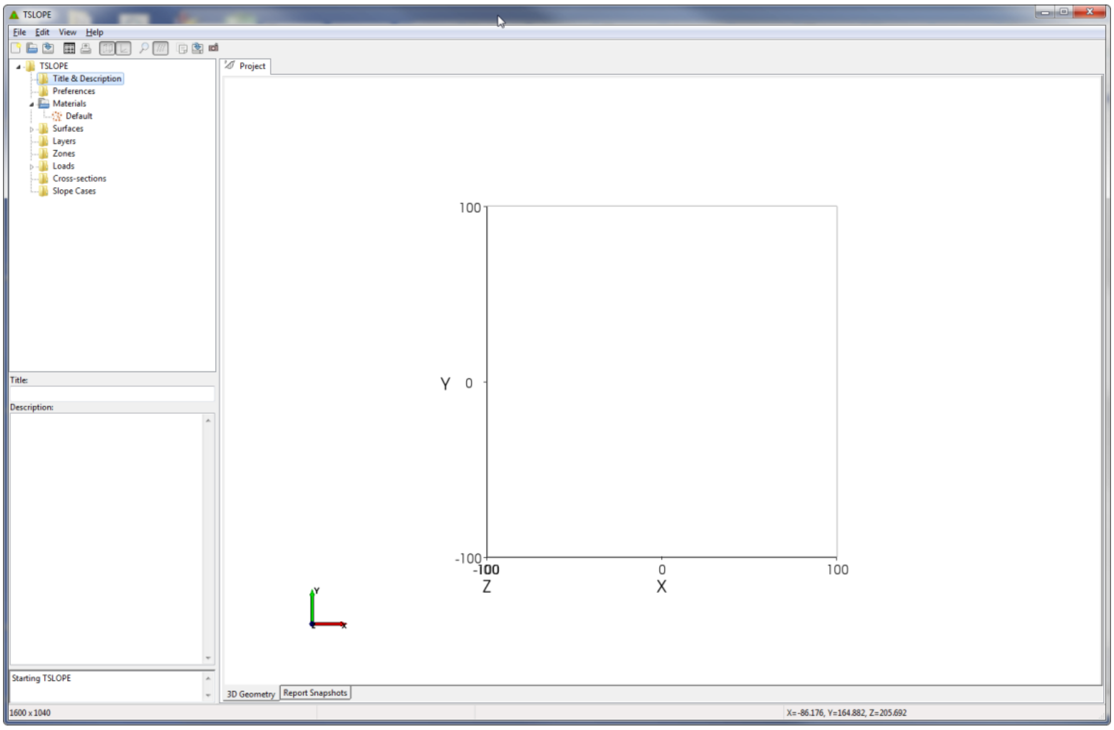
TSLOPE

Getting started Guide

1. Introduction

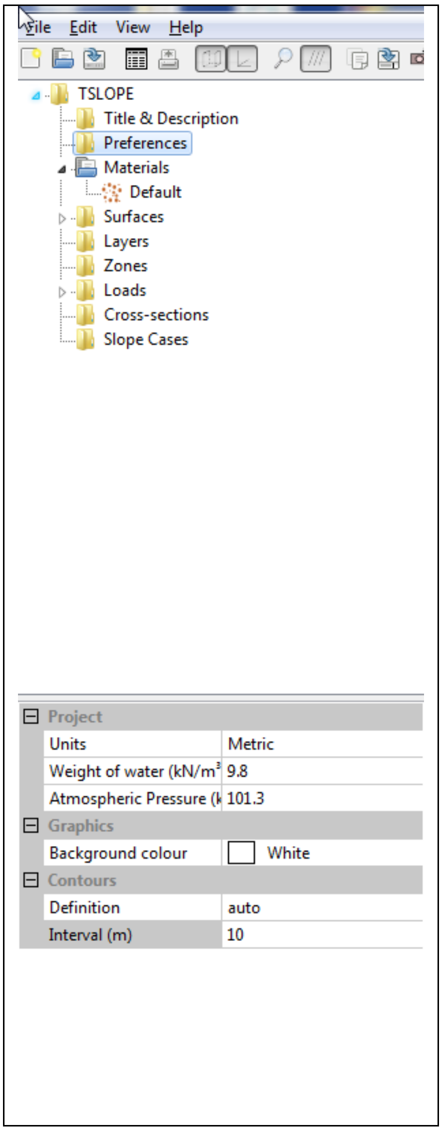
On starting TSLOPE a five panel interface allows for a project directory tree (top left) two analysis input panels (middle left) and a geometry panel (right) which provides 3D interactive graphics. The bottom left panel provides you with information on TSLOPE processing activity.

The ***File*** tab enables a new project to be setup or an existing project to be opened.



If starting a new project check each item in the ***directory tree*** panel. Beginning with the first item - ***Title & Description***. This is a useful place to document the location and reason for the analysis especially if sharing with colleagues.

A small black command window will also open behind the main TSLOPE window. That may show occasional error messages. Should you encounter a problem with operation of TSLOPE and you need to share it with the TAGAsoft development team, you should copy and paste the information in that window to an email message to info@tagasoft.com

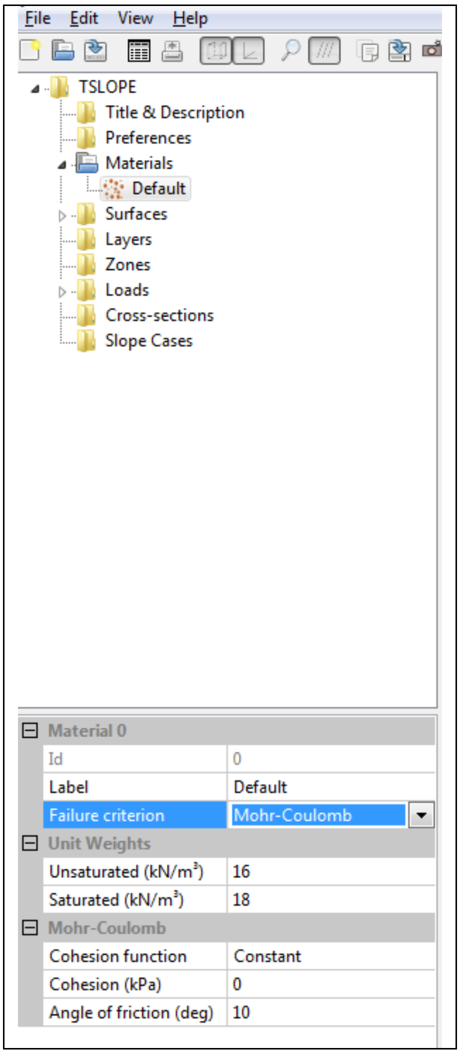


2. Setup

In the ***preferences input panel*** (lower left) the Units may be defined as metric, US Customary , or user defined by activating the tag on the right side the Units cell. There is no automatic conversion of values when the units label is changed, but it is a reminder to the user to enter appropriate values e.g. metric values will be kPa, m, and kN/m^3 while U.S. Customary will be psf, ft, pcf.

There are default values for the ‘Weight of water’ and ‘Atmospheric Pressure’. The unit weight of water is required if a phreatic surface is defined and the atmospheric pressure is required for the curved Mohr-Coulomb failure criterion. Both these values can be changed if required.

By default the background colour in the 3D geometry panel is white but this can be changed by left mouse clicking in colour cell to activate the Color Chooser dialogue box.



Proceeding down the ***directory tree*** the next part of the setup involves defining the Material properties.

Clicking on the ***Default*** tag shows the preset values for slope stability analysis.

