Oasys

The software house of Arup

Frew Tutorial Manual
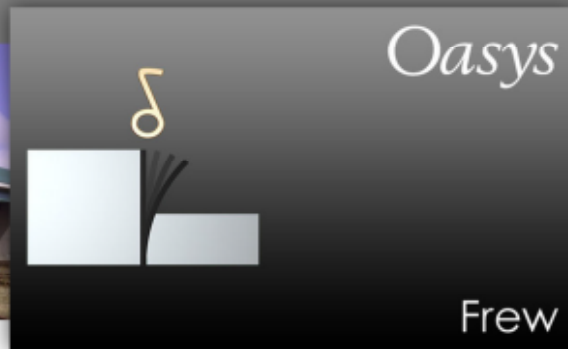
Oasys Frew

Copyright © Oasys 2013

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 users responsibility to validate the program for the proposed design use and to select suitable input data.

Contents

1.   Introduction

Frew (Flexible REtaining Walls) is a program that analyses flexible earth retaining structures such as sheet pile and diaphragm walls. The input wizard enables the quick generation of complex staged construction sequences, including the modelling of props and changes in water levels. The outputs enable the user to study the deformations of, and stresses within, the structure through a specified sequence of construction. The calculation itself is based on verified and robust numerical methods and the simplicity of calculation enables new users to set up and run a retailing wall problem within hours of training.

The various tutorial examples deal with a range of practical retaining wall and shaft applications but this Tutorial Manual is intended to familiarise the user with Frew. The examples should therefore not be used as a basis for practical projects.

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

It is important to realise that Frew is an advanced program analysing a complex problem and the user must be fully aware of the various methods of analysis, requirements and limitations discussed in the User Manual before use. The Tutorial Manual will not provide theoretical background information on the soil structure interaction method, 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 Frew. This also contains detailed information on the available program features. Short courses are also regularly organised and should you be interested in more hands-on experience you can contact oasys@arup.com for dates and program content.

2.   Frew analysis methods

Frew supports limit equilibrium and soil structure interaction methods for retaining wall analysis. Consequently, it can be used for simple problems such as calculating the toe depth of a cantilever wall to analysing the performance of a retaining wall through a number of construction sequences involving multiple props and dealing with different soil conditions (i.e. drained and undrained).

Consequently, different analysis can be carried out depending on the users design needs. These are listed below with a brief description

1. Stability Check Only
   For this analysis, the user can calculate the toe depth of the retaining wall using the limit equilibrium method. Frew allows the user to enter the details of the single stage. As only one stage has been inputted and the construction sequence is not present, the soil structure analysis will not be carried out.

2. Staged Construction with Stability Check
   For this analysis, the user can enter the details of the multiple stages and Frew will calculate the toe depth in the worst case using the Stawal limit equilibrium method.

   The toe depth from this analysis will be used to generate the nodes for the soil structure analysis.

3. Staged Construction without Stability Check
   For this analysis, the user can manually input the toe depth of the wall (e.g. for an existing construction or based on hand calculations) and Frew will generate the nodes for the soil structure analysis.

For the Soil Structure Interaction Analysis methods, more advanced features can be modelled, such as wall relaxation, Minimum Effective Fluid Pressure (MEFP), passive softening and automatic undrained pore pressure calculation. These are described in detail in the Frew Manual and some features will be utilised in the Frew Tutorial manual.

3.   Frew User Interface

The principal components of the user interface are illustrated below and these will be referred to in the tutorial manual.
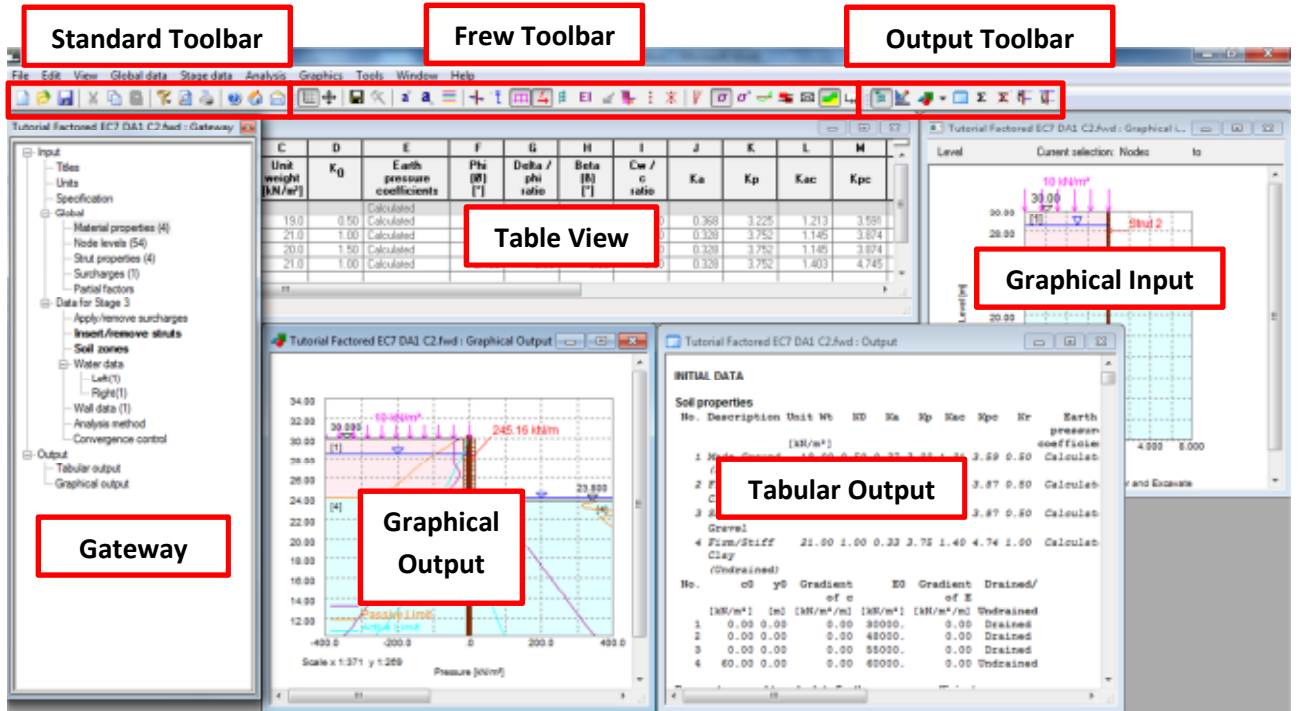
4. Cantilever Retaining Wall

The simplest retaining wall analysis can be carried out for a cantilever wall. The first tutorial example is a quick introduction to the layout of the Frew program and shows you how to carry out a very quick and simple limit equilibrium analysis. It only considers one stage.

This is the first step in becoming familiar with Frew as the toe level obtained is used in the Soil Structure Interaction analysis. The information in this chapter will be utilised in later lessons so it is important to complete this lesson prior to attempting further tutorials.

Objectives

- Start a new project
- Create soil stratigraphy and input material properties using the *Model Wizard* feature
- Use the *Graphical Input* interface
- Apply water data and surcharges
- View calculation results
- Export tabular outputs
- Print and adapt graphical outputs

4.1   Geometry

For the stability analysis, only one stage is analysed. For this specific problem, it is assumed that the wall has been installed without a prop, and the excavated side of the wall has been dewatered. The site investigation should be used to determine the stratigraphy data and the rock layer is not included in the model as an appropriate rigid boundary is applied at the bottom of the Glacial Till layer.
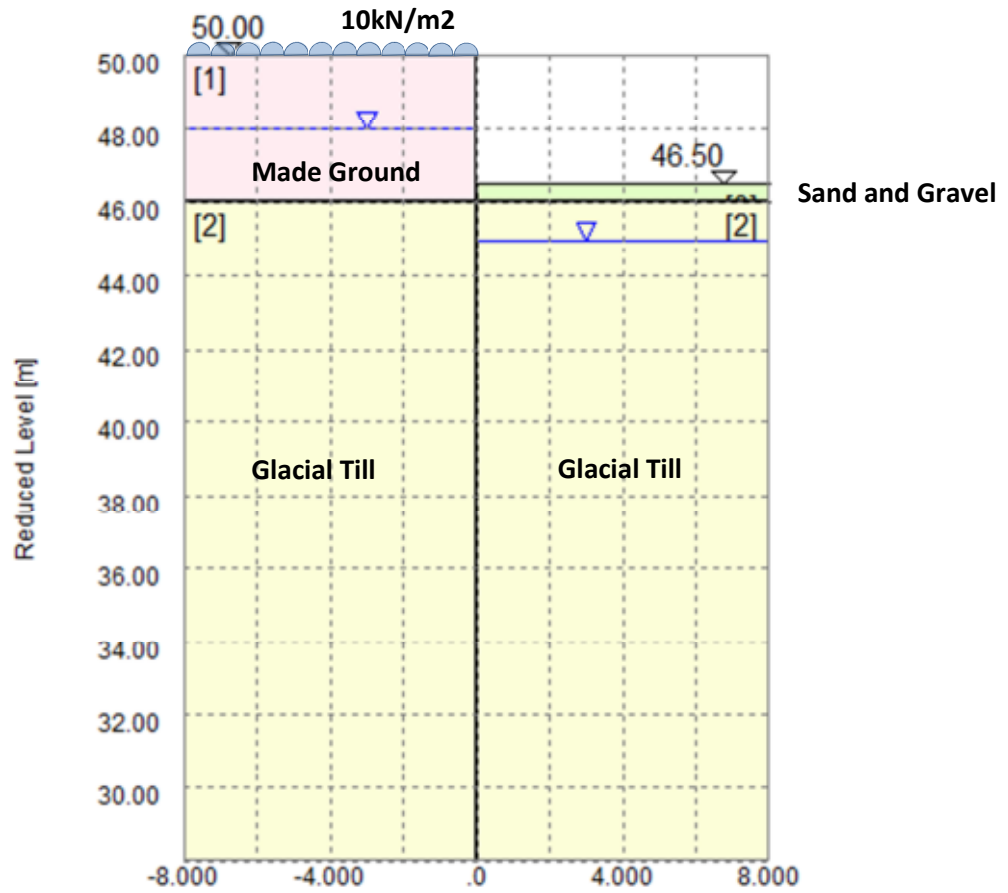
4.2   Stability Analysis

4.2.1 Creating the Input

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

1. Fill the *Titles and Units* dialog box:

(Hint: Should you want to change the Units from the metric default, click the *Units* box here before inputting soil and boundary values).

2. Complete the *Basic Data* dialog box with the following:
    Input the *Level of top node* (+50mOD), which is the top of the wall.
    Input the *Level of rigid boundary* (30mOD), which is often the level of rock as this is considered to act as 'rigid'.
    Specify the *Problem Type* as Stability Check Only.
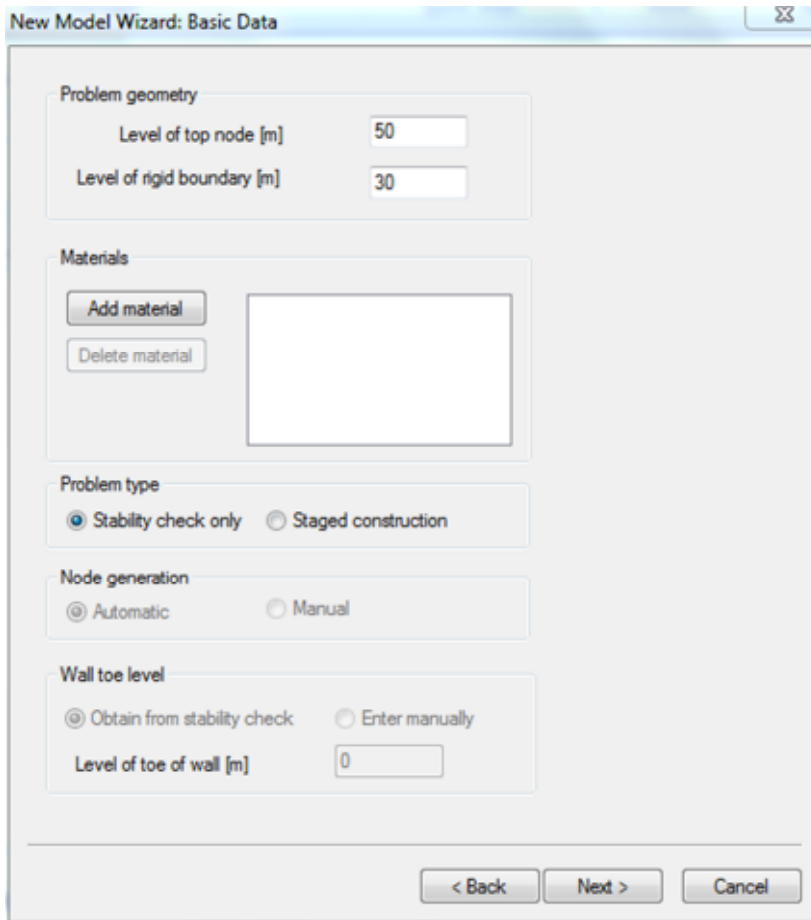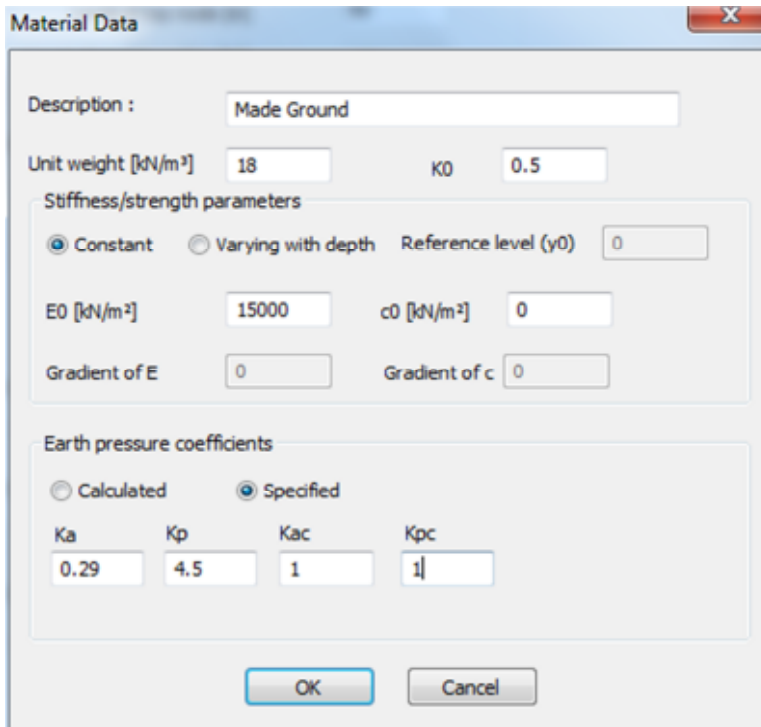
3. Within the *Basic Data* box, click on the *Add Material* box. This box allows you to enter the basic material data.
    Enter the Material Properties for Made Ground, Glacial Till and Sand and Gravel as specified in the tables below.

