Soil Profiles 1 and 2.
   New soil profiles can be entered using the *Add Soil Profile* tab at the bottom of the table.

   Poisson's ratio is 0.25 for all the soil strata.

   For each soil strata, specify displacement calculation points at every 0.5m using appropriate values for the *Number of intermediate displacement levels*.

   (Hint: Use section 3.2.3 of the User Manual to help you understand the required inputs)



An example for Soil Profile 1 is shown below:

Tutorial 1 Final.pdd : Soil Profiles

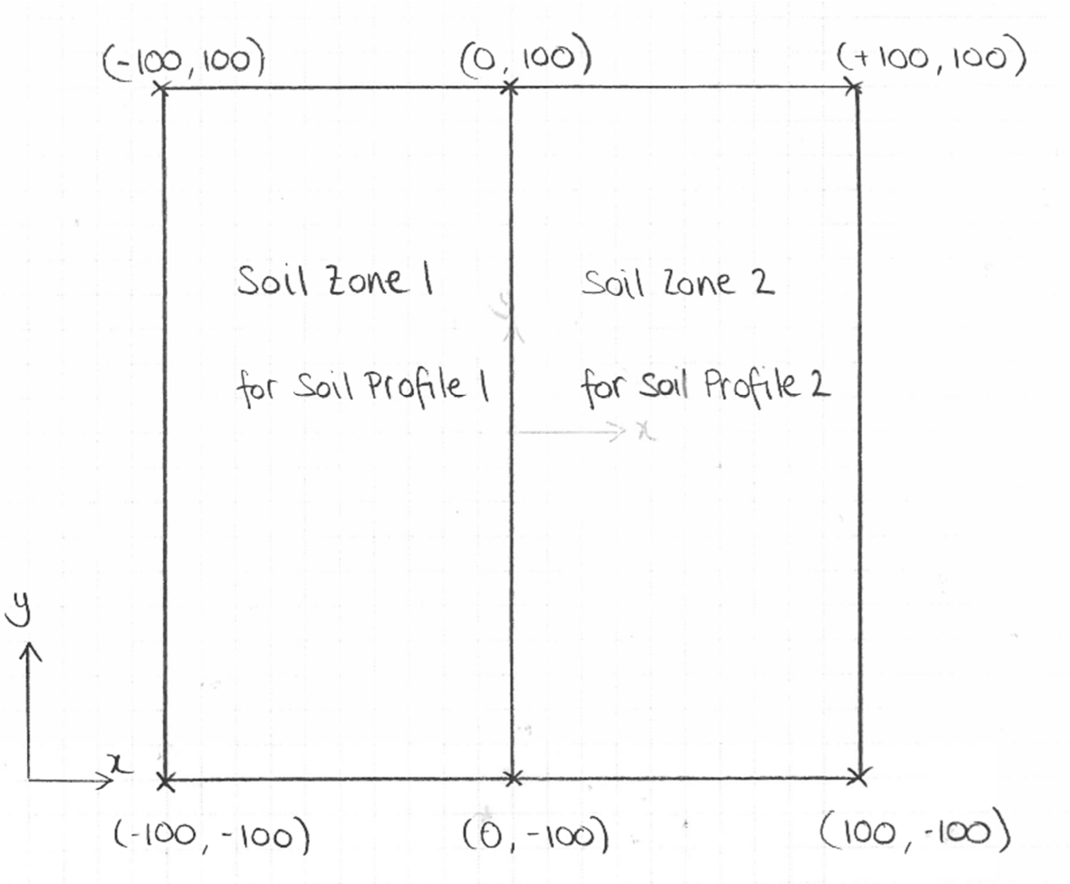
	A	B	C	D	E	F	G
Layer	Level at top [m]	No of intermediate displacement levels	Young's modulus [kN/m²] Top	Bottom	Poisson's ratio	Colour	Non-linear curve
Defaults					0.20		None
1	0.000	21	20000	20000	0.25		None
2	-10.000	21	40000	40000	0.25		None
3	-20.000	21	60000	60000	0.25		None
4							

1: Soil Profile 1 | 2: Soil Profile 2 | Add Soil Profile

Press <TAB> to start a new record

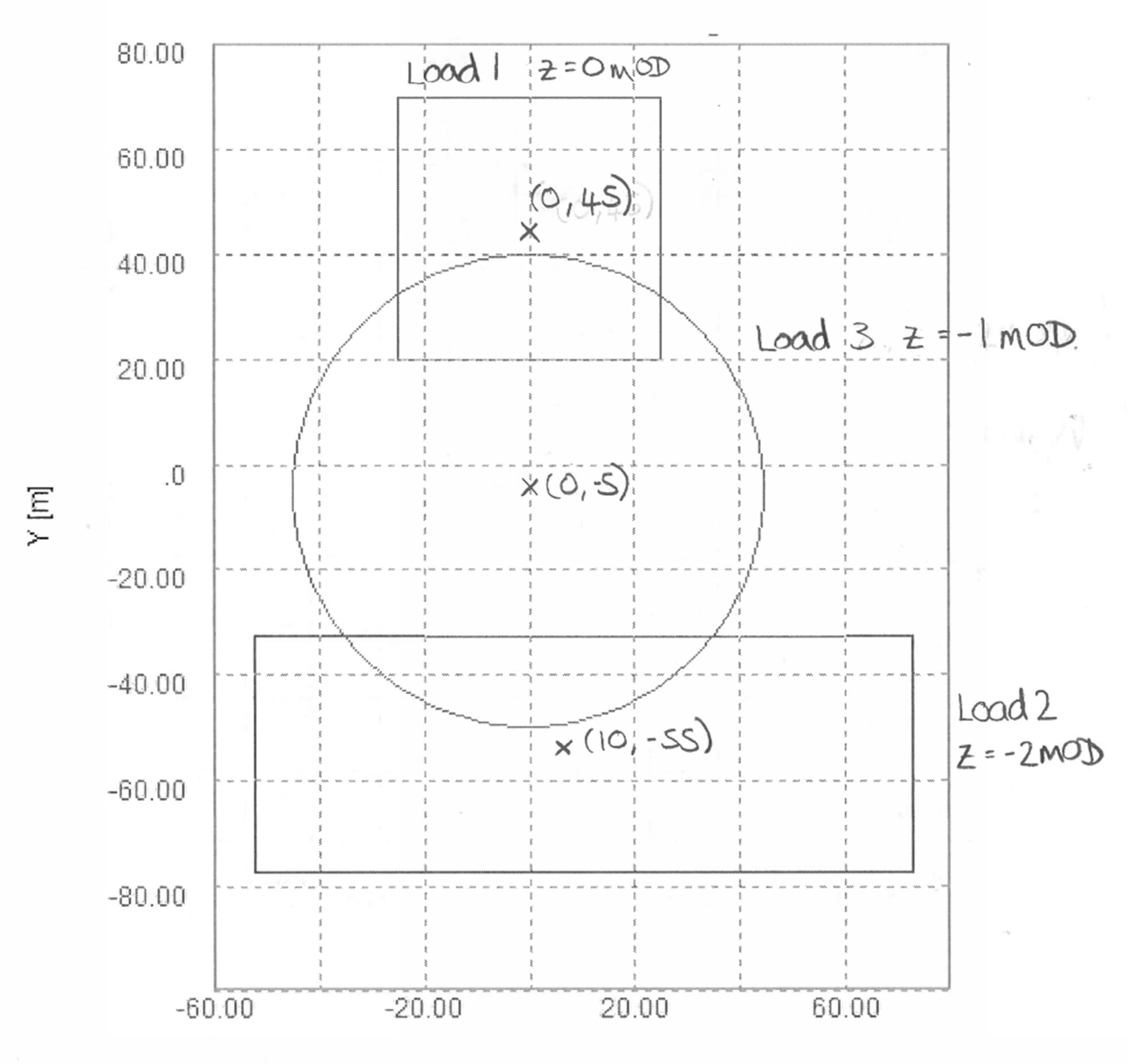
4. Double click on *Soil Zones* in the Gateway

   Enter data for the soil zones shown in the diagram below for Soil Zones 1 and 2.



5. Double click on *Load Data* on the Gateway.

   Enter the loads shown in the diagram below.



The loads are:

- Load 1 = 60 kN/m2
- Load 2 = 70 kN/m2
- Load 3 = 50 kN/m2

Circle radius = 45m
Represent the circular load with 7 rectangles.
An odd number of rectangles is always specified due to the geometry of representing a circle using rectangles.