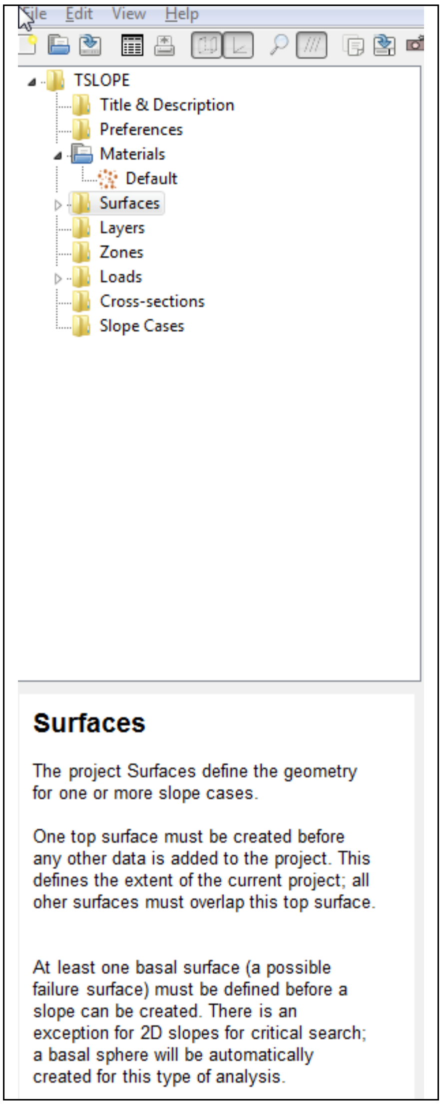
Clicking on the ***Label*** tag enables the material name to be changed to the appropriate unit in the slope model. Where there are different materials in a slope model, the first material entered will be the overlying material, or highest stratigraphic unit.

The ***Failure criterion*** can be toggled between Mohr-Coulomb or bedrock. Bedrock is a unit that limits the extent of slope failure.

The unit weights for ***Dry*** and ***Saturated*** scenarios can be adjusted.

Likewise the Mohr-Coulomb settings for ***Cohesion*** and ***Angle of friction*** can also be modified as required.

The material properties for other units in the slope model are added by right mouse clicking on the ***Materials*** tag and chosing ***Add material.***



3. Surface data input

Up to three physical ***Surfaces*** may be modelled for a slope stability problem, but the topographic and basal surfaces must be defined. The phreatic surface is optional.

To import the ground surface (topography) right click the ***Top*** label in the Directory Tree panel for the options :

- Add Extruded section
- Add Cloud of points (ASCII xyz)
- Add Plane
- Add Cone
- Add Biplanar surface
- Add surface file import (obj, dxf, vtp, xml)
- Add Composite

The topographic surface is likely to have been modelled in another software package. These data may be contour or gridded values but require X Y Z components. If data import is via an external file then use the ***Cloud of points*** or the ***Surface file*** options. A file input dialogue box will appear so the user can select the desired file.

It is recommended that you limit the area of the surfaces that you import into TSLOPE to the area of interest for your slope stability problem. The project dimensions are set to accommodate the full range of X Y Z values that are loaded into the system.

Data imported are rendered using a Delaunay triangulation and displayed as a surface in the Slope Geometry panel on the right side of the TSLOPE window.

When adding a surface as an ***Extruded section,*** (as in the case of an embankment), you can manipulate the section plane by moving the purple nodes. The outer node will lengthen/shorten the section, the middle node will rotate the section and the inner node will move the section up/down. Once the plane is in the desired position it is advisable to tick the check box for ***Is locked*** and assigned material properties for the surface under the ***Material*** tab in the Data Panel (lower left).

The ***Plane*** and ***Cone*** derived surfaces are manipulated as for the Extruded section by moving the coloured nodes to expand, rotate, and move the model into position.

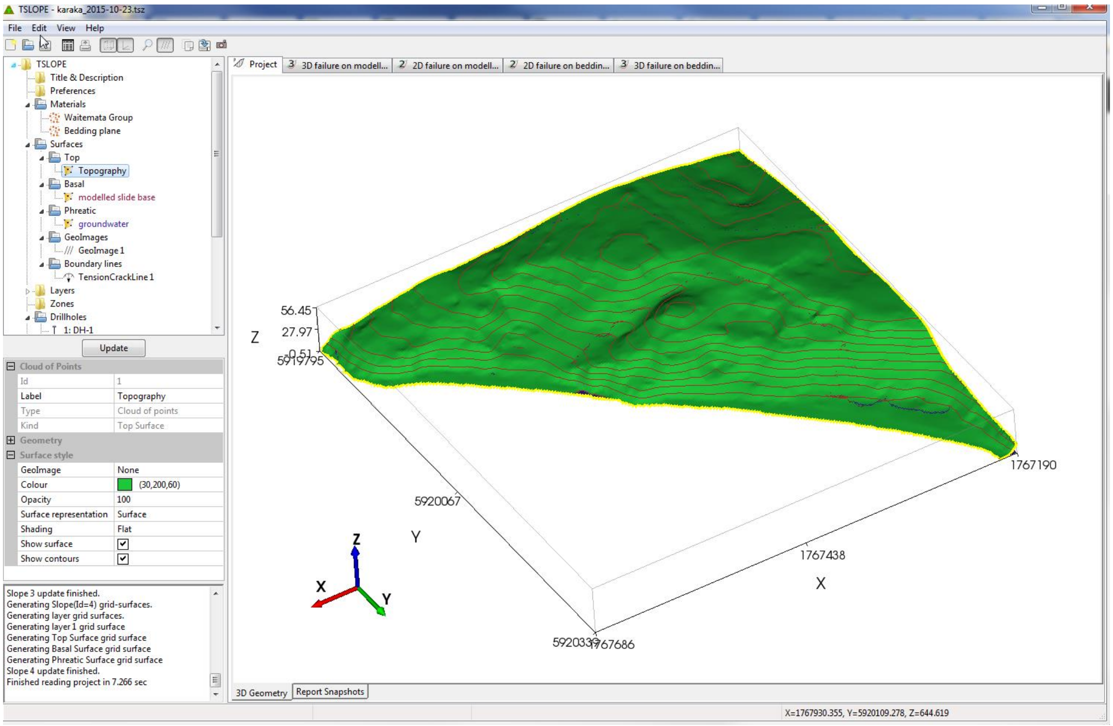
The ***Biplanar surface*** produces a hinged surface which can also be expanded by manipulating the coloured nodes. The outer nodes control the size of the bi-plane while the centre node raises and lowers the surface.

In addition to the nodes, the bi-planar surface dip and bearing direction can be adjusted by changing the default values in the Data panel.

A ***Composite*** surface allows for combination of two or more surfaces, such as a topographic surface, and a designed cut slope.

When the position of the surface is finalised click the Is locked tick box.

The surfaces can be removed at any time by highlighting the surface, then right mouse click to select the Delete function.



In the image above the directory tree is highlighted at

> Surfaces

> Top

> Topography   (the icon on the left indicates import mode of the data)