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients			
					Ka	Kp	Kac	Kpc
Made Ground	18	0.5	15000	0	0.29	4.5	1	1
Glacial Till	21	1.5	45000	0	0.3	5	1	1

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients			
					Phi (deg)	Delta/Phi	Beta	Cw/c ratio
Sand and Gravel	20	1.5	55000	0	27	0.6	0	0

For Made Ground and Glacial Till, tick the *Specified* option in the *Earth Pressure Coefficient* box, as shown in the example below:

Material Data

Description :   Made Ground

Unit weight [kN/m³]   18   KO   0.5

Stiffness/strength parameters

◉ Constant   ○ Varying with depth   Reference level (y0)   0

E0 [kN/m²]   15000   c0 [kN/m²]   0
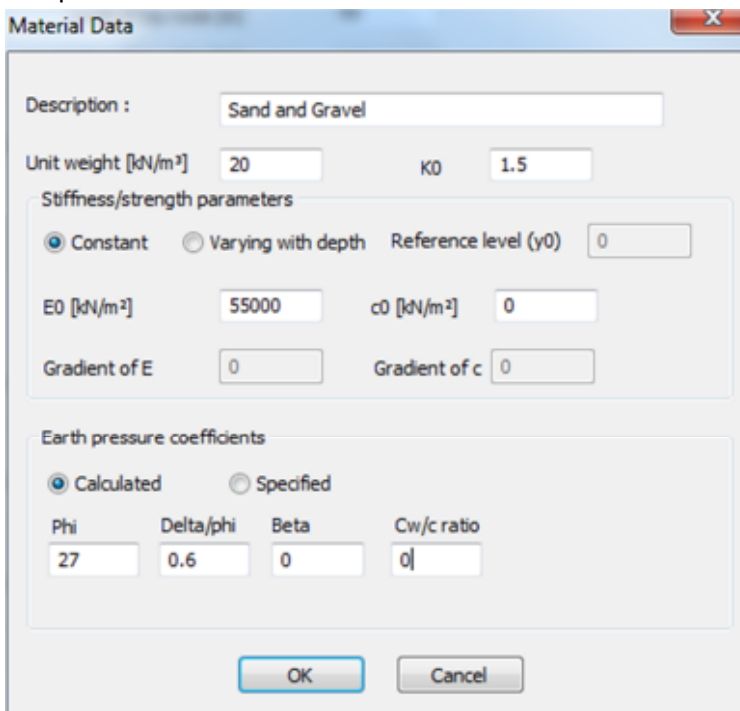
Gradient of E   0   Gradient of c   0

Earth pressure coefficients

○ Calculated   ◉ Specified

Ka	Kp	Kac	Kpc
0.29	4.5	1	1

OK   Cancel

For Sand and Gravel, tick the *Calculated* option in the Earth pressure coefficients box. Once completed click the OK button.

Material Data

Description :   Sand and Gravel

Unit weight [kN/m³]   20   KO   1.5

Stiffness/strength parameters

◉ Constant   ○ Varying with depth   Reference level (y0)   0

E0 [kN/m²]   55000   c0 [kN/m²]   0

Gradient of E   0   Gradient of c   0

Earth pressure coefficients

◉ Calculated   ○ Specified

Phi	Delta/phi	Beta	Cw/c ratio
27	0.6	0	0

OK   Cancel

Click the Next button in the *Basic Data* box.

4. The *Soil Interface* box allows the user to enter the basic stratigraphy as specified in the problem geometry.

New Model Wizard: Soil Interfaces

Initial material layers

Left of wall: Right of wall:

	A	B			A	B
	Level	**Material**			**Level**	**Material**
Defaults		None		Defaults		None
1	50.00	Made Ground		1	46.50	Sand and Gra
2	46.00	Glacial Till		2	46.00	Glacial Till
3				3		
4				4		

Click "Finish" to exit the wizard. The Graphical view will open to allow further editing of soil zones (and optionally node levels if entering nodes manually).

Remember to enter strut, surcharge and groundwater data before proceeding to define more stages or to do stability analysis.

| < Back | Finish | Cancel |

Note: the user must press the tab button at the end of each entry (e.g. after entering Glacial Till in the Materials Column) to ensure the line is saved by the program.

(Hint: Should a mistake be made with the materials or stratigraphy in the Model Wizard, these properties can be easily amended using the Gateway interface in the program once the user has left the Model Wizard.)

5. Click the Finish button and the *Model Wizard* box will close. The Graphical Input will show the stratigraphy:

STAGE 0 : Initial condition
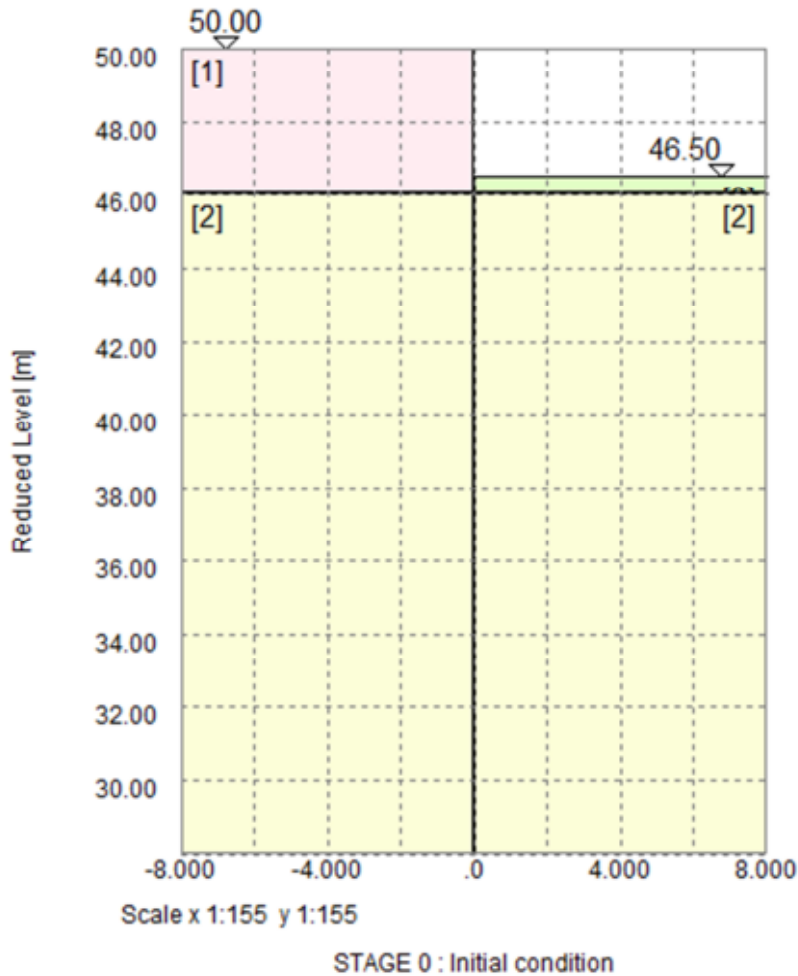
(Hint: Should the user want to amend the stratigraphy, they can do so by right clicking their mouse at the level they want to add a new soil boundary. The level is shown in the top left of the *Graphical Input* box.
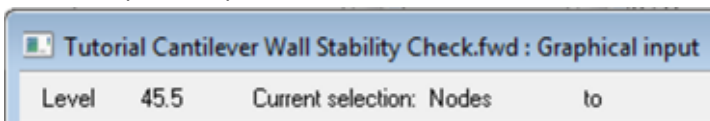
Alternatively, they can minimise the *Graphical Input* box and manually enter the level and material in the *Material Layer* box. The tabs at the bottom allow the user to change material properties on either side of the wall.)
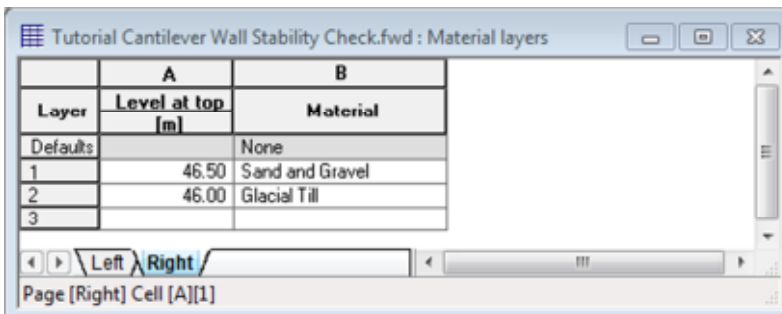
Press the Soil Zones button to call up the *Graphical Input* box.

Press the Water button. Left click on either side of the wall. The *Add water table* box will appear. Enter the water levels as specified in the Geometry section.

Press the Surcharges button and input the surcharges specified in the Geometry section.

Tutorial Cantilever Wall Stability Check.fwd : Surcharges

	A	B	C	D	E	F	G	H
Surcharge	Side	Level [m]	Type	Pressure [kN/m²]	Partial Factor	Offset [m]	Width [m]	Ks
Defaults	Left		UDL		1.00			
1	Left	50.00	UDL	10.00	1.00			
2								

Select which side of the wall

4.2.2 Performing the calculation

1. Prior to calculation, all inputs should be checked (e.g. material properties should be checked by double clicking on Material Properties in the Gateway. In particular, one material parameter (Kr) is not set in the Wizard, and needs to be specified in the table.)

```
Input
    Titles
    Units
    Specification
    Material properties (3)
    Material layers
        Left (2)
        Right (2)
    Strut properties
    Surcharges (1)
    Water data
        Left (1)
        Right (1)
    Partial factors
Output
    Tabular output
```
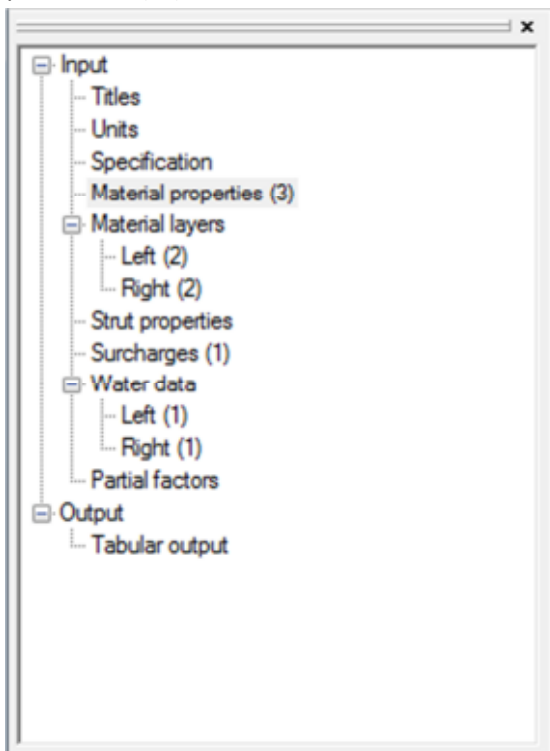
2. The user should save the file using the Save button or pressing Ctrl + S.

Click the Stability Check button. The *Stability Check* box will appear. Click the OK button.

4.2.3 Viewing Results

1. The tabular output will appear. This gives a detailed breakdown of the calculation. The toe depth and success of analysis can be viewed at the end of the tabular output.

```
   1 48.00      0.0

Water data : at Right side of Wall
Unit weight of water  10.00 kN/m³
Point Level Pressure
      [m]  [kN/m²]
   1 45.00      0.0

Surcharge Loads
No.   Side Level Pressure Partial Offset Width
                         Factor
           [m]  [kN/m²]       [m]    [m]
   1 Left 50.00   10.00   1.000

Results of Analysis
The required toe level of the wall [m] = 43.40
Ground level active side [m] = 50.00
Ground level passive side [m] = 46.50
Level         Left                Right          Bending  Shear
                                                 Moment   Force

         Pe      u     Soil Soil   Pe      u
   [m]  [kN/m²] [kN/m²]          [kN/m²] [kN/m²] [kNm/m]  [kN/m]
  50.00   0.0    0.0    1    0    0.0     0.0     0.0      0.0
  49.00   8.120  0.0    1    0    0.0     0.0    -2.320    5.510
  48.00  13.34   0.0    1    0    0.0     0.0   -12.76    16.24
  47.00  15.66  10.00   1    0    0.0     0.0   -37.72    35.74
  46.00  18.60  20.00   2    2   60.00    0.0   -86.78    68.18
  45.00  21.90  30.00   2    2  155.0     0.0  -124.0     0.9297
  44.00  25.20  40.00   2    2  210.0    10.00  -64.74  -128.0
  43.76 433.0   42.35   2    2   13.38   12.35  -30.19  -166.9
  43.64 439.6   43.56   2    2   13.77   13.56  -13.48  -111.4
  43.52 446.2   44.77   2    2   14.17   14.77  -3.384   -56.06
  43.40 452.8   45.97   2    2   14.57   15.97    0.0  0.007281
```

```
Toe depths from stability analysis
     Stage      Calculated Calculation   Analysis                    Reason for failure
                toe depth   interval     successful?
                [m] CD
Stage 0         43.403      1.000        Yes       N.A.
```

Cell [A][1]   NUM   Stage 0

(Hint: Should the user wish to carry out further analysis on the outputs (e.g. in Excel), they can export the file by clicking File > Export)

Click the Graphical output button or double click *Graphical output* in the Gateway menu. This view shows the shear forces, bending moments, pressures and toe level:
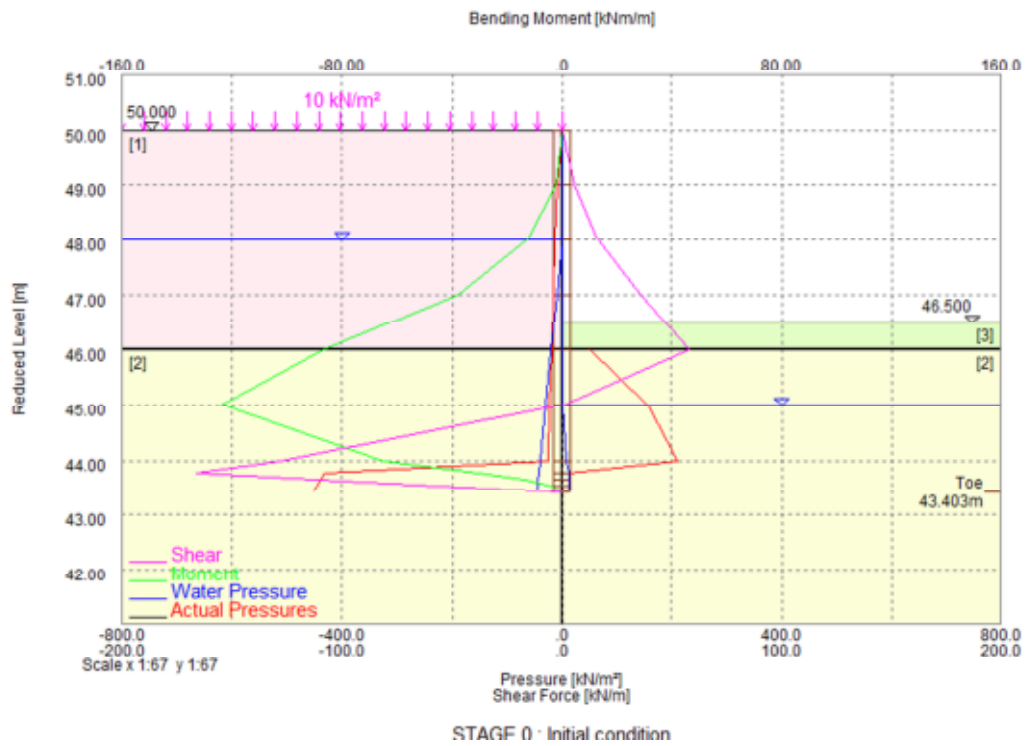
Different parameters can be toggled on and off using the buttons on the Output bar, representing pressures, bending moments and shear forces:

2. The user can print the Graphical Output for reports and calculations. The program allows the user to set the scale and adapt the printout to their company standards. The user can input company information and change the type of printout from Tools > Preferences in the program toolbar.

