

MIDAS Academy UK

Soil-structure interaction

Part 2 : Modelling Tutorial in MIDAS Civil and GTS NX

Pere Alfaras
Principal Bridge Engineer
ARCADIS

Contents

1. Introduction
2. Example #1 – Integral bridge on piled bank seats
3. Example #2 – Integral bridge with full height abutments supported on a single row of piles

Case Study: Fundamentals of Soil Structure Interaction Analysis for Integral Bridges

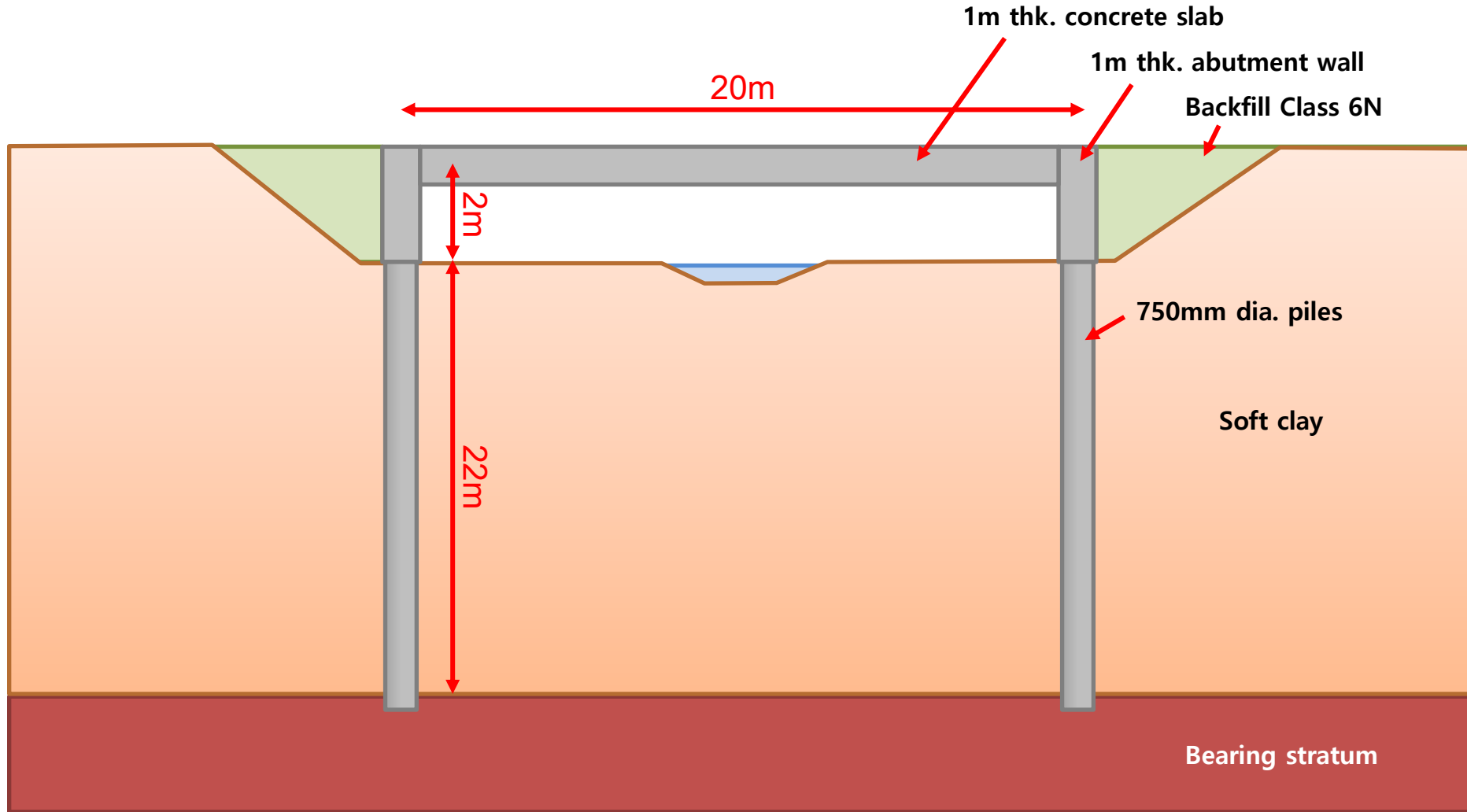
Pere Alfaras
Arcadis, UK

Main outcomes

1. Limit equilibrium method
2. Spring analogy: method of subgrade reaction, p-y curves
3. Continuum media approach
4. Requirements of the PD 6694-1

