Xdisp Tutorial Manual
Oasys Xdisp

Copyright © Oasys 2013

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

This document has been created to provide a guide for the use of the software. It does not provide engineering advice, nor is it a substitute for the use of standard references. The user is deemed to be conversant with standard engineering terms and codes of practice. It is the user's responsibility to validate the program for the proposed design use and to select suitable input data.

Printed: May 2013
Contents

1. Introduction p1
2. Components of Xdisp p2
3. Program Features p3
4. Tunnel and Excavation Tutorial p6
5. Building and Utility Damage Assessment Tutorials p22
6. Importing CAD Drawings p48
7. Graphical Input p50
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. Furthermore, the simplicity of interface enables new users to set up and run a retaining wall problem in a few minutes and 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 problems 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. Program Features

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. Program Features

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. Tunnel and Excavation Tutorial

4.1 Creating the input

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

1. Fill the *Titles* dialog box:

   ![Titles dialog box](image)

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

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)
4. Tunnel and Excavation Tutorial

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.
4. Tunnel and Excavation Tutorial

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)

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)

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

4.2 Calculation and Outputs

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

   Click on the analyse button to carry out the analysis

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

   ![Progress of Checks and Analysis]

   (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. Tunnel and Excavation Tutorial

4.2.2 Viewing and Manipulating Graphical Results

**Contours**

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

   ![Grid icon](image)

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

   ![Contours icon](image)

   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

   ![Down icon](image)

   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 ( )
4. Tunnel and Excavation Tutorial

**Exporting Contours co-ordinates to Excel**

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

![CSV Results File Output Selection](image)

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.)
4. Tunnel and Excavation Tutorial

**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.
4. Tunnel and Excavation Tutorial

**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:
5. Building and Utility Damage Assessment Tutorials

5.2 Checking the base file

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. Building and Utility Damage Assessment Tutorials

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:

   ![Strain Ranges](image1.png)

   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.

   ![Specific Structures](image2.png)

   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.
5. Building and Utility Damage Assessment Tutorials

The Burland method of building damage assessment assumes that a building's façade behaves as a beam in bending. The following 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 Young's 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 12.6, rather than 2.6 used for masonry structures".

*Melis and Rodriguez 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.xls, Specific Structures

<table>
<thead>
<tr>
<th>Ref.</th>
<th>Structure Name</th>
<th>Sub-Structure Name</th>
<th>Displacement Line</th>
<th>Line Length</th>
<th>Start Distance Along Line</th>
<th>End Distance Along Line</th>
<th>Vertical Offsets from Line to Vertical Movement Calculation</th>
<th>Vertical Displacement Limit</th>
<th>Vertical Displacement Sensitivity</th>
<th>Damage Category</th>
<th>Strains</th>
<th>Poison's Ratio</th>
<th>E/G</th>
<th>Height</th>
<th>Default Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Building #</td>
<td>Sub #</td>
<td>Surface</td>
<td>1.00</td>
<td>0.00</td>
<td>0</td>
<td>Internal</td>
<td>0.75</td>
<td>0.100</td>
<td>Buried Strain Limits</td>
<td>0.24</td>
<td>2.500</td>
<td>10.000</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Building 1</td>
<td>Sub #</td>
<td>Surface</td>
<td>99.99</td>
<td>100.00</td>
<td>0</td>
<td>Internal</td>
<td>0.75</td>
<td>0.100</td>
<td>Buried Strain Limits</td>
<td>0.200</td>
<td>2.500</td>
<td>10.000</td>
<td>Yes</td>
<td></td>
</tr>
</tbody>
</table>

Press <TAB> to start a new record
5. Building and Utility Damage Assessment Tutorials

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. Building and Utility Damage Assessment Tutorials

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:

![Image of Tabular Output]

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

![Image of Specific Building Damage Results]

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. Building and Utility Damage Assessment Tutorials

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)
5. Building and Utility Damage Assessment Tutorials

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. Building and Utility Damage Assessment Tutorials

5.3.4 Exporting Building Damage Assessment Results

XDsp 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:

<table>
<thead>
<tr>
<th>Folders</th>
<th>Files</th>
<th>Content/Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt;Run filename&gt;</td>
<td>Building_Response.csv</td>
<td>The folder name is XDisp data file name.</td>
</tr>
<tr>
<td>&lt;StructureName&gt;</td>
<td>Classification_of_Damage.csv</td>
<td>One folder is created for each structure. The folder name is the structure name.</td>
</tr>
<tr>
<td>&lt;StructureName&gt;</td>
<td>Critical_Sub_Structure.csv</td>
<td></td>
</tr>
</tbody>
</table>

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.

![Displacement Graph](image.png)
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. Building and Utility Damage Assessment Tutorials

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.

Interval = 50m
Length = 100m

Transects (which represent structures)
Polyline
5. Building and Utility Damage Assessment Tutorials

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.

<table>
<thead>
<tr>
<th>Transect</th>
<th>Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10m</td>
</tr>
<tr>
<td>2</td>
<td>20m</td>
</tr>
<tr>
<td>3</td>
<td>40m</td>
</tr>
</tbody>
</table>

(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)

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

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

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

5.4.2 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 Specific Structure](image)

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:

![Graph of Maximum Tensile Strain](image)

(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. Building and Utility Damage Assessment Tutorials

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.xls: Acceptance Criteria/Parameters

<table>
<thead>
<tr>
<th>Ref.</th>
<th>Name</th>
<th>Acceptance Criteria</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Young's Modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Allowable Strain</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Tension</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Check</td>
<td>Check</td>
</tr>
<tr>
<td>Defaults</td>
<td>Acceptance Criteria</td>
<td>Yes</td>
<td>98.000</td>
</tr>
<tr>
<td>1</td>
<td>Verified Clay</td>
<td>Yes</td>
<td>100.000</td>
</tr>
</tbody>
</table>

Press <TAB> to start a new record
5. Building and Utility Damage Assessment Tutorials

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

6. Click on the analyse button to carry out the analysis

The solution progress window will automatically appear and the analysis will run.
Click the OK button and the tabular outputs will automatically appear.
5. Building and Utility Damage Assessment Tutorials

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:

![Tabular Output Image]
5. Building and Utility Damage Assessment Tutorials

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 Utilities 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

   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. Building and Utility Damage Assessment Tutorials

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 Utilities icon and select ‘Generic Utility’
   ![Generic Utility](image)
   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:
   ![Combined Strain vs Distance](image)
   (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.
5. Building and Utility Damage Assessment Tutorials
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:
6. Importing CAD Drawings

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
7. Graphical Input

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)
7. Graphical Input

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

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

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 \( z=10 \) m. To finish the tunnel, double left click on the last point.

(Hint: Check the Grid Plane before sculpting.)
7. Graphical Input

6. Enter the following data for the tunnel

Test yourself

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