5.   Propped Retaining Wall

Most retaining wall analyses in Frew analyse the installation of the wall through a number of stages and these often involve props or anchors in the construction. This tutorial example builds on the previous example and demonstrates to the user how to carry out a limit equilibrium and soil structure analysis on various propped retaining wall sequences.

This example demonstrates how Frew can be used, but the data and sequence should not be used as the basis of design for any Geotechnical models the user may plan to model in the future.

5.1   Case A – Fully Drained Sequence

This example requires the user to set up a retaining wall sequence involving drained materials only.

Objectives

- Prepare geometry sequence for reliable analysis using soil structure analysis
- Use the graphical input feature
- Input Material Properties via the Gateway interface
- Input construction stages
- Run a limit equilibrium analysis for a number of stages to find the minimum toe level
- Generate nodes automatically
- Toggle through graphical outputs for a number of stages
- Manipulate graphical outputs (e.g. view results envelopes and zoom into results)

5.1.1 Geometry

Planning the geometry for each stage is critical and should be done before beginning the Frew analysis. The following image shows the construction stages for this particular problem. For each stage, soil boundaries, surcharges, water levels strut properties etc. need to be specified. These values are given in detail at the different points of input in the tutorial.
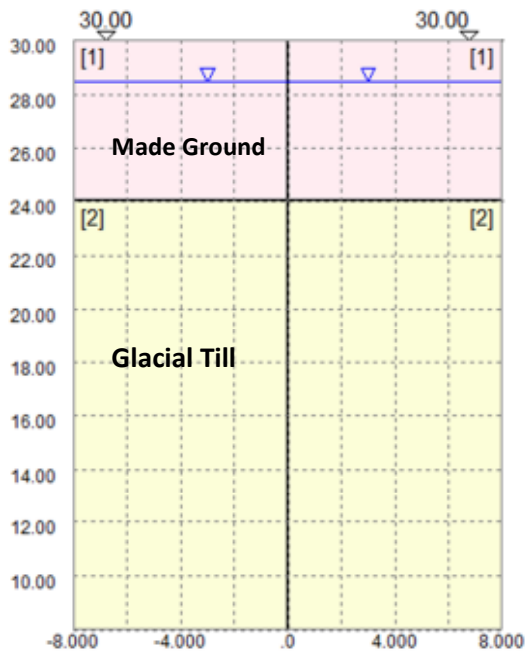
Stage 0 – Initial Condition
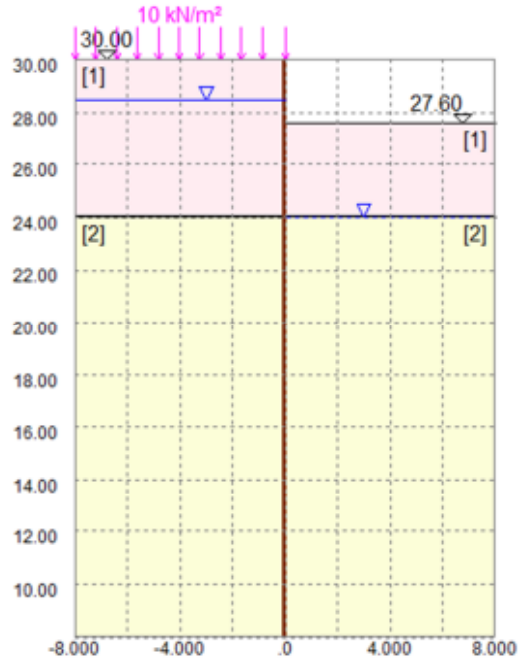
Stage 1 – Install Wall, Dewater and Excavate
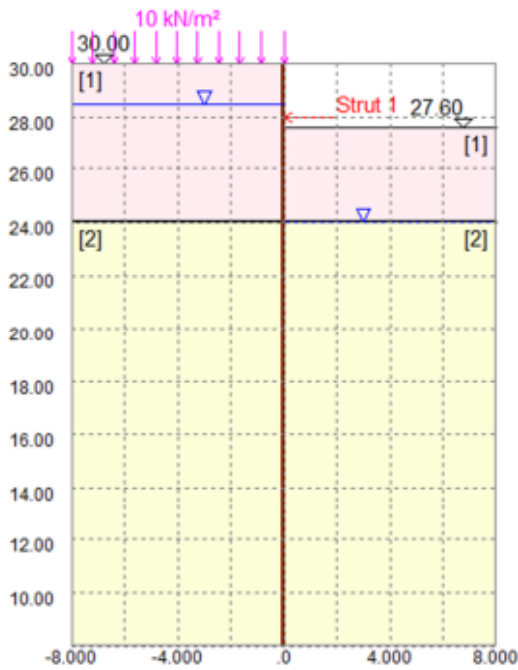
Stage 2 – Anchor Installation

Stage 3 – Lock Anchor and Excavate

Stage 4 – Final Condition

5.1.2 Creating the Input

1. Set up a new Frew file (as outlined in Section 4.2).
2. In the *Basic Data* box, set up the Geometry shown in the Geometry section and in the *Problem Type* section, select staged construction.

3. In the *Basic Data* box, add the materials with the following properties using the *Add Materials* box.
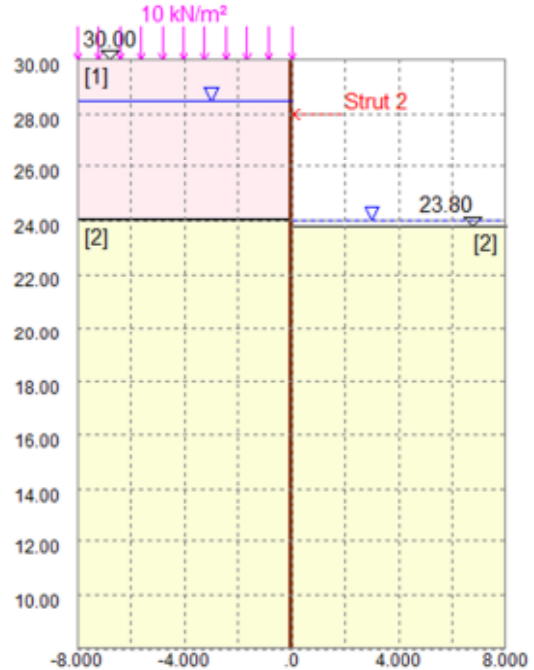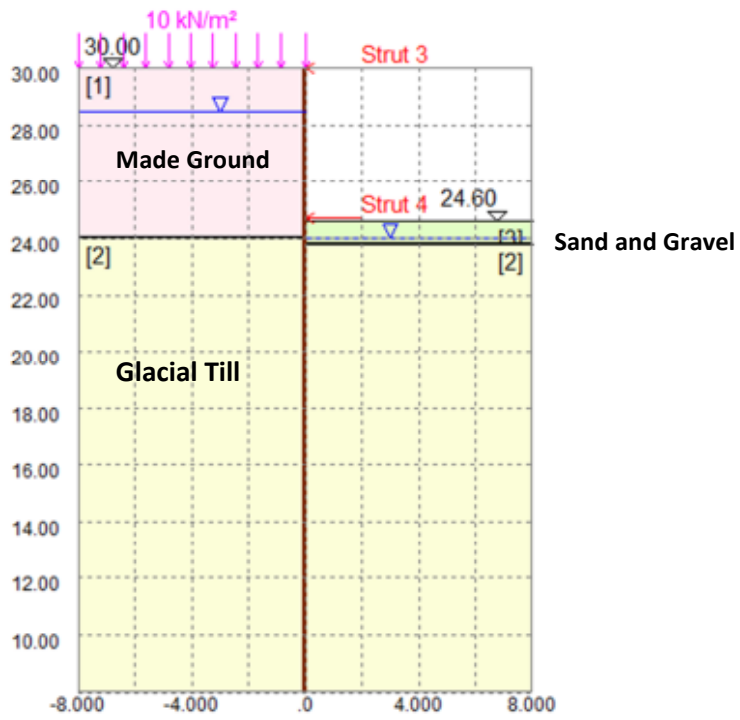
   (Note: the Kr value cannot be input at this stage and will have to be input after exiting the Wizard)

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients				
					Ka	Kp	Kac	Kpc	Kr
Made Ground	18	0.5	15000	0	0.29	4.5	1	1	0.5
Glacial Till	21	1.5	45000	0	0.3	5	1	1	0.3

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients				
					Phi (deg)	Delta /Phi	Beta	Cw/c ratio	Kr
Sand and Gravel	20	1.5	55000	0	27	0.6	0	0	0.5

4. Click Next in the *Basic Data* box. In the *Stage Defaults* box, select Fixed in the Wall/soil interface section.

    Note: The Wall/soil interface option relates to how displacements are calculated in the Safe method and details of the method and interface options can be found in the user manual.

5. In the *Soil Interface* box, input the stratigraphy specified in Stage 0 of the Geometry section.

6. On completion, the *Graphical Input* box will show the following:
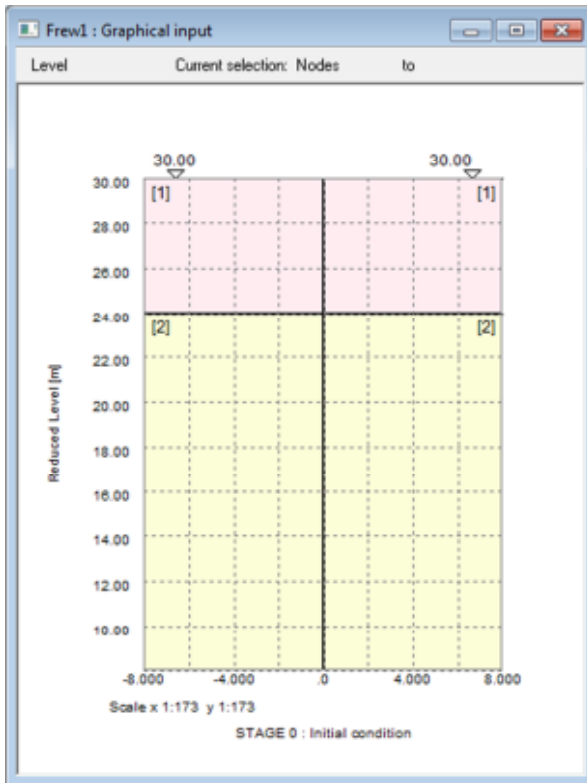
7. Double click on *Material properties* in the Gateway menu. Enter the Kr values for each of the soils specified in the previous table.

Click on the Stage Operations icon to open the *Stage Operations* box

8. Click on the Add Stage button in the *Stage Operations* box and enter "Install Wall, Dewater and Excavate", then click the OK button.
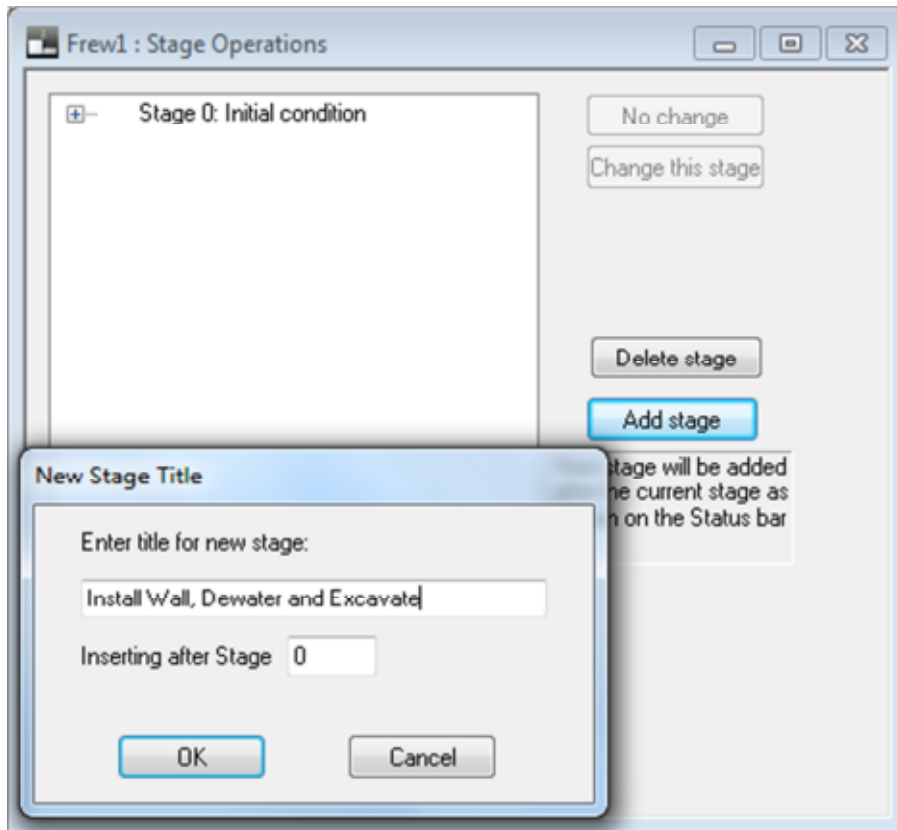
9. The *Graphical Input* box will open at Stage 1.

    (Hint: should the user wish to toggle between stages for Graphical Input, use the Previous and Next icons. The user must always check they are on the correct stage before entering water levels or changing stratigraphy by using the graphical input.)