(Hint: Check the input is correct using the Graphical Input window, accessible through the Gateway)

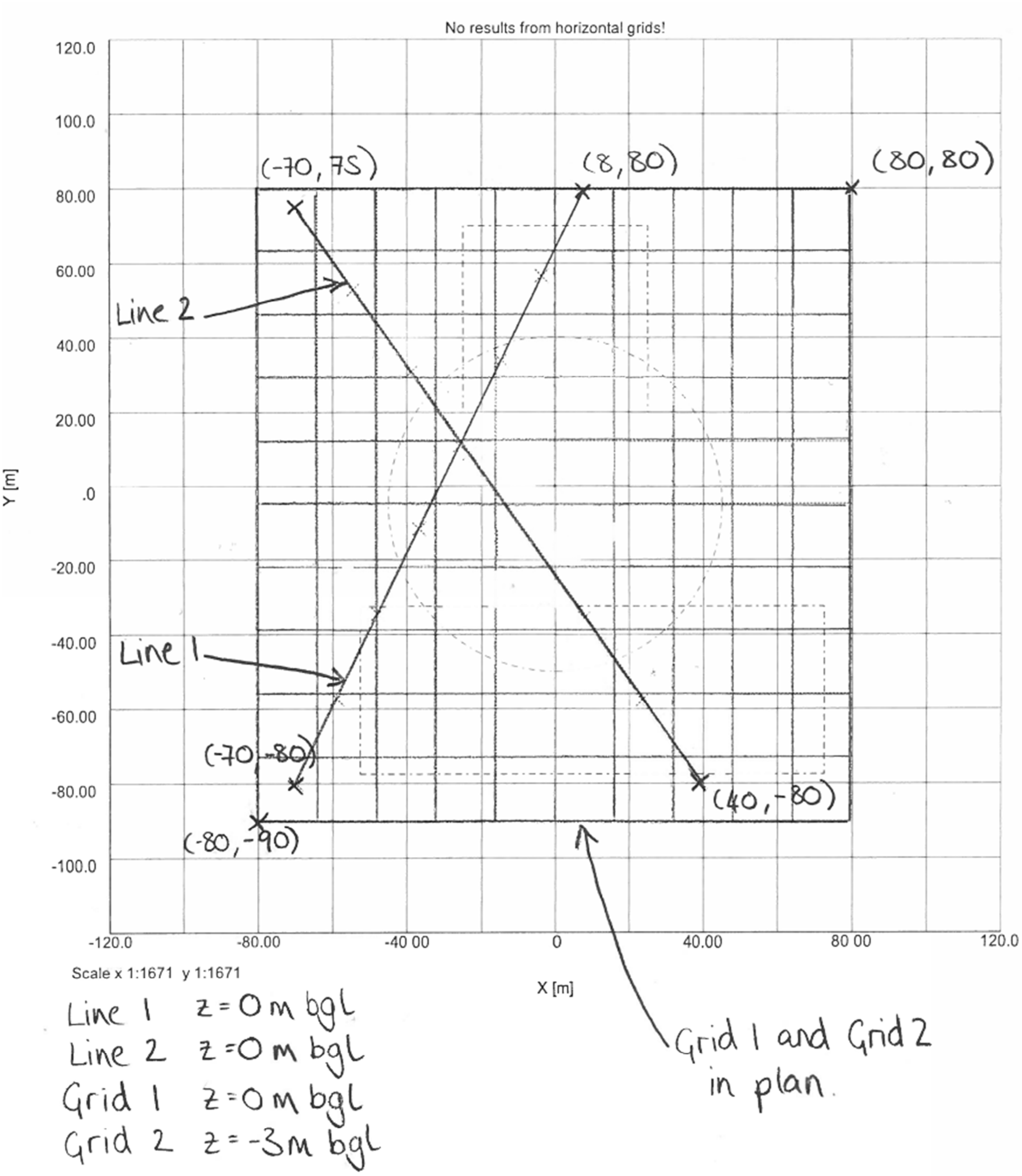
6. Double click on *Displacement Data* in the Gateway.

   Enter the displacement lines and grid shown in the diagram below.

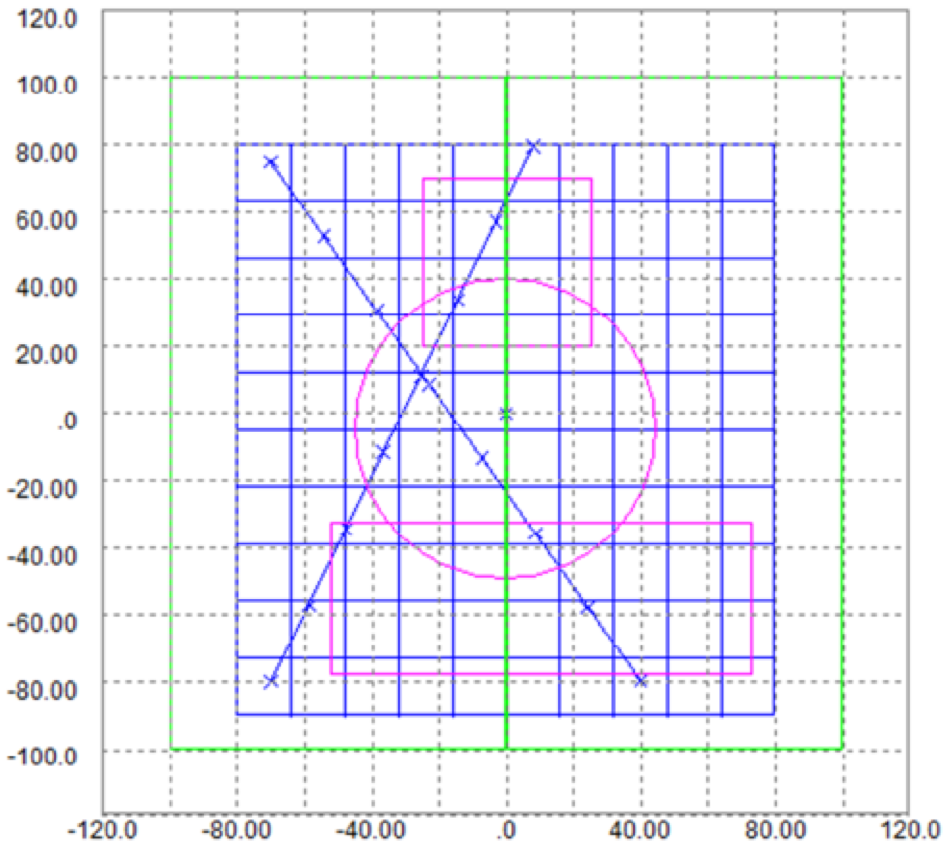
   The grids have 10 extrusion intervals and the lines have 7 extrusion intervals.

   Try inputting the Grids in the Global X and Global Y direction to better understand the process.

   (Hint: Use Section 3.2.6 of the User Manual to assist you with the inputs)



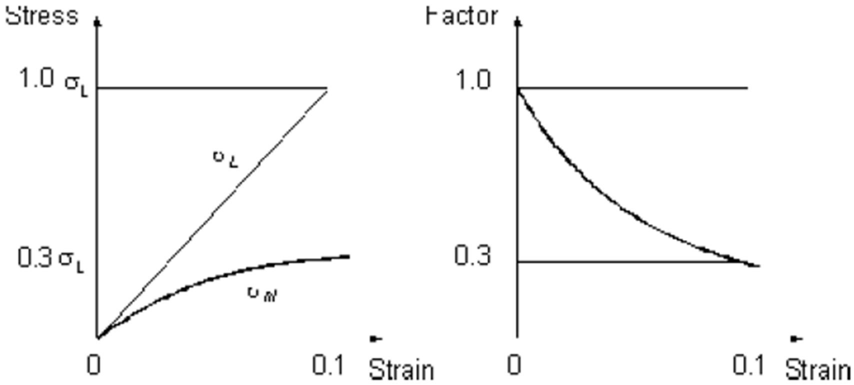
7. Double click on *Graphical Input* in the Gateway to check the load, displacement grid and displacement line geometry.



5.1.1 Non-linear curves

1. Using the following description, derive the non-linear curve from the data below.

Strain (%)	Non Linear Stress (kPa)	Equivalent Non Linear Stress (kPa)
0.0247	0	0
0.0798	0.000822126	0.003546703
0.8348	0.723993197	3.735775008
22.88	72.89719626	421.3710767
94.16	300	1734.104046



For increasing values of strain, the factor (F) by which Young's modulus is to be modified, is defined by:

$$\frac{\sigma_{nl}}{\sigma_L}$$

where σL is the equivalent stress for a linear curve at a given strain (ε) and σ_{nl} is the stress for the non-linear curve.

