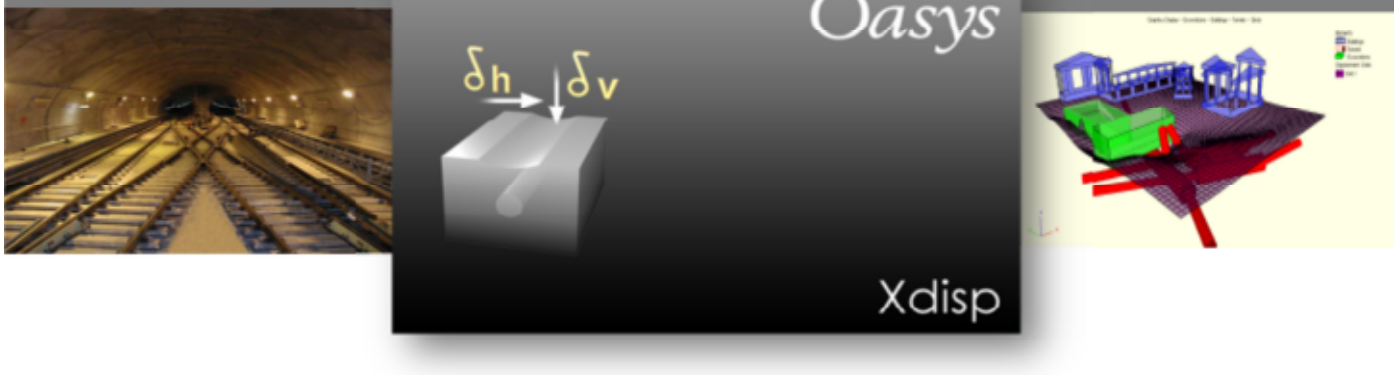
Oasys

The software house of Arup

Xdisp Tutorial Manual



Oasys Xdisp

Printed: May 2013

Contents

1. Introduction

Xdisp (Excavation induced ground displacements) is a program that calculates displacements due to excavation in the ground from tunnelling, retaining walls and mining. The program allows the user to model complex 3D problems with ease. Graphical inputs enable the user to set up a problem quickly, whilst a number of options are provided to enable users to map settlements and building damage. The outputs enable the user to study the settlements in detail, or plot contours and deformations in plan and in three dimensions. The calculation itself is based on verified and robust empirical methods, which have been shown to compare favourably to Finite Element analysis[1]. Furthermore, the simplicity of interface enables new users to set up and run a retaining wall problem in a few minutesa nd with minimal training.

The various tutorial examples deal with a range of practical tunnel and excavation applications but this Tutorial Manual is intended to familiarise the user with Xdisp. The examples should therefore not be used as a basis for practical projects. Mining problems calculate surface displacements only and are therefore geometrically simpler. Consequently, a mining example has not been included in this tutorial manual.

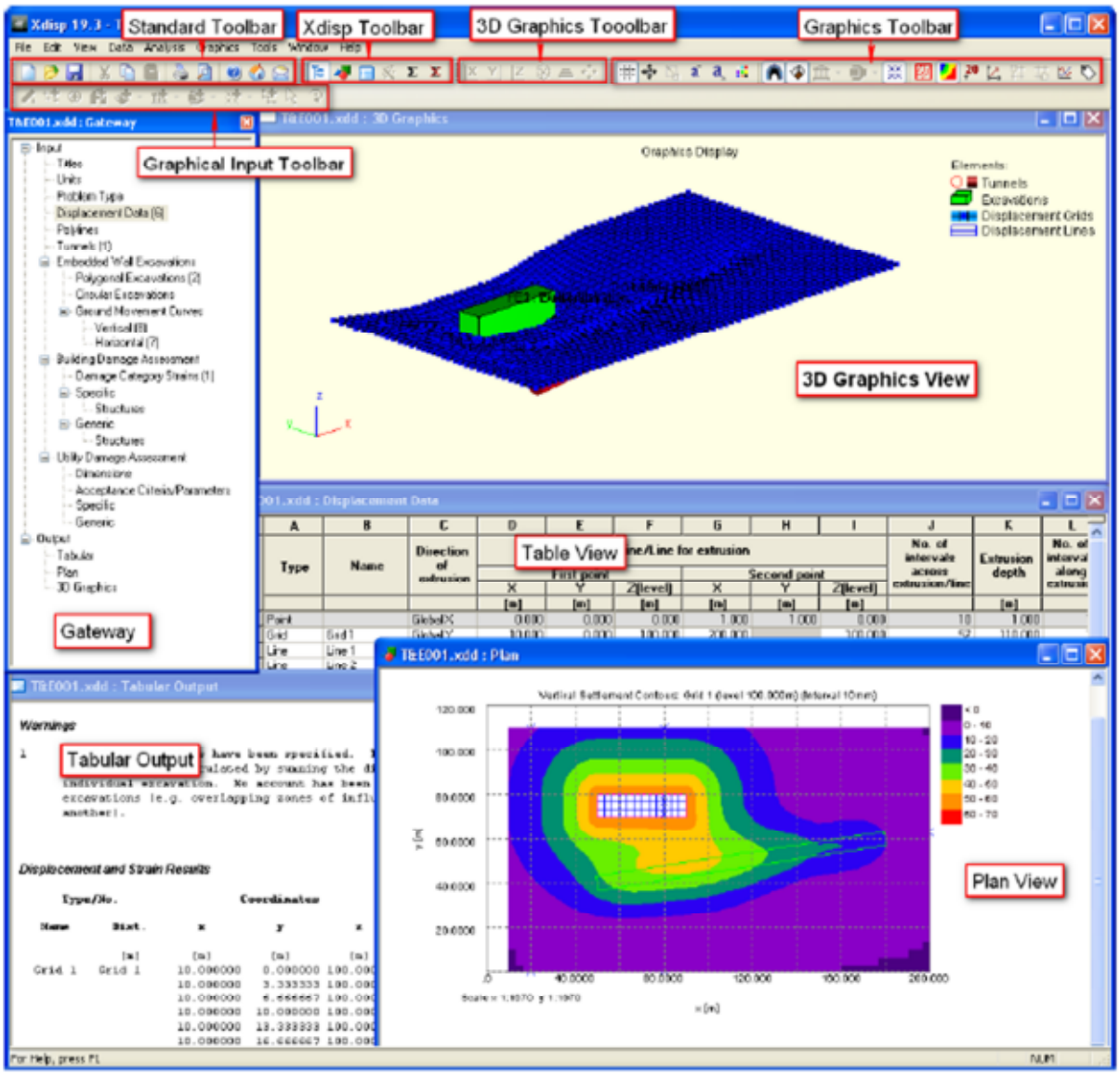
Users are expected to have an understanding of soil mechanics and geotechnical theory, and should be able to work in a Windows environment. The tutorial lessons are also available in the examples folder and can be used to check your results.

It is important to realise that Xdisp is an advanced program analysing complex problesm 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, 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 Xdisp. 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 programme content. Further examples of analysis, video tutorials and webinars are available from the Oasys website.

[1] **Addenbrooke & Potts St James's Park FE analysis described in CIRIA 2001**

2. Components of XDisp



3. Program Features

Users should familiarise themselves with the features and capabilities of the program prior to carrying out the tutorials. This will assist them in understanding the analysis being carried out. If further clarification is required, the User Manual should be referred to.

The following features are separated into those applicable either to tunnels or mines and those applicable to both.

3.1 Tunnels

A tunnel is taken as an excavation of circular cross-section in soil. Several methods of solution are available to create the profile of settlement above the tunnels. These include methods for the following.

- Analysis methods to model settlements in both fine (cohesive) and coarse (granular) grained soils
- Two-layer systems with level or inclined soil interfaces
- Settlement profiles due to multiple tunnels
- Deformation and strain data plots for lines of any orientation and level above tunnel axis level
- Sub-surface displacement methods

3.2 Embedded Wall Excavations

An embedded wall excavation is defined by a polygon or circle describing its plan area, top and bottom levels, and its associated vertical and horizontal ground movement curves. It is used to model soil displacements caused by installation of, or excavation in front of, embedded walls. The following features are available.

- Ground movement curves chosen from a library of pre-programmed curves, or specified by the user explicitly
- Soil displacements arising from installation of and/or excavation beside, retaining walls by selecting appropriate ground movement curves
- Multiple embedded wall excavations
- Both embedded wall excavations and tunnels
- Deformation data plots for lines of any orientation and level

3.3 Mines

A mine is taken as an excavation of rectangular cross-section in rock. The following features are available.

- Overlying strata may form a two-layer system, but with a horizontal interface
- Only one method of solution is available and results are only available at ground surface level
- Deformation data may only be plotted for horizontal lines at ground level

3.4 Combined Features

The following features can be applied to tunnels, embedded wall excavations and mines.

- Tunnel and mine end points, and embedded wall excavations' plan positions can have any spatial location
- The program calculates the three-dimensional displacements (and strains for pure tunnelling problems)
- Vertical displacements may be positive or negative (settlement or heave)
- Displacements are calculated for a grid of points, a line of points, or individual points
- Output is available in tabular and graphical forms, and may be exported to e.g. csv file
- Displacements from other programs can be imported for inclusion in the building damage assessment calculation.

3.5 Building Damage Assessment

Building Damage Assessment is performed using the Burland (1995) assessment method.

- Each 'sub-structure' wall or facade is given a location by association with a displacement line.
- Horizontal displacements are calculated for the position of that displacement line.
- Vertical displacements are calculated for any number of vertical offsets of the displacement line – e.g. representing pile depths.
- Building damage calculations are performed for each of those vertical offsets, using the horizontal displacements calculated at the level of the displacement line combined with each level of vertical displacement results.
- Each 'sub-structure' is given a set of damage category strains to define the threshold of 5 damage categories based upon the geometry defined for the structure.
- Either user-defined damage category strains, or pre-defined values from Burland (1995), may be chosen.
- Damage categories are calculated for each hogging and sagging segment along the length of each sub-structure.
- Adjacent hogging and sagging segments may be combined for damage category assessment as one segment.
- Graphs of vertical and horizontal displacement may be viewed for each sub-structure.
- Damage category interaction charts may be viewed for each segment of each sub-structure.

3.6 Utility Damage Assessment

Xdisp performs detailed damage assessment of a utility by calculating the pullout, rotation, axial strain and flexural strain values at different points along the utility's length. It then checks whether these values are falling within acceptance criteria. Utilities are composed of one or more straight sub-utilities.

- Each 'sub-utility' is given a location by association with a displacement line.
- Each 'sub-utility' is given a set of cross-sectional dimensions (internal diameter and wall thickness).
- Each 'sub-utility' is given a set of parameters and acceptance criteria to determine the damage limits.
- Displacements are calculated for the position of the displacement line.
- 'Sub-Utility' damage is calculated using those displacements to determine joint rotations, pullout and axial and flexural strains along the length of the utility.
- Graphs of rotation, pullout and strains along the 'sub-utility' length may be viewed for each 'subutility'.

4. Tunnel and Excavation Tutorial

Objectives

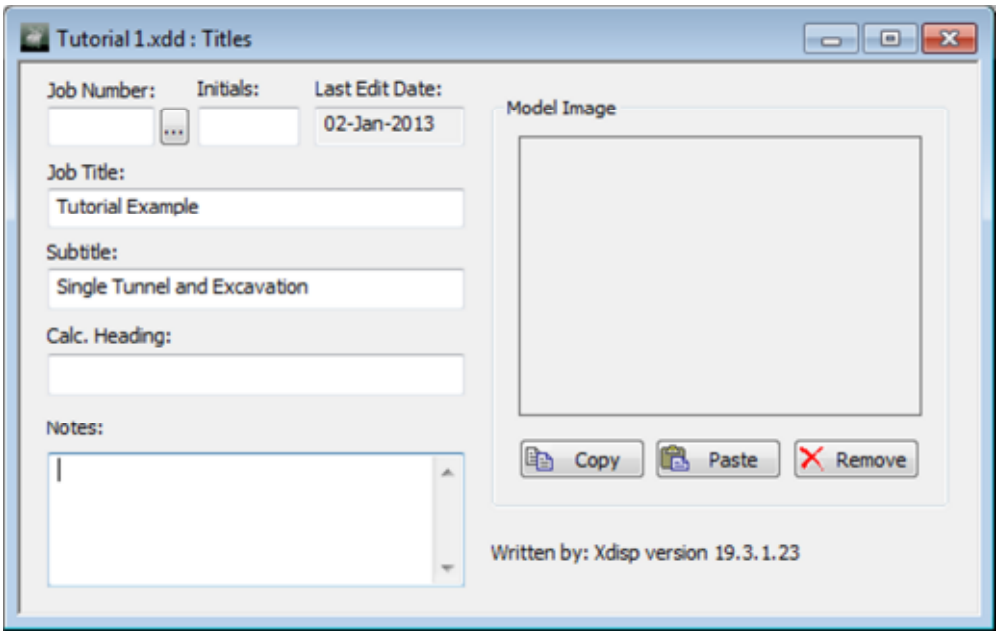
By the end of the session the user should be able to:

- Navigate the Xdisp Interface
- Input tunnel and excavation data
- Enter surface and sub-surface curve data
- Input Displacement data
- Run an analysis
- Navigate the Graphical Output
- Create simple graphs
- Export tabular outputs for further analysis

4.1 Creating the input

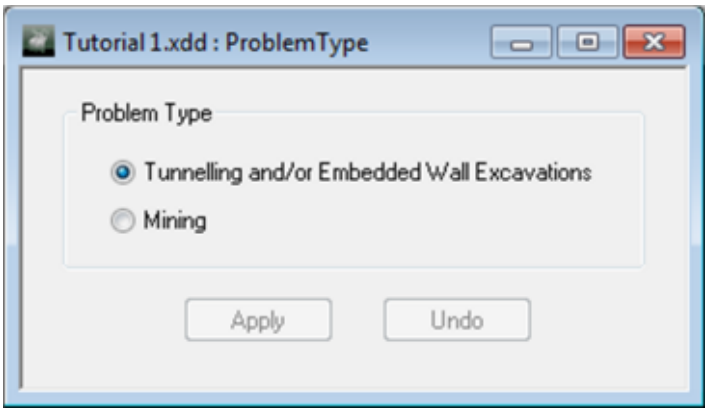
Once opened, create a new file by clicking the ‘New File’ icon on the Standard toolbaror clicking Ctrl + N

1. Fill the *Titles* dialog box:



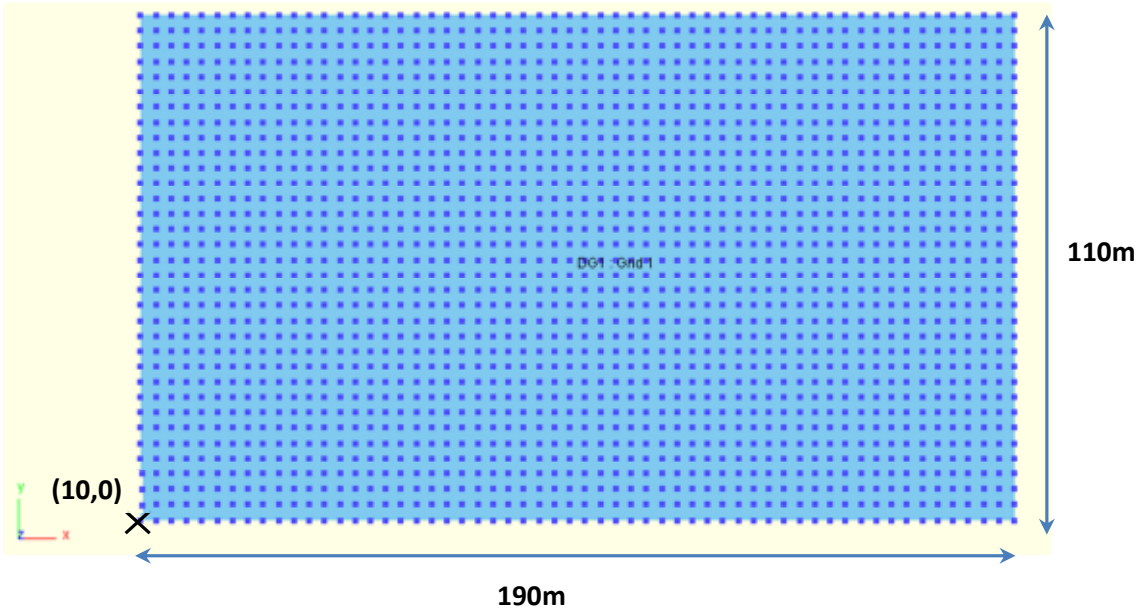
(Hint: Should the user need to change the Units, they should do so at this point by double clicking on *Units* in the Gateway)

2. Double click on *Problem type* in the Gateway.
   Select Tunnelling and/or Embedded Wall Excavations



4.1.1 Displacement Data

1. Double click on *Displacement Data* in the Gateway
   Enter the data for the grid shown below which lies at z=100m.



The grid should have 57 intervals in the x direction and 33 intervals in the y direction.

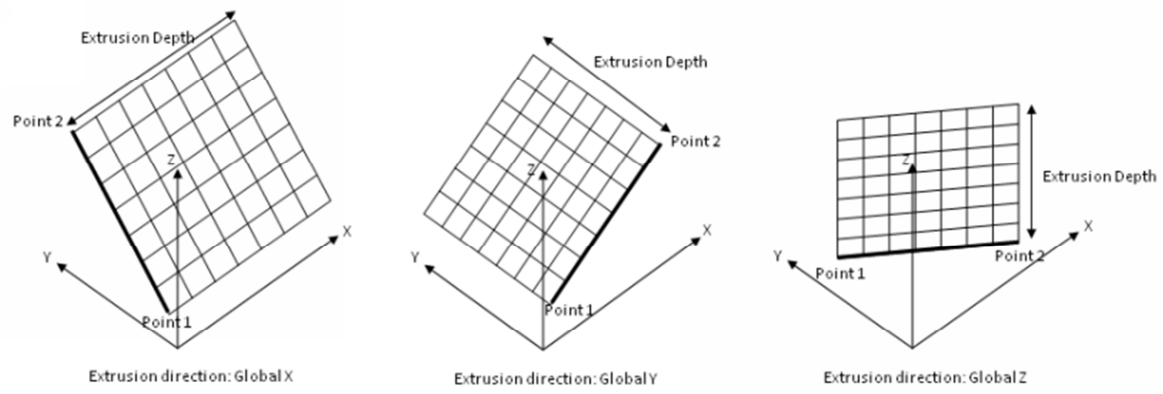
Note:

The intervals determine the points of calculation, and displacements etc. will be shown for each point in the grid in the tabular outputs. Too many points in the grid will slow the analysis and the display of graphical results.