    Click on the Water icon and set the water level on the right of the wall to 24mOD.

Click on the Dig/Fill icon and right click on the right of the wall. Enter the dig level as shown in Stage 1 of the Geometry Section.

Frew1 : Graphical input

Level   Current selection: Nodes   27.6   to

STAGE 1 : Install Wall, Dewater and Excavate

Scale x 1:173 y 1:173
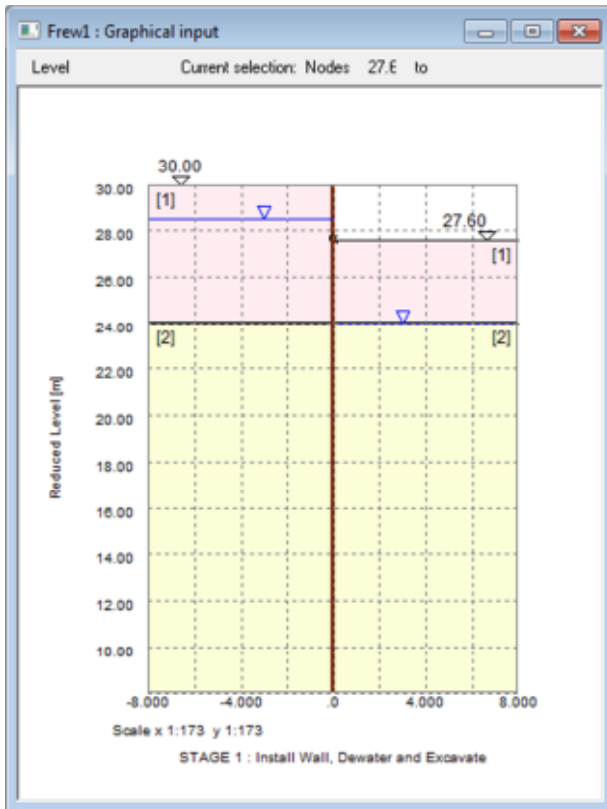
EI   Click on the Wall icon and input the wall stiffness of 90,000 kNm2/m.

Frew1 : Wall EI profile

	A	B
	Level [m]	EI [kNm²/m]
Defaults		
1	30.00	90000.00
2		
3		

Level of node - nodes must be entered in decreasing level order

Click on the Stage Operations icon and add Stages 2 and 3 as specified in the Geometry section.
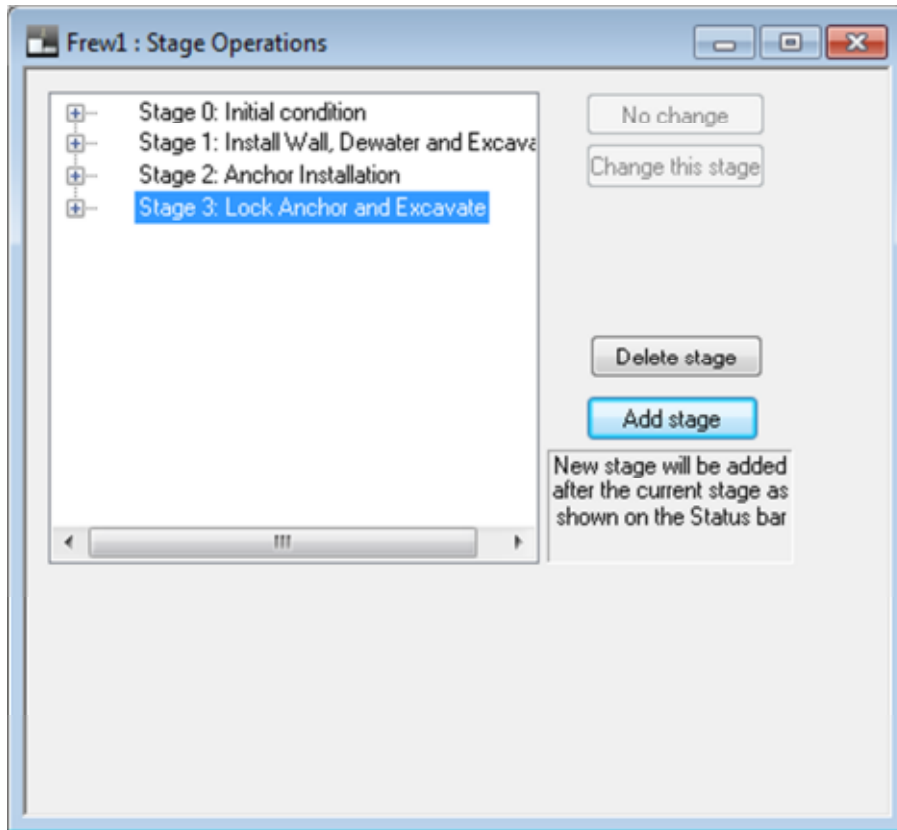
10. The Graphical Input box will open in Stage 3. Use the Dig/Fill icon to excavate to 23.8 mOD as shown in the Geometry section.

Click on the Stage Operations icon and add Stage 4 as specified in the Geometry Section.

11. In Stage 4, double click on *Soil Zones* in the Gateway.
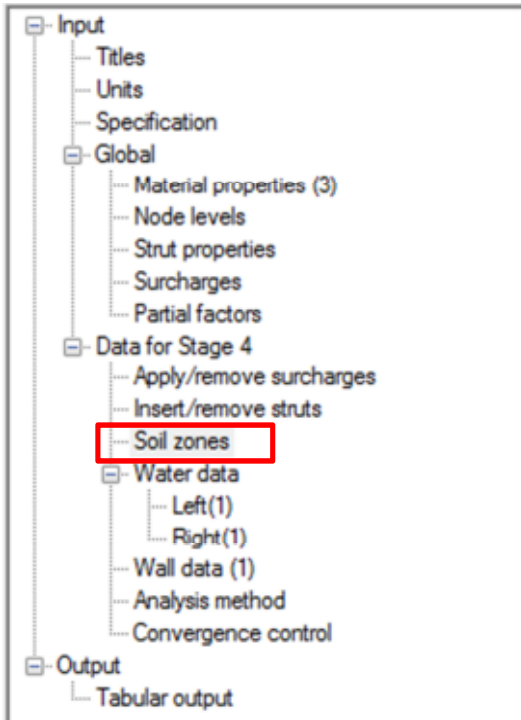
Input the stratigraphy as shown in the Geometry Section in the *Material Layers* table.

Layer	A Level at top [m]	B Material
Defaults		None
1	24.60	Sand and Gravel
2	23.80	Glacial Till
3		

◄ ► \ Left ⋀ Right /

Press <TAB> to start a new record
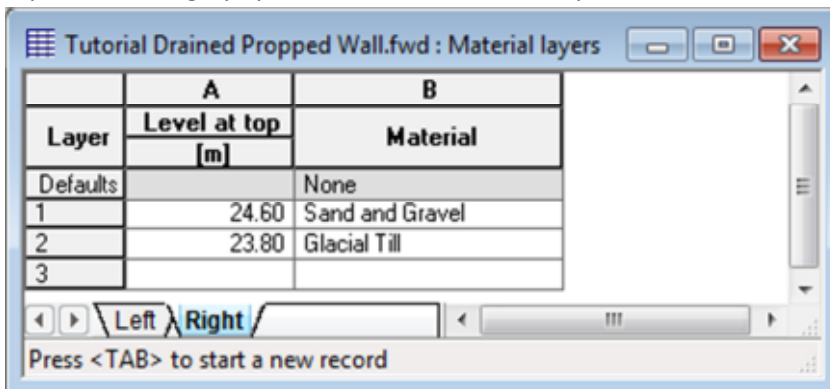
(Note: should the above table not appear automatically, minimise the Graphical Input view and the table will be accessible.)

Click on the Surcharge icon and input a UDL of 10kN/m to the left of the wall.

Surcharge	A Stage In	B Stage Out	C Side	D Level [m]	E Type	F Pressure [kN/m²]	G Partial Factor	H Offset [m]	I Width [m]	J Ks
Defaults			Left		UDL		1.00			
1	1		Left	30.00	UDL	10.00	1.00			
2										

Enter stage number to remove surcharge (blank if never removed)
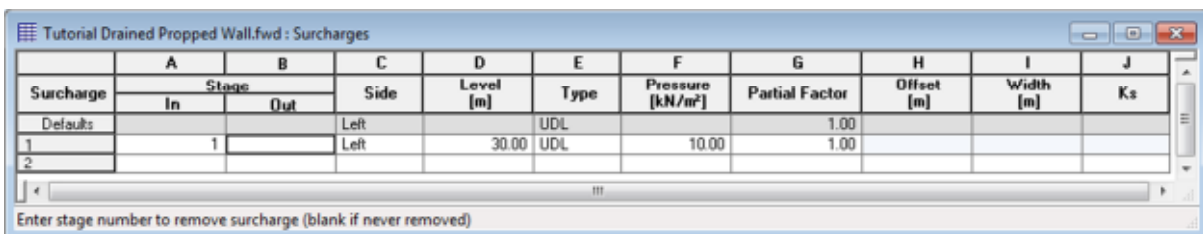
(Note: partial factors can be applied to the surcharges in accordance with various design codes, should this be required by the user. This will be covered in more detail in Section 5.4)

Click on the Strut icon and input the strut properties as specified in the Geometry section.

Strut	A Stage In	B Stage Out	C Level [m]	D Prestress [kN/m]	E Stiffness [kN/m/m]	F Angle [°]	G Lever arm [m]
Tutorial Drained Propped Wall.fwd : Struts							
Defaults							
1	2	3	28.000	200.00	0.00	40.00	0.30
2	3	4	28.000	200.00	5000.00	40.00	0.30
3	4		30.000	0.00	30000.00	0.00	0.00
4	4		24.650	0.00	30000.00	0.00	0.00
5							

Press <TAB> to start a new record

(Hint: press = key to copy a value from the above row and press == keys to copy an entire row.)

12. Check stratigraphy and properties at the different stages and save your model. The model is now ready for analysis

5.1.3 Stability Analysis

This Stability Analysis has been split into two sections; Stability Analysis (many nodes) and Stability Analysis (optimal). The purpose of this is to demonstrate how nodes are generated in Frew and the methods that can be used to optimise the number of nodes. Consequently, it is important the user considers this when devising the construction sequence.

It also introduces the user to amending their input file.

Stability Analysis (Many Nodes)

Click on the Stability check icon.

1. In the *Stability check* box, uncheck the option which allows you to automatically generate nodes from the calculated toe level. This allows the user to review the stability check results and then progress to generating the nodes.

Stability check

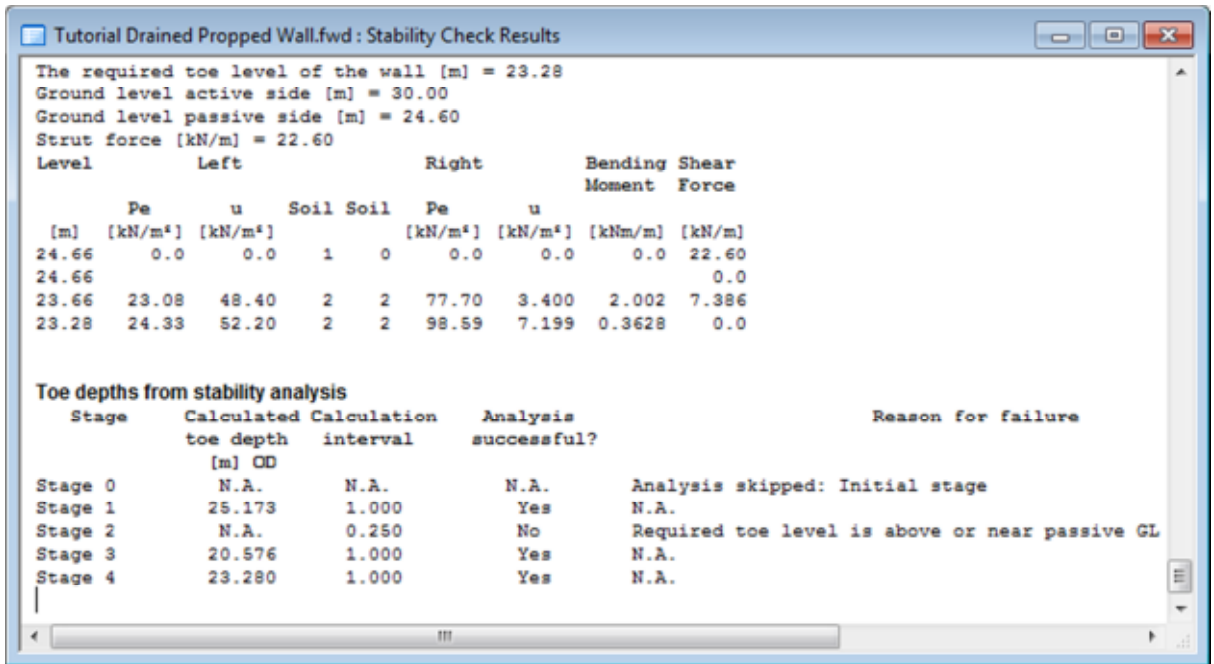|	A	B	C	D	
	Stage	Failure mechanism	Rotation strut index	Balance water pressure	
Defaults		Fixed Earth		Yes	
1	Stage 1	Fixed Earth		No	
2	Stage 2	Free Earth	Strut 1	No	
3	Stage 3	Free Earth	Strut 2	No	
4	Stage 4	Free Earth	Strut 4	No	

Default calculation interval [m] :   1

Iteration limit: 1000

☐ Generate nodes from the lowest calculated toe depth from successfully analysed stages

[OK]   [Cancel]