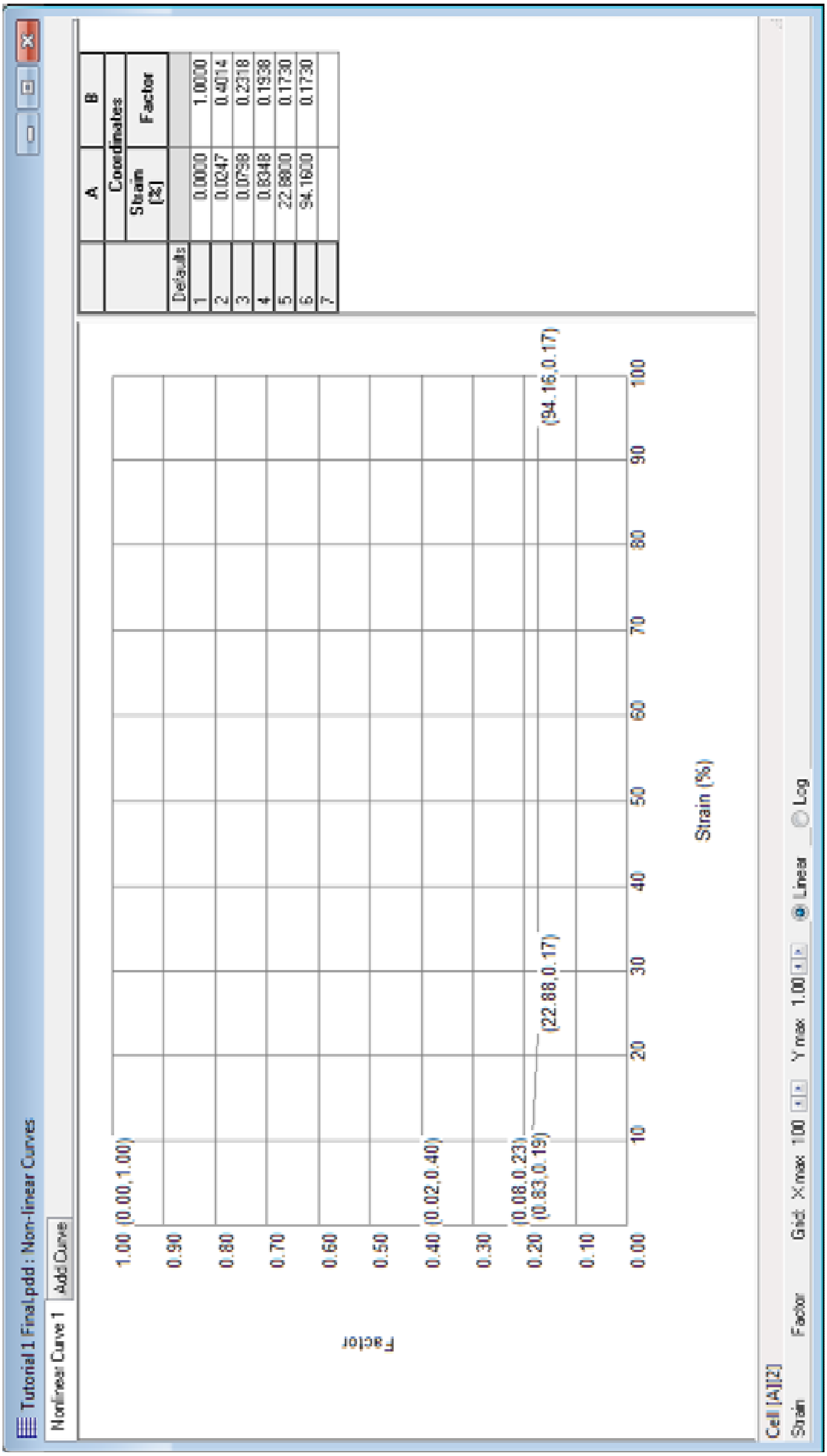
All stresses referred to in this manual are stress changes due to the application of loads. No account is taken of stresses existing in the ground before loading.

2. Double click on Non-Linear Curves in the Gateway

   Click on the tab titled Add Curve on the top right of the table.

   Choose the linear option at the base of the curve.

   Create the curve from the values derived at Stage 1. The expected curve and values are shown on the following page.


Tutorial 1 Final.pdd : Non-linear Curves
Nonlinear Curve 1
Add Curve
A
B
Coordinates
Strain [%]
Factor
Defaults
1 0.0000 1.0000
2 0.0247 0.4014
3 0.0758 0.2318
4 0.8348 0.1938
5 22.8800 0.1730
6 94.1600 0.1730
7
Factor
Strain (%)
(0.00,1.00)
(0.02,0.40)
(0.08,0.23)
(0.83,0.19)
(22.88,0.17)
(94.16,0.17)
Cell [A][2]
Strain
Factor
Grid: X max 1.00
Y max 1.00
Linear
Log

(Note: The user can also approximate the curve by clicking points on the graph using their mouse)

3. Double click on *Soil Profiles* in the Gateway.

   Assign the non-linear curve to Soil Stratum 2 in Soil Profile 2.

Tutorial 1 Final.pdd : Soil Profiles

	A	B	C	D	E	F	G
Layer	Level at top [m]	No of intermediate displacement levels	Young's modulus [kN/m²] Top	Young's modulus [kN/m²] Bottom	Poisson's ratio	Colour	Non-linear curve
Defaults					0.20		None
1	0.000	9	10000	15000	0.25		None
2	-4.000	17	20000	40000	0.25		Nonlinear Curve 1
3	-12.000	19	30000	50000	0.25		None
4	-21.000	19	50000	60000	0.25		None
5							

1: Soil Profile 1 | 2: Soil Profile 2 | Add Soil Profile

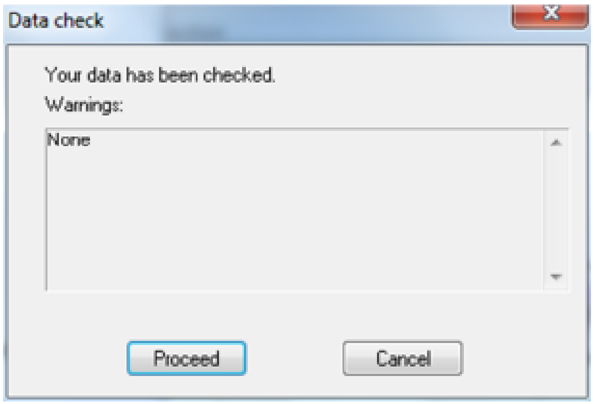
Soil Profile [Soil Profile 2] Cell [A][1]

5.2 Analysis and Output

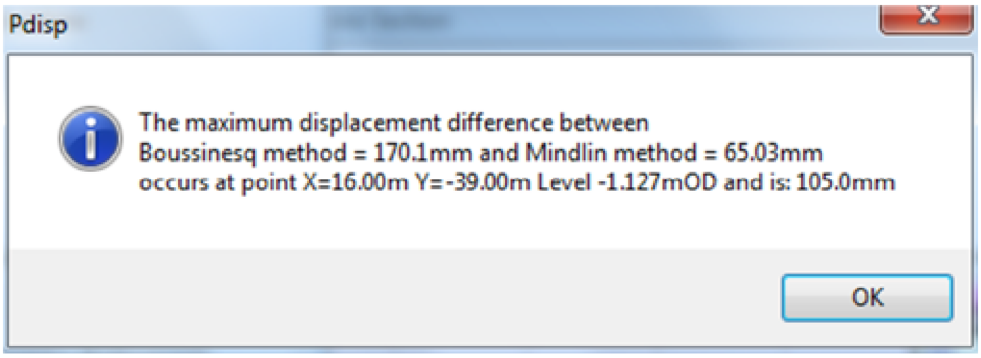
1. Prior to analysis, check all inputs and save the file.

Σ Click on the analyse button to carry out the analysis

2. The data check window will automatically appear. Select **Proceed**.



3. The following window will appear showing the difference between the Mindlin and Boussinesq results.

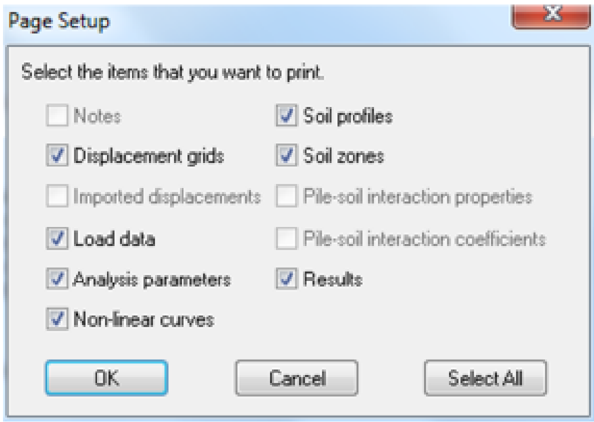


Note:
There is a significant difference between the Mindlin and Boussinesq methods because:

a) This analysis involves non-linear soils, which are not considered in a Mindlin analysis.
b) This analysis involves loads below the surface, which is not represented as accurately using the Boussinesq method

The point at which the greatest difference occurs is where the loads below the surface intersect. This is as expected due to the differences in the methods for loads acting below the surface.

4. The *Page Setup* window will appear. Select all the options to view the results and inputs in the Tabular Output.



5.2.1 Tabular Output

1. View the Tabular Output.

   What is the vertical stress and displacement at:

   a. The centre of each load
   b. -64, 39, 0
   c. -64, 30, 3
   d. -70, -80, 0

   Hint:

   Section 4.2 of the User Manual outlines where displacements and stresses are shown in the Tabular Output.

   The points listed are shown on the displacement grids. A quicker way to find these displacements would be to enter displacement points in the Displacement Data input table.

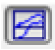
2. Choose **File > Export > Tabular Output**

3. Save the file with a .csv extension and open the Excel file.

5.2.2 Plan View

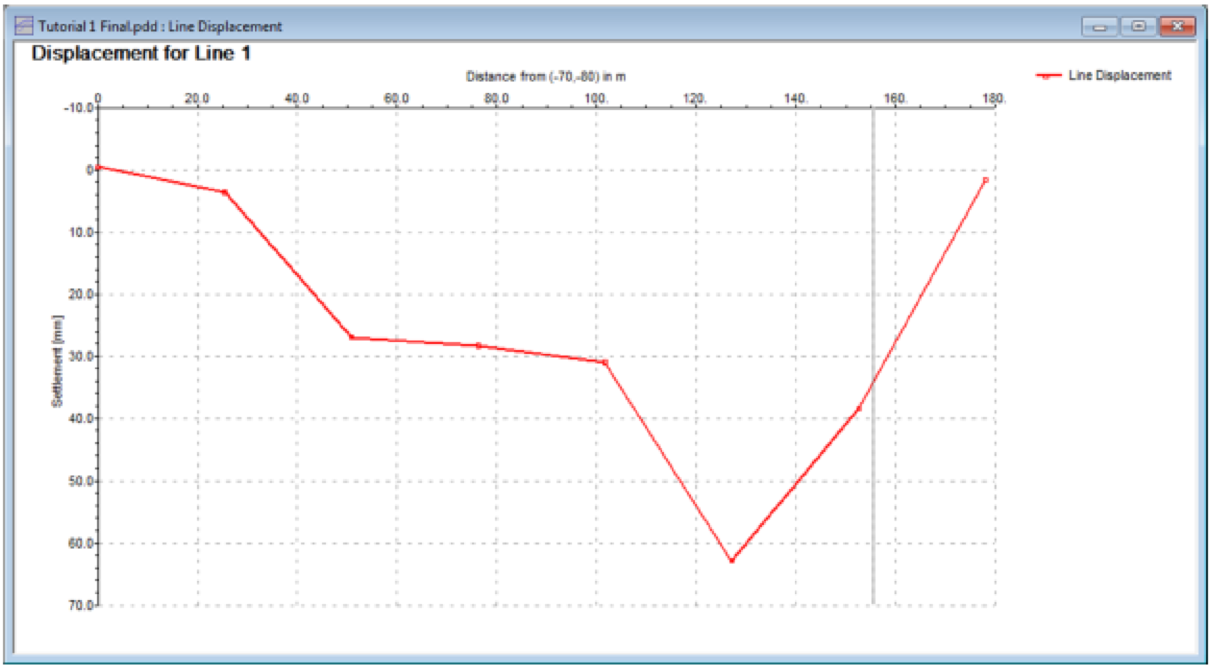
Double click on *Plan* in the Output section of the Gateway

Line Graph



Click on the *Line Graph* icon.