Details of how the grid should be input is shown in the excerpt of the user manual, Section 3.5, below.

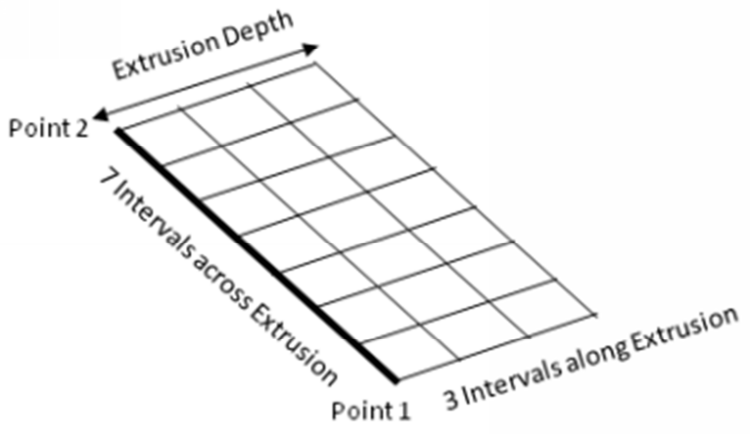
For the grid specified, the user can enter the grid using either Global X or Global Y.

Grids are specified by extruding a line. The **Direction of extrusion** is specified as one of the Global axes (X, Y or Z). A **Line for extrusion** must be entered by specifying its end coordinates. For example, if 'Global X' is the direction of extrusion then the table allows the specification of a line in the YZ plane.



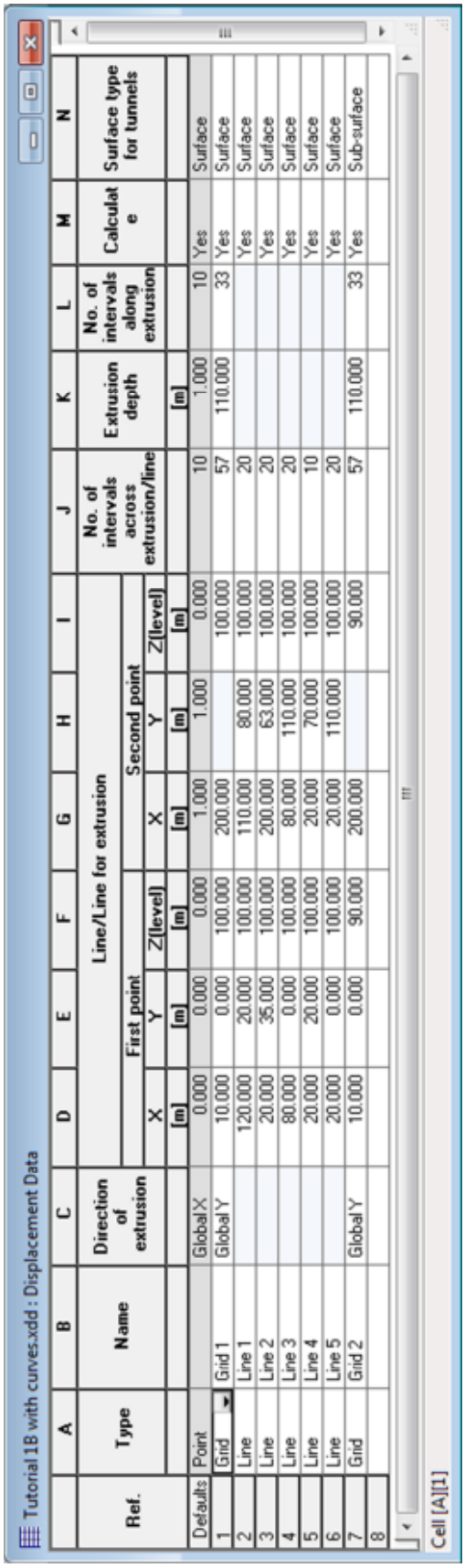
The extrusion depth should not be zero. Negative extrusion depth extrudes in the opposite direction to the global directions.

The number of intervals is specified across and along the extrusion as shown below.



Check the entered grid details with those shown on the following page.

2. Copy the cells for the displacement grid and lines from the table below.
   (Note: the extra grids and lines are to be used later in the tutorial)



Tutorial 1B with curves.xdd : Displacement Data

Ref.	A Type	B Name	C Direction of extrusion	D Line/Line for extrusion – First point X [m]	E First point Y [m]	F First point Z[level] [m]	G Second point X [m]	H Second point Y [m]	I Second point Z[level] [m]	J No. of intervals across extrusion/line	K Extrusion depth [m]	L No. of intervals along extrusion	M Calculate	N Surface type for tunnels
Defaults	Point		Global X	0.000	0.000	0.000	1.000	1.000	0.000	10	1.000	10	Yes	Surface
1	Grid	Grid 1	Global Y	10.000	0.000	100.000	200.000		100.000	57	110.000	33	Yes	Surface
2	Line	Line 1		120.000	20.000	100.000	110.000	80.000	100.000	20			Yes	Surface
3	Line	Line 2		20.000	35.000	100.000	200.000	63.000	100.000	20			Yes	Surface
4	Line	Line 3		80.000	0.000	100.000	80.000	110.000	100.000	20			Yes	Surface
5	Line	Line 4		20.000	20.000	100.000	20.000	70.000	100.000	10			Yes	Surface
6	Line	Line 5		20.000	0.000	100.000	20.000	110.000	100.000	20			Yes	Surface
7	Grid	Grid 2	Global Y	10.000	0.000	90.000	200.000		90.000	57	110.000	33	Yes	Sub-surface
8														

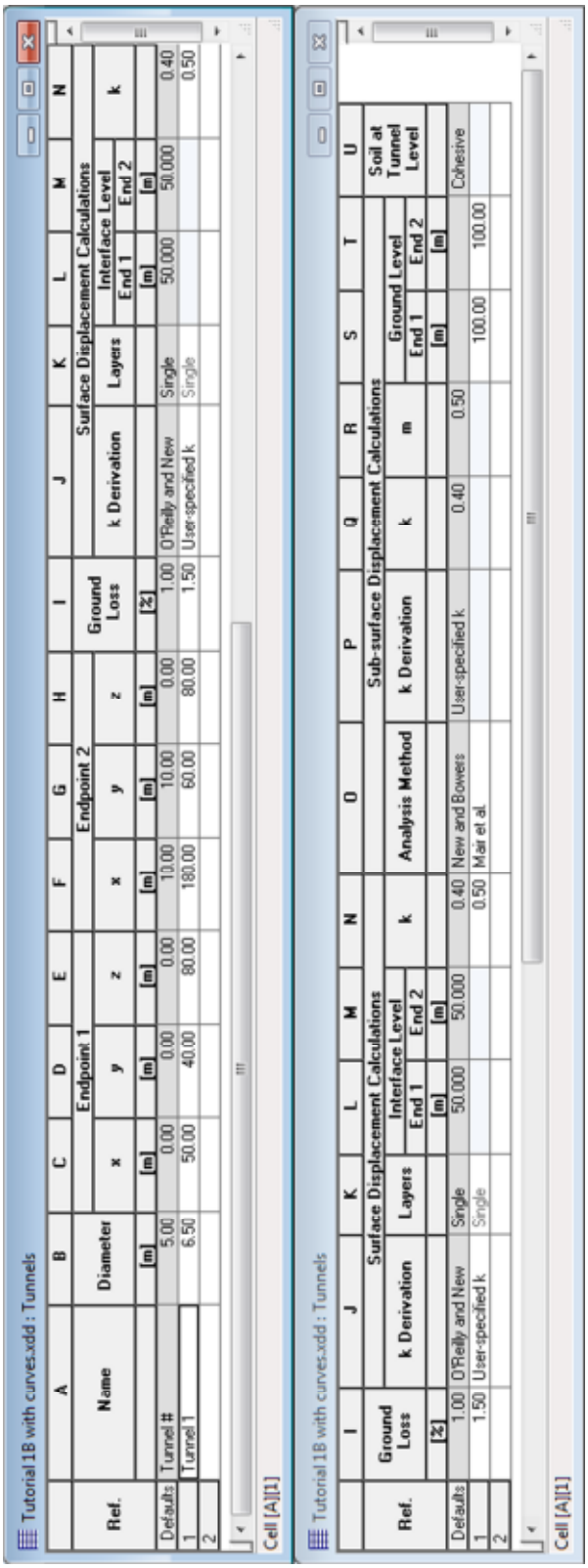
Cell [A][1]

4.1.2 Tunnel Data

1. Double click *Tunnels* in the Gateway.
   Enter the data for the tunnel outlined below:

   Diameter =6.5m
   Start point: (50, 40, 80)
   End point: (180, 60, 80)
   Ground Loss (VL) = 1.5
   Surface k = 0.5 (user specified)
   Sub surface k derivation method is Mair et al.

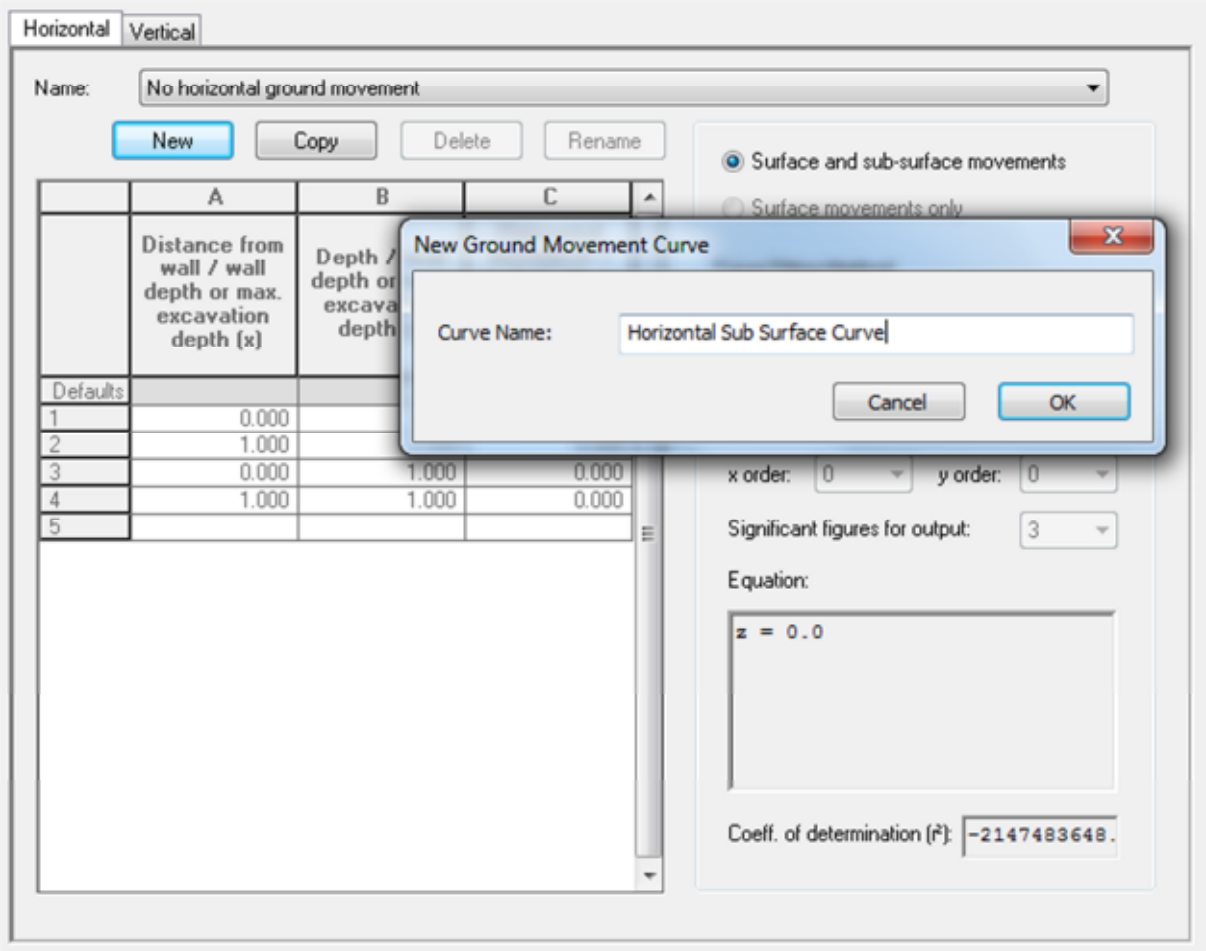
   Check your inputs with those shown on the following page.


Tutorial 1B with curves.xdd : Tunnels
Cell [A][1]
Tutorial 1B with curves.xdd : Tunnels
Cell [A][1]

4.1.3 Embedded Wall Data

1. Double click *Ground Movement Curves* in the Embedded Wall Excavations section of the Gateway.

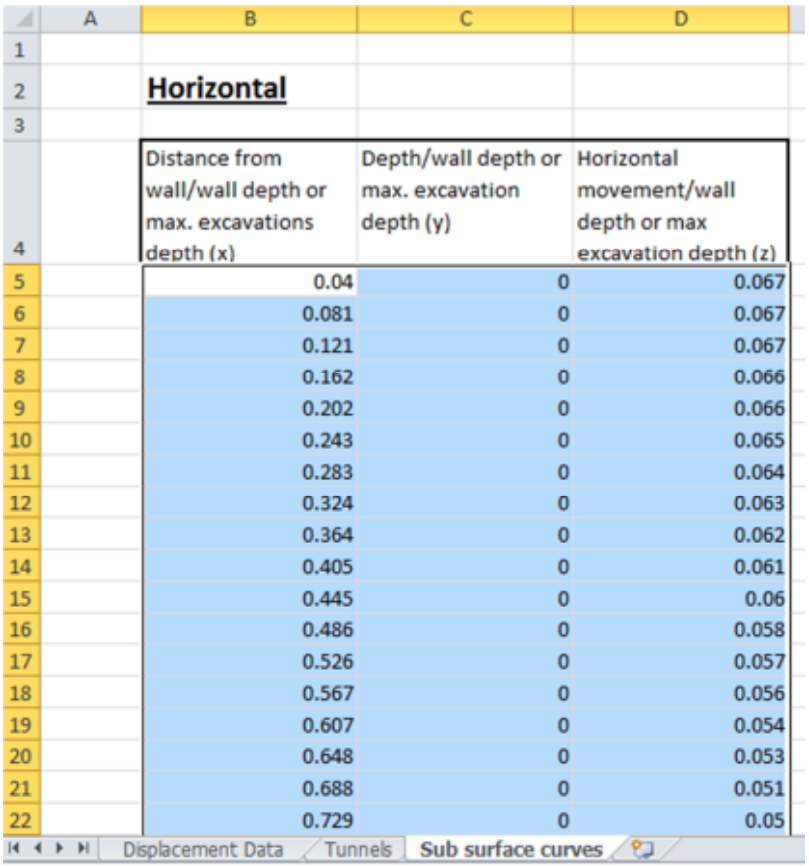
2. In the Horizontal tab, click on the *New* button and enter the name of the new curve, 'Horizontal Sub Surface Curve'



3. Select the *Surface and sub-surface movements* and *Linear Interpolation* options.

   Copy the values from the curve provided in the Excel file and paste into the cells in the Ground Movement Curve table.