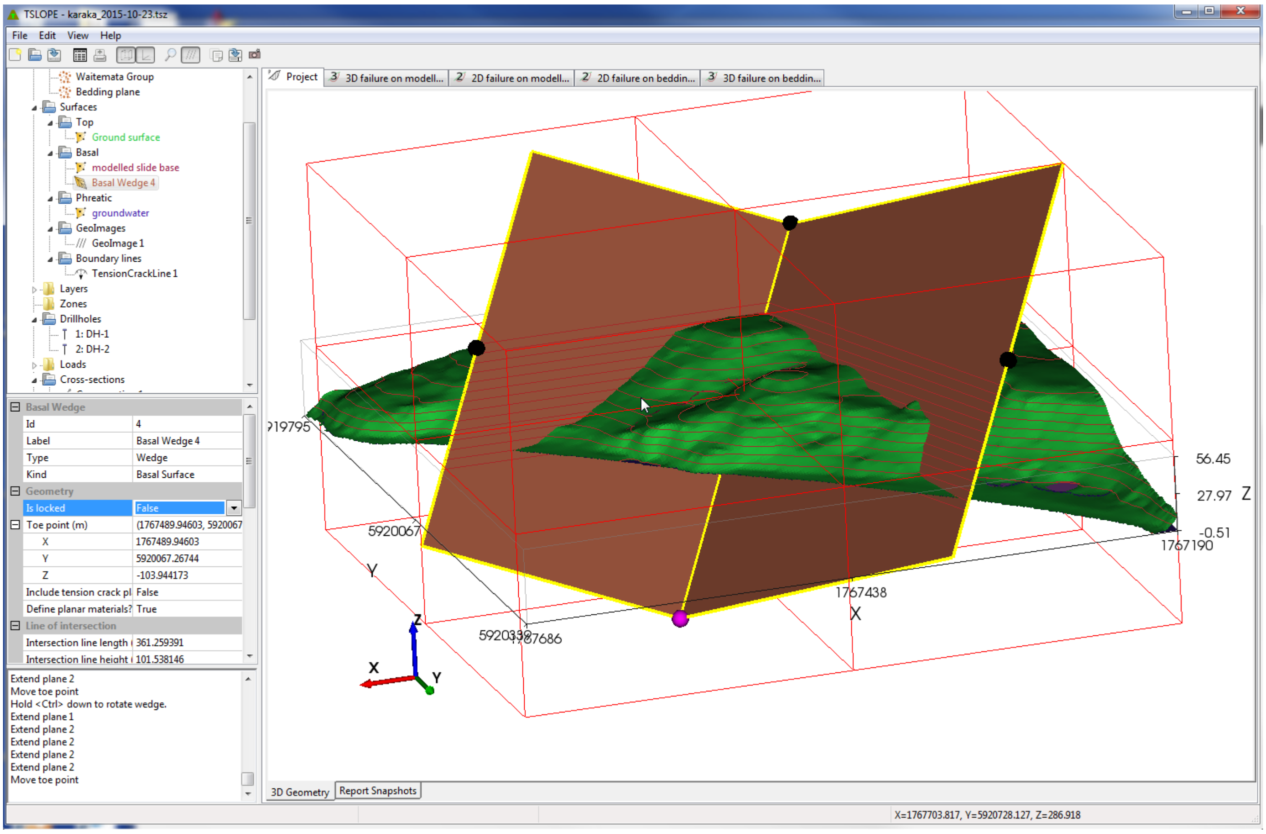
In the data panel below is more information about the Top surface. The ***Label*** can be renamed.   The ***Is Locked*** tick ensures when rotating the model in the right panel the geographic position remains fixed. The ***Colour*** can be reset by clicking the mouse button on the 3-dot icon at the right and selecting from a separate colour chooser panel or adjusting the numerical RGB components. The options for ***Surface representation*** are: Flat, Gourand, Phong (which are all similar) and Wireframe. Again a dropdown box of these options is available with a mouse click in the right cell. The next two options in the data panel control the surface or contour visualisation. These displays are controlled by toggling the tick icon.

The ***Basal*** (failure) ***surface*** can be defined or imported in a similar manner to the Top surface options. They include:

- Add Extruded section
- Add Cloud of points (ASCII X Y Z)
- Add Plane
- Add Ellipsoid
- Add Log-Spiral
- Add Parabaloid
- Add Wedge
- Add Surface file import
- Add Composite

The ***Ellipsoid, Log Spiral and Parabaloid*** all follow the same methodology as for the Extruded section. The choice of using one of these types of basal surface will depend on the slope problem that is to be analysed.

In the case of the ***Basal wedge*** option the coloured nodes allow scaling of the wedge and movement up and down, but the orientation of each side of the wedge is controlled from the Data panel. This is so measurements taken in the field can be applied. Separate material properties can be set for each side of the wedge.



A ***Composite*** surface allows for combination of two or more surfaces to form a basal shear surface that will define a 3D potential failure volume.

Phreatic surface/s can be added in a number ways. These include:

- Add plane
- Add Cloud of points
- Add Extruded section
- Add Surface file import
- Add Biplanar surface

GeoImages

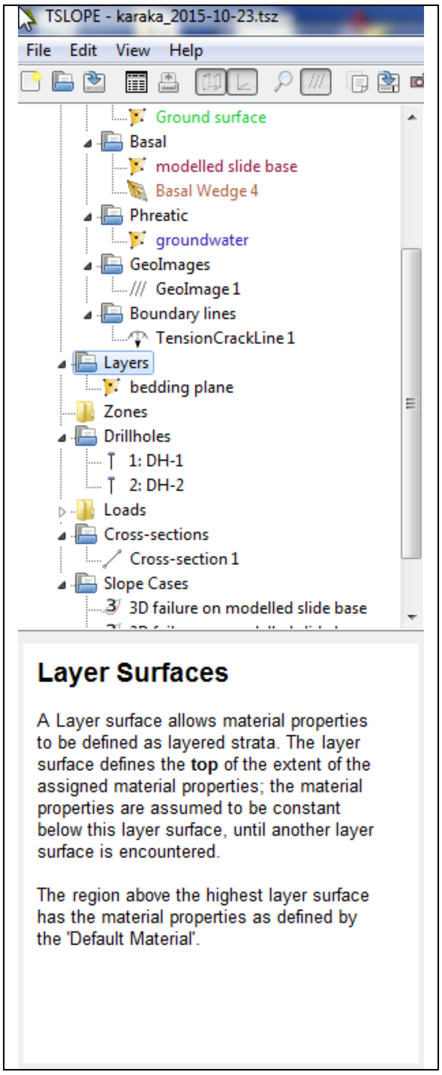
A GeoImage is an image (JPEG, PNG, TIFF), such as an aerial photograph, with associated coordinate information that may be used to drape over a surface.

Boundary lines

A Boundary line is a 2D polyline that is associated with a top surface.
The boundary line may be a tension-crack line that has been observed in the field, and may not be closed. For some slopes, movement may be sufficient to define a closed boundary line that combines: a tension-crack, a side shear line, and a thrust line at the base of the slope.

The use of a boundary line for a slope-case is optional.

4. Layer data input



Depending on the slope stability problem there are options to refine the analysis by adding specific ***Layers*** to the model that can have different material properties.

For the definition of ***Layers*** TSLOPE works with the top of each layer. We assume that the user will have used a geological modelling system for complex problems, and the appropriate checks will have been made that there are no stratigraphic anomalies.

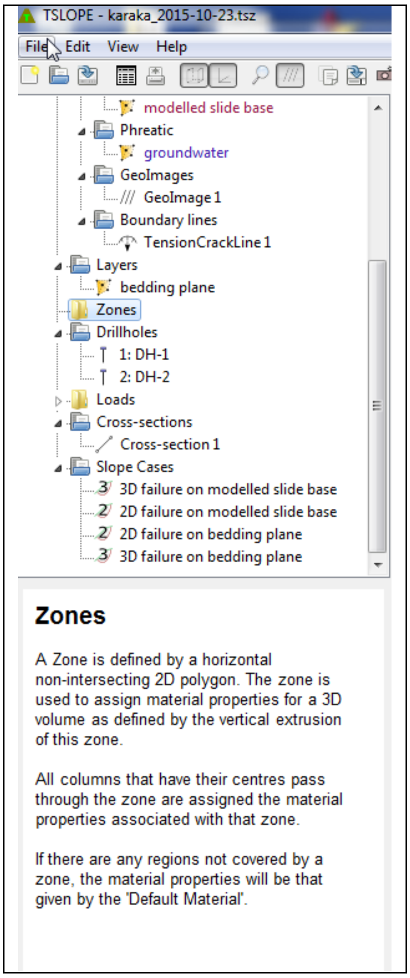
TSLOPE assumes that the properties associated with each layer extend with depth below the top of each layer until the top of the next lower layer is reached.