Example #1

■ General Arrangement



Example #2

■ Methods of analysis (according to PD 6694-1)

Limit equilibrium methods

- Depth of soil affected by abutment movement **can be identified without soil-structure interaction analysis**
- thermal movement ≤ 40 mm;
- skew $\leq 30^\circ$

PD 6694-1:2011 9.4.4

- Horizontal soil pressures on end screen and abutments that accommodate thermal movements by translation without rotation

Soil-structure interaction

- Depth of soil affected by abutment movement **cannot be easily identified** unless a soil-structure interaction analysis is performed
- thermal movement > 40 mm;
- skew $> 30^\circ$
- over-consolidated backfill material
- layered soils

Example #1

■ Earth pressure

Backfill

→ Class 6N (MCHW specs)

Material properties

→ $\gamma = 20 \text{ kN/m}^3$

→ $\phi = 35^\circ$

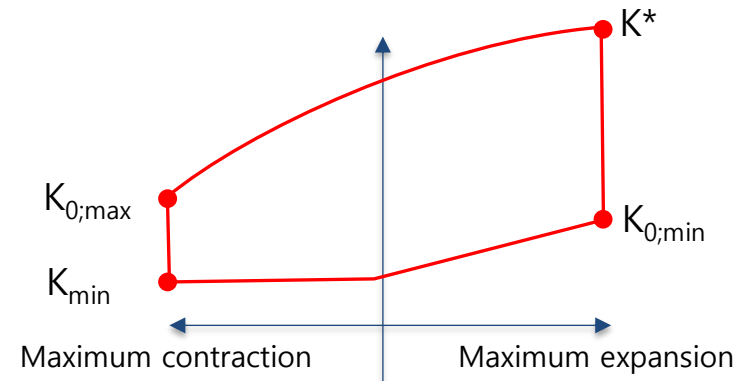
Lateral earth coefficients

→ $K^* = ?$

→ $K_{0;\max} = 0.43$

→ $K_{\min} = 0.15$

→ $K_{0;\min} = 0.29$



Example #1

Model

The screenshot displays the MIDAS/Civil software interface. The main window shows a 3D model of a bridge structure. The model is composed of a deck, abutments, and piles. Red arrows point to specific parts of the model with text boxes:

- Deck modelled as shell elements
- Abutment modelled as shell elements
- Piles modelled as beam elements
- Pile tips assumed fully fixed

The software interface includes a menu bar (View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, PSC, Pushover, Design, Rating, Query, Tools) and a toolbar. The tree menu on the left shows the project structure:

- Works
 - Structures
 - Nodes : 641
 - Elements : 640
 - Properties
 - Material : 1
 - 1 : C40/50
 - Section : 2
 - 1 : Pile750
 - 2 : Edge beam
 - Thickness : 1
 - 1 : 1
 - Boundaries
 - Supports : 12
 - Type 1 [1110000]
 - Static Loads
 - Static Load Case 1 [T+ :]
 - Static Load Case 2 [T- :]
 - Static Load Case 3 [K' :]
 - Static Load Case 4 [K0,max :]
 - Static Load Case 5 [K0,min :]
 - Static Load Case 6 [Kmin :]

The message window at the bottom displays the following text:

```
-----SOLUTION TERMINATED-----
YOUR MIDAS JOB IS SUCCESSFULLY COMPLETED.....C:\Users\paa76924\Desktop\MIDAS\SSI\SSI_INITIAL
TOTAL SOLUTION TIME... 3.37 [SEC]
```

Example #1

- Pile springs – user-defined p-y curves (not used on this example)

Click 'Point Spring'.