(Hint: Remember to click the 'Apply' button before moving onto the next curve)



	A	B	C	D
1				
2		**Horizontal**		
3				
4		Distance from wall/wall depth or max. excavations depth (x)	Depth/wall depth or max. excavation depth (y)	Horizontal movement/wall depth or max excavation depth (z)
5		0.04	0	0.067
6		0.081	0	0.067
7		0.121	0	0.067
8		0.162	0	0.066
9		0.202	0	0.066
10		0.243	0	0.065
11		0.283	0	0.064
12		0.324	0	0.063
13		0.364	0	0.062
14		0.405	0	0.061
15		0.445	0	0.06
16		0.486	0	0.058
17		0.526	0	0.057
18		0.567	0	0.056
19		0.607	0	0.054
20		0.648	0	0.053
21		0.688	0	0.051
22		0.729	0	0.05

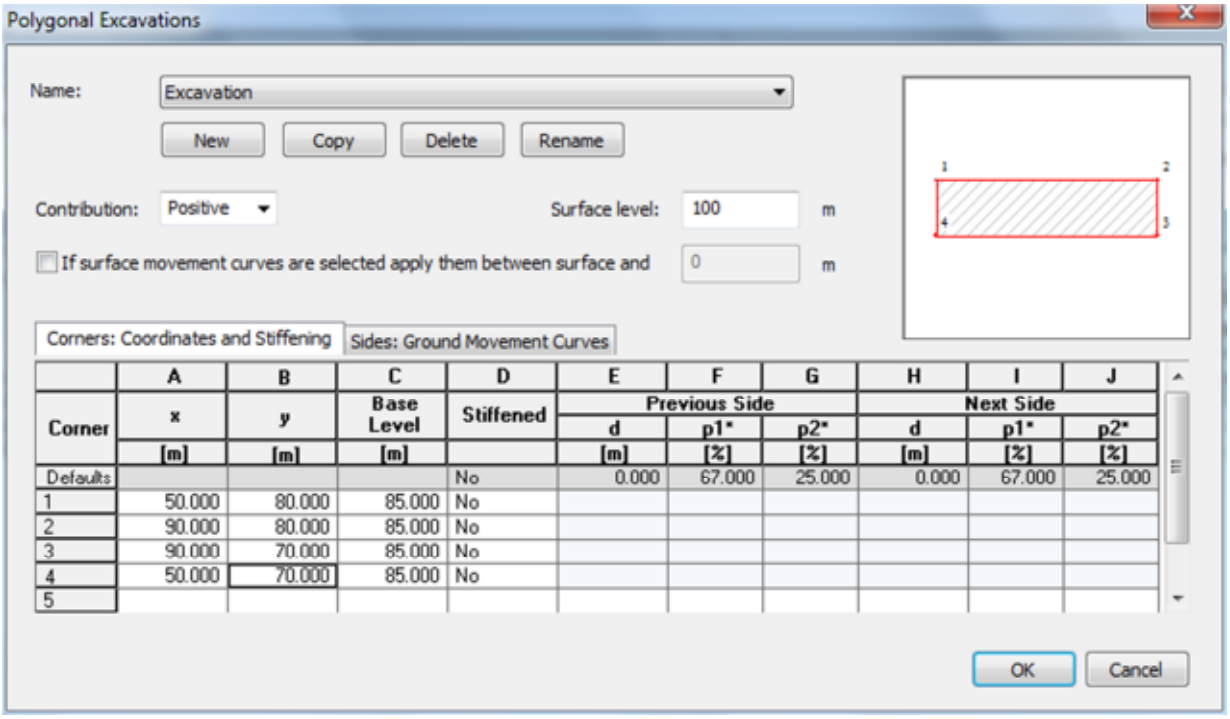
Displacement Data | Tunnels | Sub surface curves

Repeat the previous steps to enter the 'Vertical Sub-surface Curve'

4. Double click on *Polygonal Excavations* in the Embedded Wall Excavations section of the Gateway.

   Enter the co-ordinates for the excavation shown below

   (Shortcut: = copies the value from the cell in the row above)



	A	B	C	D	E	F	G	H	I	J
Corner	x	y	Base Level	Stiffened	Previous Side			Next Side		
					d	p1*	p2*	d	p1*	p2*
	[m]	[m]	[m]		[m]	[%]	[%]	[m]	[%]	[%]
Defaults				No	0.000	67.000	25.000	0.000	67.000	25.000
1	50.000	80.000	85.000	No						
2	90.000	80.000	85.000	No						
3	90.000	70.000	85.000	No						
4	50.000	70.000	85.000	No						
5										

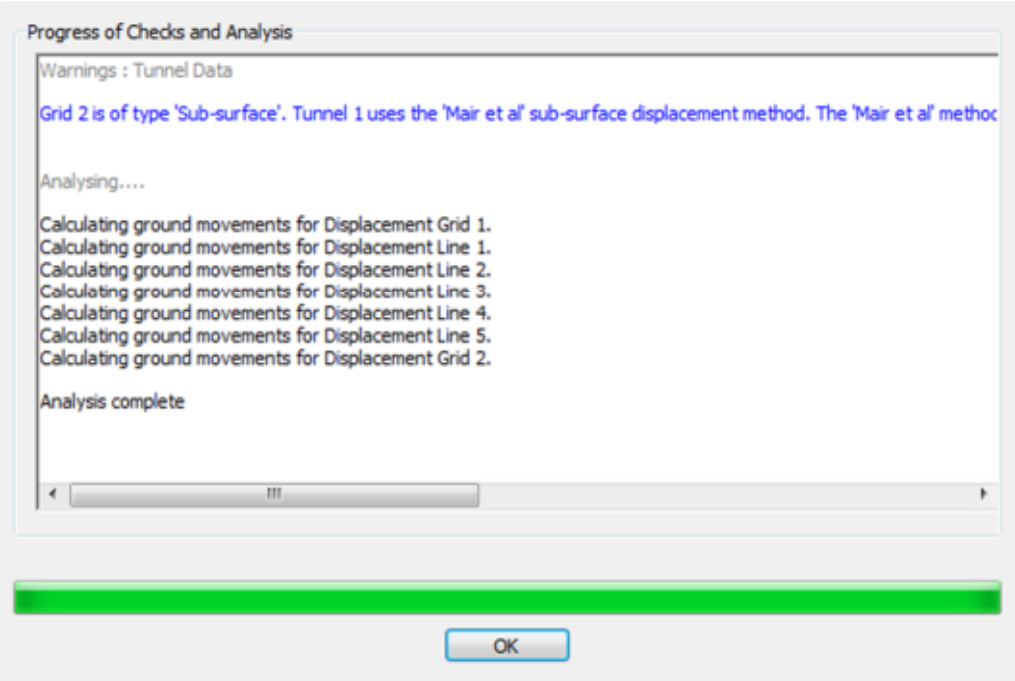
5. Click on the *Sides: Ground Movement Curve* tab.
   Select Vertical and Horizontal Sub-Surface curves for the Vertical and Horizontal columns respectively.

4.2 Calculation and Outputs

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

Σ Click on the analyse button to carry out the analysis

2. The solution progress window will automatically appear and the analysis will run.



(Note: the Warnings shown are a reminder for the user to check that their inputs are appropriate. For example, Grid 2 would not be suitable if the user had only used surface ground movement curves)

3. Click the *OK* button and the tabular outputs will automatically appear.

4.2.1 Exporting tabular results

It is often useful to export results to Excel for further examination, to search for data or plot user defined graphs.

1. Select **File > Export > Tabular Output**
2. Save the file in an appropriate format.
   For Excel files, the user should save the file with the .csv extension

4.2.2 Viewing and Manipulating Graphical Results

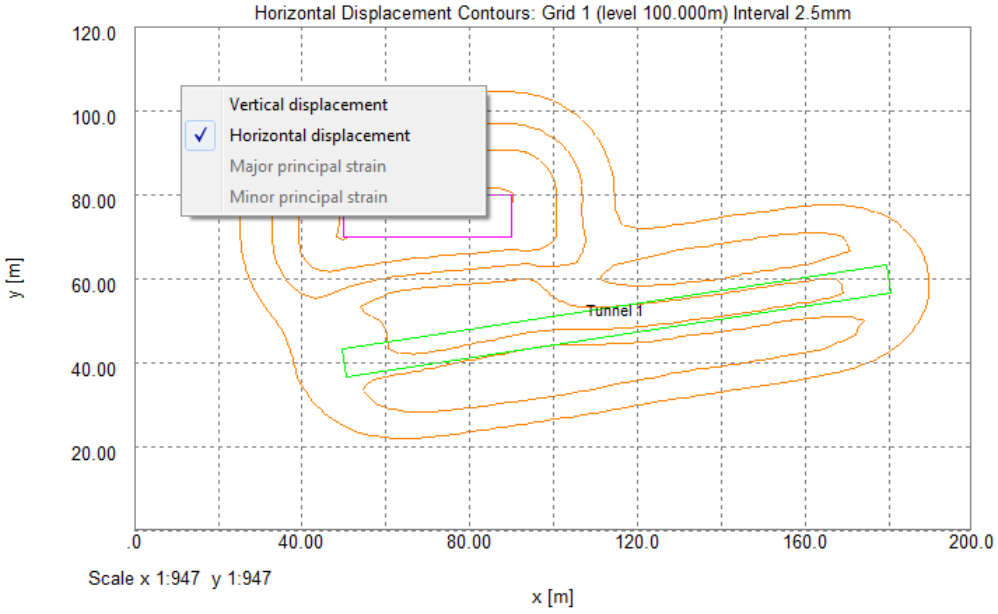
Contours

1. Select *Plan* from the Output section in Gateway

Click on the *Grid* icon to switch off the Grid

Click on the *Contours* icon to view the displacement contours. Should the user wish to view filled contours, they can click on the *Fill Contours* icon.

2. Right click on the Plan View to toggle between vertical and horizontal displacement contours

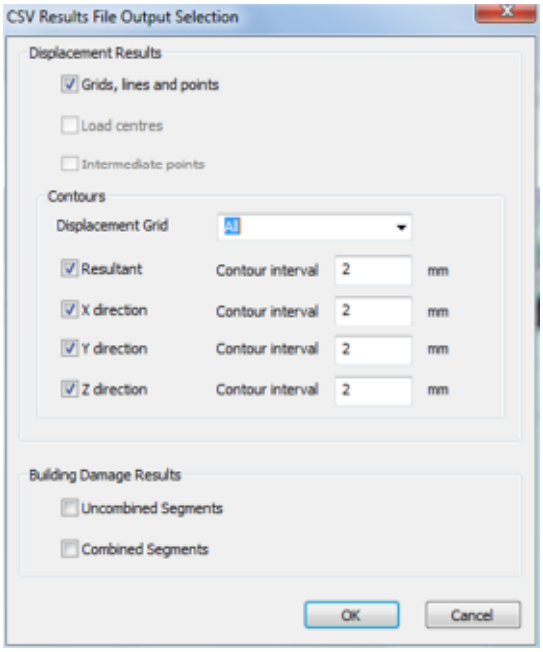


Click on the *Down* icon to view the results for Grid 2

3. To Zoom into an area, left click on the top left point of the zoom window and drag. To zoom out, select the *Unzoom* icon ()

Exporting Contours co-ordinates to Excel

1. Select **File > Export > CSV Results File**
2. Select the options shown below:



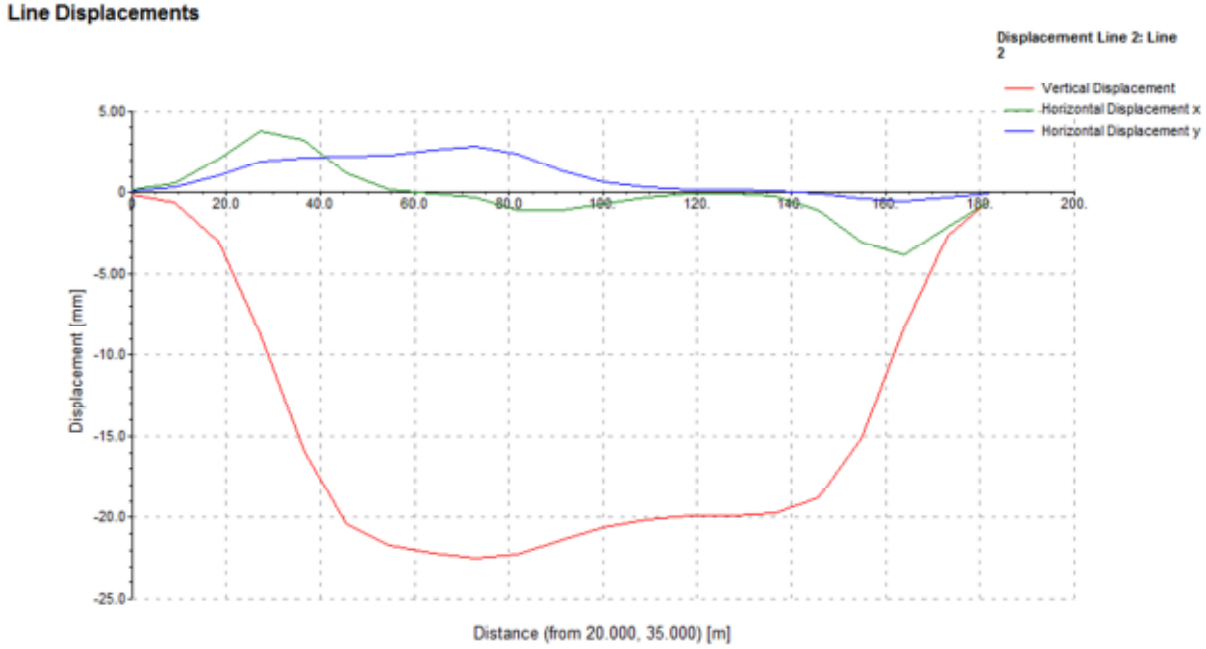
3. Save the csv file. These co-ordinates can be imported into CAD files to overlay contours

   (Note: the csv file is comprised of a series of points for each contour only for ease of input into CAD. Section 4.4 of the user manual covers the format of the output in more detail.)

Graphs

Click on the *Grid* icon to switch on the Grid

Click on the Graph icon and hover the mouse over a displacement line until a black cross appears. Right click to view the graph. An example is shown below.



Graphical Output

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

To navigate the view, the following excerpt from Section 4.2 of the User Manual has been included:

Rotate

The model can be rotated by holding left-click and dragging the mouse. Horizontal drag rotates the model with respect to its z axis. Vertical drag rotates the model with respect to the axis parallel to a horizontal line through the centre of the view.

Zoom

The model can be zoomed in or out by scrolling the mouse wheel or by <ctrl> + drag up or down. The model can be zoomed to its original scale by pressing 'z' from keyboard.

Pan

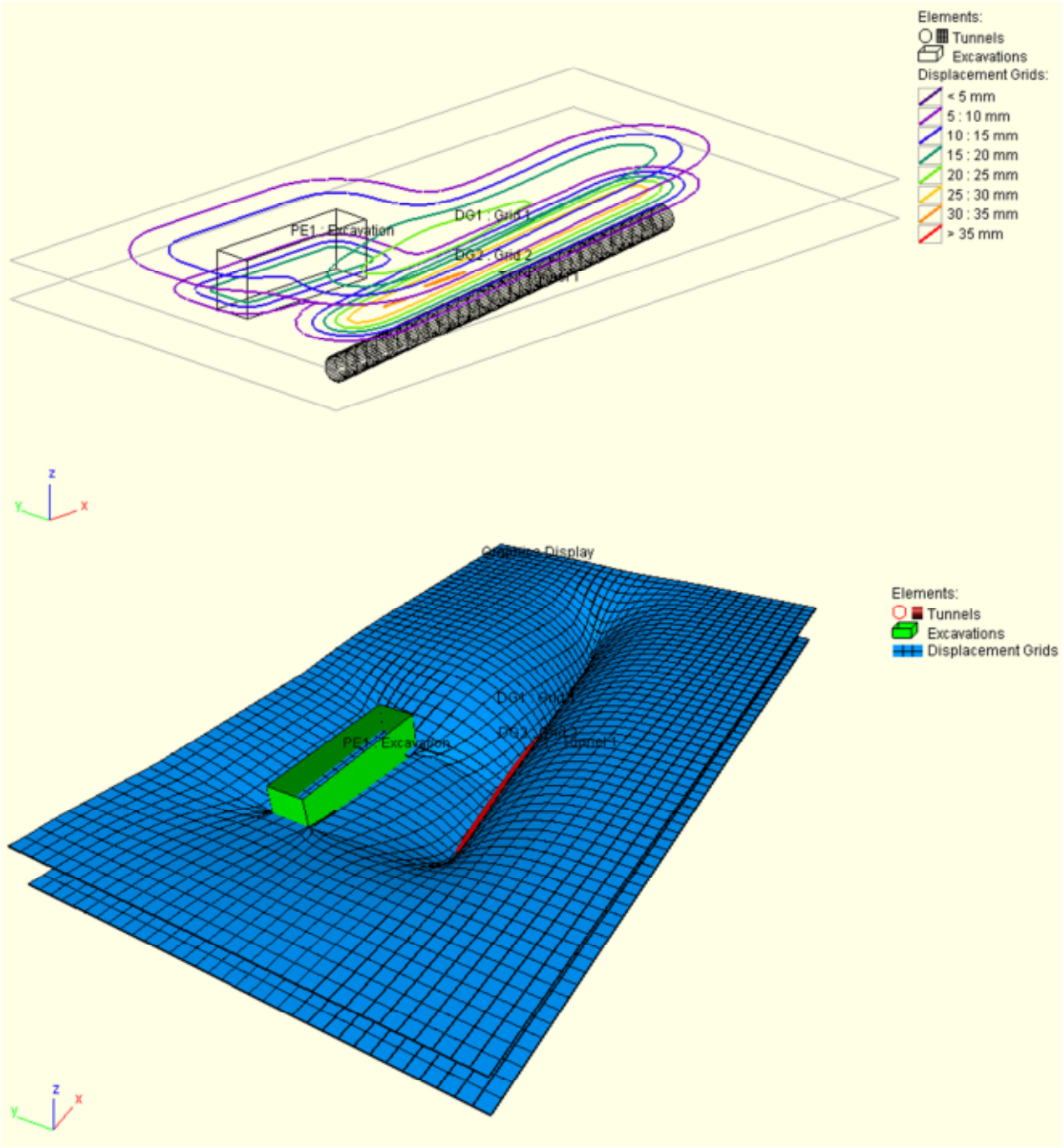
The model can be panned by dragging the mouse with the mouse wheel (or middle button) held down.

Click on the *Wizard* icon or right click on the graphical output view to access the Wizard.

The wizard enables the user to manipulate the output to show inputs, such as tunnels, excavations and displacement points, alongside outputs. These different options can be selected to produce images for reports.

The user should toggle different options to create a suitable image. For example, the displaced grid could be shown and the user should attempt this output.