To add a Layer right-click with the mouse to reveal the options for importing a layer surface

- Add Extruded Section
- Add Cloud of points (ASCII XYZ)
- Add Surface file import
- Add Plane
- Add Composite

When adding a layer as an ***Extruded section,*** (as in the case of an embankment), you can manipulate the section plane by moving the purple nodes. The outer node will lengthen/shorten the section, the middle node will rotate the section and the inner node will move the section up/down. Once the plane is in the desired position it is advisable to tick the check box for ***Is locked*** and assigned material properties for the surface under the ***Material*** tab in the Data Panel (lower left). The material properties for the layer will apply to the area below the plane while the area above the plane will adopt the overlying layer properties.

5. Zones data input



Zones may also be included in the model. To activate the Zone input right mouse click on the **Zone** label in the Directory tree and Add zone …

In the data panel the ***Label*** can be renamed. The points defining the zone can be initiated by clicking in the Boundary points (m) cell which produces the ***Zone Boundary Points*** dialogue box. Points can be entered manually or imported via The ***File Import*** option. The file is ASCII format (X Y (ID is optional)). Imported points will appear in the dialogue box. Click OK to proceed.

The Colour, Opacity, and visibility parameters can be set. Material properties for each zone can then be defined under the Material tab in the Data panel box.

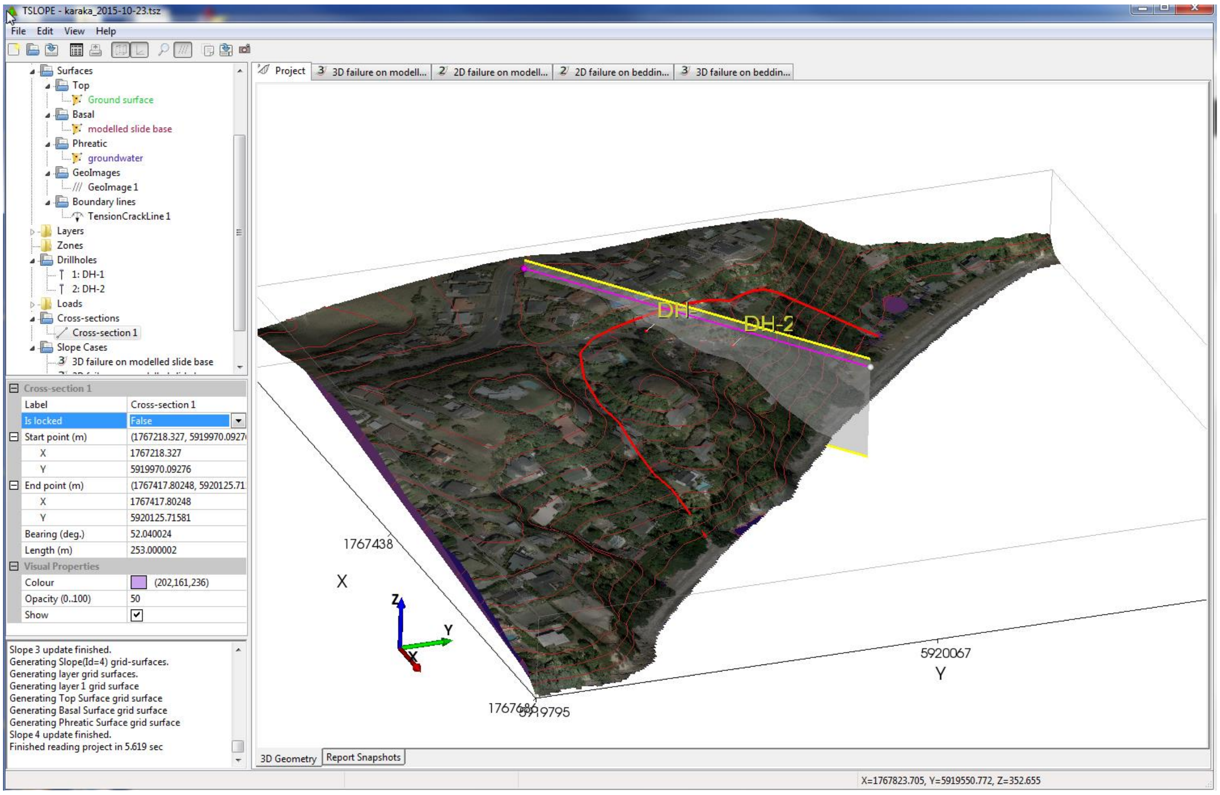
The height of the zone/s in the Slope Geometry panel (right view) are controlled by the coloured node which is visible when the ***Zones*** label in the Directory panel is highlighted.

6. Loads

Under development.

7. Cross-sections

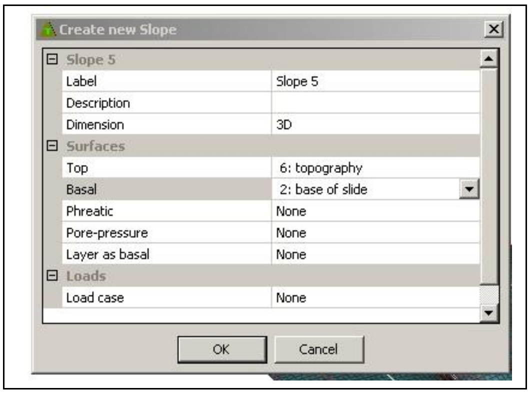
TSLOPE also provides 2D slope analysis. To define a cross-section highlight ***Cross section*** in the directory tree and right mouse click to Add section.   To orient the section through the model at the desired location move the end point nodes in the Slope Geometry panel. One node acts as an anchor and the other changes the length and orientation.



8. Slope Analysis

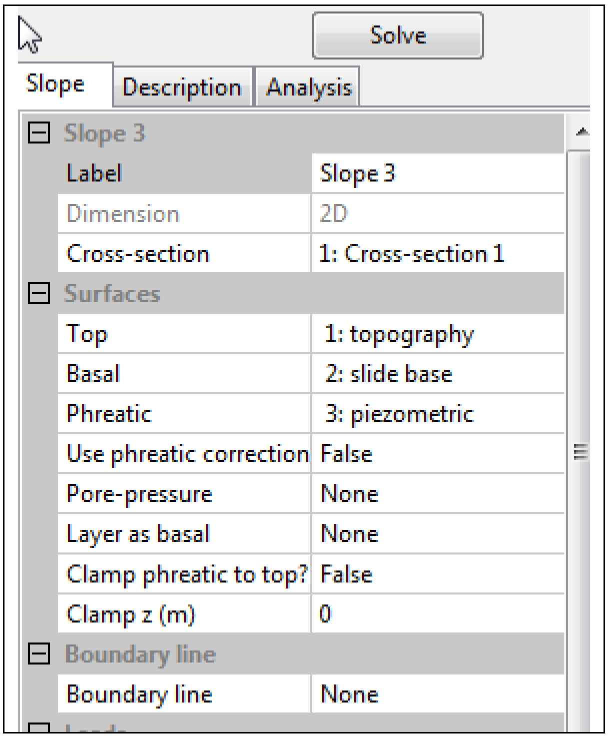
A project may consist of one or more slopes. Multiple slopes allow parameteric studies to investigate the stability of slopes when aspects of the slope vary, eg material properties, pore pressure and phreatic surface changes, and any other parameter of interest. Once the Material properties, Layers and Zones have been defined and the Surfaces included the slope analyses can be conducted. Phreatic and pore-pressure surfaces are optional.

To start a slope analysis, highlight the Slope Cases label in the directory tree and right mouse button to add a new slope.



The ***Create New Slope*** dialogue box provides the setup for the slope analysis components. The Dimension is toggled to be either 3D or 2D.