1. Hover the mouse over Line 1 and left click to plot the displacements over the line.

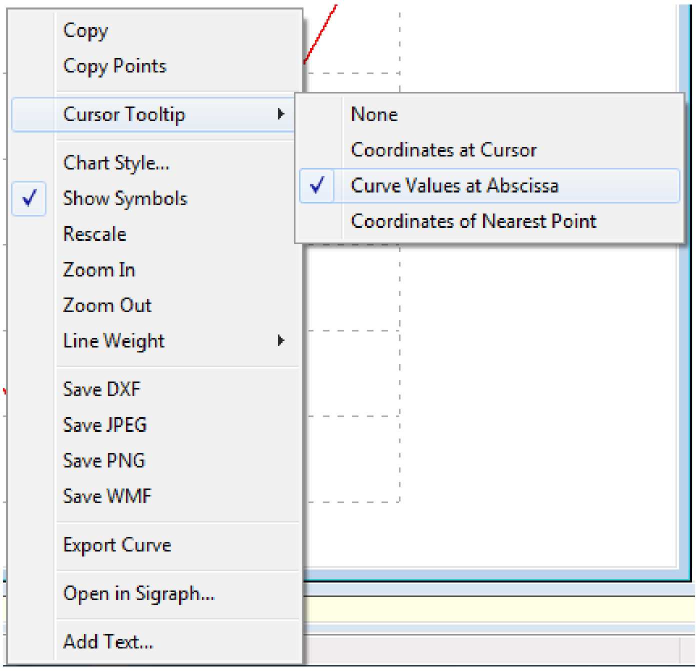


Note:

To plot more points in the graph, amend the Displacement Data input table by adding more intervals to the displacement line.

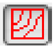
It is advised that the distance between points on the line should be less than the width of the loads to depict settlements accurately.

2. Right click on the graph.
   The following options will appear:



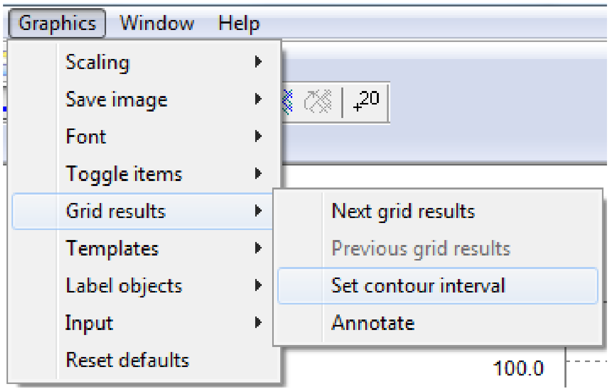
3. Explore the different options to amend the graph view, appearance and to save the graph. Use *Chart Style* to label axes.

Contours

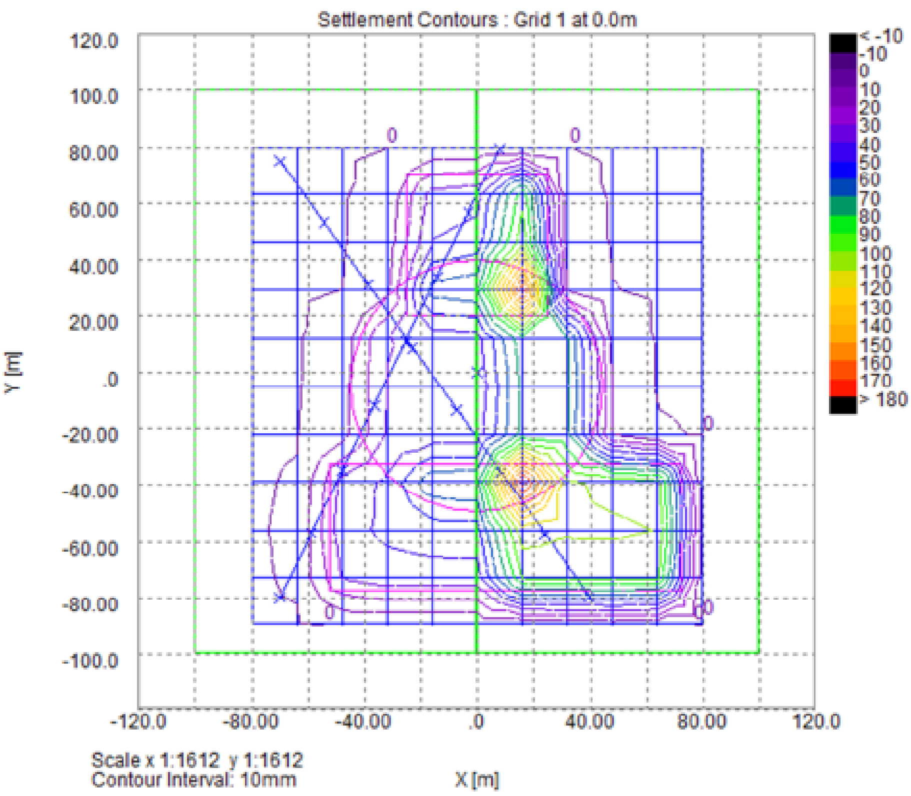


Select the *Contours* icon. View the line contours for Grid 1.

1. Choose **Graphics > Grid Results > Set Contour Interval** and set the contour interval to 10mm.

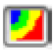


The following view will show.

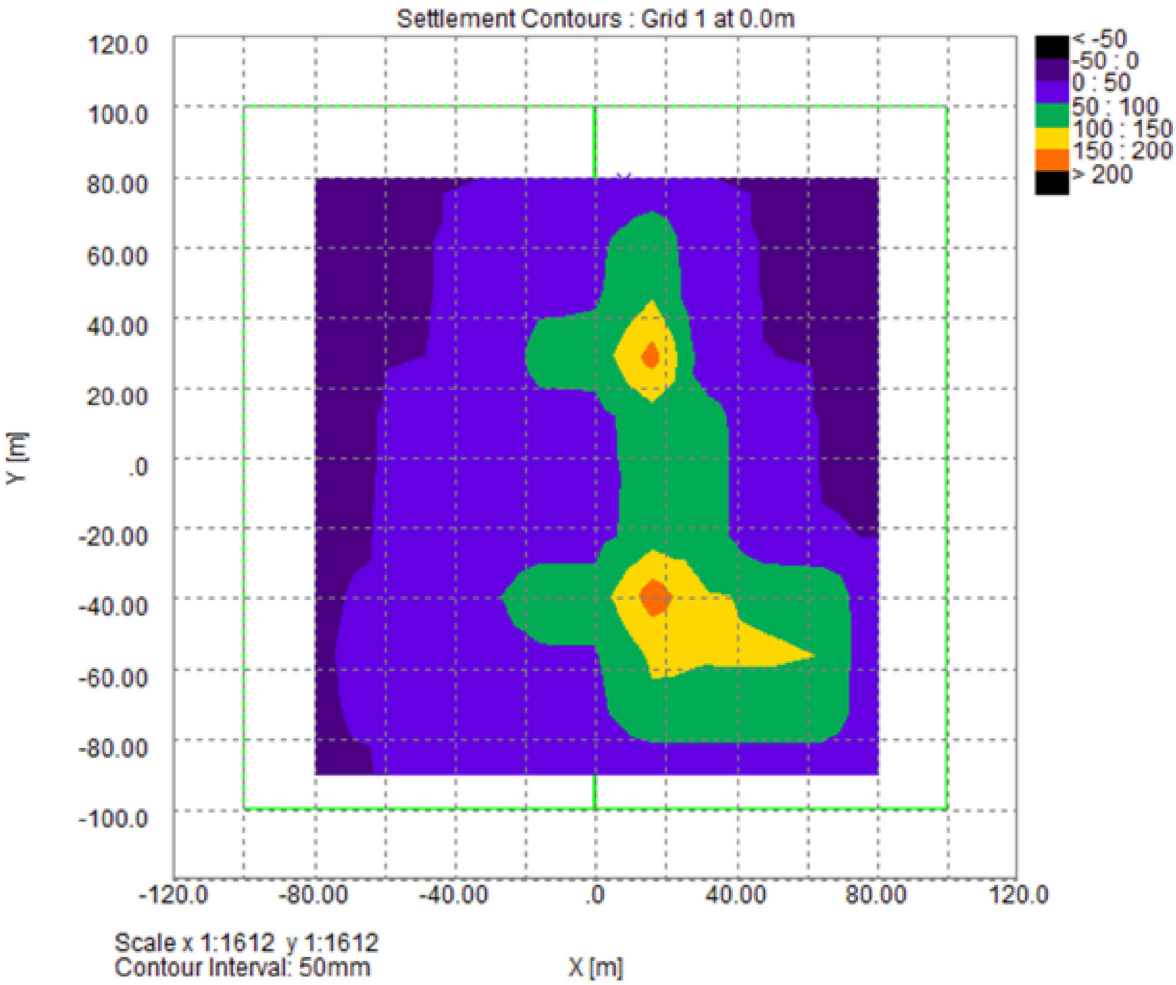


2. Toggle between grid results by using the *Grid Up* and *Grid Down* icons ()

Show settlement at specific points using the *Annotate* icon.