The user can now view the results of the limit equilibrium analysis and, at the end of the tabular output, the toe levels for the different construction stages are shown.
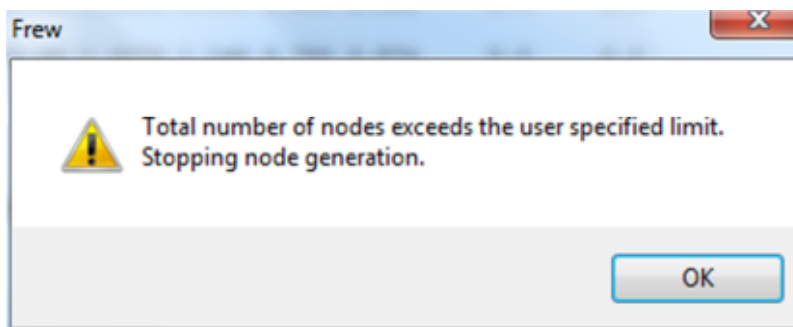
```
Tutorial Drained Propped Wall.fwd : Stability Check Results                    ⊟ ▫ ✕

The required toe level of the wall [m] = 23.28
Ground level active side [m] = 30.00
Ground level passive side [m] = 24.60
Strut force [kN/m] = 22.60
Level          Left                    Right           Bending Shear
                                                       Moment  Force

         Pe       u      Soil Soil    Pe       u
  [m]  [kN/m²]  [kN/m²]              [kN/m²]  [kN/m²]  [kNm/m]  [kN/m]
 24.66   0.0     0.0       1    0     0.0      0.0      0.0    22.60
 24.66                                                          0.0
 23.66  23.08   48.40      2    2    77.70    3.400    2.002   7.386
 23.28  24.33   52.20      2    2    98.59    7.199   0.3628    0.0


Toe depths from stability analysis
     Stage        Calculated Calculation    Analysis                      Reason for failure
                  toe depth   interval      successful?
                   [m] OD
 Stage 0            N.A.        N.A.           N.A.      Analysis skipped: Initial stage
 Stage 1           25.173      1.000           Yes      N.A.
 Stage 2            N.A.        0.250           No       Required toe level is above or near passive GL
 Stage 3           20.576      1.000           Yes      N.A.
 Stage 4           23.280      1.000           Yes      N.A.
|
                              ⠇⠇⠇
◄                                                                              ►
```

2. To carry out the soil structure interaction analysis, nodes need to be generated. Frew can generate nodes using the toe depth from the stability analysis and the data for the construction sequences already inputted. The following will demonstrate why the user will need to be familiar with how Frew creates nodes prior to generating the nodes using the automatic node generation function.
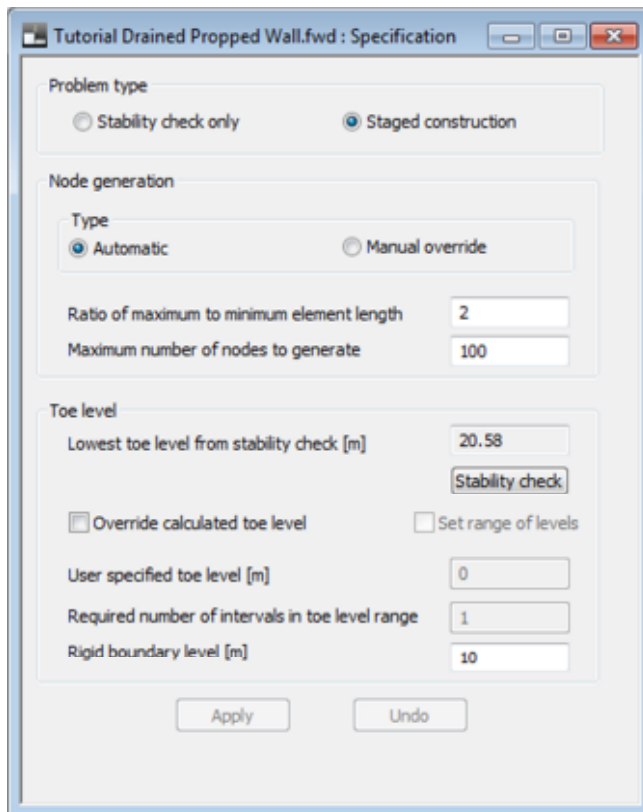
⋮ Click on the Generate node icon.
   The following warning sign will appear:

```
Frew                                            ✕

   ⚠   Total number of nodes exceeds the user specified limit.
        Stopping node generation.


                                        [  OK  ]
```

(Note: the number of nodes have been limited in Frew as too many nodes will take longer to converge and slow the analysis process. Instances where more nodes would be necessary would be when fixed node locations are in close proximity and therefore node spacing needs to be very small.)

3. The user can increase the number of nodes generated by double clicking *Specification* in the Gateway menu.

The Maximum number of nodes to generate should be increased to 200. Click the Apply button and generate the nodes using the Generate node icon.

4. The number of nodes is shown in Node levels in the Gateway menu. For this problem, 106 nodes have been generated.

   The reason for this number of nodes being generated for a simple problem is that the strut level (+24.65mOD) is very close to the soil boundary (top of Sand and Gravel at 24.6mOD).

Stability Analysis (Optimal)

1. Although the soil structure interaction analysis will run for the previous stability analysis, it is optimal to change either the rigid boundary level or the strut level. In this case, the user should change the strut level to +24.7mOD.

✖   Click on the Delete node icon . This will delete the generated nodes and allow the user to change their Input data.

↱   Click on the Strut icon to change the strut properties.

Tutorial Drained Propped Wall.fwd : Struts							
	A	**B**	**C**	**D**	**E**	**F**	**G**
Strut	**Stage**		**Level**	**Prestress**	**Stiffness**	**Angle**	**Lever arm**
	In	**Out**	**[m]**	**[kN/m]**	**[kN/m/m]**	**[°]**	**[m]**
Defaults							
1	2	3	28.000	200.00	0.00	40.00	0.30
2	3	4	28.000	200.00	5000.00	40.00	0.30
3	4		30.000	0.00	30000.00	0.00	0.00
4	4		24.700	0.00	30000.00	0.00	0.00
5							

Press <TAB> to start a new record

▮   Click on the Stability check icon to re-run the analysis with the amended strut level.

Tutorial Drained Propped Wall.fwd : Stability Check Results

```
24.71                                              0.0
23.71   22.91   47.90   2   2   74.95   2.900   2.539  8.850
23.25   24.44   52.55   2   2   100.5   7.546 0.05126   0.0
```

Toe depths from stability analysis

| Stage	Calculated toe depth [m] OD	Calculation interval	Analysis successful?	Reason for
Stage 0	N.A.	N.A.	N.A.	Analysis skipped: Initial stage
Stage 1	25.173	1.000	Yes	N.A.
Stage 2	N.A.	0.250	No	Required toe level is above or r
Stage 3	20.576	1.000	Yes	N.A.
Stage 4	23.245	1.000	Yes	N.A.

⋮   Click on the Generate node icon. 54 nodes will be generated and it should be noted that this is significantly less than the previous analysis with the strut 0.5m lower.

5.1.4 Soil Structure Interaction Analysis
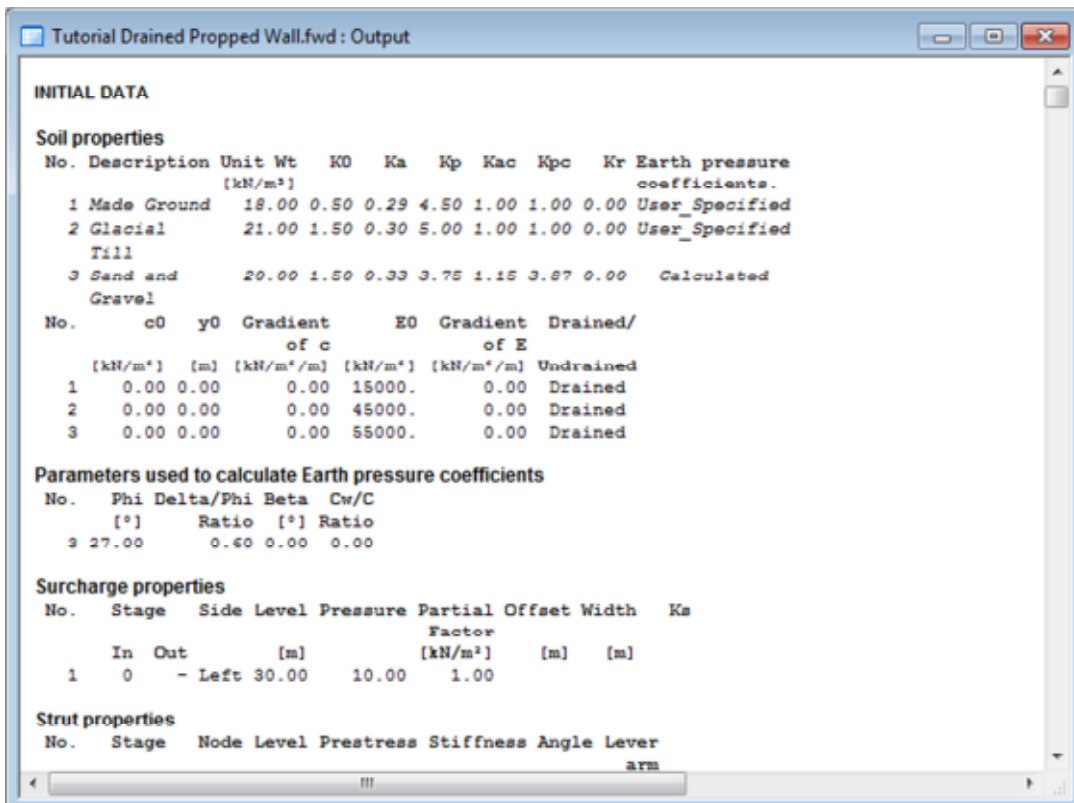
Σ

Click on the analyse button to carry out the Soil Structure Interaction Analysis.

The solution progress window will automatically appear and the analysis will run.
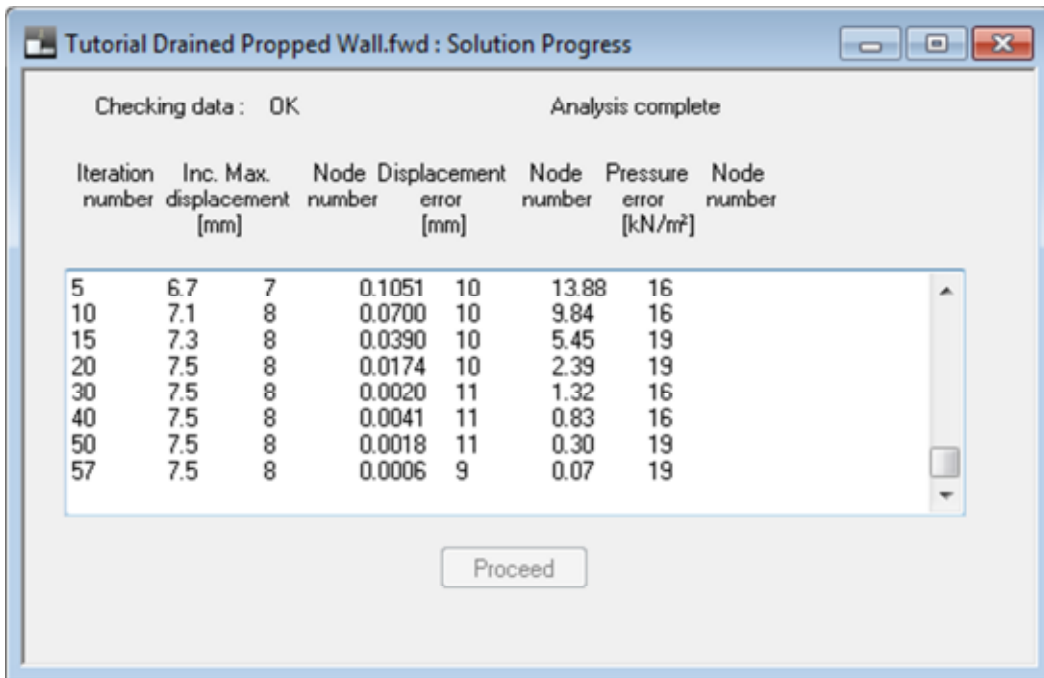
Upon completion, the tabular output will appear. As previously shown, this can be exported to excel for further analysis.
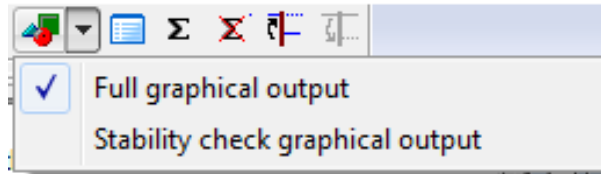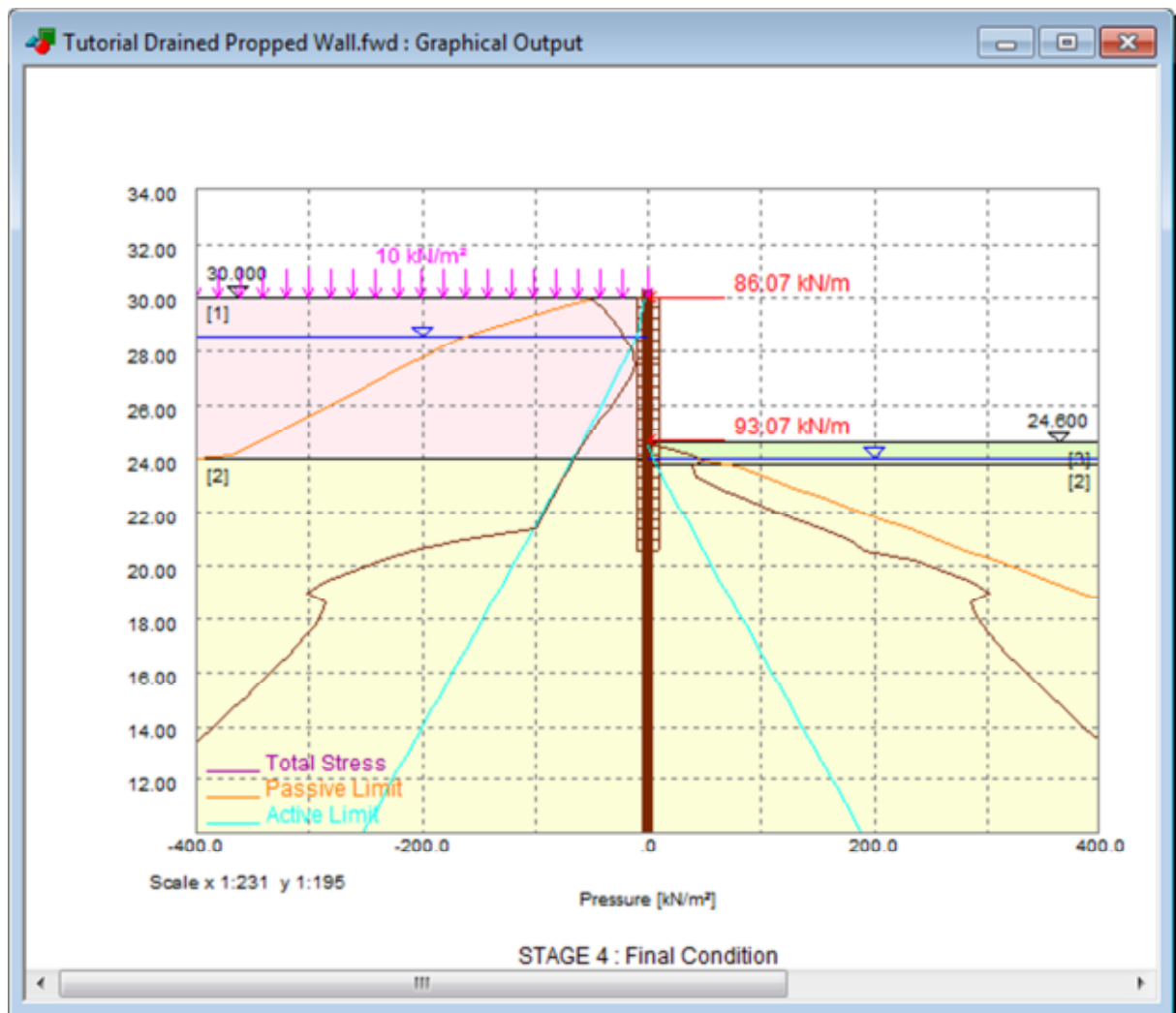
5.1.5 Viewing Results

Click on the black arrow next to the Graphical output button.
The program gives you the option to view the Graphical Output from the Full analysis (Soil Structure Interaction) or the Stability check.