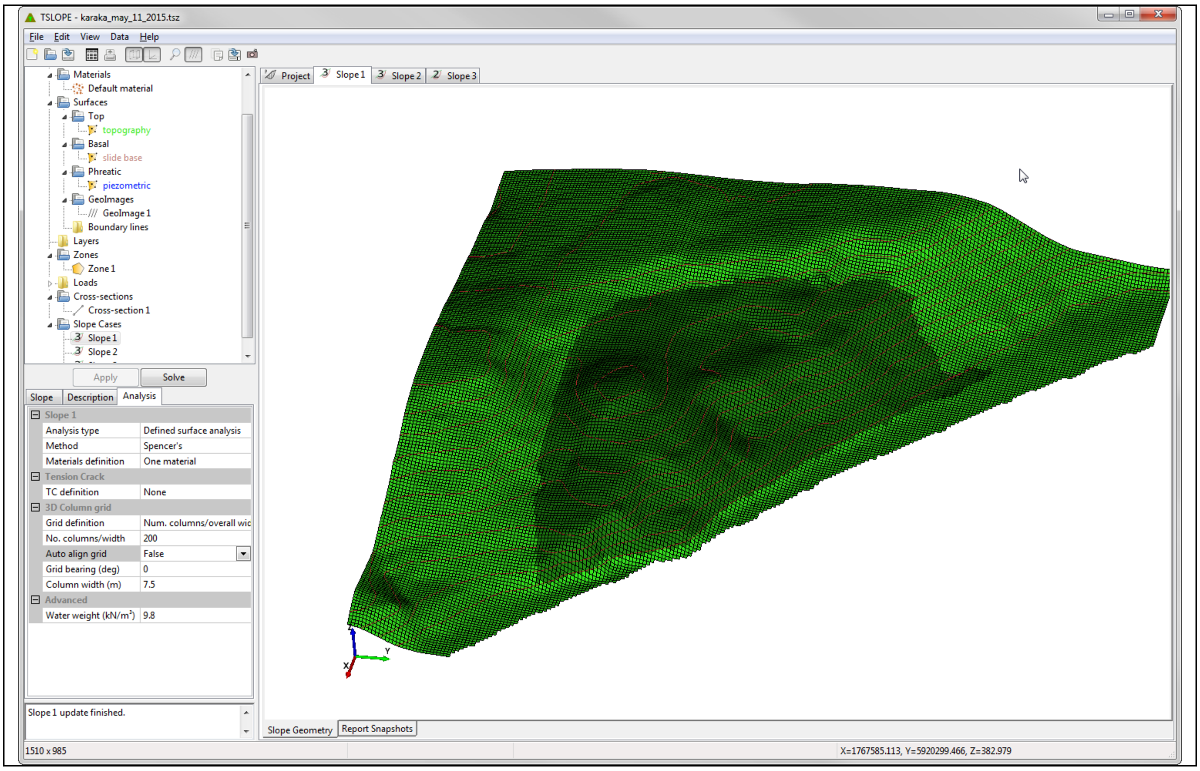
For 3D analysis the compulsory surfaces are included from the Directory tree Surfaces for Top and Basal. Phreatic surface or pore pressures can be included.



Where the 2D option is selected the menu automatically includes the facility to select one of the pre-defined cross sections and to specify the number of slices that will be used in the analysis. The larger the number the longer the processing time.

When the Slope Cases are active the Data Panel (lower left view) has two tabs: one has details about the ***Slope*** (surfaces used) and the other has the ***Analysis*** parameters. There is also a Solve tab under the Directory tree.

The 3D Slope Geometry panel (right view) now has a new tab for each slope case. See Figure below. The gridded view of the model shows the columns involved in the stability analysis in a darker shade.



The Analysis menu has default settings but each parameter has a number of options.

Analysis Type:	Standard, Critical, Back Analysis, Solution-space analysis, Yield acceleration analysis
Method:	Spencer's, Ordinary
Material Definition:	One Material, Layers, Zones

If a layer or zone has been defined and is to be included in the slope case analysis then the appropriate ***Material definition*** option needs to be selected in the Analysis Tab.

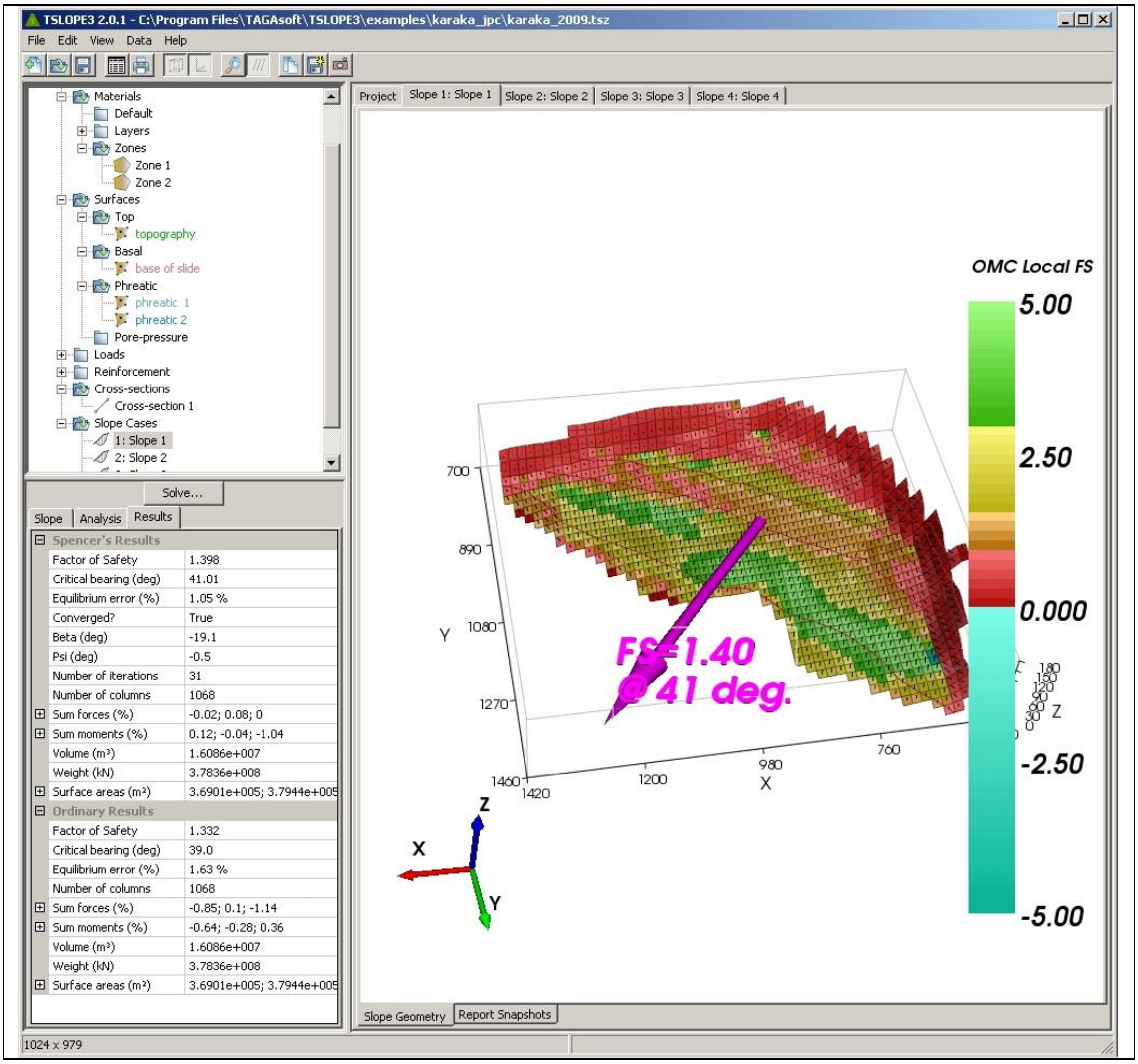
To commence the analysis for a particular slope case click on the ***Solve*** tab at the base of the Directory tree panel (upper left). A new ***Results*** tab will appear in the data panel (lower left). When any of the Material properties are changed then click the ***Solve*** tab for an updated analysis.