2. To assess the user's ability with the graphical output, attempt to produce the two outputs shown below.



5. Building and Utility Damage Assessment Tutorials

Objectives

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

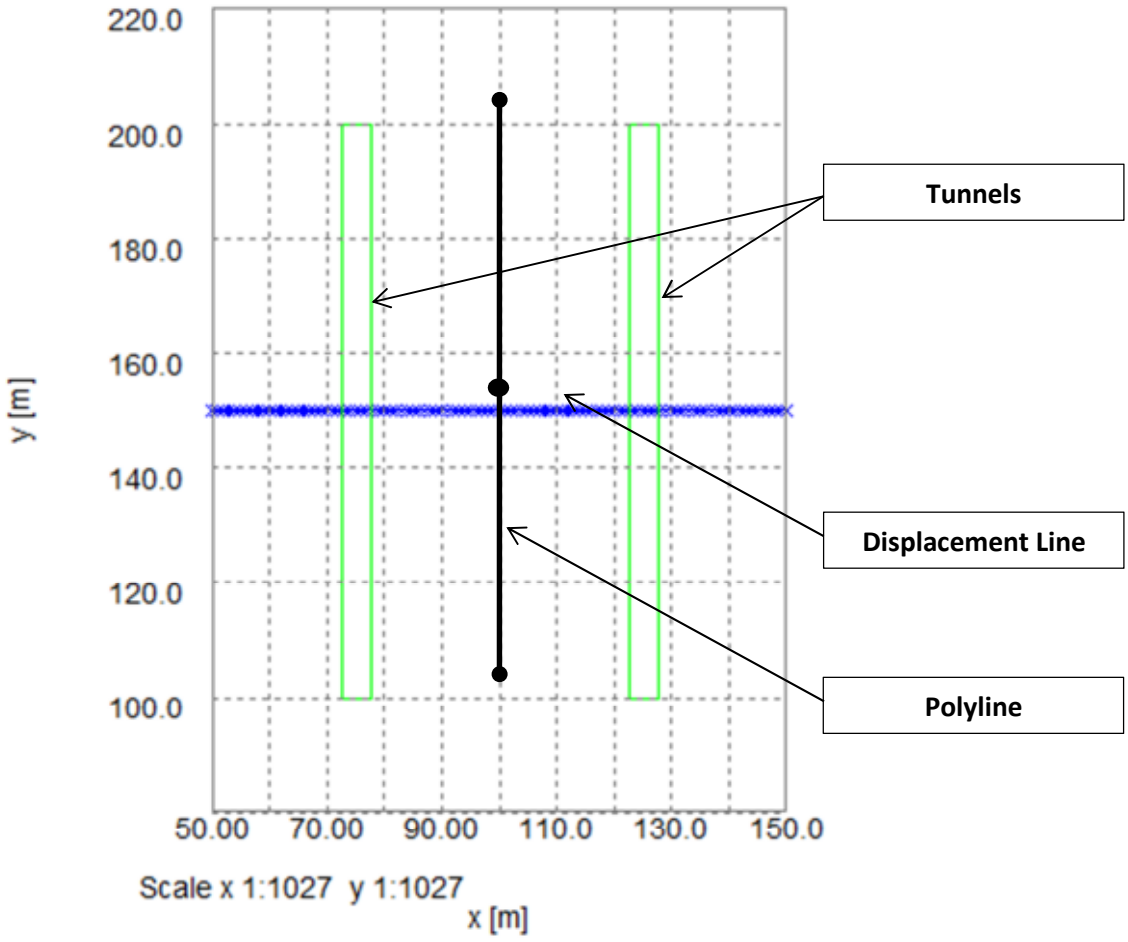
- Enter inputs for a general and specific building and utility damage analysis
- Examine the graphical outputs for a building and utility damage assessment
- Export a Building Damage Assessment report for further analysis

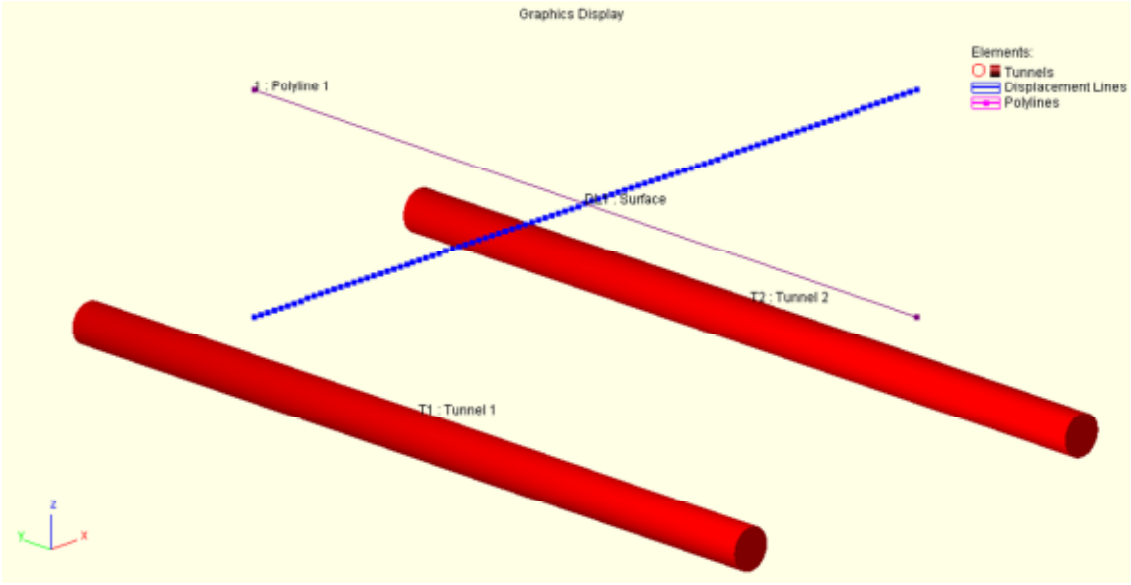
5.1 Base XDisp file for analysis

The base file will be used as a starting point for the various exercises introducing the building and utility damage assessments.

Base File.xdd is provided with this tutorial

The geometry is shown in the diagrams below:





(Note: the polyline is only used for the generic building and utility damage assessments. The displacement line specifies the position of a specific building or utility.).

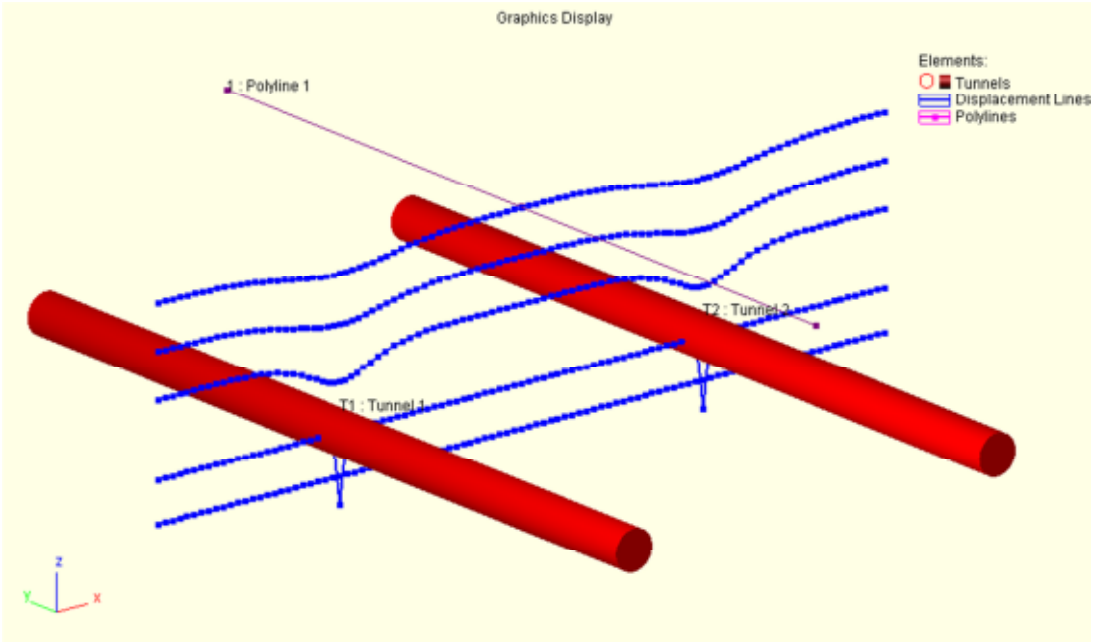
5.2 Checking the base file

Σ Click on the analyse button to carry out the analysis

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

Click on the *Wizard* icon or right click on the graphical output view to access the Wizard.

View the deflected displacement lines to check the results visually:

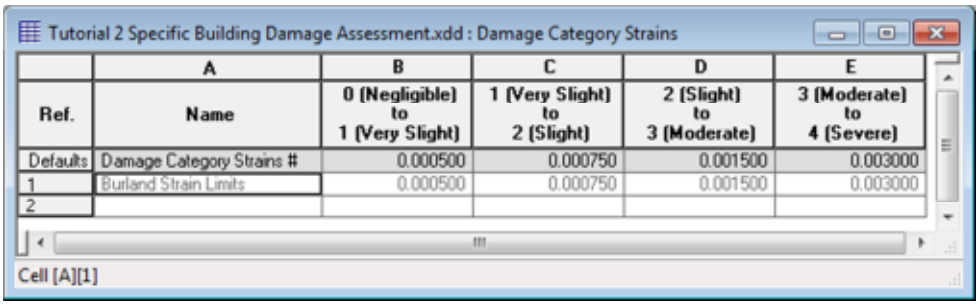


5.3 Exercise 1 – Specific Building Damage Assessment

1. Open Base file.xdd and save as Specific BDA.xdd

5.3.1 Inputs

1. Double click on *Damage Category Strains* in the Building Damage Assessment section of the Gateway.
   View the strain ranges for the categories according to Burland:

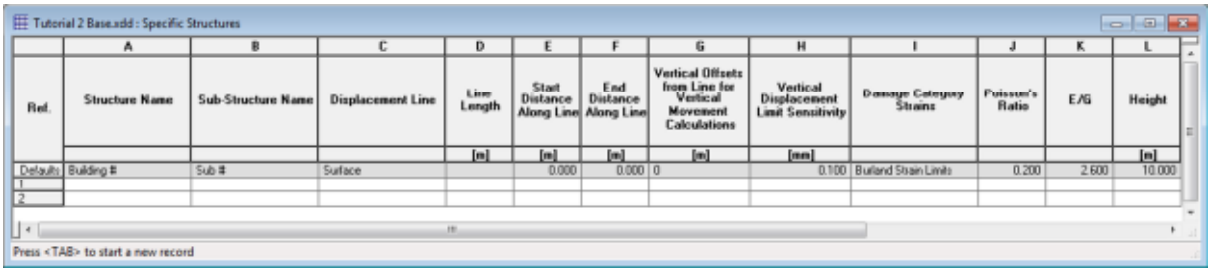


	A	B	C	D	E
Ref.	Name	0 (Negligible) to 1 (Very Slight)	1 (Very Slight) to 2 (Slight)	2 (Slight) to 3 (Moderate)	3 (Moderate) to 4 (Severe)
Defaults	Damage Category Strains #	0.000500	0.000750	0.001500	0.003000
1	Burland Strain Limits	0.000500	0.000750	0.001500	0.003000
2					

   Note:

   If required, the user can change the strain ranges and these will be plotted in the Building Damage Assessment charts to show where the structure lies within the strain categories.

2. Double click on *Specific Structures* in the Building Damage Assessment section of the Gateway.



Ref.	A Structure Name	B Sub-Structure Name	C Displacement Line	D Line Length [m]	E Start Distance Along Line [m]	F End Distance Along Line [m]	G Vertical Offsets from Line for Vertical Movement Calculations [m]	H Vertical Displacement Limit Sensitivity [mm]	I Damage Category Strains	J Poisson's Ratio	K E/G	L Height [m]
Defaults	Building #	Sub #	Surface		0.000	0.000	0	0.100	Burland Strain Limits	0.200	2.600	10.000
1												
2												

   Enter the data for the structure specified below where, other than the position and vertical offsets, all other values are at default.

   Building Position: Along Surface Displacement Line (Section 4.2 of Tutorial Manual)

   Vertical offsets: At surface, 5.5m and 11m below ground level

   (Hint: the vertical offsets can be inputted into one cell by using a comma between values)

   (Note: an example of where vertical offsets are useful would be when the user wants to assess the impact of displacements on the building at pile base, basement level and ground slab level)

To assist the user, the following definitions from Section 3.14.1 of the manual are listed:

Structure Name - a name to identify structure e.g. a building's name.

Sub-Structure Name - a name to identify a sub-structure e.g. one facade of a building.

Displacement Line - the Displacement Line that is to be used to describe the plan alignment of the sub-structure.

Line Length - the length of the Displacement Line that is the maximum length that the sub-structure can have.

Start Distance Along Line - the distance along the Displacement Line that defines the start point of the sub-structure.

End Distance Along Line - the distance along the Displacement Line that defines the end point of the sub-structure.

Vertical Displacement Limit Sensitivity - the minimum value that is to determine the extent of regions of settlement or heave for the sub-structure's building damage assessment calculations. This is an absolute value. It allows settlement or heave profiles, that tend towards zero very gradually, to be curtailed for the purposes of establishing the end hogging or sagging zones in building damage assessment.

Height - the height of the sub-structure from foundation to eaves' level.

Vertical Offsets from Line for Vertical Movement - the vertical offset to be applied to the displacement line before calculating vertical displacements for use in building damage assessments.

A series of offsets may be specified in order to compare building damage results for different elevations e.g. for a piled building: basement level; equivalent pile level; and pile toe level. These vertical movements are used with horizontal movements at the level of the displacement line. See Points of Inflexion, Gradient and Radius of Curvature for further information. More than one value may be entered separated by commas ",". A negative value represents a reduction in elevation.

The Burland method of building damage assessment assumes that a building's façade behaves as a beam in bending.
The follwing parameters in the table provide the information that is required by the Burland method to effect this approximation:

Damage Category Strains - the set of Damage Category Strains that this sub-structure is to adopt to describe the thresholds of each of the 5 damage categories (0 to 4)

Poisson's Ratio - the Poisson's ratio of the beam that is to represent the sub-structure. Values in the range of 0.2 to 0.3 are commonly adopted.

E/G - the Youngs modulus : shear modulus ratio of the beam that is to represent the sub-structure (if the sub-structure is solid, isotropic, linear and elastic then a typical value would be based on Poisson's Ratio, ν, as $2(1 + \nu)$ so ranging from 2.4 to 2.6, if values of 0.2 to 0.3 are used for the Poisson's Ratio.

Burland and Wroth (1974) discuss the effect of E/G ratios but draw no conclusions about appropriate values to use for 'typical' masonry or concrete structures. Mair, Taylor and Burland (1996) state, "For the purposes of assessment of potential damage, framed buildings on shallow foundations can be considered using the same methodology as for masonry structures. It is more appropriate to adopt an E/G ratio of 12.5, rather than 2.6 used for masonry structures".

Melis and Rodríguez Ortiz (2001) suggest "for flexible buildings with big spans or steel structure, the ratio E/G can be as high as 12 or 15".

Default Properties - set 'Yes' for Xdisp to calculate default values for 2nd Moment of Area and neutral axis distances as discussed below, or 'No' to provide specific values.

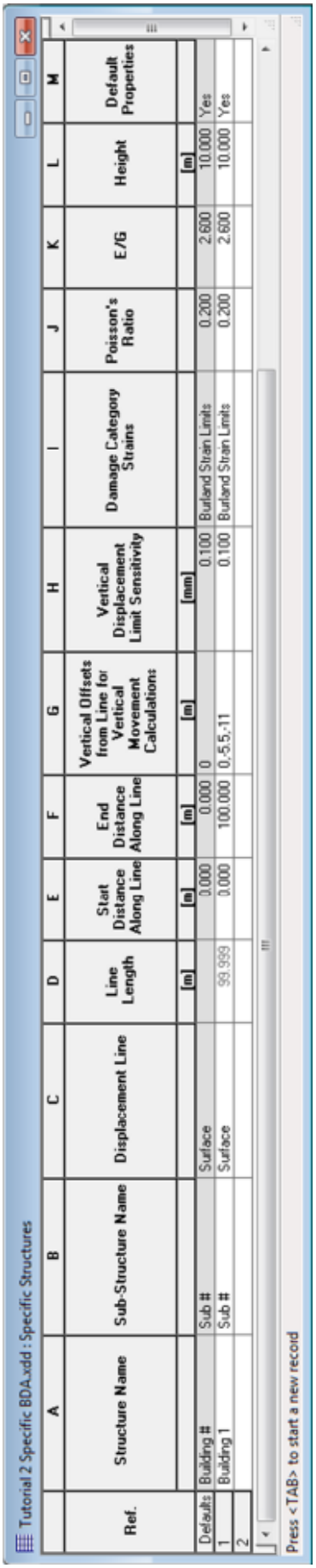
The following data is required for hogging and sagging zones of the building.

Distance of Bending Strain from N.A. - the distance of bending strain to be calculated from the neutral axis. For sagging of a linear isotropic elastic beam a value equal to the height/2 is commonly used. For hogging of a building with a rigid base slab a value equal to the height is commonly used.

Distance of N.A. from Edge of Beam in Tension - distance of the neutral axis from the edge of the beam in tension. For sagging of a linear isotropic elastic beam a value equal to the height/2 is commonly used. For hogging of a building with a rigid base slab a value equal to the height is commonly used.

2nd Moment of Area (per unit width) - adopting the above for Distance of Bending Strain from N.A. and for Distance of N.A. from Edge of Beam in Tension - conventionally for an element of a structure undergoing hogging a value of $d^3/3$ is adopted. For an element of a structure undergoing sagging a value of $d^3/12$ is adopted, see Mair et al (1996).