✓	Full graphical output
	Stability check graphical output

1. Click on the Full graphical output.

Tutorial Drained Propped Wall.fwd : Graphical Output

STAGE 4 : Final Condition

(Note: should the Full graphical output window not open, the user should check the opened windows for the Stability check graphical output. This needs to be closed for the program to allow the user to open the Full graphical output.)

✉   Click the Envelope icon to view the maximum values for all the construction stages for various outputs for the retaining wall.


Envelope for stages 0 to 4
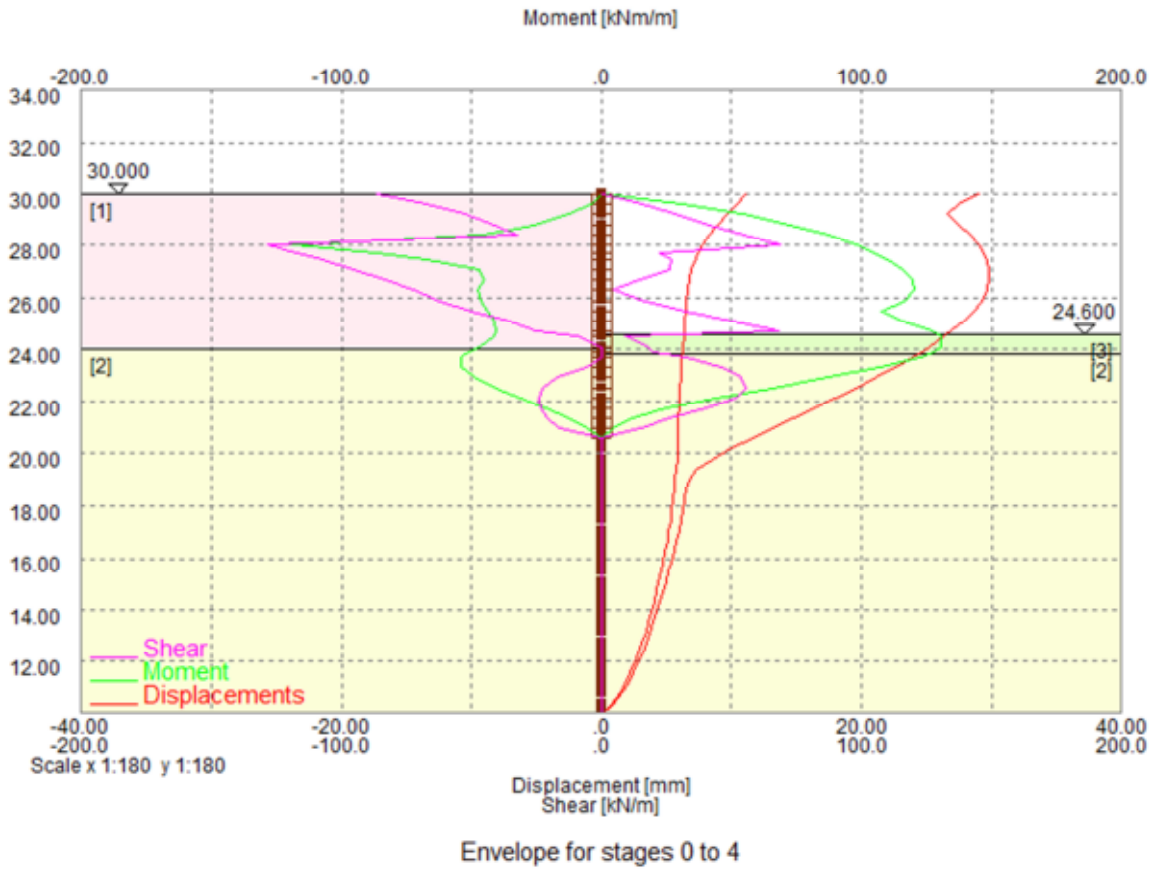
2. As with the previous tutorial example, the user can switch toggle different parameters on and off. These can be examined and compared for the different stages by using the Previous and Next icons (⊩ ⊪).

3. The user can zoom into sections of the graphical output for viewing or as an output for reports by right clicking and dragging in the required area.

Moment [kNm/m]
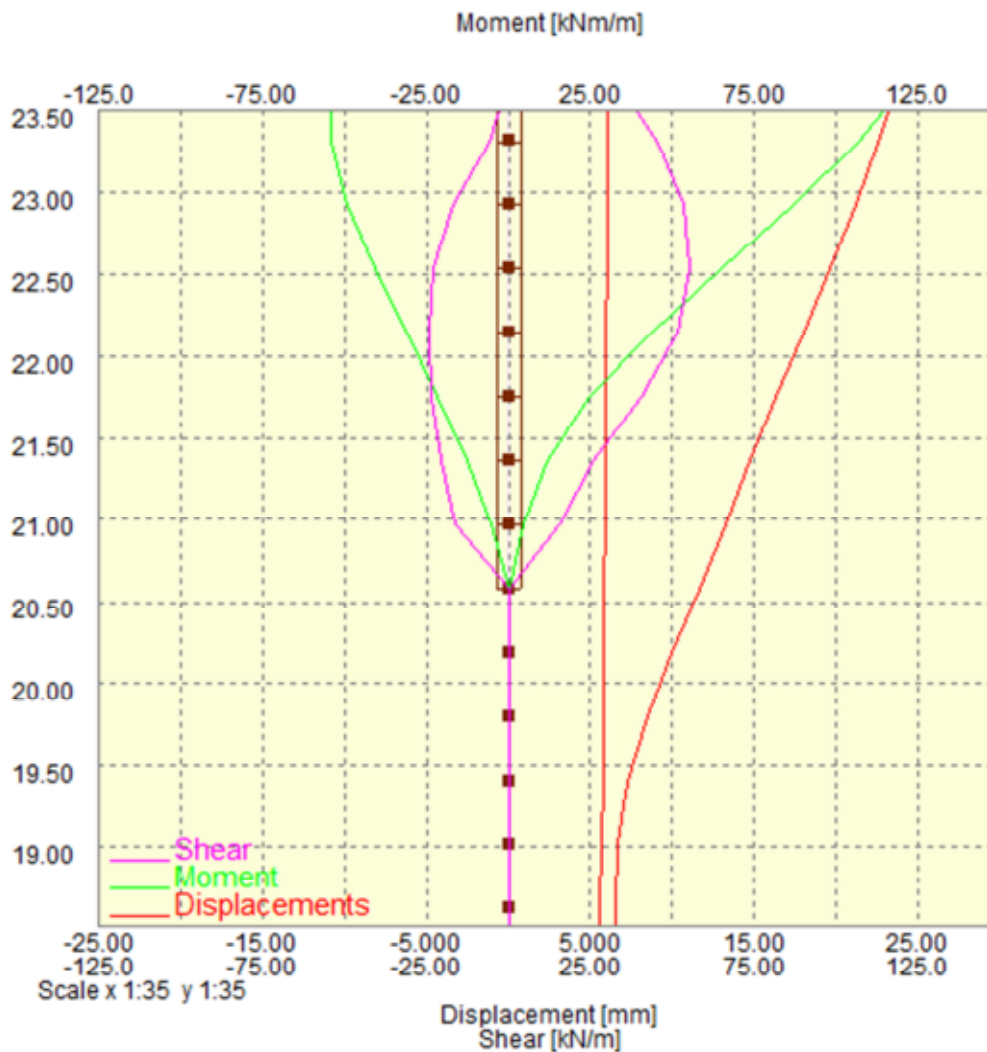
Click on the Unzoom icon to view the results for the full wall.

4. The *Graphics* option allows the user to change the format of the Graphical output for reports and for calculation purposes. The user should make themselves familiar with the different options shown:
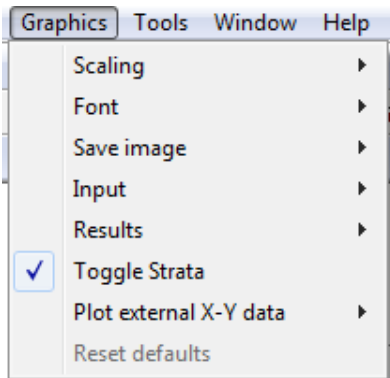
5.2   Case B –Undrained and Drained Sequence Analysis

Most retaining wall problems involve the analysis of soils in the drained and undrained conditions.

There are two ways of modelling undrained soils in Frew; using user-defined pressures or calculated pressures. Frew simplifies the calculated pressure method by allowing the user to use the drained soil properties for calculation of the undrained pressures.

This example shows how undrained pressures can be modelled in Frew. It allows the user to input different calculated and user defined earth pressure coefficients.

Objectives:

- Input undrained material properties and understand the different methods to do so
- Amend construction stages within a Frew file
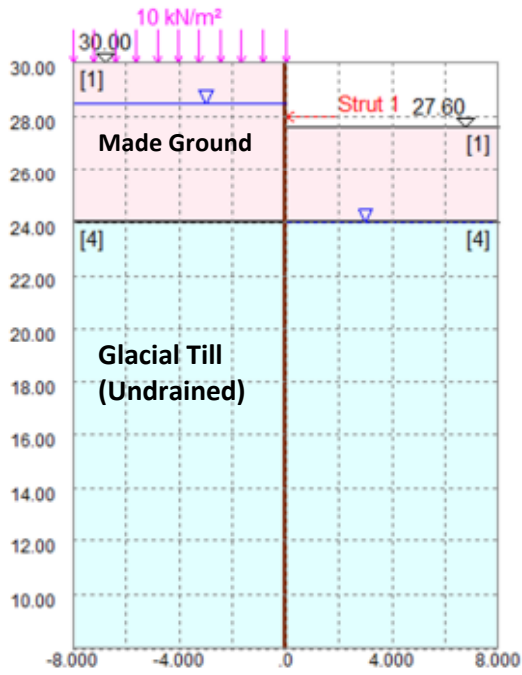
5.2.1 Geometry

The geometry for both examples is identical to the previous example but the soil properties are different for both examples. Consequently, the user can adapt the file from the previous example. This file should be saved with a new filename to avoid confusion.
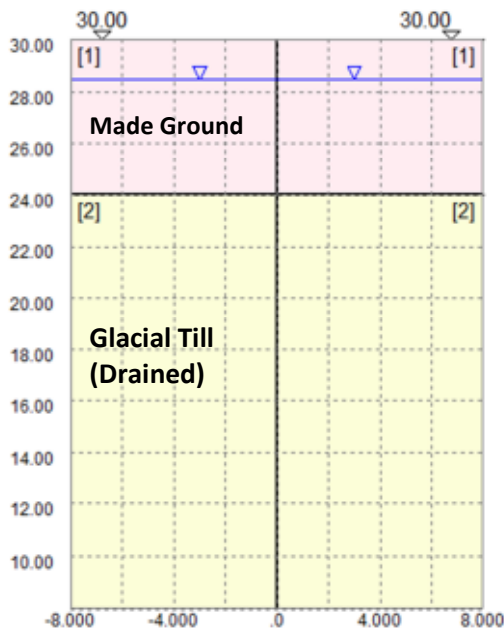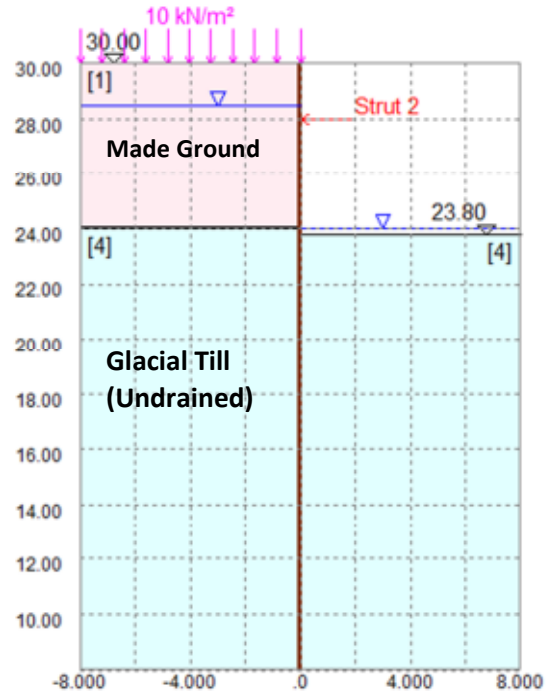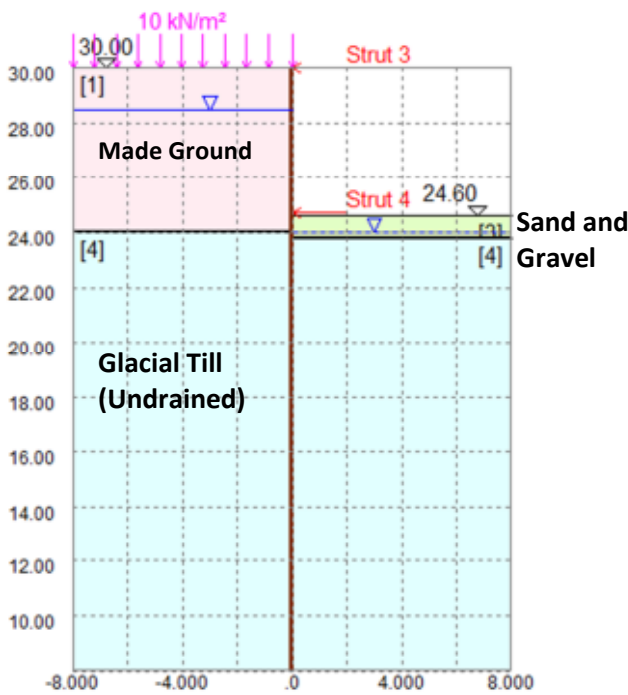