Select the Fill Contour icon to view the filled contours:



Note:

Assess the impact of changing the number of intervals in the Grids on the contour output:

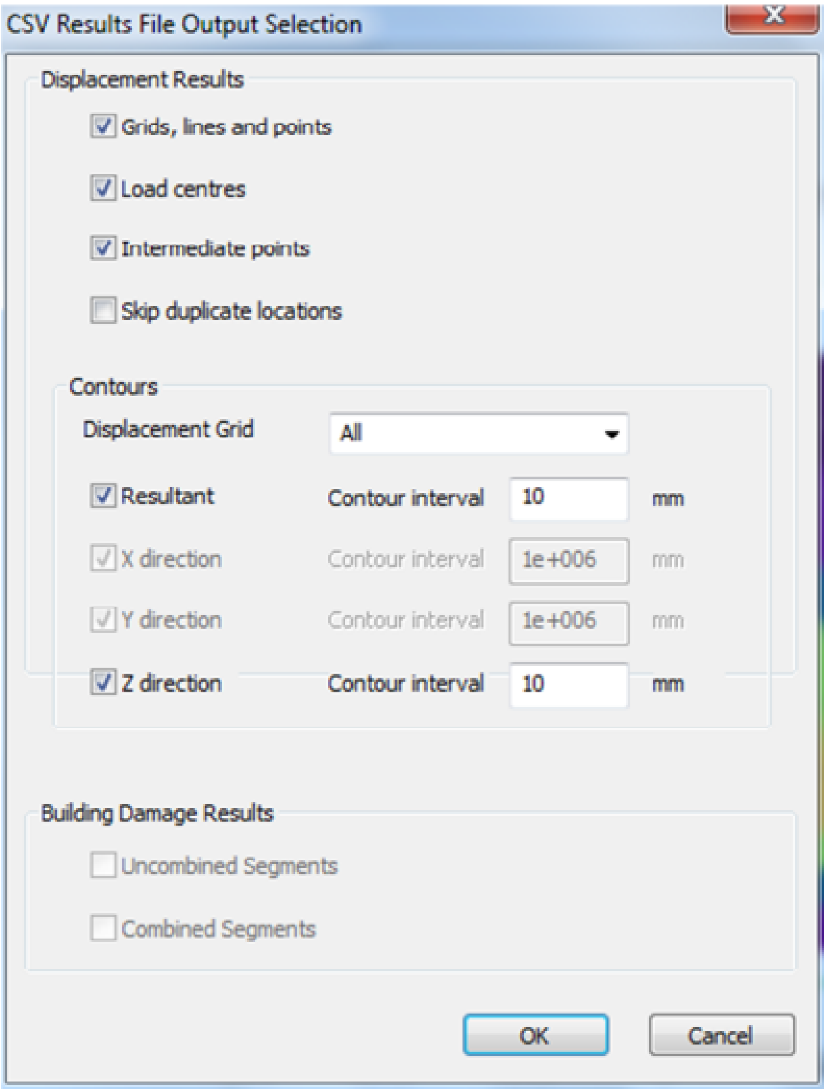
a) Use **File > Save As** to save a test file
b) Change the grid intervals in the x and y direction to 3 to assess the impact of decreasing the grid intervals on the output file.
c) Change the grid intervals in the x and y direction to 30 to assess the impact of increasing the grid intervals on the output file.

Having looked at the file, it is evident that the file with fewer intervals does not show maximum values for settlement.

If the grid interval produces distances between grid points which are larger than the width of the load, the grid points may not pick up the point of largest settlement.

Exporting Contours

1. Go to **File > Export > CSV Results File**
2. Choose the following option to output the contours and select OK:



(Note: X and Y settlements are not shown as these are not calculated for the Boussinesq method)

3. Save the csv file and review the outputs.

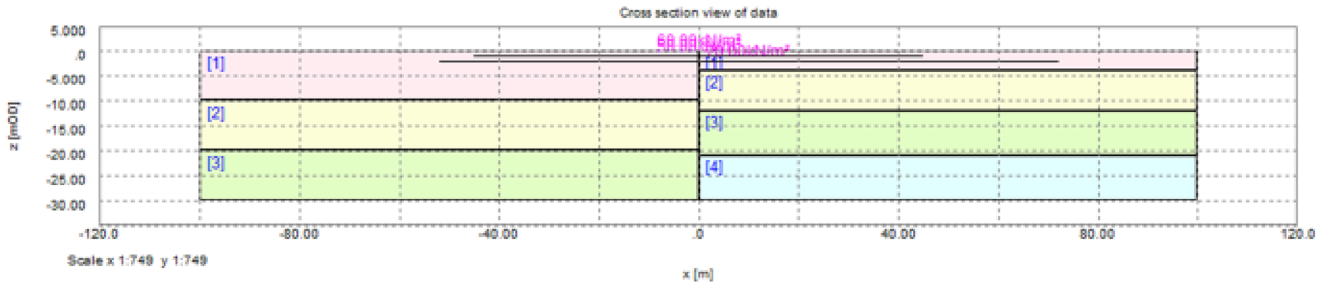
(Note: Section 4.4 of the User Manual will explain how the csv results file could be interpreted for a CAD drawing.)

5.2.3 Cross Section

1. Double click on *Cross Section* in the Output section of the Gateway
2. Use the toolbar to view the loads in relation to the soil stratigraphy.

Toolbar:

View:

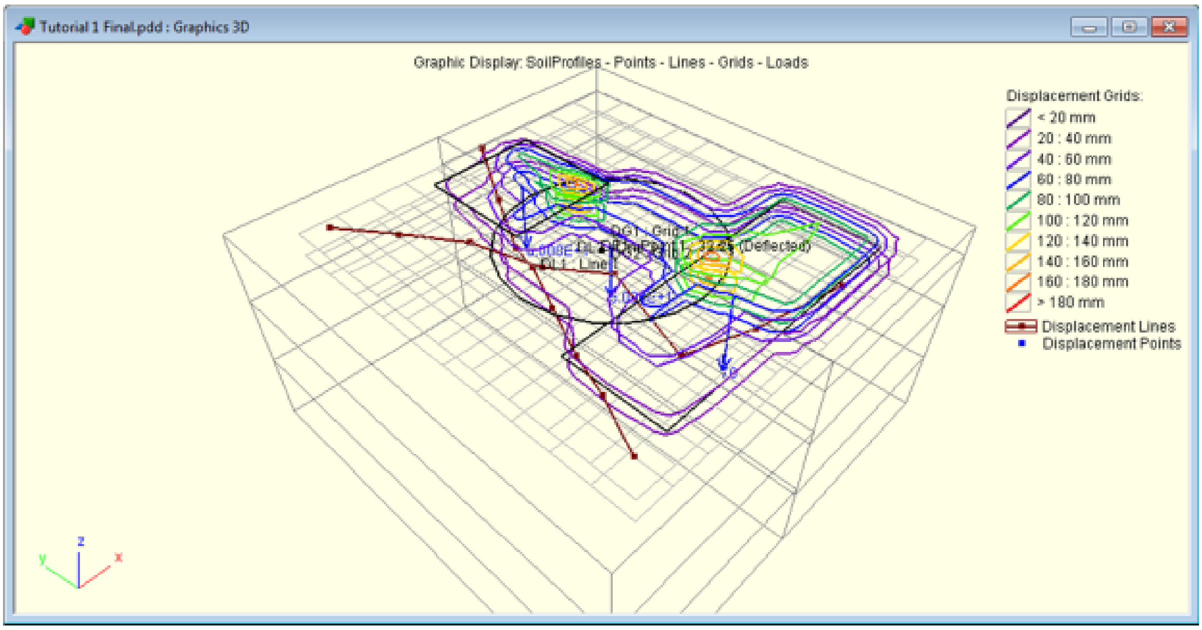


5.2.4 Graphical Output

1. Double click on *3D Graphics View* in the Output section of the Gateway

Select the *Wizard* icon or right click on the view and select **Settings Wizard**