9. Analysis Outputs

To view the various outputs select from the ***Data*** label in the upper left TSLOPE3 window. Outputs, which are only valid for the 3D cases, include:

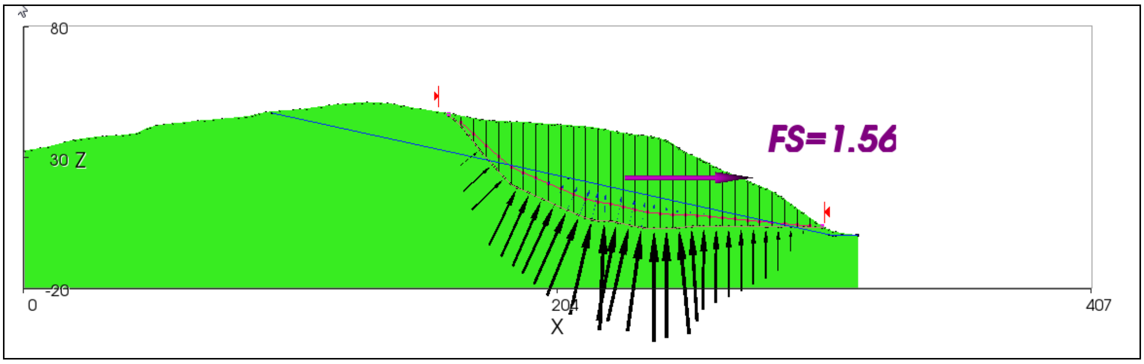
- Total Normal Stress
- Effective Normal Stress
- Pore Pressure (where a Phreatic surface has been included)
- Mobilised Shear Strength
- OMC Local FS

All graphical results appear in the Slope Geometry panel on the right when the Data type has been selected. In the case of a 3D analysis the Local Factors of Safety output produces a colour-coded model based on the number of columns used for the analysis and a directional vector with an overall FS value (see Figure below).



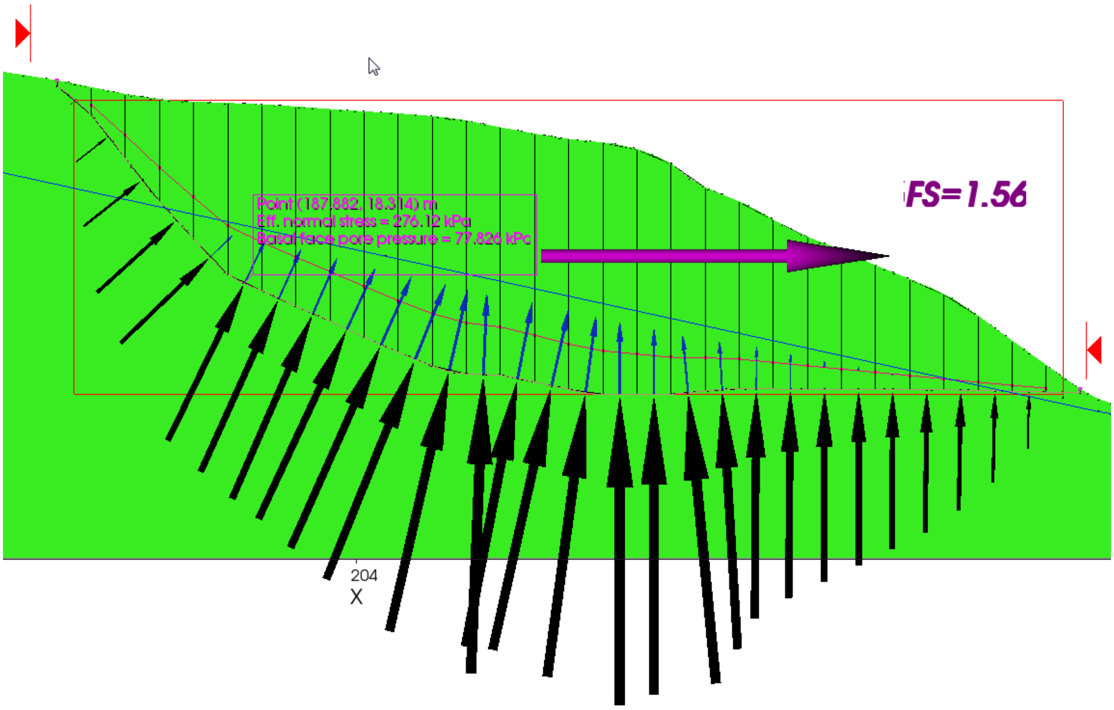
Each slope case can be selected from the TABs on the Slope Geometry panel or from the Directory tree panel. Once a case has been solved the Local FS model can be viewed by turning off the top and / or Phreatic surfaces from the View menu. The coloured cells are mapped to the basal surface.   The ***Results*** tab in the Data panel itemises the key analysis parameters.

In the case of a 2D slope analysis the cross section must have been selected for the slope case. The bearing direction is calculated when the Cross section is defined and can be found in the Data panel (lower left) when the ***Cross section*** is highlighted in the Directory tree (upper left).



The graphical display of the 2D slope case shows the thrust line (red), the effective normal stress (black arrow), and the basal pore pressure (blue arrows). These may be toggled off, or on as required. If there are tensile interslice forces, then those slices will be coloured red as a warning.

To interrogate the data graphically, position the mouse at a relevant point, and use Cntrl left mouse click to get the display below. A further Cntrl left mouse click will remove that information.



10. Reporting

There are a number of features which enable the user to easily capture the graphical outputs for inclusion in a report.

The background colours can be set and the image can be saved as a PNG by clicking the ***'Save image to file'*** tag from the TSLOPE window. The graphic can then be inserted into other documents.

Next to the Save image to file tag is the ***'Take image for report'*** tag. Whatever is active in the Slope geometry panel will be saved into a ***Report snapshot*** tab that appears at the bottom of that panel. Options are available to add a figure caption.

A summary report is automatically generated and can be previewed under

> File

> Print Preview Report

OR the **Preview** icon on the tool toolbar.

Any report snapshots associated with the slope cases will be included in the summary report which lists all the surfaces and material properties as well as all the slope cases analysed.

The TSLOPE3 summary report can be reproduced from the options under the File toolbar

> File

> Print Report

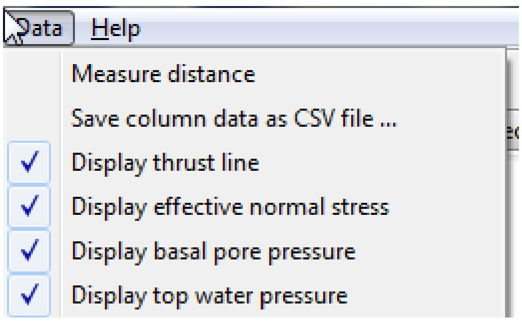
or exported as a HTML file

>File

> Export

See Appendix 2.