Click 'Boundary' tab.

Click 'Add'

Select 'Forces-Deformation Function'

Type name

Click 'Symmetric'

Enter values

OK

	d(x) (m)	F(y) (kN)
1	0.00000	0.000000
2	0.00100	800.0000
3	0.01000	1000.0000
4	0.02000	1000.0000
5		

Graph of F(y) vs d(x) showing a p-y curve. The curve starts at (0, -1100), rises to (0, 1100), and then levels off at F(y) = 1000 kN for d(x) > 0.01 m.

Example #1

- Pile springs – user-defined p-y curves (not used on this example)

The screenshot displays the MIDAS/Civil software interface with the following steps highlighted by red boxes and numbers:

1. Click 'Boundary' tab.
2. Click 'Point Spring'.
3. Select 'Point spring'.
4. Select 'Multi-linear'.
5. Click 'Symmetric'.
6. Choose direction.
7. Apply.

The interface includes a Tree Menu on the left showing the project structure, a main workspace, and a right-hand panel for defining the boundary condition. The 'Boundary' tab is active, and the 'Point Spring Supports' dialog is open, showing options for 'Boundary Group Name', 'Options' (Add, Replace, Delete), 'Point Spring (Local Direction)' (Type: Multi-Linear), 'Forces-Deformation Function' (p-y (z=2m)), and 'Direction' (Dx(+)). The 'Apply' button is highlighted.

Example #1

Pile springs – built-in p-y curves (not used on this example)

2 Click 'Boundary' tab.

3 Click 'Integral bridge'.

1 Double Click 'Pile750' to select all piles.

Note.
Suitability of the built-in p-y curves should be confirmed in a case-by-case basis.

MIDAS/Civil

Message Window

-----SOLUTION TERMINATED-----
YOUR MIDAS JOB IS SUCCESSFULLY COMPLETED.....C:\Users\paa76924\Desktop\MIDAS\SSI\SSI_INITIAL
TOTAL SOLUTION TIME...: 3.37 [SEC]

4 Select 'Pile spring'

5 Define soil parameters

6 Apply

Soil Spring Type

☐ Abutment Spring

☒ Pile Spring

Reference Figure

Deck

Girder

Pile

Pile Spring Data

Soil Type: Soft Clay

Ground Level: 0 m

Pile Diameter (D): 0.75 m

Unit Weight of Soil (r): 19 kN/m³

Earth Pressure Coeff. at rest (K₀): 0.4

Coeff. of Subgrade Reaction (K_h): 10000 kN/m³

1/2 Strain at Max. Stress Point in Tri. Comp. Test (e₅₀): 0.01

Undrained Cohesion (c_u): 5 kN/m²



Example #1

Pile springs – Method of subgrade reaction

1 Double Click 'Pile750' to select all piles.

2 Click 'Boundary' tab.

3 Click 'Surface spring'.

4 Distributed Spring

5 Type: Frame

6 Local Axis: Local z

7 Width: .75 m

8 Modulus of subgrade reaction 10000 kN/m³

9 Apply

Note. We will be using the method of subgrade reaction (linear springs)

Example #1

■ Pile springs – Method of subgrade reaction

The screenshot displays the MIDAS/Civil software interface. The main window shows a 3D model of a bridge deck supported by pile springs. The pile springs are represented by dashed lines connecting the deck to the ground. A red box highlights these dashed lines, and a text label "Surface springs are shown as dashed lines" with an arrow points to them.

The left sidebar contains the Tree Menu with tabs for Node, Element, Boundary, Mass, and Load. The Boundary tab is selected, showing the Surface Spring Supports section. The Boundary Group Name is set to Default. The Surface Spring section has the Distributed Spring option selected. The Element Selection section shows Type: Frame, Selection: Element, Local Axis: Local z, and Width: .75 m. The Spring Properties section shows Type: Linear and Modulus of Subgrade Reaction: 10000 kN/m³.