3. Check your inputs with those shown on the following page.



Tutorial 2 Specific BDA.xdd : Specific Structures

Ref.	A Structure Name	B Sub-Structure Name	C Displacement Line	D Line Length	E Start Distance Along Line	F End Distance Along Line	G Vertical Offsets from Line for Vertical Movement Calculations	H Vertical Displacement Limit Sensitivity	I Damage Category Strains	J Poisson's Ratio	K E/G	L Height	M Default Properties
				[m]	[m]	[m]	[m]	[mm]				[m]	
Defaults:	Building #	Sub #	Surface		0.000	0.000	0	0.100	Burland Strain Limits	0.200	2.600	10.000	Yes
1	Building 1	Sub #	Surface	99.999	0.000	100.000	0,-5:5,-11	0.100	Burland Strain Limits	0.200	2.600	10.000	Yes
2													

Press <TAB> to start a new record

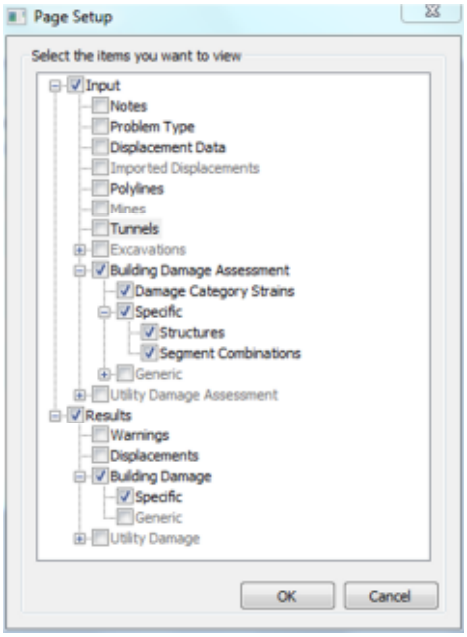
4. Prior to analysis, check all inputs and save the file

Click on the analyse button to carry out the analysis

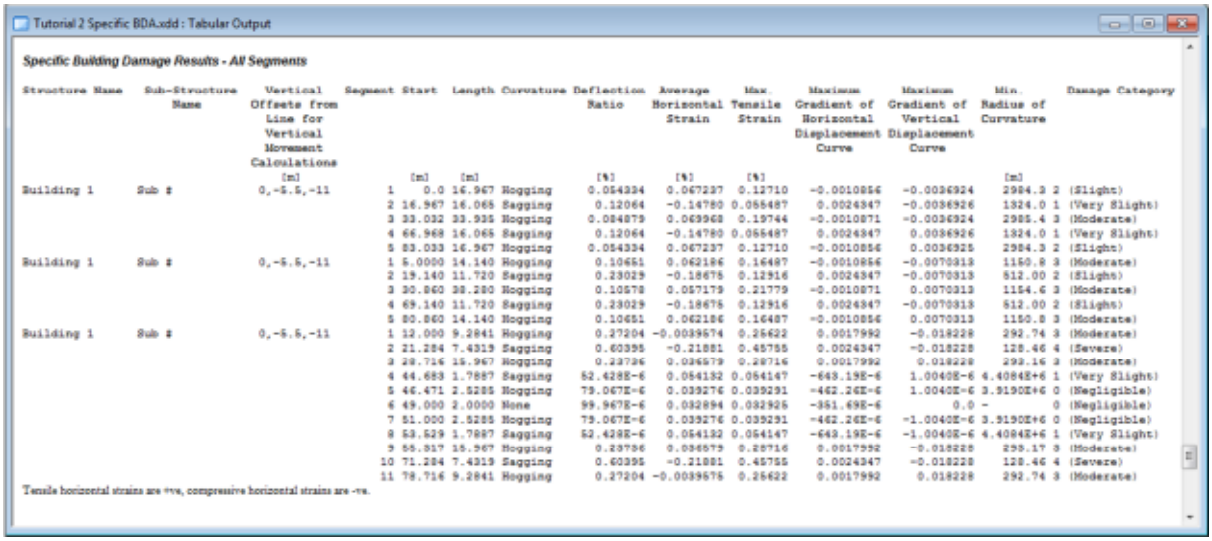
5. The solution progress window will automatically appear and the analysis will run. Click the OK button and the tabular outputs will automatically appear.

5.3.2 Viewing Tabular Results for Building Damage Assessment

1. Double click on *Tabular* in the Output section of the Gateway.
2. Choose the following options to view only the Building Damage Assessment results:



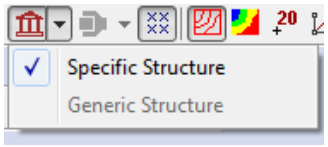
3. As well as giving displacements, the final section outlines the specific results for each segment:



4. This should be examined by the user to understand how the structures are performing. For example, the user can check the damage categories for the different segments of the structure.

5.3.3 Viewing Graphical Outputs for Building Damage Assessment

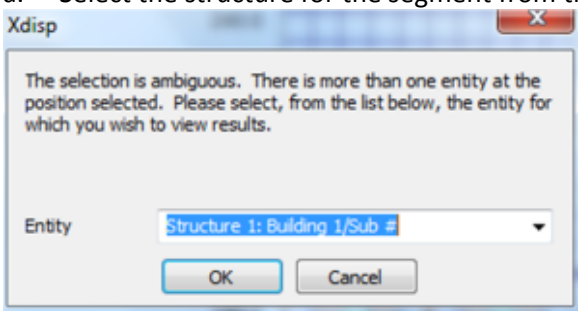
1. Double click on *Plan* in the Output section of the Gateway
2. Click on the *Structures* icon and select 'Specific Structure'



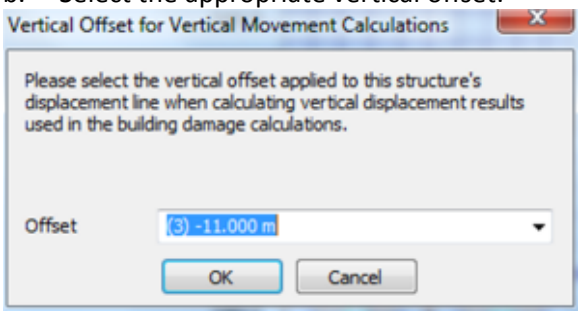
3. Check the Specific Building Damage Results section of the tabular output. Choose a segment which lies in the 'severe' damage category. To view the graph for this segment, follow the steps below:

Select the *Graphs* icon and hover the mouse over the line at y=150m until a cross appears. Left click and a number of options will appear as various graphs can be plotted.

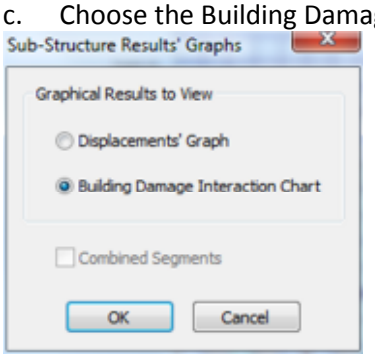
a. Select the structure for the segment from the entity options:



b. Select the appropriate vertical offset:

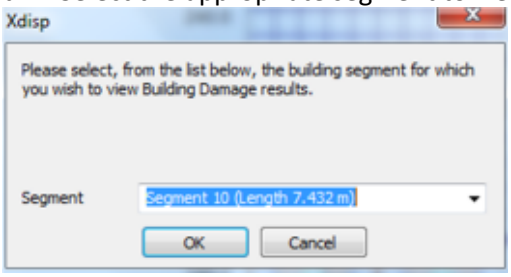


c. Choose the Building Damage Interaction Chart option:

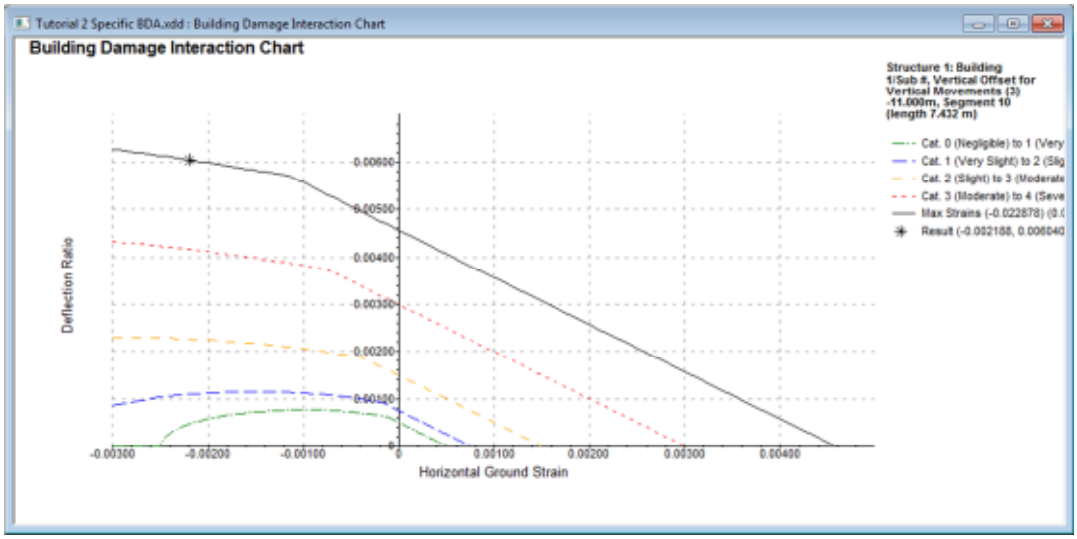


(Note: selecting the Displacement Graph at this point allows the user to view the displacement of the structure)

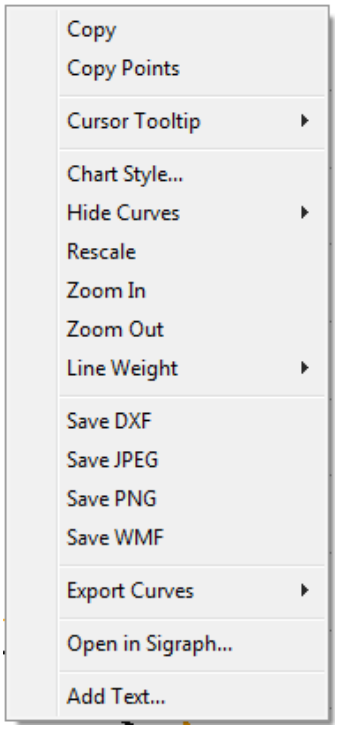
d. Select the appropriate segment to view the chart:



As shown in the tabular output, the segment lies in the Severe Burland damage category.



Right click on the graph to access useful options to view values, change graph styles, save the graph and export curves:



5.3.4 Exporting Building Damage Assessment Results

XDisp allows the user to export the results and graphs for each segment in one click. This is a valuable tool for reports.

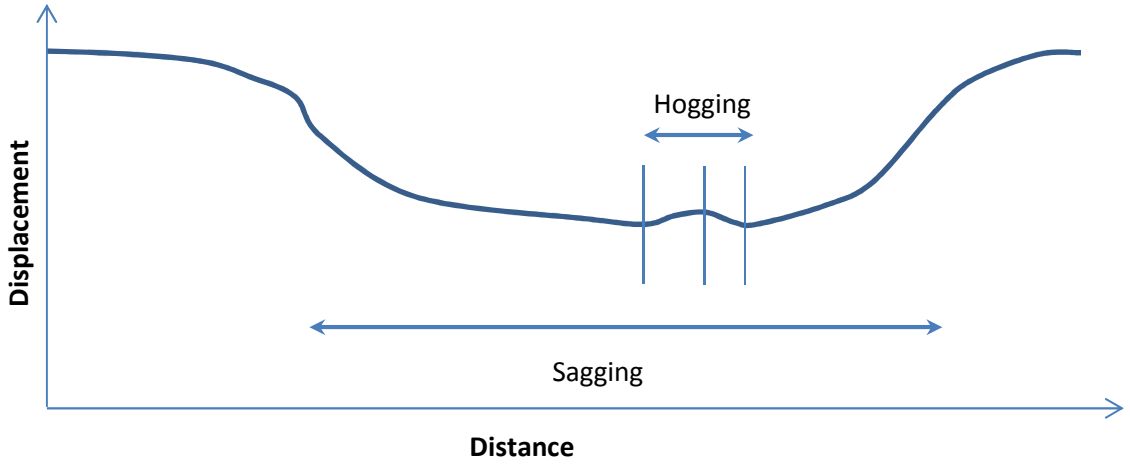
1. Select **File > Export > Building Damage Assessment Data**
2. Save the Folder in an appropriate area and open to view the different files.
   The folder will contain the results in excel and .jpg form with the structure outlined below:

Folders	Files	Content/Comments
<Run Filename>		The folder name is Xdisp data file name
<Structure Name>		One folder is created for each structure. The folder name is the structure name
	Building_Response.csv	
	Classification_of_Damage.csv	
	Critical_Sub_Structure.csv	
<Sub-structure Name>		One folder is created for each sub-structure. The folder name is the sub-structure name
	Classification_of_Damage.csv	
	Strains_in_Sub_Structure_Offset_1.csv	
	Strains_in_Sub_Structure_Offset_2.csv	
	Strains_in_Sub_Structure_Offset_3.csv	
	Sub_Structure_Details.csv	
	Sub_Structure_Response.csv	
	Horizontal_Displacement.csv	
	Vertical_Displacement_Offset_1.csv	
	Vertical_Displacement_Offset_2.csv	
	Vertical_Displacement_Offset_3.csv	
	BD_Interaction_Chart_Offset_1_Segment_n.jpg	One building damage interaction chart file is created for each vertical offset and segment where n is the segment number is created. *Note: chart title refers to "structure and sub-structure with segment details"*
	BD_Interaction_Chart_Offset_2_Segment_n.jpg	
	BD_Interaction_Chart_Offset_3_Segment_n.jpg	
	Sub_Structure_Displacement_Graph_Offset_1.jpg	Graph of horizontal and vertical sub-structure displacements *Note: chart title refers to "structure and sub-structure"*
	Sub_Structure_Displacement_Graph_Offset_2.jpg	
	Sub_Structure_Displacement_Graph_Offset_3.jpg	

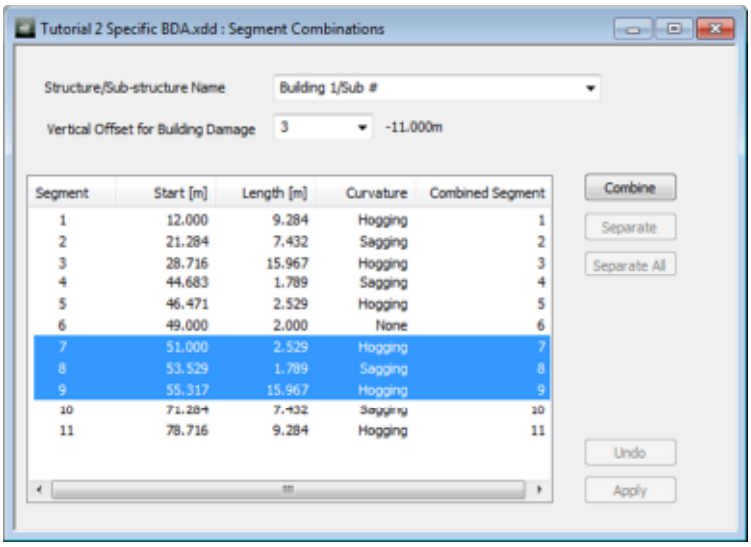
Combining Segments

For more complex files, the displacement graph for the structure may have small segments of hogging or sagging which the user may not wish to consider in the assessment.

For example, in the graph shown below, the sagging segments could be combined with the hogging section to have one sagging section overall.



This is not covered in the tutorial manual but, should the user wish to combine segments once the results have been viewed, they could do so by double clicking on *Segment Combination* on the Specific Building Damage Assessment section of the Gateway.

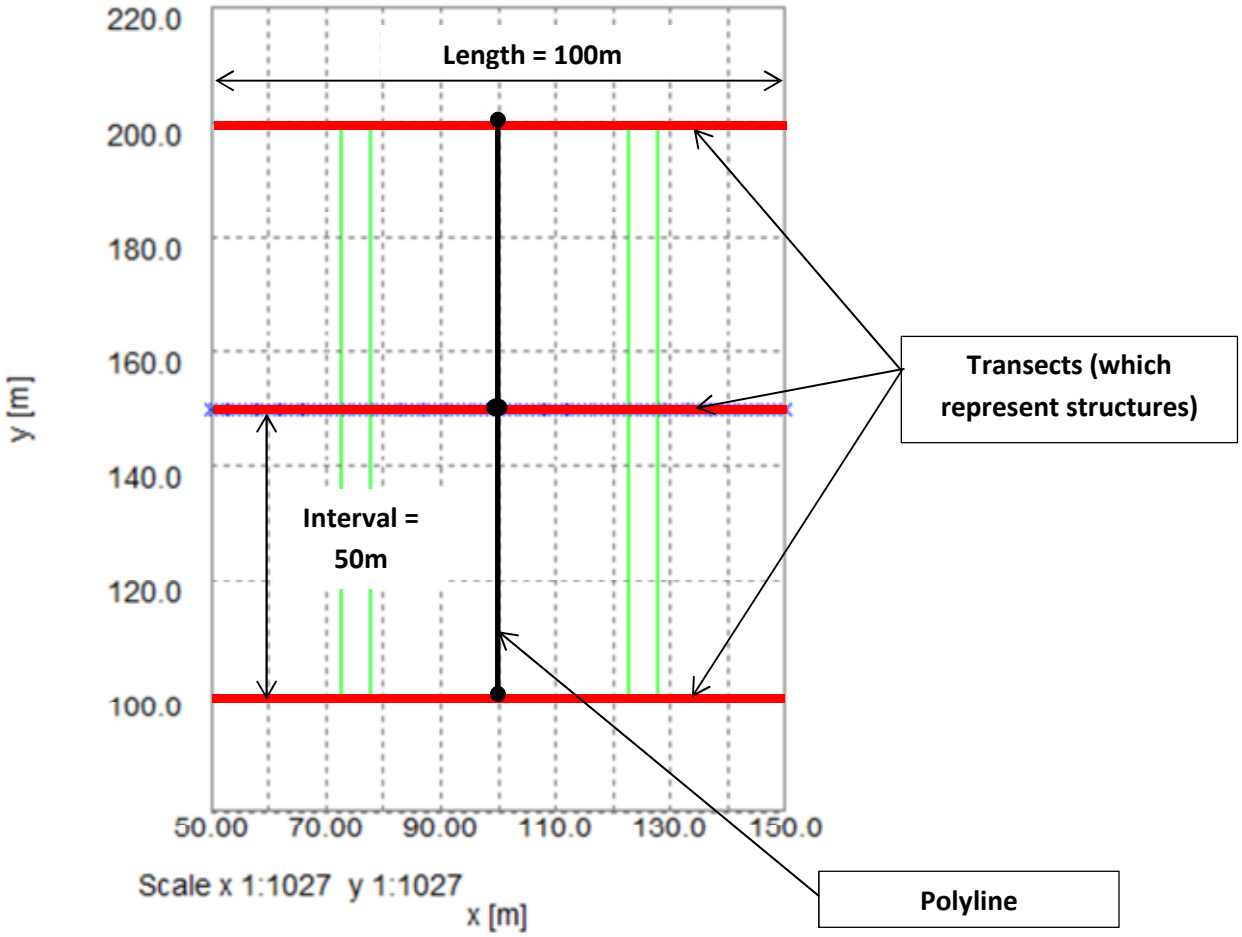


Once combined the results can be re-analysed to give different assessment results.