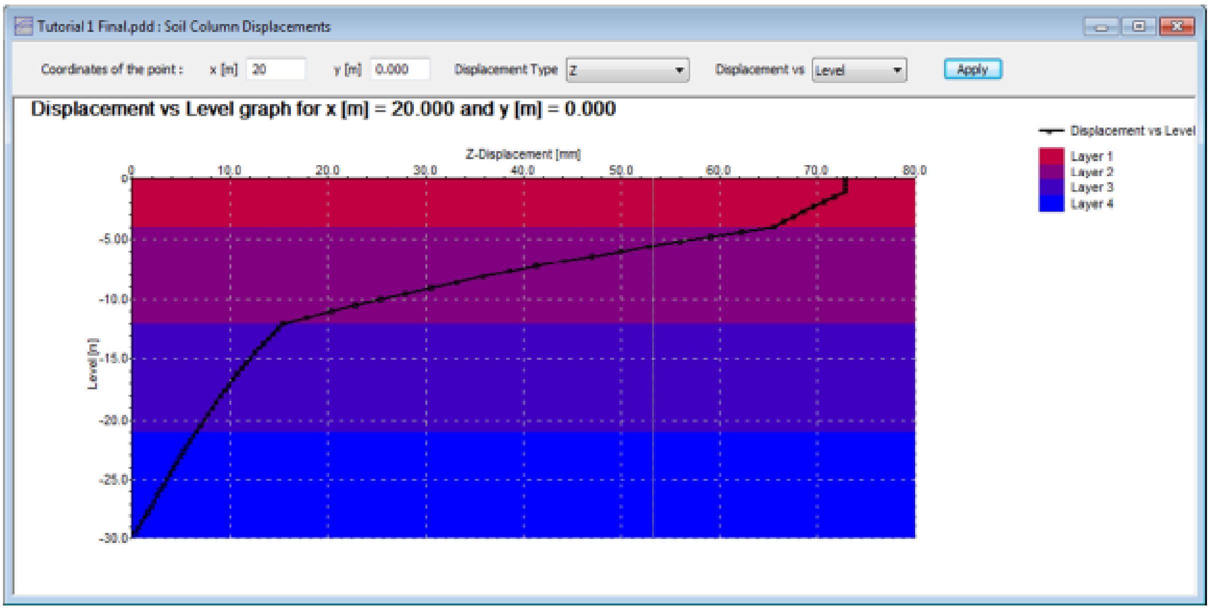
2. Choose the appropriate options to get the following graphical view:



Hint:
Selecting the Wire Frame grid on the Soil property page allows the user to view loads and contours.
The outputs include the deflected displacement line and line contours of settlements

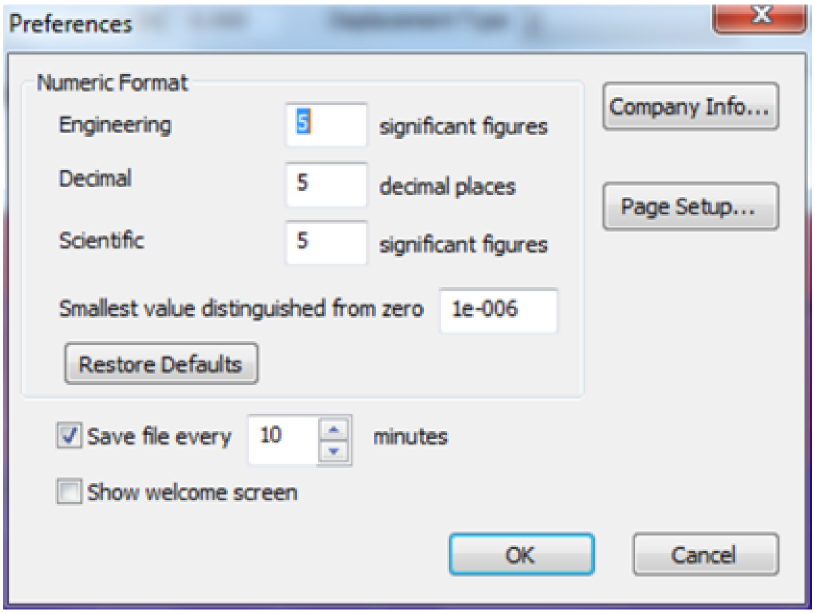
5.2.5 Soil Column Displacement

1. Double click on *Soil Column Displacement* in the Output section of the Gateway
2. Choose to view the displacement at point (20, 0) in plan.



5.2.6 Printing Outputs

1. Select **Tools > Preferences**
2. Apply your company details



3. View your output using **Print Preview**

6. Graphical Input Tutorial

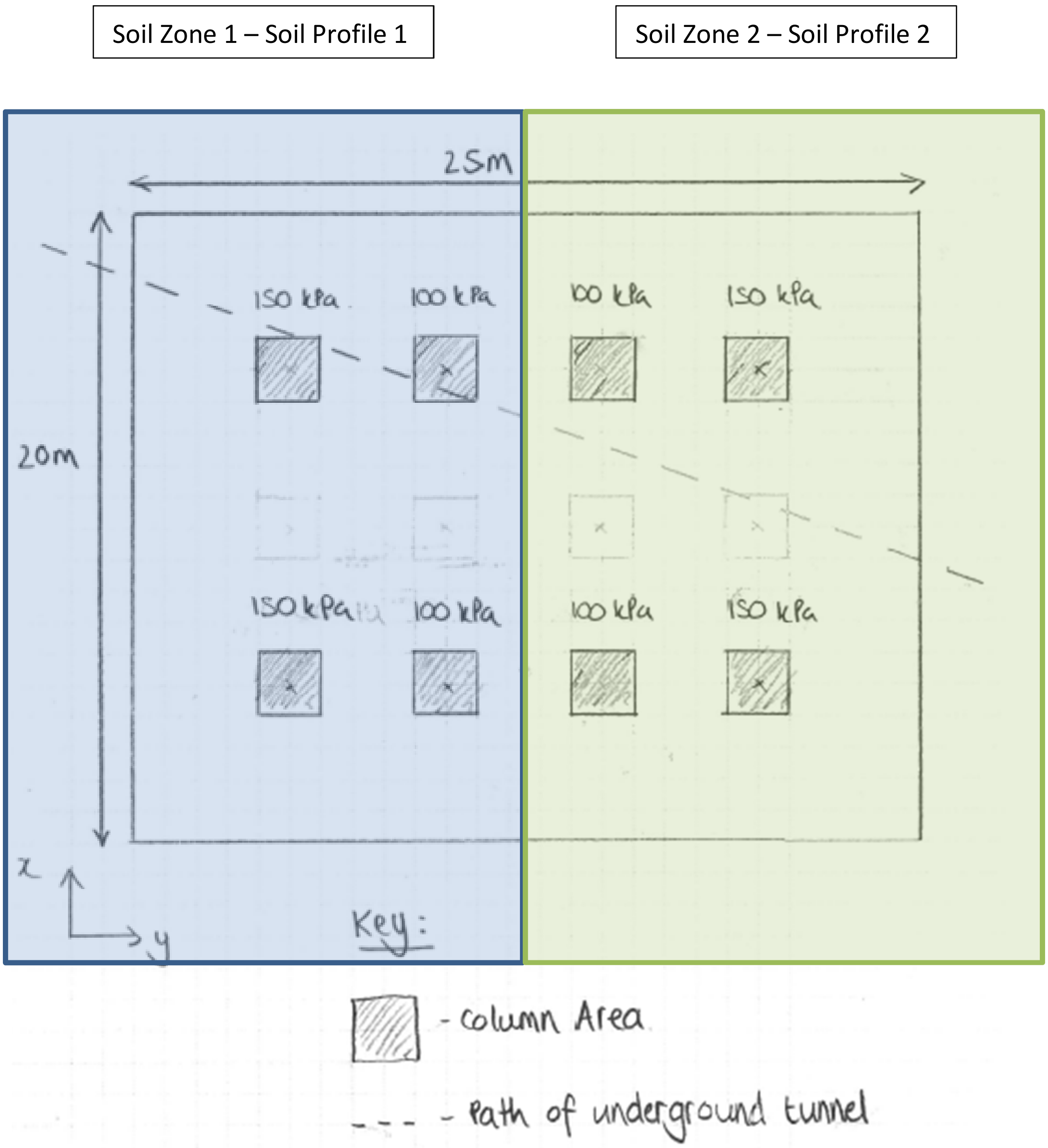
By the end of the session the user should be able to:

- Input background bitmaps to inputs
- Graphically input loads, soil zones and displacement points
- Run a Mindlin analysis

3.1 Analysis Description

The following is a scaled drawing of a building with column loadings at surface level (10mOD) and the path of an underground train tunnel below the building (crest at z=5mOD).

The building is due to be demolished and the Asset Owners require prediction of the actual and differential settlements that will occur at the crest of the tunnel.