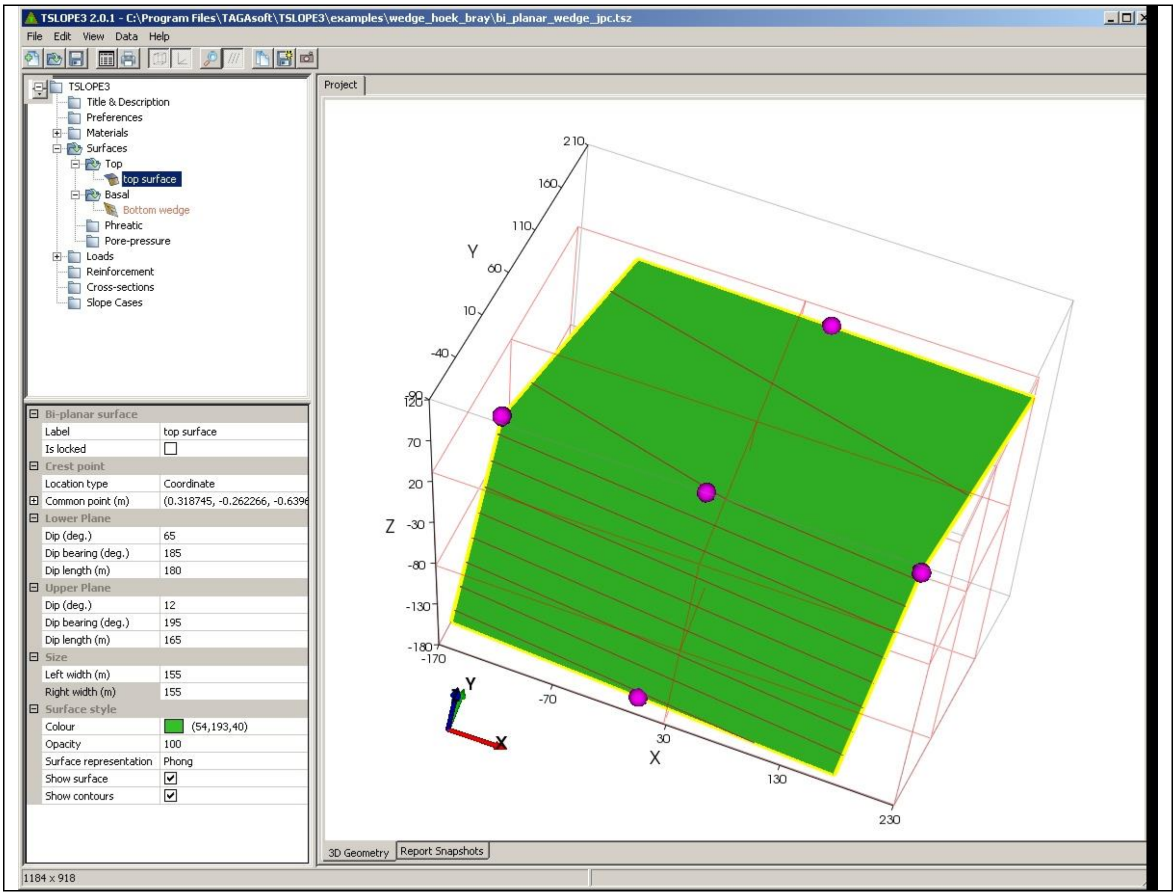
The ***Data*** tab opens to allow selection of ***Save column data as CSV file…***



A CSV file will be opened in the current directory and allow the user access to all the column (or slice) data.

Appendix 1. Wedge and Tension Crack

This a theoretical example to demonstrate how to construct a bi-planar top surface and a basal wedge with a tension crack and then analyse the slope stability of this model.

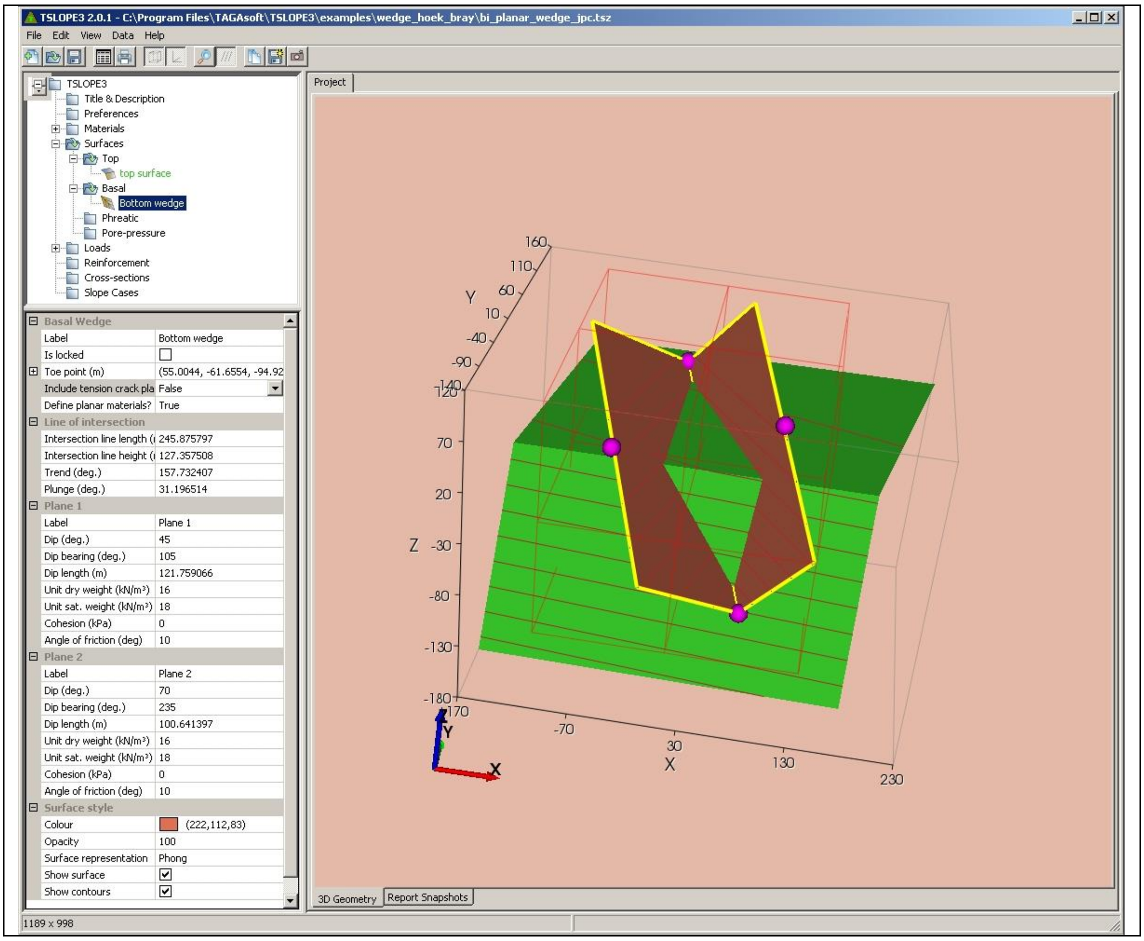


The default material properties have been accepted and no layers or zones are used.

A Surface for the Top is defined by adding a bi-planar surface. The surface automatically appears in the **3D geometry** panel. The shape of the surface can be manipulated using the coloured nodes which control the length and width of each plane but these parameters can also be set manually in the **data input** panel on the left. The ***Dip*** of each plane is set manually. In this case the parameters of the surface have been adjusted to match the Heok and Bray example

Once the surface dimensions are finalised it is advisable to click the ***Is Locked*** box before adding the Basal surface.

The Basal surface in this theoretical example is a Wedge.