5.4 Exercise 2 – Generic Building Damage Assessment

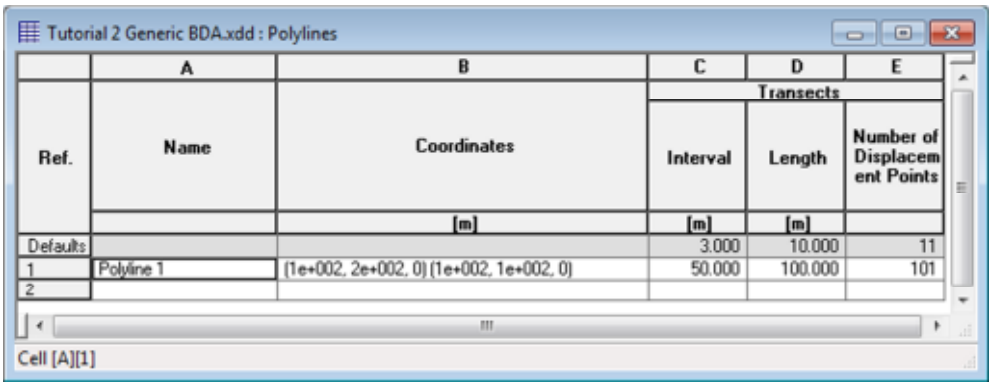
1. Open Base file.xdd and save as Generic BDA.xdd

5.4.1 Inputs



The above image shows how the inputs for Polyline 1 in the *Polyline* Table translate to Transects.

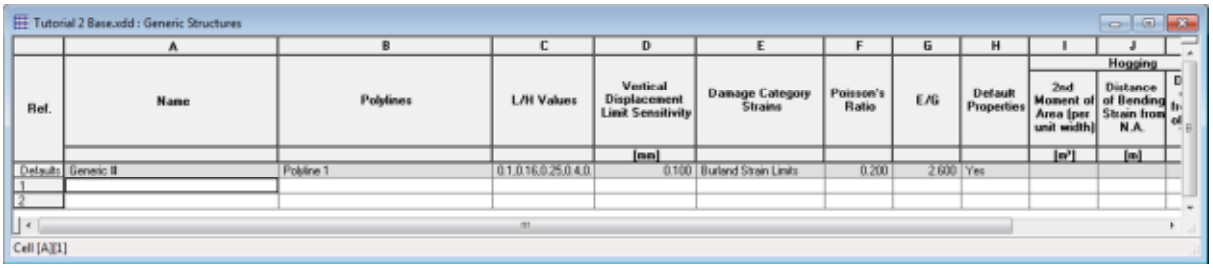
Each of these transects represent Structures and the generic assessment analyses each structure for a range of length : height ratios applied to each hogging and sagging segment along its length.


Tutorial 2 Generic BDA.xdd : Polylines

	A	B	C	D	E
			Transects		
Ref.	Name	Coordinates	Interval	Length	Number of Displacement Points
		[m]	[m]	[m]	
Defaults			3.000	10.000	11
1	Polyline 1	(1e+002, 2e+002, 0) (1e+002, 1e+002, 0)	50.000	100.000	101
2					

Cell [A][1]

2. Double click on *Structures* in the Generic Building Damage Assessment Section of the Gateway.


Tutorial 2 Base.xdd : Generic Structures

	A	B	C	D	E	F	G	H	I	J
									Hogging	
Ref.	Name	Polylines	L/H Values	Vertical Displacement Limit Sensitivity	Damage Category Strains	Poisson's Ratio	E/G	Default Properties	2nd Moment of Area (per unit width)	Distance of Bending Strain from N.A.
				[mm]					[m³]	[m]
Defaults	Generic II	Polyline 1	0.1,0.16,0.25,0.4,0.	0.100	Burland Strain Limits	0.200	2.600	Yes		
1										
2										

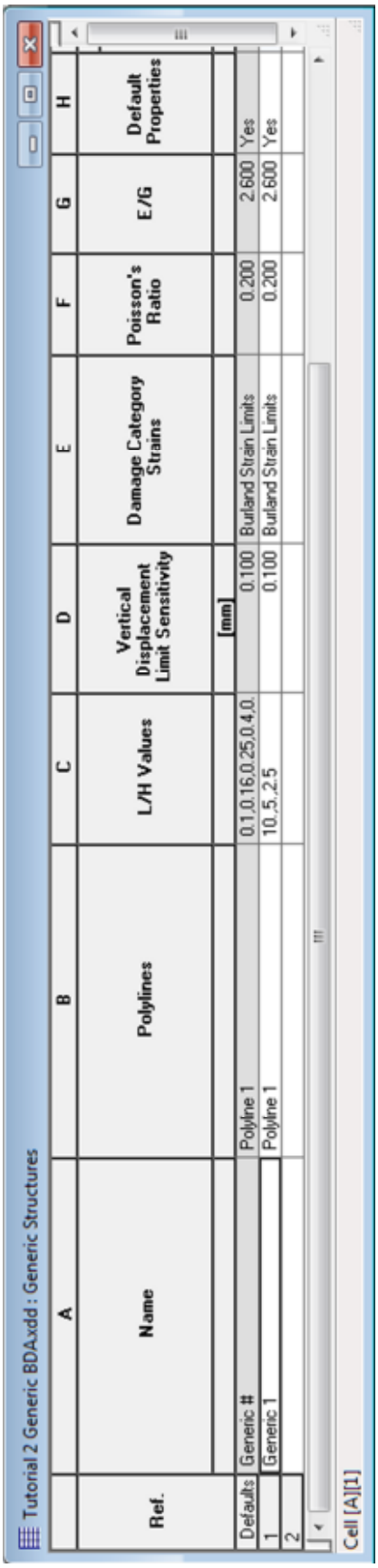
Cell [A][1]

Enter the data for the below for Polyline 1 where, other than the height, all other values are at default.

Transect 1 height = 10m
Transect 2 height = 20m
Transect 3 height = 40m

(Hint: the height is defined using the L/H value in the table. Consequently, for each transect, input the L/H value separated by a comma in the L/H cell)

3. Check your inputs with those provided on the following page.


Tutorial 2 Generic BDAxdd : Generic Structures
Ref.
A
Name
B
Polylines
C
L/H Values
D
Vertical Displacement Limit Sensitivity
[mm]
E
Damage Category Strains
F
Poisson's Ratio
G
E/G
H
Default Properties
Defaults
Generic #
Polyline 1
0.1,0.16,0.25,0.4,0.
0.100
Burland Strain Limits
0.200
2.600
Yes
1
Generic 1
Polyline 1
10..5..2.5
0.100
Burland Strain Limits
0.200
2.600
Yes
2
Cell [A][1]

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

Click on the analyse button to carry out the analysis

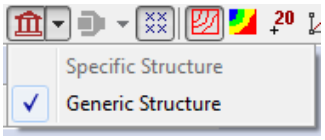
5. The solution progress window will automatically appear and the analysis will run.
   Click the OK button and the tabular outputs will automatically appear.

5.4.2 Viewing Tabular Results for Building Damage Assessment

1. Double click on *Tabular* in the Output section of the Gateway.
2. The results give the displacements for each structure on the transect and damage categories for segments along the transect

5.4.3 Viewing Graphical Outputs for Building Damage Assessment

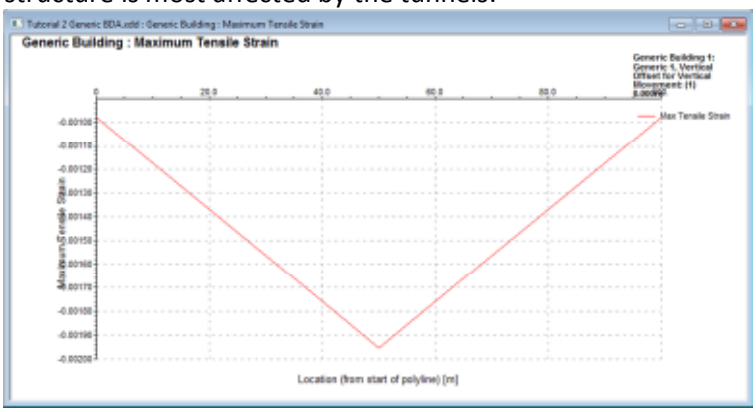
1. Double click on *Plan* in the Output section of the Gateway
   Click on the *Structures* icon and select 'Generic Structure'



Select the *Graphs* icon and hover the mouse over the polyline (x=100m) until a cross appears.

Left click and the graph for Maximum Tensile Strain will be plotted

The graph shows maximum tensile strain along the polyline and demonstrates which structure is most affected by the tunnels:



(Note: should the user wish to view more points on the graph, they should reduce the interval value in the Polyline table and re-analyse. This requires more calculations and the analysis progress may be slower)

5.5 Exercise 3 – Specific Utility Damage Assessment

1. Open Base file.xdd and save as Specific UDA.xdd

5.5.1 Inputs

1. Double click on *Dimensions* in the Utility Damage Assessment section of the Gateway.

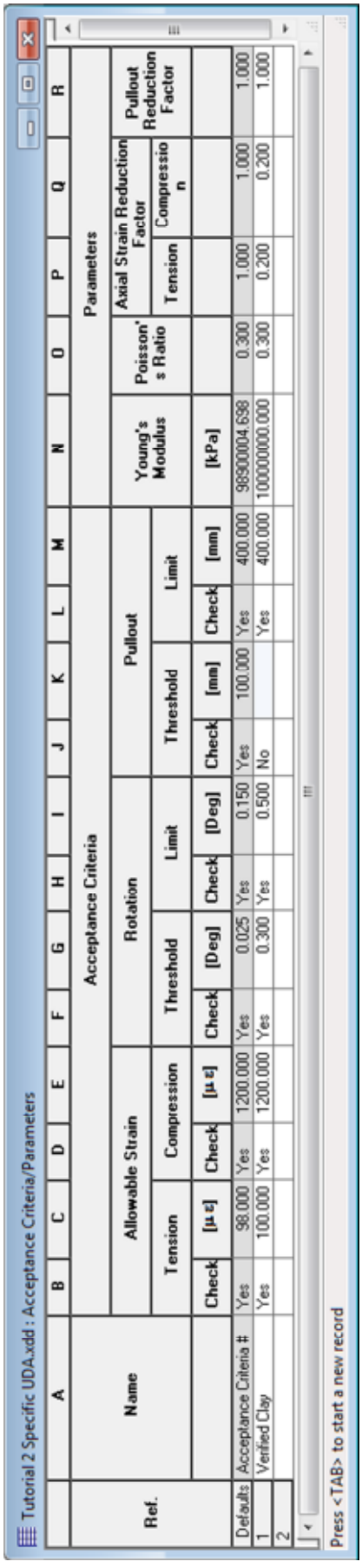
   Input the following for Utility Dimension 1:

   Internal Diameter = 1219mm
   Wall Thickness= 15mm

2. Double click on *Acceptance Criteria/Parameters* in the Utility Damage Assessment section of the Gateway.

   Input the values shown from the example on the next page.

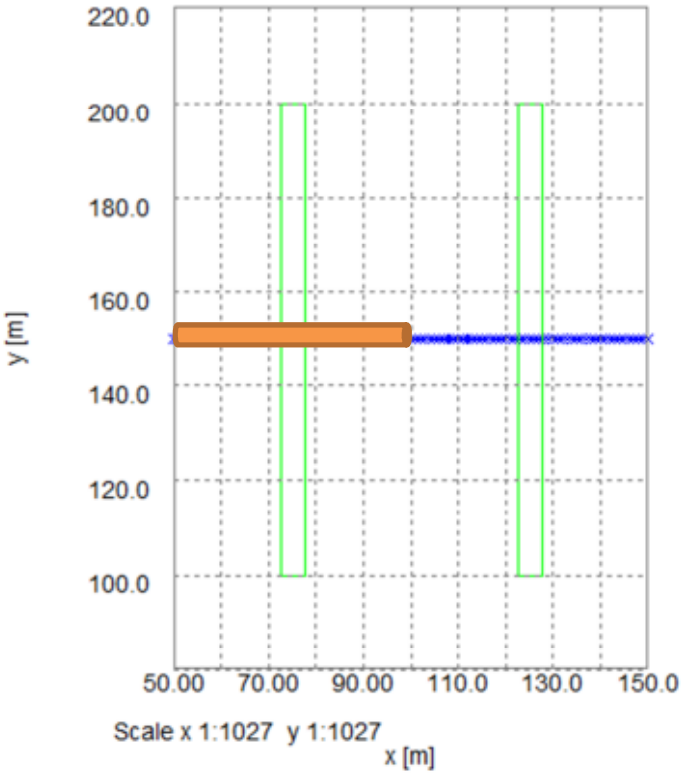
   (Note: The Check option specifies whether the program should check the value against the limit specified in the table for failure.
   If the property is checked and the calculated value is beyond the limit, the segment will be shown to fail in the tabular output)


Tutorial 2 Specific UDA.xdd : Acceptance Criteria/Parameters
Ref.
Name
Acceptance Criteria
Allowable Strain
Tension
Compression
Rotation
Threshold
Limit
Pullout
Threshold
Limit
Parameters
Young's Modulus
Poisson's Ratio
Axial Strain Reduction Factor
Tension
Compression
Pullout Reduction Factor
Check
[με]
[Deg]
[mm]
[kPa]
Defaults
Acceptance Criteria #
Yes
98.000
Yes
1200.000
Yes
0.025
Yes
0.150
Yes
100.000
Yes
400.000
98900004.698
0.300
1.000
1.000
1.000
1
Verified Clay
Yes
100.000
Yes
1200.000
Yes
0.300
Yes
0.500
No
Yes
400.000
10000000.000
0.300
0.200
0.200
1.000
2
Press <TAB> to start a new record

3. Double click on *Specific* in the Utility Damage Assessment section of the Gateway.

   Input the values for a pipe in Verified Clay with the following properties and calculation requirements.

Starts at	(50, 150, -5-5)
Ends at	(100, 150, -5.5)
Segment Length =	20m
Assessment Interval =	1m



4. Check your inputs with those provided in Tutorial 2 Answers.xls

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

Σ Click on the analyse button to carry out the analysis

6. The solution progress window will automatically appear and the analysis will run. Click the OK button and the tabular outputs will automatically appear.

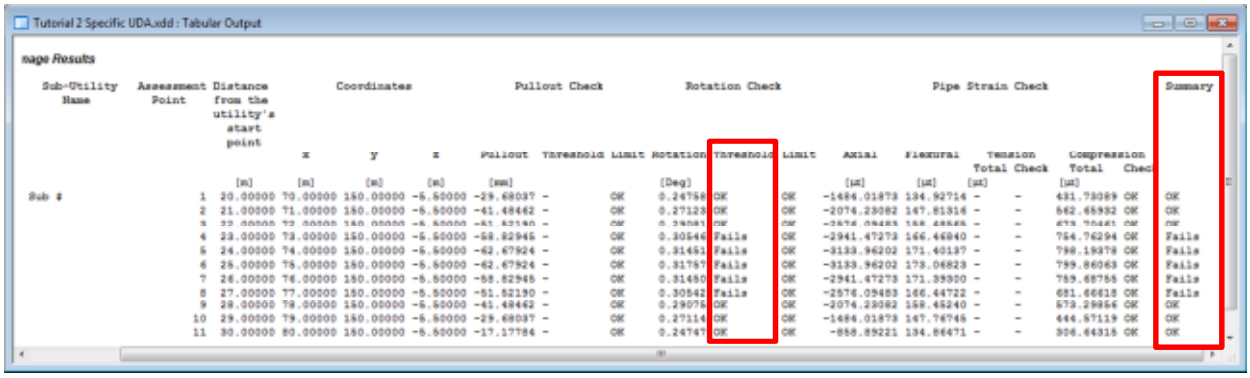
5.5.2 Viewing Tabular Results for Building Damage Assessment

1. Double click on *Tabular* in the Output section of the Gateway.

2. In Page Setup, select the Specific Utility Damage Assessment results only

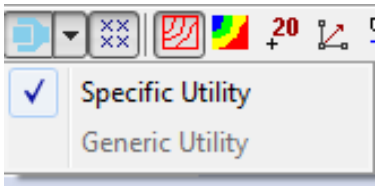
3. The user should examine the tabular output to determine if their utility fails on any of the limits they had checked in the *Acceptance Criteria/Parameters* section of the Gateway.

   The output shows that some segments of the utility are failing on the threshold values for Rotation:



5.5.3 Viewing Graphical Outputs for Building Damage Assessment

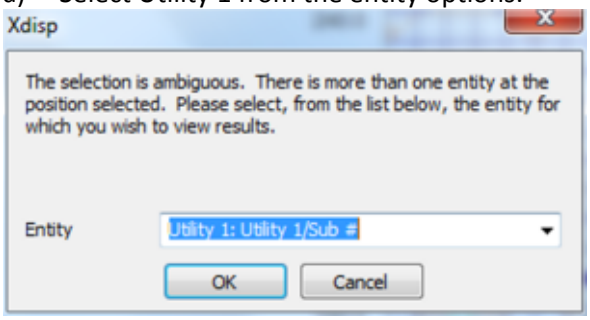
1. Double click on *Plan* in the Output section of the Gateway
   Click on the *Utilites* icon and select 'Specific Utility'



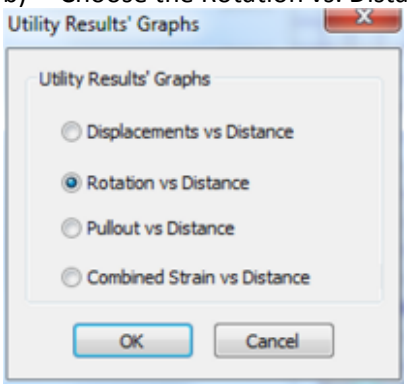
Select the *Graphs* icon and hover the mouse over the utility (y=150m) until a cross appears. Left click and a number of options will appear as the utility also lies on the displacement lines so various graphs can be plotted.

The following steps will demonstrate how to plot the graph that shows failure of the utility due to Rotation.