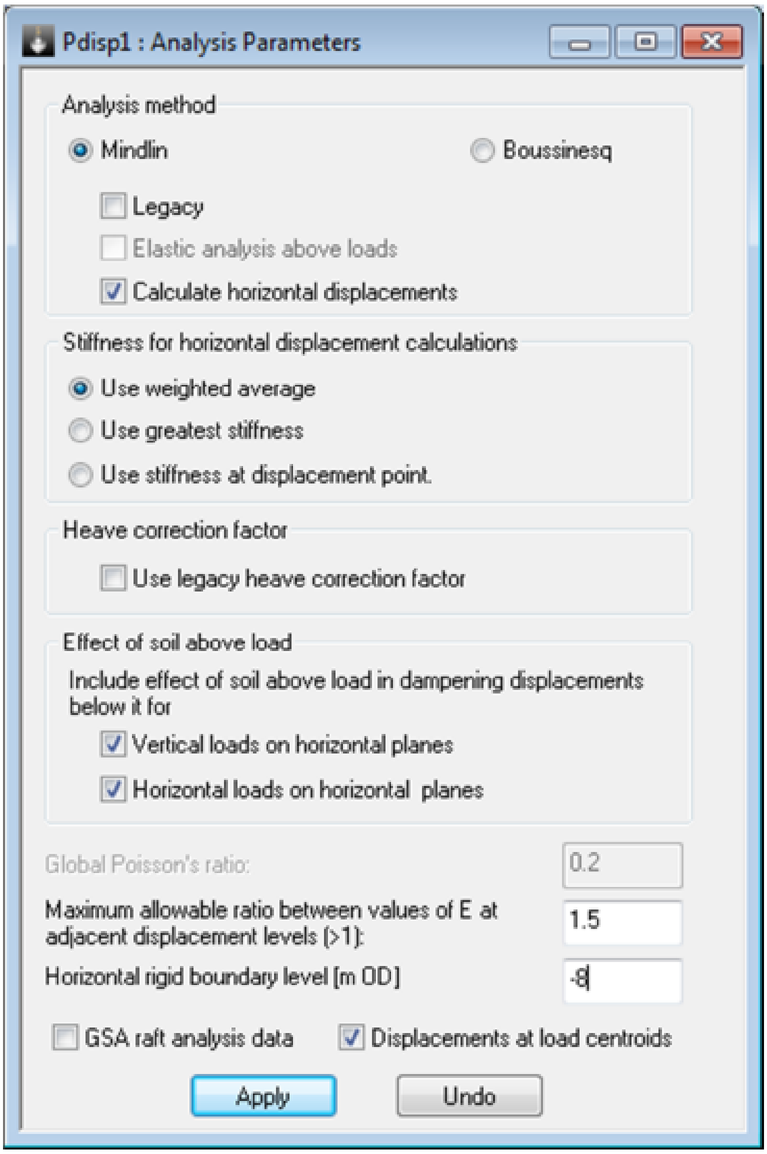


3.2 Graphical Input

1. Start a new Pdisp file

2. Fill the **Titles and Units** section

3. Complete the **Analysis Parameters** section as follows:



(Note: The legacy correction factor has not been chosen as the program defaults to the new correction factor, which is outlined in Section 2.3.2 of the User Manual)

4. Enter Soil Profiles as shown below:

Soil Profile 1

	A	B	C	D	E	F
Layer	Level at top [m]	No of intermediate displacement levels	Young's modulus [kN/m²] Top	Bottom	Poisson's ratio	Colour
Defaults					0.20	
1	10.000	5	19000	19000	0.20	
2	5.000	5	25000	25000	0.20	
3	2.500	5	20000	20000	0.20	
4	0.000	5	25000	25000	0.21	
5	-5.000	5	30000	30000	0.22	
6						

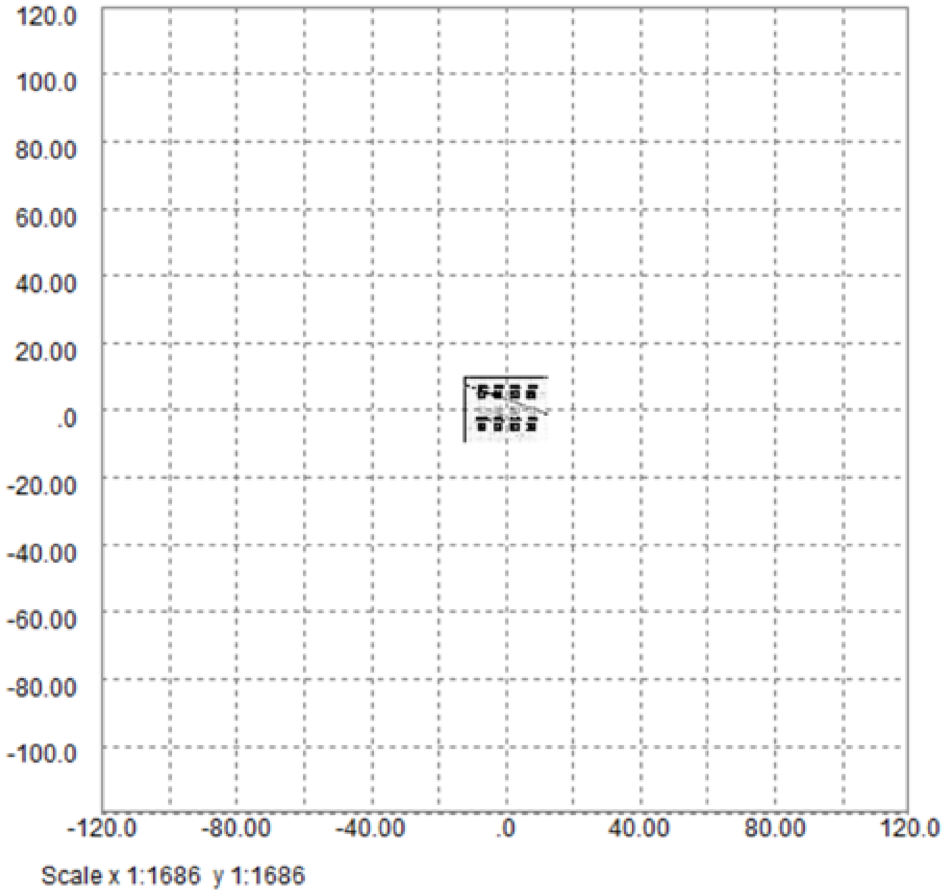
Soil Profile 2

	A	B	C	D	E	F
Layer	Level at top [m]	No of intermediate displacement levels	Young's modulus [kN/m²] Top	Bottom	Poisson's ratio	Colour
Defaults					0.20	
1	10.000	5	22000	22000	0.21	
2	8.000	5	18000	18000	0.20	
3	6.000	5	25000	25000	0.20	
4	0.000	5	17000	17000	0.20	
5	-5.000	5	24000	24000	0.20	
6	-6.000	5	12000	13000	0.20	
7						

5. Double click on *Graphical Input* in the Gateway

bmp

Click on the *bitmap* icon and insert Graphical Input.bmp to the scale shown in Section 3.1.



6. Zoom into the area around the image by left clicking and pulling a window around it.

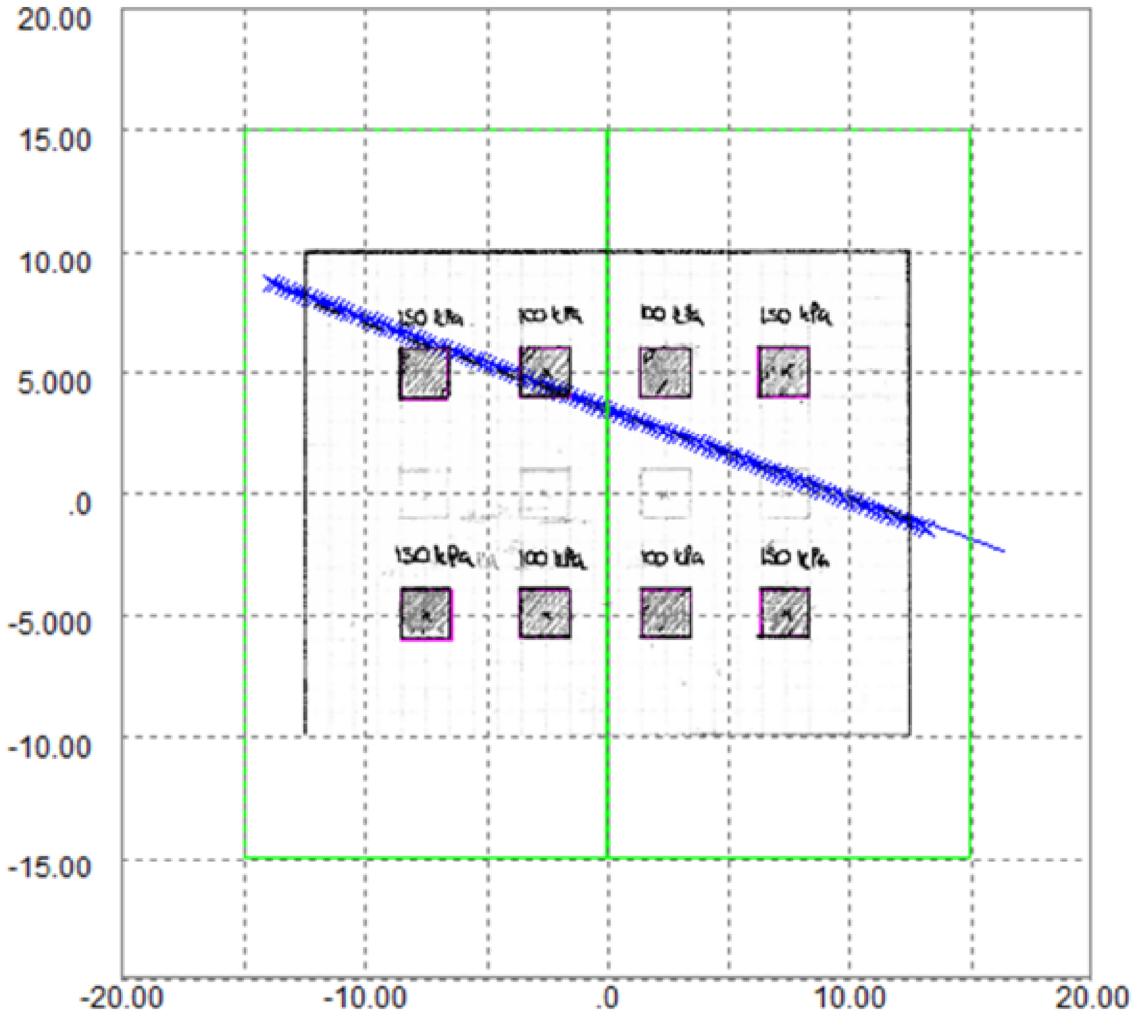
Click on the *Soil Zones* icon to plot Soil Zone 1 and Soil Zone 2.
Check the Soil Zones table to adjust the co-ordinates if necessary

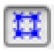
Use the *Displacement Line* icon to draw the line along the path of the tunnel. Specify the level in accordance with Section 3.1.