The Message Window at the bottom displays the following text:

```
-----SOLUTION TERMINATED-----
YOUR MIDAS JOB IS SUCCESSFULLY COMPLETED.....C:\Users\paa76924\Desktop\MIDAS\SSI\SSI_INITIAL
TOTAL SOLUTION TIME...: 3.37 [SEC]
```

Example #1

Temperature loads

The screenshot displays the MIDAS/Civil software interface. The **Boundary** tab is active in the top ribbon. The left **Tree Menu** shows a project structure with various elements like Nodes, Elements, Properties, and Static Loads. The main workspace shows a bridge model with two vertical red lines indicating the front view. Two red annotations are present:

- 1** Select front view: A red box with an arrow pointing to the front view selection icon in the top right corner.
- 2** Select all deck elements: A red box with an arrow pointing to the deck elements in the main workspace.

The bottom status bar shows the current view is **Plate-317** at **U: -10, 1, -3.5** and **G: -10, 1, -3.5**. The **Message Window** at the bottom indicates: "The project will be saved by the auto-save feature."

Example #1

Temperature loads

1 Click 'Load' tab.

2 Click 'Temp./Prestress'

3 Click 'Element Temp.'

The screenshot displays the MIDAS/Civil software interface. The top menu bar includes 'View', 'Structure', 'Node/Element', 'Properties', 'Boundary', 'Load', 'Analysis', 'Results', 'PSC', 'Pushover', 'Design', 'Rating', 'Query', and 'Tools'. The 'Load' tab is selected. Below the menu bar, the 'Load' sub-menu is open, showing options like 'Static Load', 'Using Load', 'Element Temp.', 'Temp. gradient', 'Beam Section Temp.', 'System Temp.', 'Nodal Temp.', 'Tendon Property', 'Tendon Profile', 'Tendon Prestress', 'External Type Loadcase', 'Prestress Beam Loads', 'Pretension Loads', and 'Prestress Loads'. The 'Temp./Prestress' option is selected. The 'Temp./Prestress' sub-menu is also open, showing 'Element Temp.', 'Temp. gradient', and 'Beam Section Temp.'. The 'Element Temp.' option is selected. The main workspace shows a 3D model of a structure with a red line indicating the temperature load application. The left pane shows the project tree with 'Static Load Case 1 [T+ :]' selected. The bottom pane shows the 'Message Window' with the text 'The project will be saved by the auto-save feature.'

Example #1

■ Temperature loads

The screenshot shows the 'Tree Menu' dialog box with the 'Load' tab selected. The 'Element Temperatures' dropdown is open. The 'Load Case Name' is set to 'T+'. The 'Load Group Name' is 'Default'. The 'Options' section has 'Add' selected. The 'Temperature' section shows 'Initial' as 0 and 'Final Temperature' as 30 [C]. The 'Apply' button is highlighted.