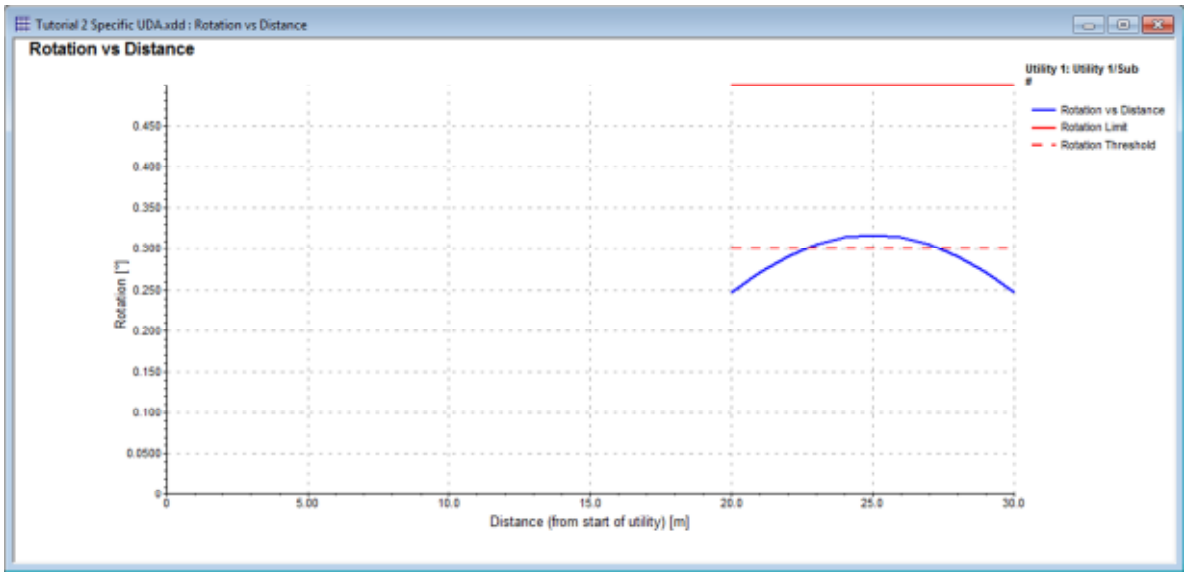
a) Select Utility 1 from the entity options:



b) Choose the Rotation vs. Distance chart option:



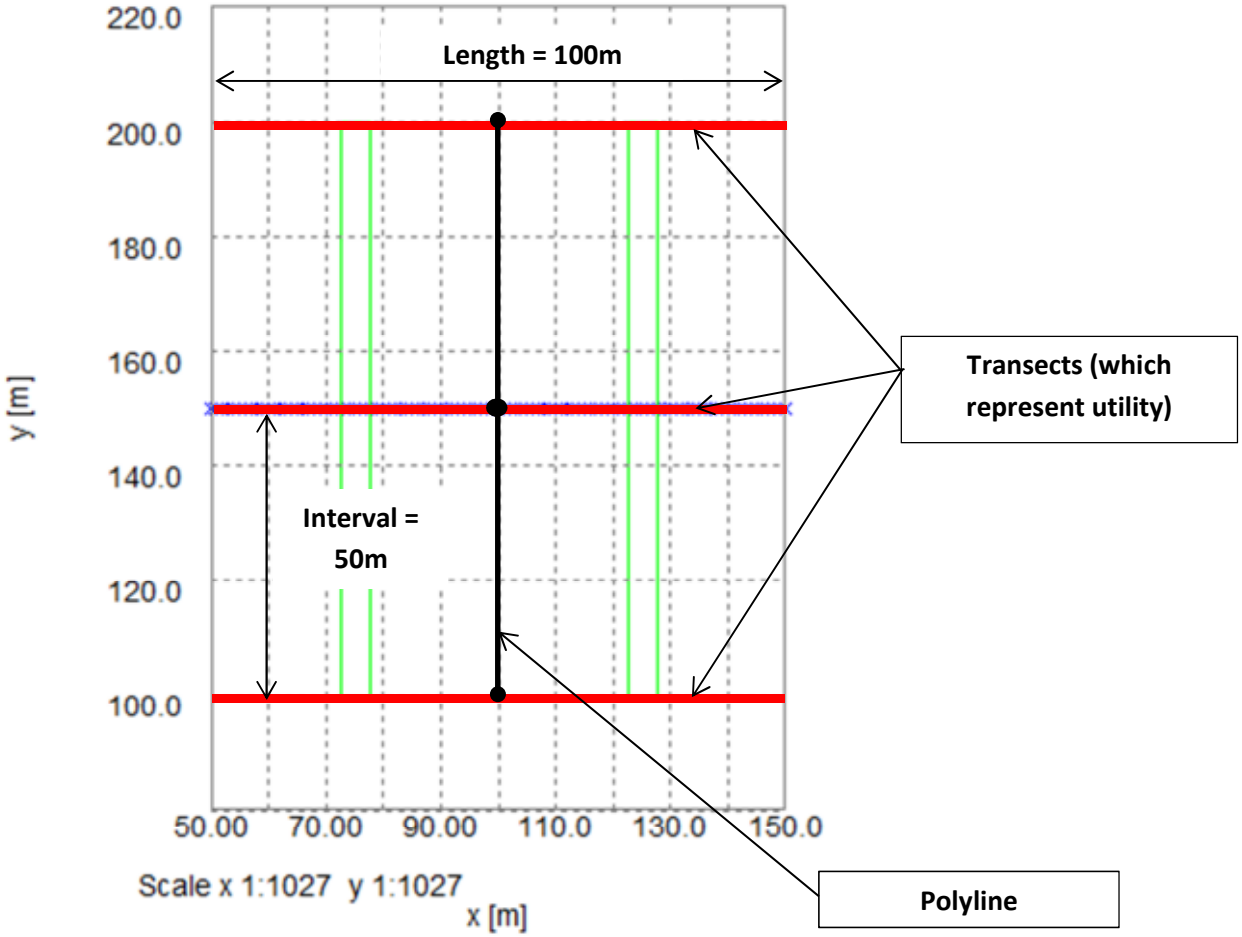
The following graph will be plotted:



5.6 Exercise 4 – Generic Utility Damage Assessment

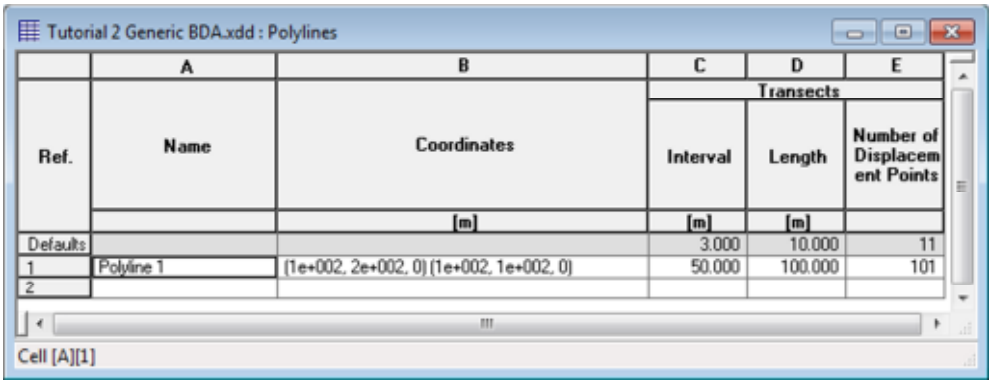
Open Base file.xdd and save as Generic UDA.xdd

5.6.1 Inputs



The above image shows how the inputs for Polyline 1 in the *Polyline* Table translate to Transects.

Each of these transects represent Utilities and the generic assessment analyses each utility.



2. Copy the values for *Dimensions* and *Acceptance Criteria/Parameters* from Specific UDA.xdd

3. Double click on *Generic* in the Utility Damage Assessment section of the Gateway.

   For Polyline 1, specify utilities in Verified Clay which has a pipe segment length of 20m and calculation interval of 1m.

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

Σ Click on the analyse button to carry out the analysis

5. The solution progress window will automatically appear and the analysis will run. Click the OK button and the tabular outputs will automatically appear.

5.6.2 Viewing Tabular Results for Building Damage Assessment

1. Double click on *Tabular* in the Output section of the Gateway.

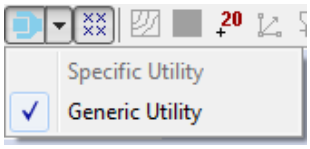
2. In Page Setup, select the Generic Utility Damage Assessment results only

3. The user should examine the tabular output to determine if their utility fails on any of the limits they had checked in the *Acceptance Criteria/Parameters* section of the Gateway.

   The output shows that, for Transect 2, more segments are failing due to reaching the Tensile Strain threshold.

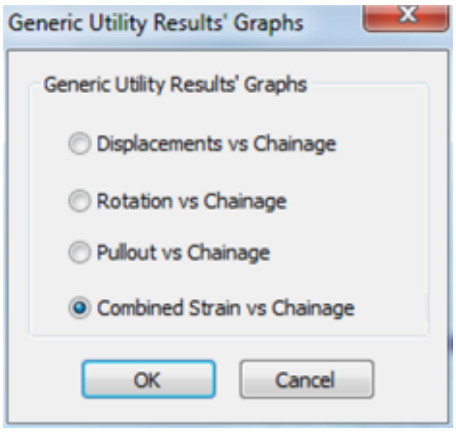
5.6.3 Viewing Graphical Outputs for Building Damage Assessment

1. Double click on *Plan* in the Output section of the Gateway
2. Click on the *Utilites* icon and select 'Generic Utility'



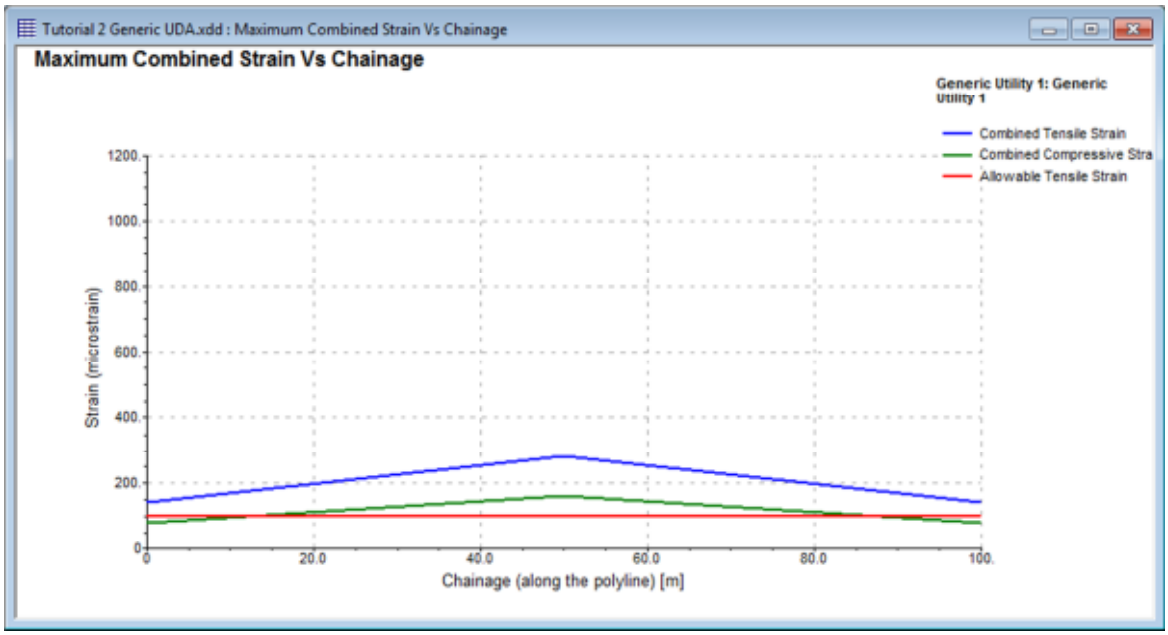
Select the *Graphs* icon and hover the mouse over the polyline (x=100m) until a cross appears.

3. Left click and choose the Combined Strain vs. Distance chart option:



(Note: if the user clicks on x=100m, near the displacement lines, they will have to choose the utility instead of the displacement lines before being prompted to choose the Utility results graph)

The graph shows that the utility at the centre of the polyline has the most significant strain. Also, all three utilities lying on the transects have maximum tensile strains over the threshold (allowable) strain. Consequently, all three utilities fail.


Tutorial 2 Generic UDA.xdd : Maximum Combined Strain Vs Chainage
Maximum Combined Strain Vs Chainage
Generic Utility 1: Generic Utility 1
Combined Tensile Strain
Combined Compressive Stra
Allowable Tensile Strain
Strain (microstrain)
1200.
1000.
800.
600.
400.
200.
0
0
20.0
40.0
60.0
80.0
100.
Chainage (along the polyline) [m]

6. Importing CAD Drawings

XDisp allows the user to import displacement lines, excavations and displacements from CAD drawings.

To create a suitable file, the dxf lines should be assigned to layers named Tunnels, Buildings or Excavations. This is covered in more detail in Section 3.8 of the manual.

Objectives

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

- Import dxf files
- Consolidate their learning from previous tutorials

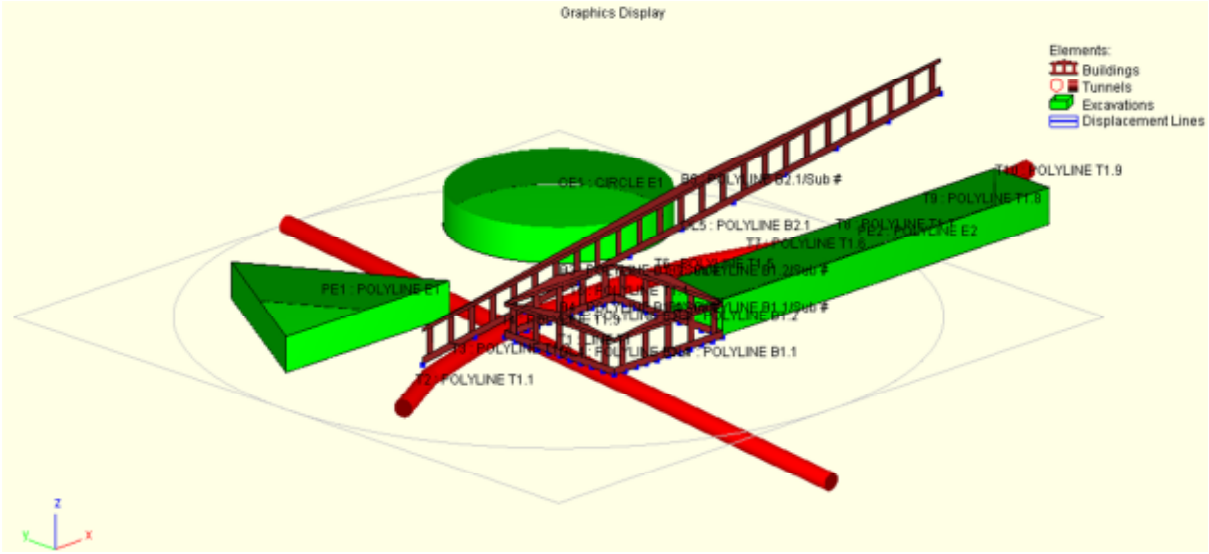
6.1 Import DXF File

Create a new file by clicking the 'New File' icon on the Standard toolbar or clicking Ctrl + N

1. Select File > Import > DXF File
2. Import the DXF elements.dxf

   (Note: DXF background.dxf has no elements assigned to Tunnels, Buildings or Excavations. These files can be used as plans to determine the orientation of the buildings, tunnels and excavations)

3. View the *3D Graphics* Output:



5.2 Test yourself

Using the file created, the user should attempt the following exercises:

1. Assign Ciria C580 curves for the Polygonal and Circular excavations
2. Change Polyline B2.1 to a utility and apply appropriate values (from Tutorial 2)
3. Analyse
4. Determine Building performance
5. Produce graphs

7. Graphical Input

Objectives

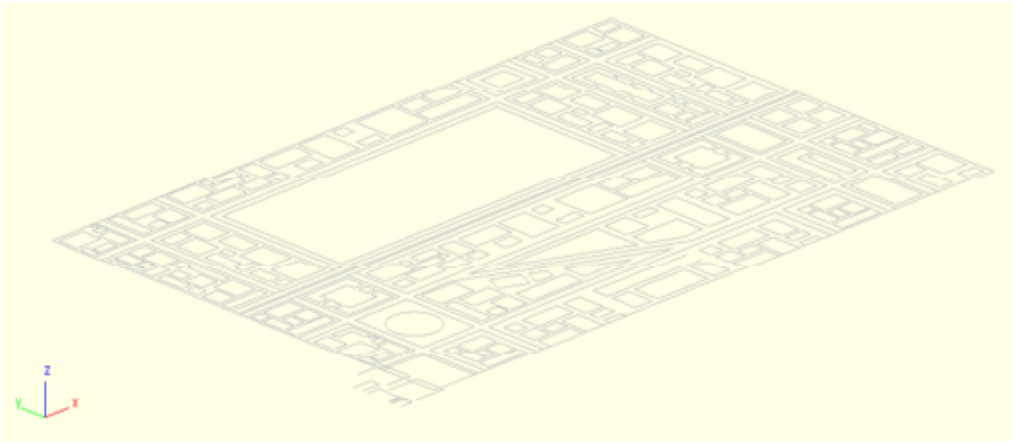
By the end of the session the user should be able to:

- Input displacement data, tunnels and building graphically

7.1 Graphical Input

Create a new file by clicking the 'New File' icon on the Standard toolbar or clicking Ctrl + N

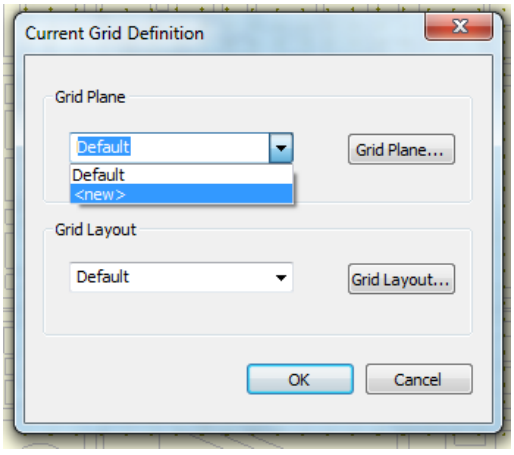
1. Import the CAD drawing site_plan_original for import.dxf



In the *Graphical Output* window, click on the 'Input/Output' icon to switch to graphical input

Click on the 'Grid Plane' to define the grids that you will use to draw.