Select the best interval for the displacement line, remembering that the load width is 2m.

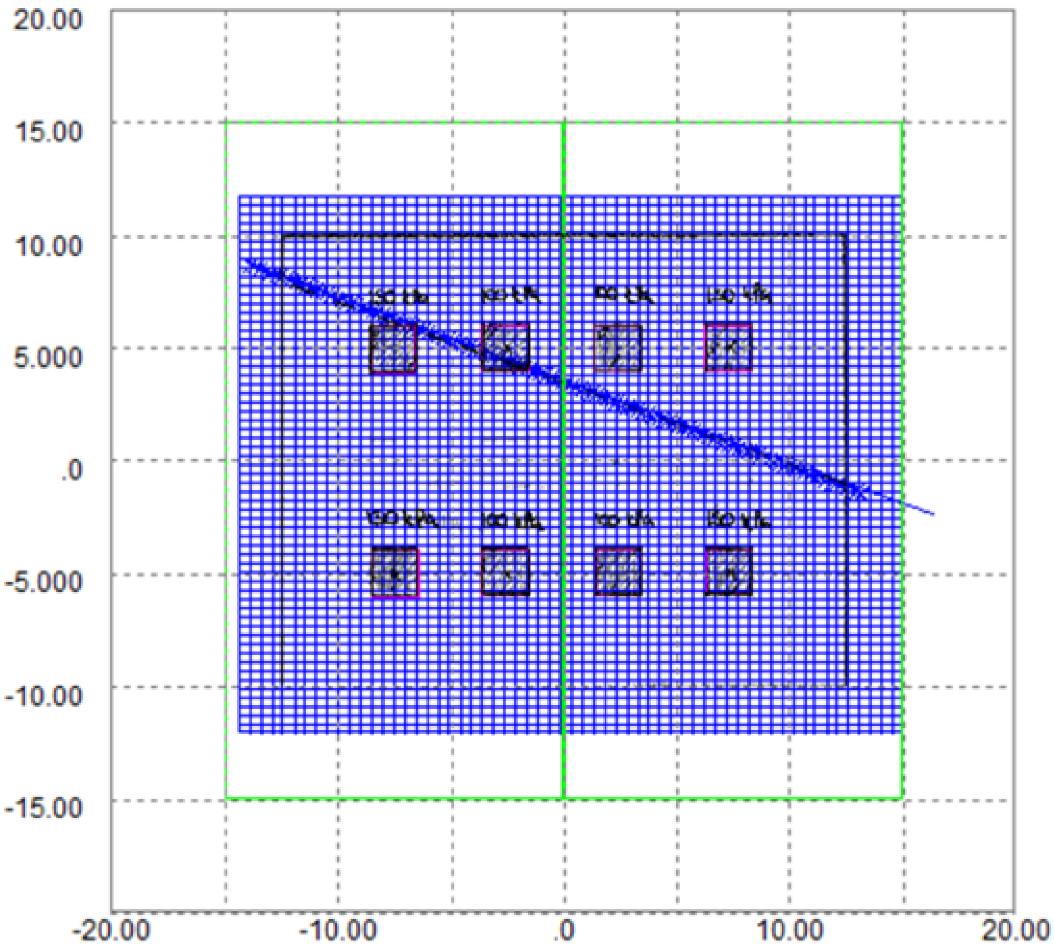
Use the *Load* icon to input the loads as shown in the diagram. Specify the level in accordance with Section 3.1.

(Hint: Consider modelling demolition loads, which act in the negative direction.)





Use the *Grid* icon to create a surface grid around the building with suitable intervals.



3.3 Analysis and Outputs

1. Check all the inputs and save

2. Run the analysis

3. Determine the following from your analysis:
    a. Maximum settlement and point of maximum settlement along the crest of the underground line

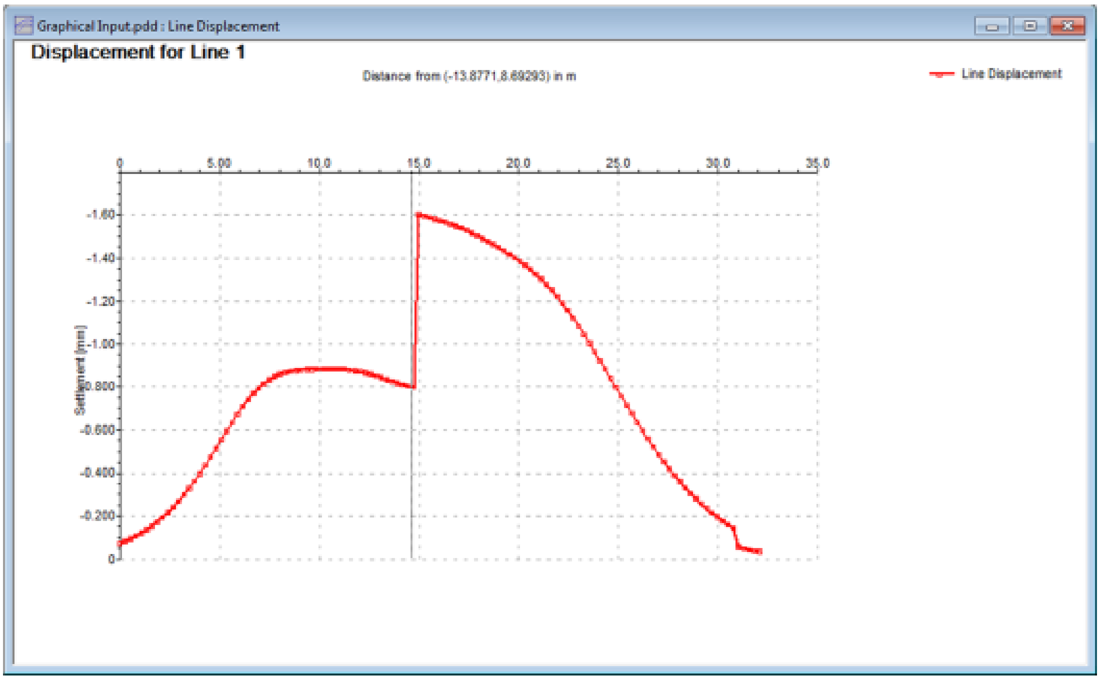
    b. Maximum gradient of settlement along the underground line

    c. Maximum settlement occurring at the surface of the demolition

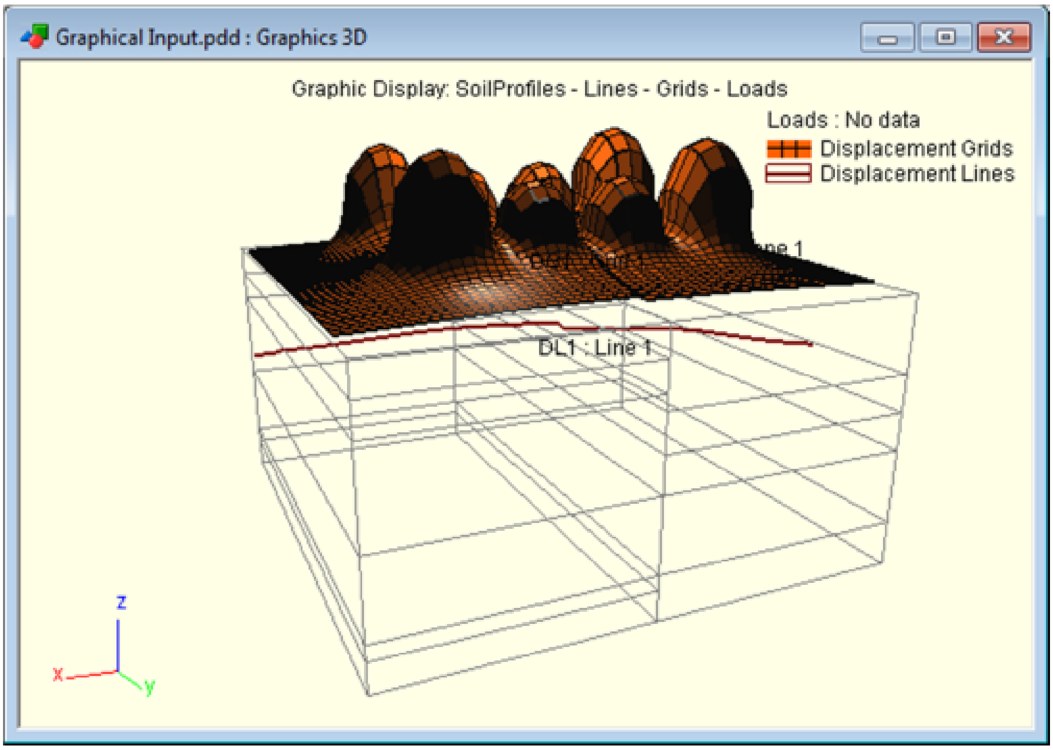
    d. Maximum settlement at a point occurring 5m from the boundary of the building

    e. Maximum x and y displacements occurring at the surface of the demolition

4. Produce the following line graph from your analysis:



5. Produce the following Graphical Output for your analysis:



(Hint: The Output shows the deflected displacement grid and displacement line)