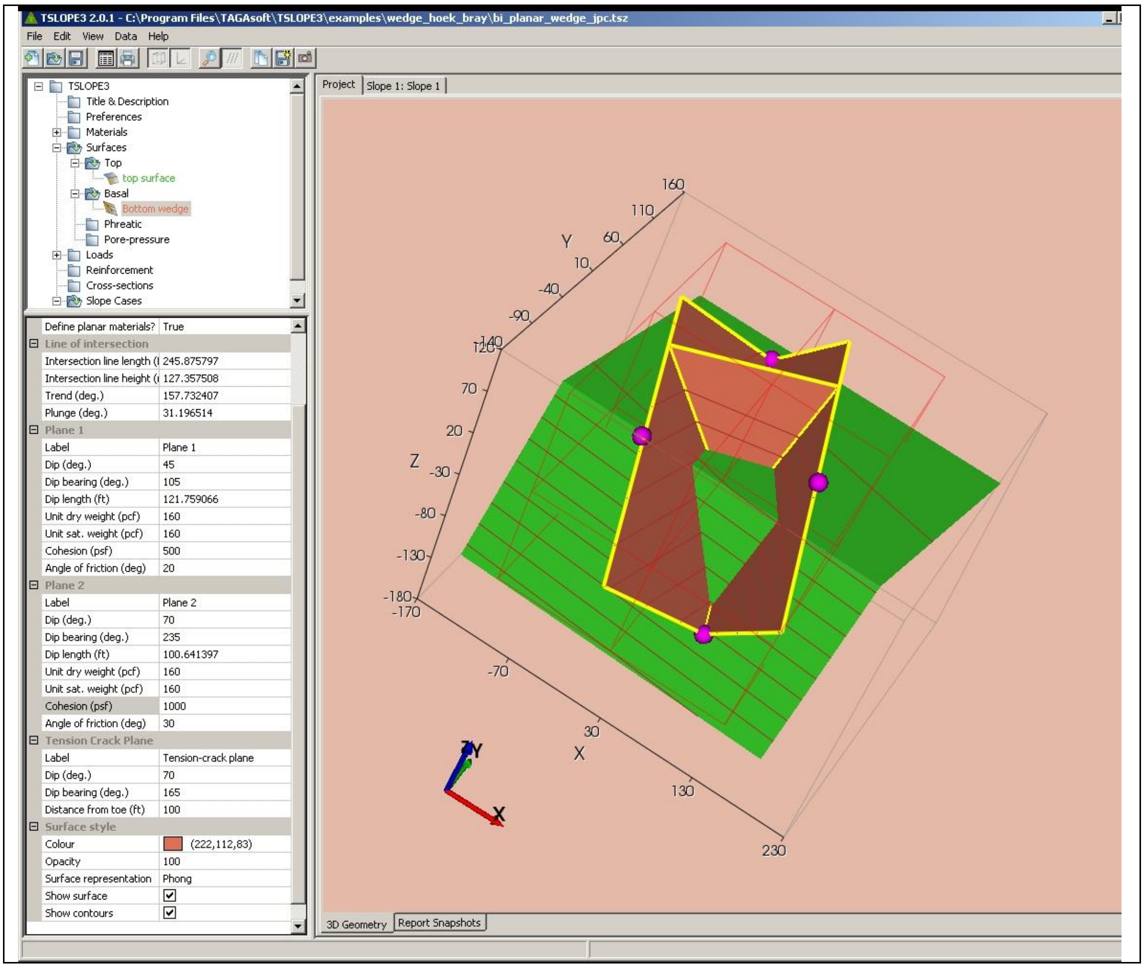


Like the bi-planar surface the wedge will automatically appear in the **3D Geometry** panel. The coloured nodes at the outer edges of the wedge control the width of each side, the lower central node nodes raises or lower the wedge and the upper/rear central node increases or decreases the length of the wedge axis.

The dip and dip bearing of each side of the wedge is set manually in the **data input** panel. The other dimensions for each side of the wedge can also be set manually as well as the material properties which are likely to vary from the default properties.

There is an option to add a tension crack to the wedge. At the top of the Data Input panel the *Include tension crack plane* by default is set to false. To add a tension crack toggle the false to true.

As with the previous constructs of the model the tension crack will automatically appear within the wedge when set to true.



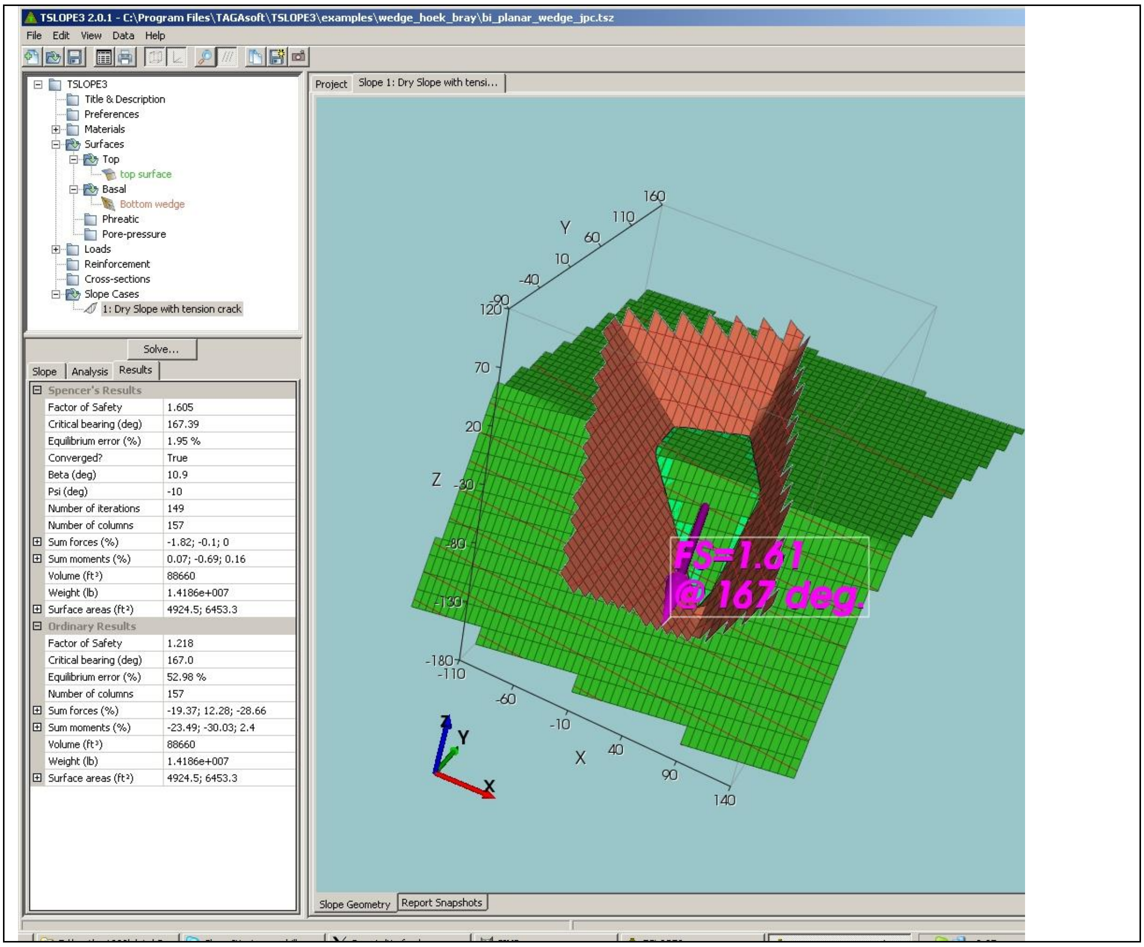
The tension crack has a node at the base of the wedge which moves the tension crack plane forward or back within the wedge. The ***Dip*** and ***Dip bearing*** can only be adjusted in the **Data Input** Panel.

Once the tension crack has been added the material properties may need revisiting. In this case the ***Cohesion*** and ***Angle of friction*** for each side of the wedge have been modified to reflect the Hoek and Bray example (1977 pg 334).

When all he surfaces and elements for the model have been defined it is time to build a slope case.

When adding a Slope Case the slope panel will adopt the Top and Basal surfaces that have already been defined. If there is more than one top or basal surface then the user will need to select the required ones. In the Data Panel under the Analysis tab there is an option to increase or decrease the number of columns. Check the Advanced parameter tag and tick the box to include partial columns if required.

Then SOLVE



The 3D panel on the right now has a tab at the top for the new slope case. The Data panel at the lower left now has a Results tab with includes both the Spencer's and Ordinary results.

References

Heok, E.; Bray, J.W. 1977. Rock Slope Engineering. Revised Second Edition. *Institute of Mining and Metallurgy*.