2. Use the 'Current Grid Definition' window to enter grids at 0mOD (Grid Plane 1) and -10mOD (Grid Plane 2).
   Select an x and y spacing of 10m using the Grid Layout button



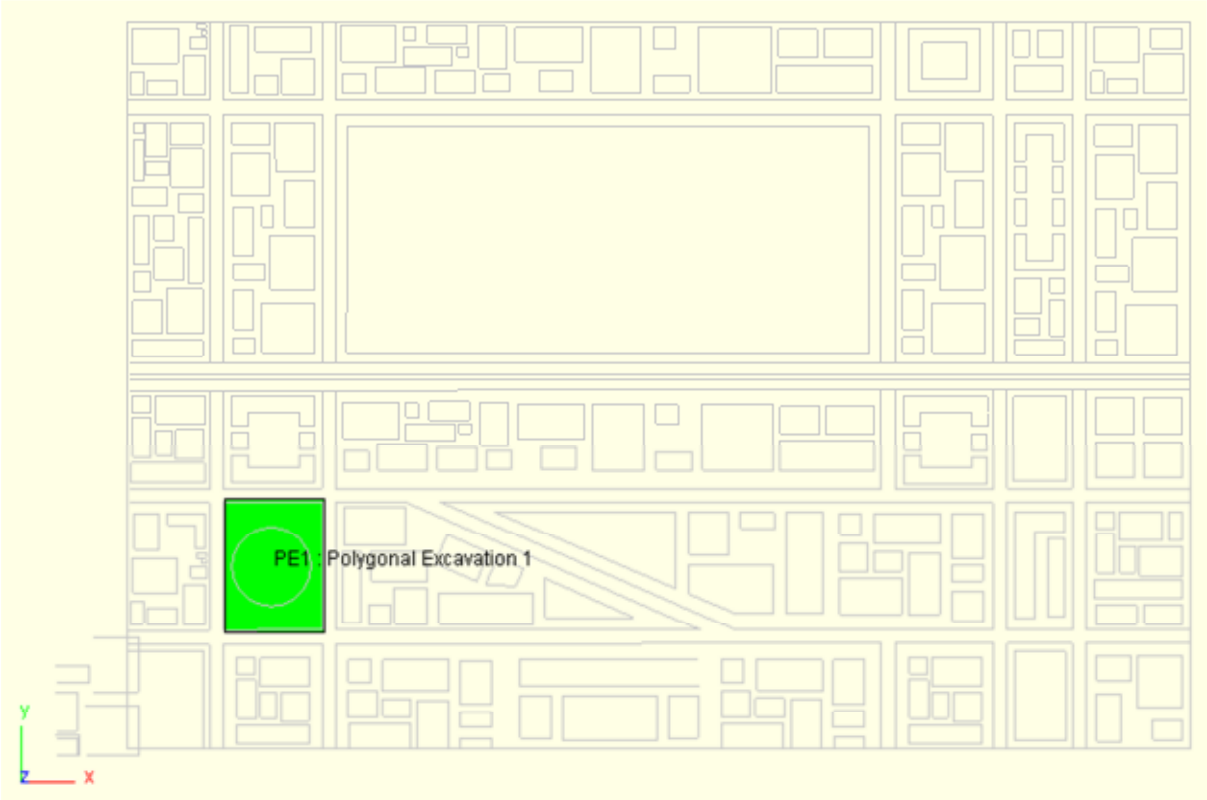
Click on the ‘Snap’ icon to enable the user to snap to points on the grid when sculpting

Click on ‘Add Polygonal Excavation’ and draw an excavation in the location shown below.

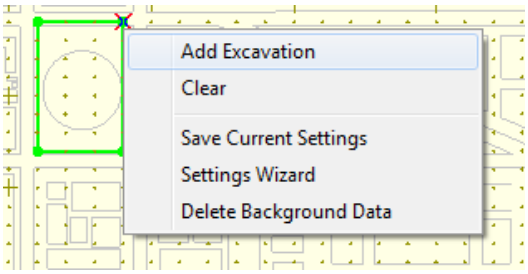
The excavation surface is at 0mOD and is 15m deep.

3. Assign the excavation surface movement curves from Ciria 580 (Excavation in front of high stiffness wall in stiff clay)

   (Hint: Check that the right Grid Plane is selected for the excavation surface)

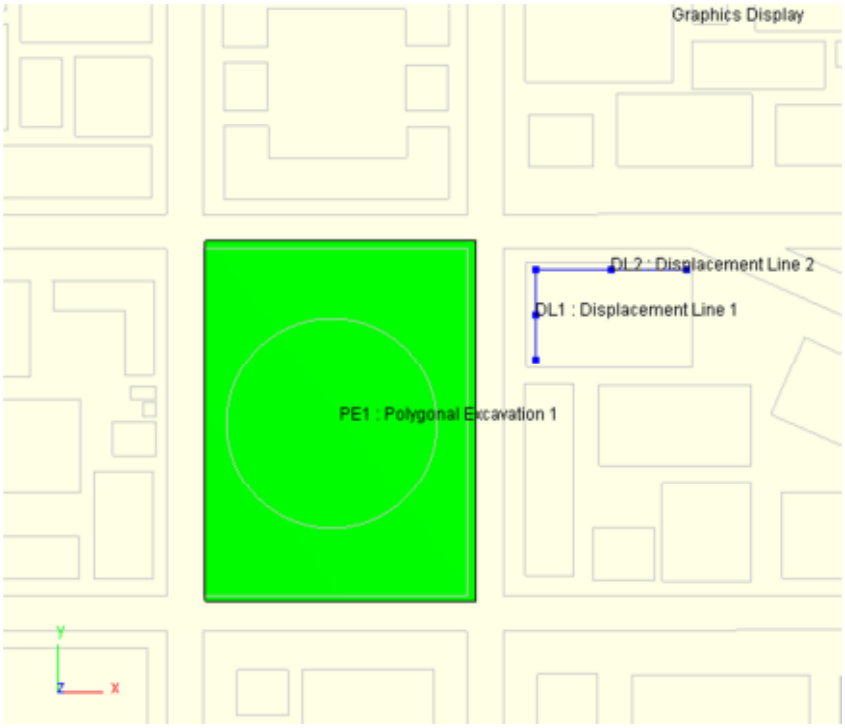


(Hint: To complete the excavation, right click on the last point and click on Add Excavation)



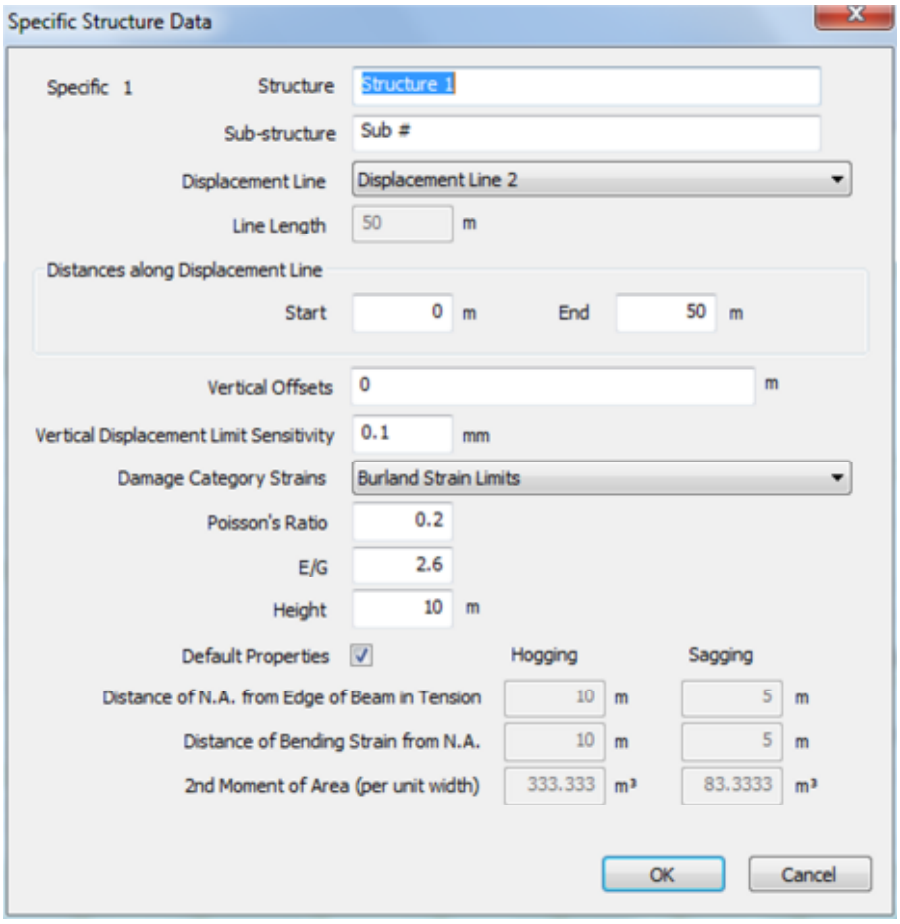
 Click on the ‘Add displacement entity’ icon and tick displacement line.

4. Draw displacement lines in the shown location at z=0mOD:

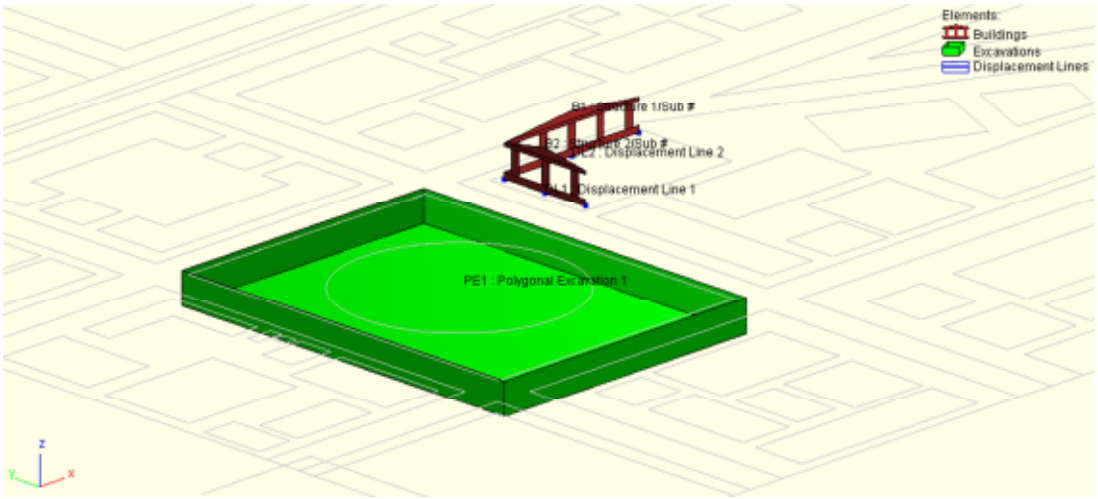


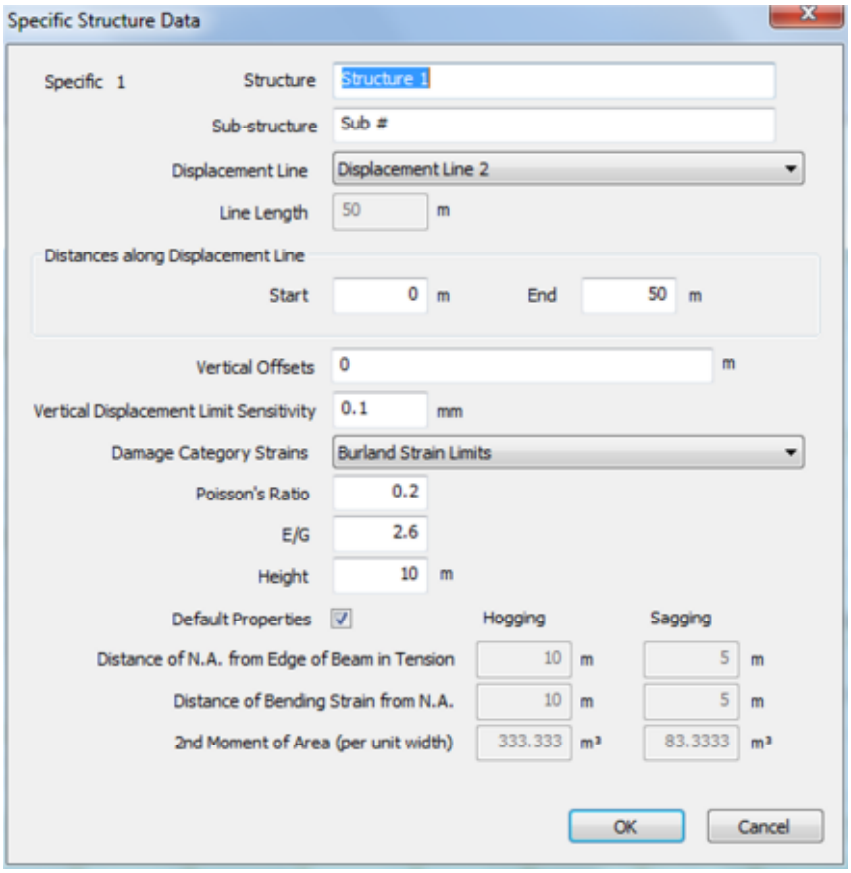
 Click on the ‘Add structure’ icon and select Specific Structure

5. Input structure with the following properties for Displacement Lines 1 and 2:



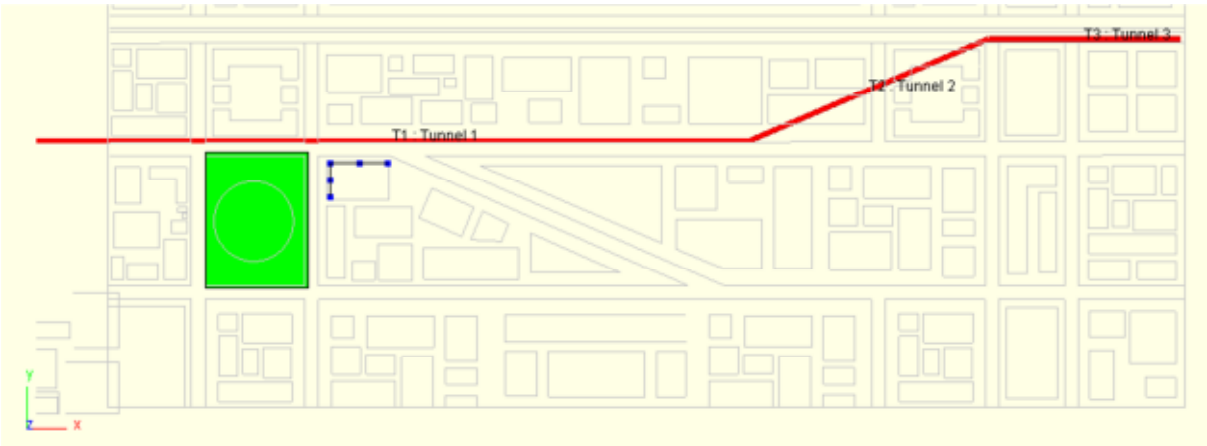
(Note: Specific Buildings can only be inputted on displacement lines)



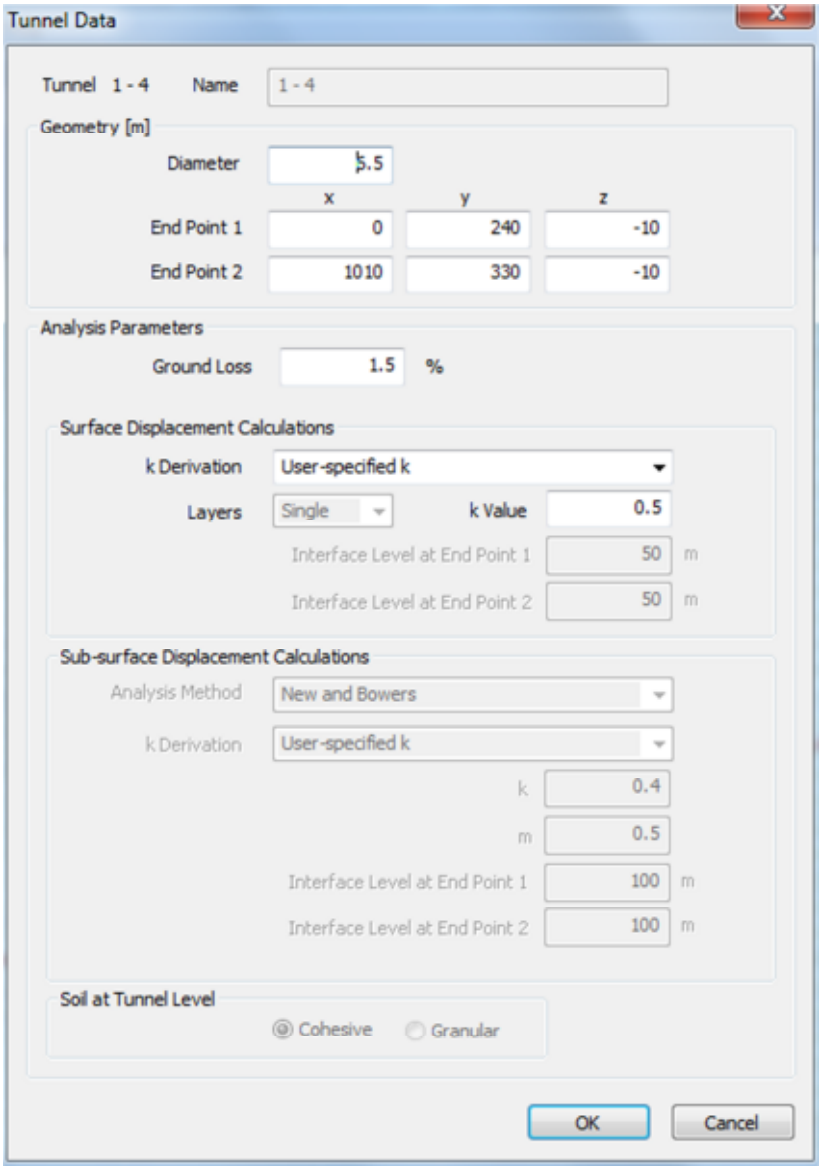


Click on the 'Add tunnel' icon to input tunnels along the path shown below at z=10m. To finish the tunnel, double left click on the last point.

(Hint: Check the Grid Plane before sculpting.)



6. Enter the following data for the tunnel



Test yourself

To consolidate your learning, draw a polyline and enter a generic building damage assessment.