Stage 0 – Initial Condition (Drained)
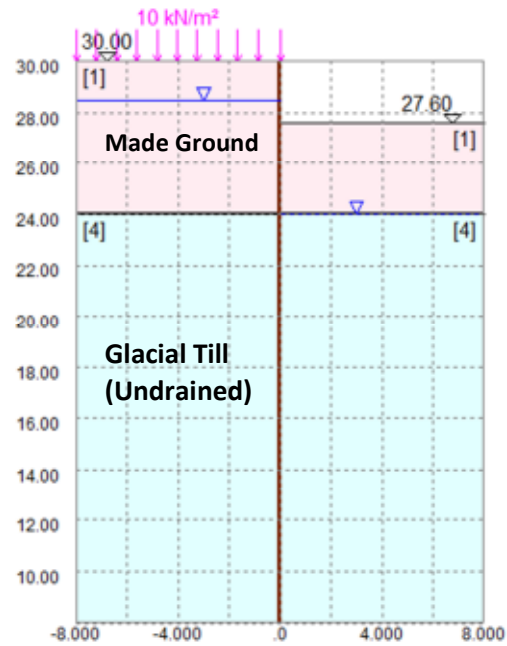
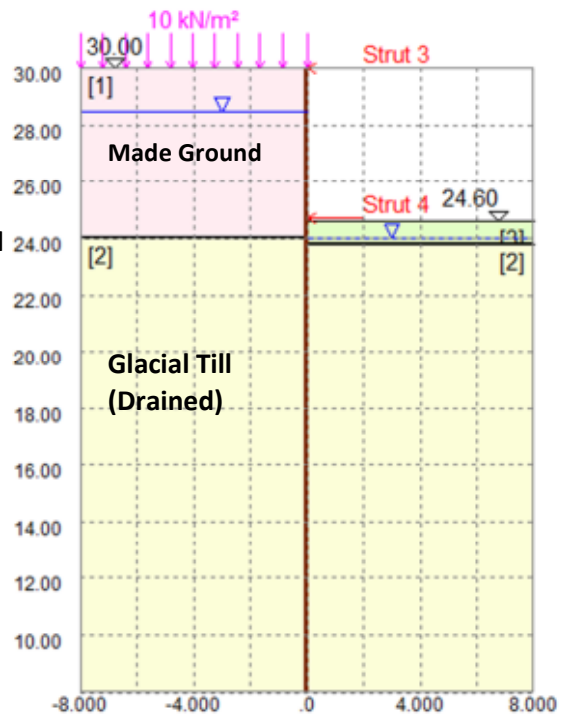Stage 1 – Install Wall (UD), Dewater and Excavate

Stage 2 – Anchor Installation (UD)

Stage 3 – Lock Anchor and Excavate (UD)

Stage 4 – Final Condition

Stage 5 – Long Term
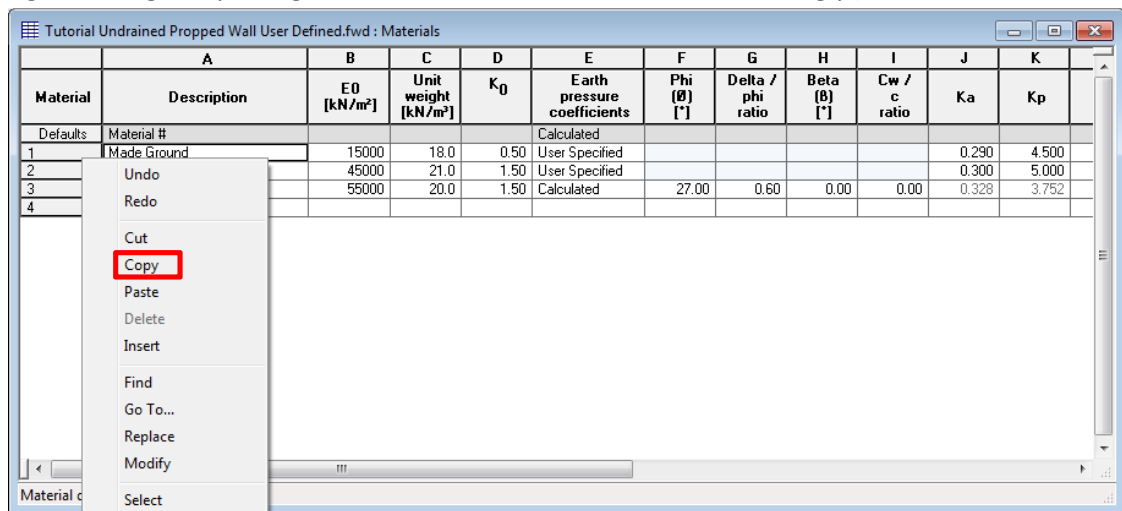
5.2.2 User defined pressures

Creating the Input

1. Save the previous 'Fully Drained Propped Wall.fwd' tutorial example as 'Undrained Propped Wall User Defined'.

2. Amend the *Titles input* in the Gateway menu to indicate that the analysis is applying user defined undrained material properties.

3. Double click on *Material Properties* in the Gateway menu
   Add the following material properties in a new row

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients				
					Ka	Kp	Kac	Kpc	Kr
Glacial Till (Undrained)	21	1.5	60000	35	1	1	2	2	1

(Hint: as the properties are similar to Glacial Till, the user can left click on the Glacial Till row number, right click and copy.

They can then paste the values into the empty row by left clicking on the empty row and right clicking and pasting. The user must amend the values accordingly.)

The user must also specify the material as Undrained with a Shape Factor of 1 and the Material number for effective stress parameters as 0 (as the pressures are user defined and not calculated).

(Note: the Shape Factor defines whether the material follows a Mohr Coulomb or Modified Cam Clay envelope.)

Tutorial Undrained Propped Wall User Defined.fwd : Materials

Material	A Description	B E0 [kN/m²]	C Unit weight [kN/m³]	D K₀	E Earth pressure coefficients	F Phi (Ø) [°]	G Delta / phi ratio	H Beta (β) [°]	I Cw / c ratio	J Ka	K Kp	L Kac	M Kpc	N Kr	O c0 [kN/m²]	P y0 [m]	Q Gradients c	R Gradients E	S Drained/ undrained	T Shape factor	U Material no. for effective stress parameters
Defaults	Material #				Calculated														Drained		
1	Made Ground	15000	18.0	0.50	User Specified					0.290	4.500	1.000	1.000	0.500	0.00	0.00	0.00	0.00	Drained		
2	Glacial Till	45000	21.0	1.50	User Specified					0.300	5.000	1.000	1.000	0.300	0.00	0.00	0.00	0.00	Drained		
3	Sand and Gravel	55000	20.0	1.50	Calculated	27.00	0.60	0.00	0.00	0.328	3.752	1.145	3.874	0.500	0.00	0.00	0.00	0.00	Drained		
4	Glacial Till (Undrained)	60000	21.0	1.50	User Specified					1.000	1.000	2.000	2.000	1.000	35.00	0.00	0.00	0.00	Undrained	1.00	0
5																					

Material from which to obtain effective stress parameters

Click on the Stage Operations icon and left click on Stage 1 to amend the stage title to "Install Wall in Undrained Condition, Excavate and Dewater".
Double left click on Stage 1 and this will open up the stage tree:
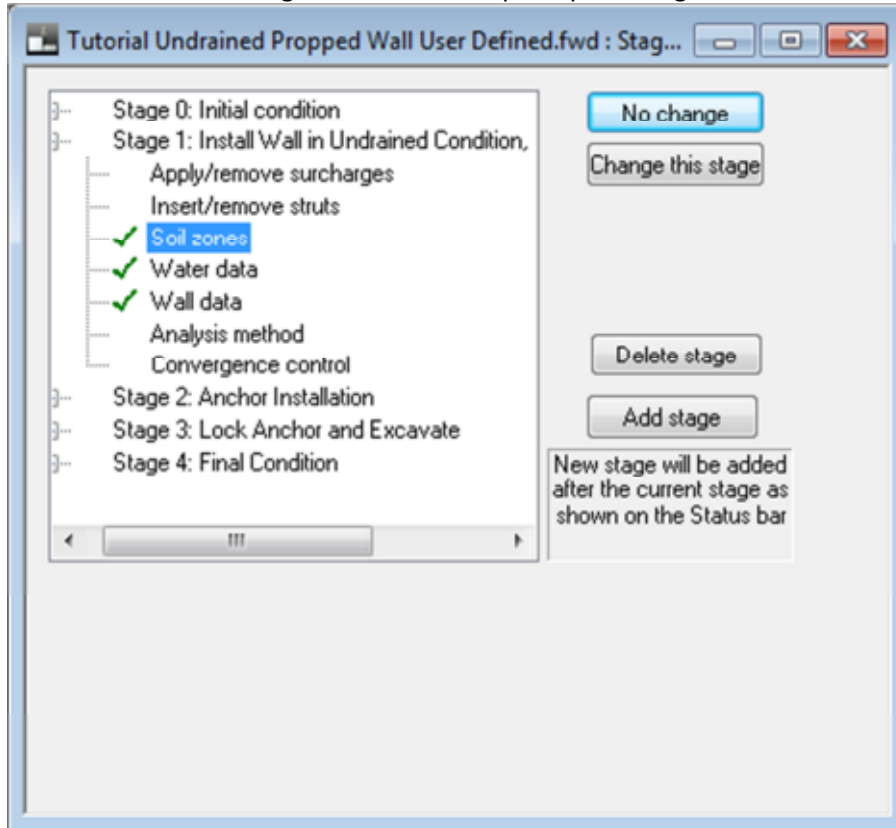
Double click on *Soil zones* to open up the Graphical input interface at this stage.

4. Hover the mouse over the Glacial Till layer to the left of the wall and right click to change the material to Glacial Till (undrained). Repeat for the right side of the wall.
   Use the next button and repeat to give subsequent stages the Undrained Glacial Till layer.

Click on the Stage Operations icon and insert Stage 5; Long term.

5. In the Graphical input window, change the Glacial till properties for Stage 5 to the drained Glacial Till (defined as Glacial Till in the users Materials menu).
   (Hint: at this point, the user should scroll through the different stages in the Graphical Input interface to ensure the stages are appropriate)

Performing the Calculation

This is the same as for the Fully Drained Propped Retaining Wall, Sections 5.1.3 and 5.1.4, but is listed below to assist the user.

1. Carry out the Limit Equilibrium stability analysis
2. Generate the nodes
3. Carry out the Soil structure interaction analysis.

5.2.3 Calculated pressures

Creating the Input

1. Save the previous 'Undrained Propped Wall User Defined.fwd' as 'Undrained Propped Wall Calculated.fwd'.
2. Amend the *Titles input* in the Gateway menu to indicate that the analysis is applying calculated undrained material properties.
3. Double click *Material properties* in the Gateway menu.
   Using the table below, change the Glacial Till Properties to Firm/Stiff Clay and the Glacial Till (Undrained) Properties to Firm/Stiff Clay (Undrained).

Soil	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients				
					Phi (deg)	Delta/Phi	Beta	Cw/c ratio	Kr
Firm/Stiff Clay	20	1	48000	0	27	0.6	0	0	0.5
Firm/Stiff Clay (Undrained)	20	1	60000	60	27	0.6	0	0.5	1

Performing the calculation

This is the same as for the Fully Drained Propped Retaining Wall, Sections 5.1.3 and 5.1.4, but is listed below to assist the user.

1. Carry out the Limit Equilibrium stability analysis
2. Generate the nodes
3. Carry out the Soil structure interaction analysis.

5.3   Case C – Wall Relaxation

Wall relaxation is often applied to model the change in stiffness of the wall in the long term. The geometry of the analysis is the same as Case B but an extra stage is incorporated into the end of the construction sequence to model relaxation.

However, changing the stiffness of the wall alone is not sufficient as this does not model the change in stresses due to wall relaxation.

This tutorial example demonstrates the simplicity of this method in Frew and will show the user how to add the stage, apply the change in stiffness and apply the relaxation factor in Frew.

Objectives:

- Change EI values in Frew
- Input wall relaxation parameters
- Apply strut relaxation

5.3.1 Creating the Input

1. Open the previous file 'Undrained Propped Wall Calculated.fwd'
2. Save the file as 'Undrained Propped Wall Calculated Wall Relaxation'
3. Amend the *Titles* in the Gateway menu to specify that the analysis will consider Wall Relaxation

Select the Delete node icon to delete the Soil Structure Interaction Analysis, the Stability Analysis and nodes.

Click on the Stage Operations icon. Add a Wall and Strut Relaxation stage after Stage 5.

4. At Stage 6, double click on *Wall Data* in the Gateway menu and change EI to 60,000 kNm2/m.

5. At Stage 6, double click on *Analysis Method* in the Gateway menu. Change the wall relaxation to 33.3% and click on the Apply button.

(Hint: the wall relaxation is calculated using the ratio of short term and long term stiffnesses. The method of calculation is specified in the User Manual)

6. Double click on *Strut properties* in the Gateway. Remove the struts from the previous stage and apply new struts with half the stiffness in Stage 6.

5.4   Case D – Application of Partial Factors

Frew has simplified the application of partial factors in the analysis. Partial factors can be applied to the whole analysis through the Gateway menu in the Global section. The user can define the partial factors. However, Frew has incorporated common retaining wall codes such as EC7 and Ciria C580, into the partial factor options to assist engineers with their design.

This tutorial will require that the user use the skills learnt in previous tutorials. They will then be led through how to compare the impact of factoring according to EC7 Design Approach 1 on the retaining wall sequence.

NOTE: This is not an exhaustive guide for the user to carry out analysis to EC7 or any other code. The user must be familiar with the code and its use prior to carrying out the analysis and all outputs must be checked to ensure the analysis shows the retaining wall behaving in an appropriate manner.

Objectives

- Apply Partial Factors to a staged construction analysis
- Compare the impact of different factors on the analysis output

5.4.1 Creating the Input

The user is expected to utilise the methods learnt in the previous tutorials to create the input file. Consequently, only basic geometry is given. However, should the user want to save time, Oasys have provided the file prior to the application of Partial Factors and analysis.

1. Create a file with the Geometry specified in Case B (Section 5.2) but with the material parameters shown below:

Soil	Condition	Unit Weight (kN/m3)	K0	E0 (kN/m2)	c0 (kN/m2)	Earth Pressure Coefficients				
						Phi (deg)	Delta/Phi	Beta	Cw/c ratio	Kr
Made Ground	Drained	19	0.5	30000	0	24	0.6	0	0	0.5
Glacial Till	Drained	21	1	48000	0	27	0.6	0	0	0.5
Glacial Till	Undrained	21	1	60000	60	27	0.6	0	0.5	1
Sand and Gravel	Drained	20	1.5	55000	0	27	0.6	0	0	0.5

Note: for the Soil Strength Partial Factor analysis, material properties need to be calculated so that appropriate factors can be applies according to code. Should the user specify user defined material properties, the program allows the user to factor Kp. This is not applicable for this tutorial example.
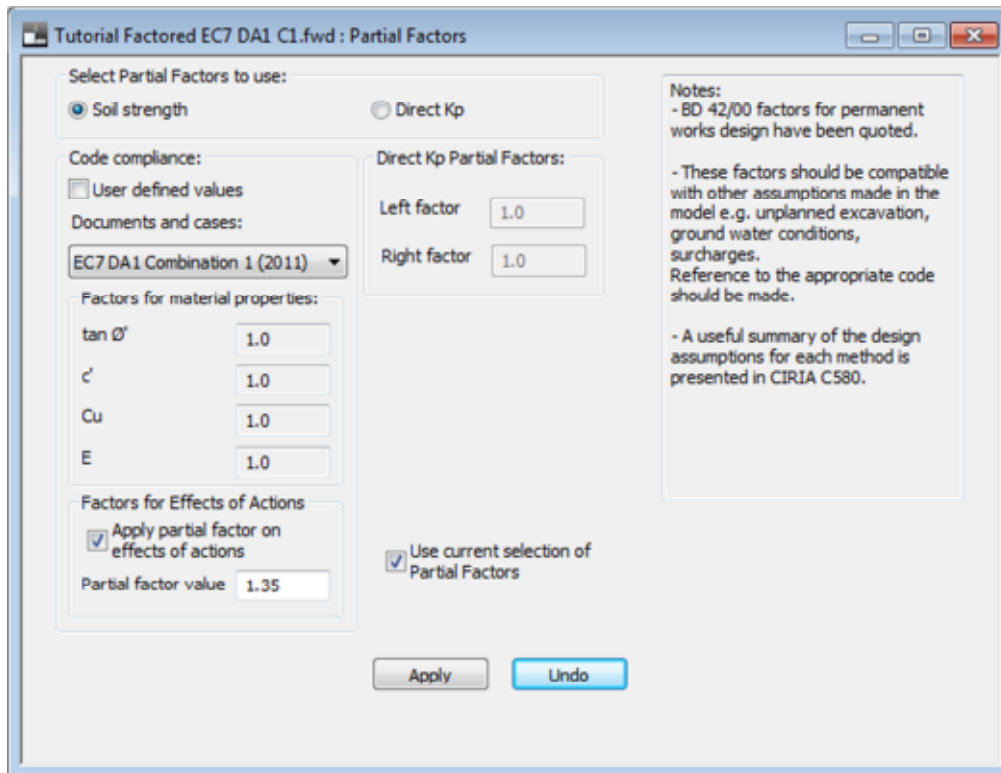
2. Apply the stability analysis and generate the nodes.
   The users toe level should be around 22.5mOD

   (Hint: Oasys has provided the file with generated nodes and the file created by the user should be similar to that provided by Oasys.)

EC7 Design Approach 1, Combination 1

1. Select Partial Factors from the Global section of the Gateway Menu.

Check the *Soil strength* option.
Under *Code compliance*, chose EC7 DA1 Combination 1 (C1) for the Documents and cases.
Check the *Factors for Effects of Actions* as this is required for EC7 DA1 C1.

To apply the partial factors, check the box titled Use current selection of Partial Factors.
Click on the Apply box.

2. Factor surcharges and strut stiffnesses in accordance to Design Approach 1

Σ   Click on the analyse button to carry out the Soil Structure Interaction Analysis.

3. Open the Tabular Output from the Gateway menu.

```
Tutorial Factored EC7 DA1 C1.fwd : Output                    ▢ ▢ ✖

Results Envelope
Node Level Displacements [mm] Moments [kNm/m] Shears [kN/m]
      [m]    Min      Max        Min      Max      Min      Max
   1 30.00 27.08     37.60      0.00     0.00 -131.06    0.00
   2 29.60 25.22     37.54     -4.14    48.28 -105.38   19.99
   3 29.20 23.37     37.42    -15.99    84.31  -73.87   39.85
   4 28.80 21.54     37.19    -36.02   107.38  -51.61   60.54
   5 28.40 19.75     36.82    -64.43   125.60  -43.65   82.20
   6 28.00 18.05     36.28   -163.41   142.30 -141.54  103.28
   7 27.73 16.98     35.82   -128.30   151.68 -123.97   38.14
   8 27.47 15.99     35.27   -102.43   158.84 -106.79   46.89
   9 27.07 14.64     34.29    -82.80   164.72  -87.10   51.44
  10 26.68 13.39     33.10    -83.38   163.85  -68.08   46.74
  11 26.28 12.23     31.69    -99.92   155.12  -51.49   36.42
  12 25.89 11.18     30.09   -108.61   137.42  -36.18   59.56
  13 25.49 10.22     28.32   -105.45   109.64  -19.26   85.52
  14 25.10  9.35     26.40    -94.77    70.67  -27.01  114.29
  15 24.70  8.57     24.40    -84.10    22.69  -40.43  142.50
  16 24.50  8.19     23.37    -80.92    24.93  -15.76   68.03
  17 24.10  7.50     21.29    -80.26    26.47    0.00   91.88
  18 23.90  7.19     20.24    -82.71    24.48   -2.60   73.39
  19 23.70  6.89     19.18    -89.31    21.33  -37.86   27.60
  20 23.33  6.37     17.18    -77.47    13.39  -68.24   21.37
  21 22.95  5.91     15.93    -44.53     5.40 -103.89   17.96
  22 22.58  5.47     15.15      0.00     0.00    0.00    0.00
  23 22.19  5.72     15.13      0.00     0.00    0.00    0.00
  24 21.80  5.81     15.02      0.00     0.00    0.00    0.00
  25 21.40  5.83     14.85      0.00     0.00    0.00    0.00
  26 21.01  5.82     14.67      0.00     0.00    0.00    0.00
  27 20.62  5.79     14.48      0.00     0.00    0.00    0.00
  ..  .. ..  . ..    .. ..      . ..     . ..    . ..    . ..
◄            III                                      ►
```

4. Select Titles from the Gateway Menu and add that the analysis is factored to EC7 DA1 C1.
5. Save the analysis as 'Tutorial Factored EC7 DA1 C1.fwd' so that the user can refer to the results for comparison.

EC7 Design Approach 1, Combination 2

This feature enables a quick comparison to determine the overriding Ultimate Limit State or could be used to compare different codes. The method for comparison is shown below.

Σ˙   Click on the Delete Results icon to delete the EC7 DA1 C1 Partial Factor analysis but keep the nodes and stability analysis results.

1. Select Partial Factors from the Global section of the Gateway Menu.
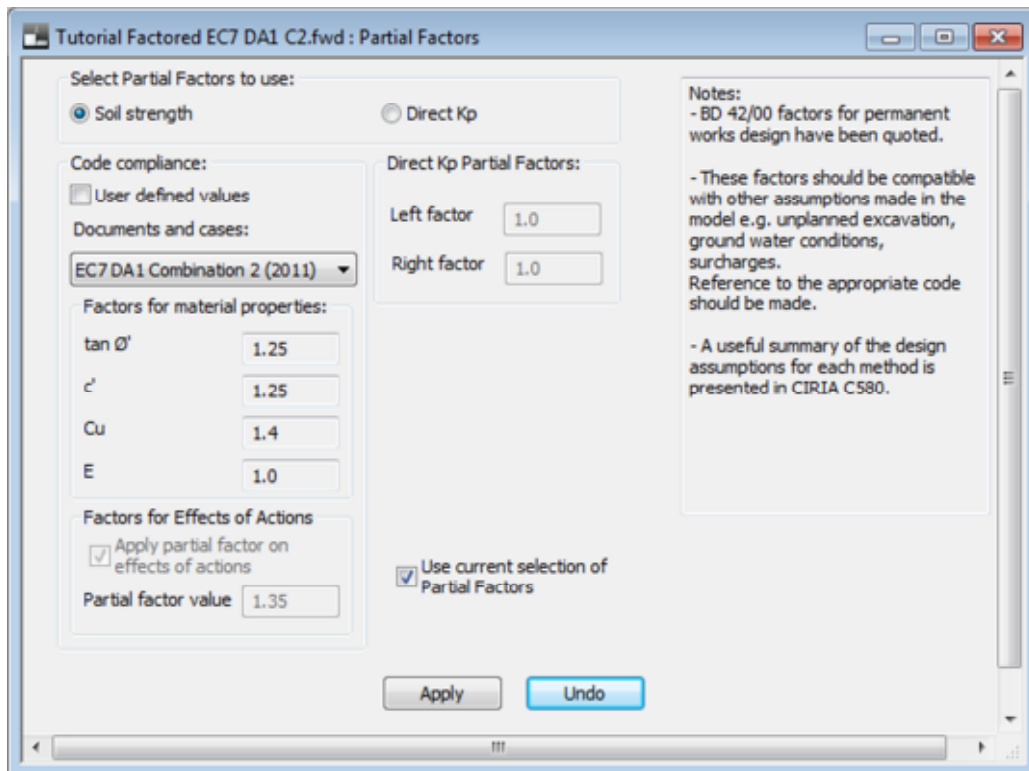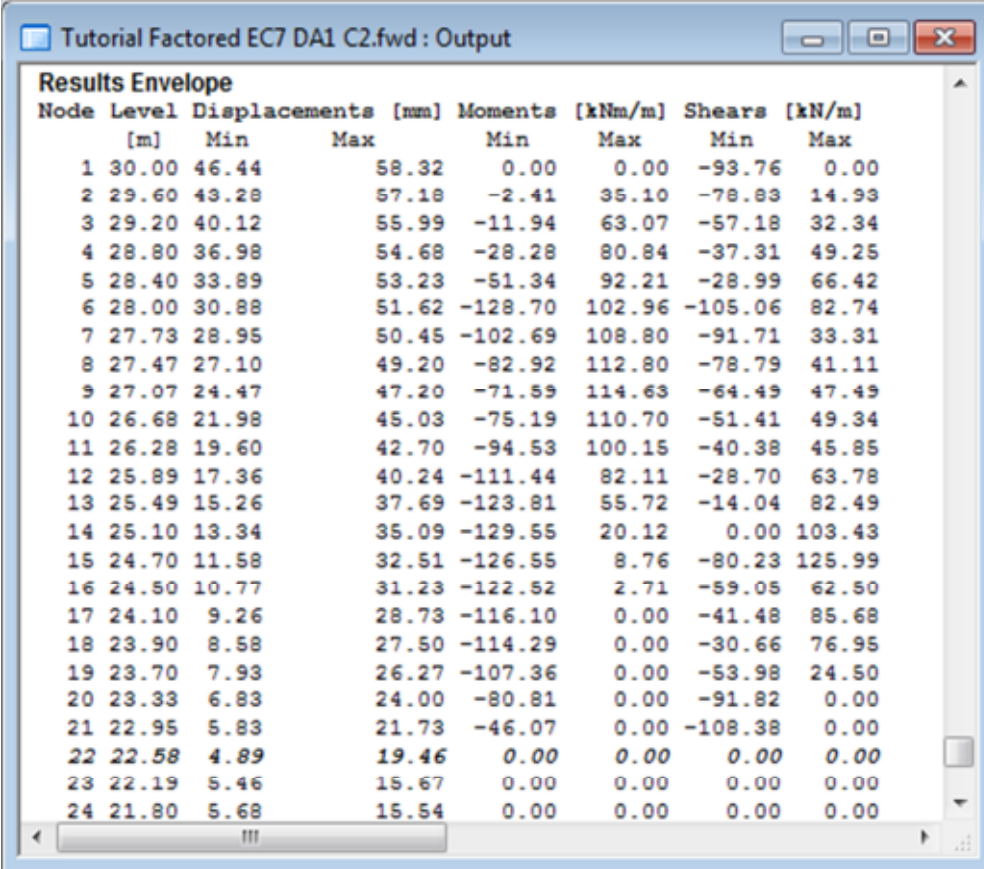

Check the *Soil strength* option.
Under *Code compliance*, chose EC7 DA1 Combination 2 (C2) for the Documents and cases.
To apply the partial factors, check the box titled Use current selection of Partial Factors.
Click on the Apply box.

2. Save this file as 'Tutorial Factored EC7 DA1 C2.fwd'
3. Select Titles from the Gateway Menu and add that the analysis is factored to EC7 DA1 C2.

Σ   Click on the analyse button to carry out the Soil Structure Interaction Analysis.

4. Open the *Tabular Output* from the Gateway menu. Read and compare the maximum bending moment to that in DA1 C1.

```
Tutorial Factored EC7 DA1 C2.fwd : Output                    ▬  ▢  ✖

Results Envelope
Node Level Displacements [mm] Moments [kNm/m] Shears [kN/m]
      [m]    Min      Max      Min      Max      Min      Max
   1 30.00 46.44      58.32     0.00     0.00   -93.76     0.00
   2 29.60 43.28      57.18    -2.41    35.10   -78.83    14.93
   3 29.20 40.12      55.99   -11.94    63.07   -57.18    32.34
   4 28.80 36.98      54.68   -28.28    80.84   -37.31    49.25
   5 28.40 33.89      53.23   -51.34    92.21   -28.99    66.42
   6 28.00 30.88      51.62  -128.70   102.96  -105.06    82.74
   7 27.73 28.95      50.45  -102.69   108.80   -91.71    33.31
   8 27.47 27.10      49.20   -82.92   112.80   -78.79    41.11
   9 27.07 24.47      47.20   -71.59   114.63   -64.49    47.49
  10 26.68 21.98      45.03   -75.19   110.70   -51.41    49.34
  11 26.28 19.60      42.70   -94.53   100.15   -40.38    45.85
  12 25.89 17.36      40.24  -111.44    82.11   -28.70    63.78
  13 25.49 15.26      37.69  -123.81    55.72   -14.04    82.49
  14 25.10 13.34      35.09  -129.55    20.12     0.00   103.43
  15 24.70 11.58      32.51  -126.55     8.76   -80.23   125.99
  16 24.50 10.77      31.23  -122.52     2.71   -59.05    62.50
  17 24.10  9.26      28.73  -116.10     0.00   -41.48    85.68
  18 23.90  8.58      27.50  -114.29     0.00   -30.66    76.95
  19 23.70  7.93      26.27  -107.36     0.00   -53.98    24.50
  20 23.33  6.83      24.00   -80.81     0.00   -91.82     0.00
  21 22.95  5.83      21.73   -46.07     0.00  -108.38     0.00
  22 22.58  4.89      19.46     0.00     0.00     0.00     0.00
  23 22.19  5.46      15.67     0.00     0.00     0.00     0.00
  24 21.80  5.68      15.54     0.00     0.00     0.00     0.00
◄          ⁞⁞⁞                                              ►
```

Note: The designer may wish to use the maximum bending moment to specify wall strength. The maximum displacement from either analysis should also be noted.