1 Select load case 'T+'
2 Assign +30 °C
3 Apply

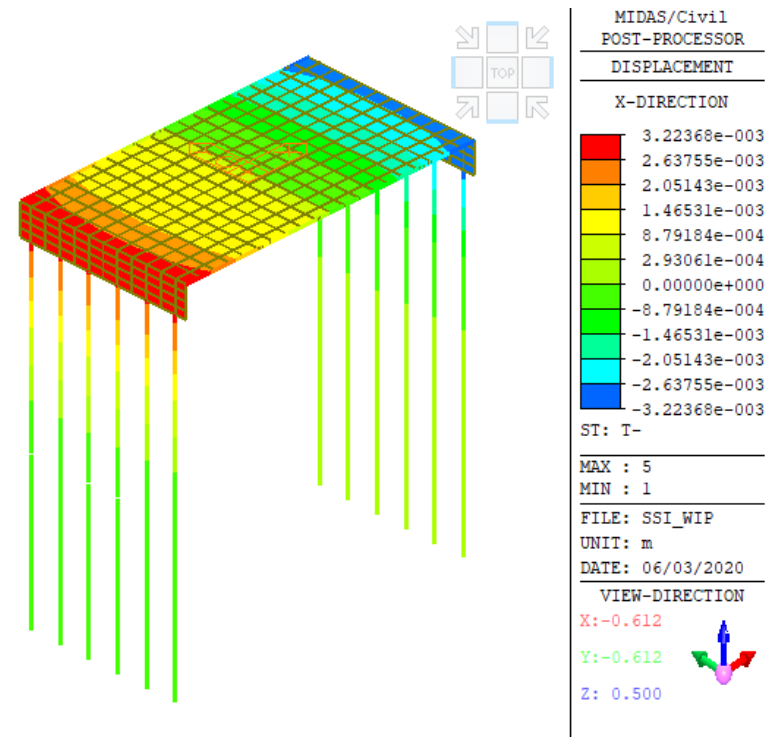
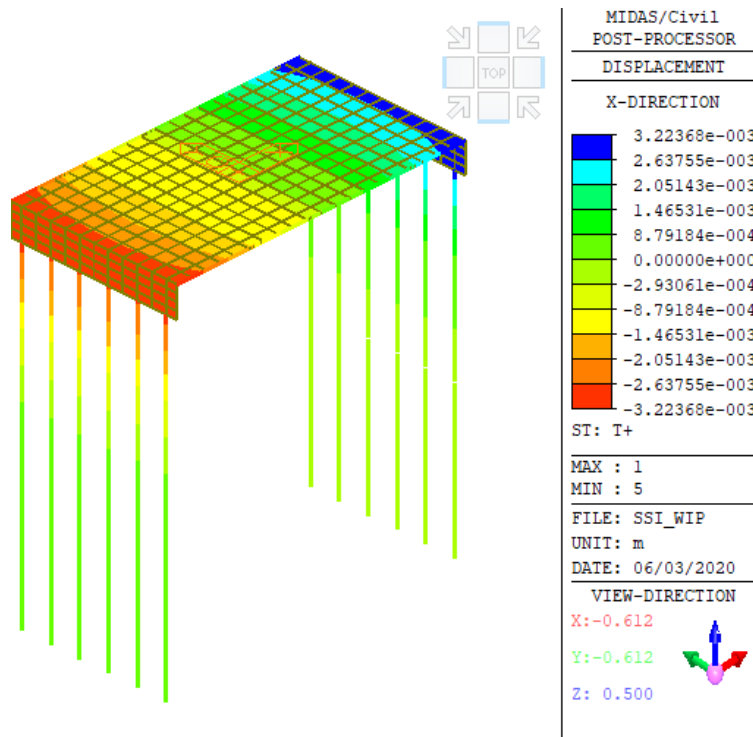
4 Select deck elements again

The screenshot shows the 'Tree Menu' dialog box with the 'Load' tab selected. The 'Element Temperatures' dropdown is open. The 'Load Case Name' is set to 'T-'. The 'Load Group Name' is 'Default'. The 'Options' section has 'Add' selected. The 'Temperature' section shows 'Initial' as 0 and 'Final Temperature' as -30 [C]. The 'Apply' button is highlighted.

5 Select load case 'T-'
6 Assign -30 °C
7 Apply

Example #1

■ Earth pressure



Thermal movement range – characteristic value: $d_k = 6 \text{ mm}$

Example #1

■ Earth pressure

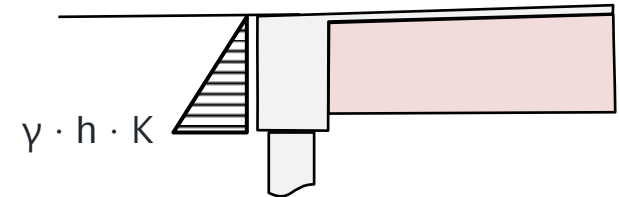
PD 6694-1:2011 9.4.3

$$K^* = K_0 + \left(\frac{40d'_d}{H} \right)^{0.4} K_{p,t}$$

$$d_d = \frac{1}{2} d_k (1 + \Psi \gamma_Q)$$

φ	K_0	d_k mm	Ψ	γ_Q	d_d mm	H mm	K_{pt}	K^*
35.0	0.43	6.0	1	1.55	7.7	2500	5.88	3.0

		γ [kN/m ³]	h [m]	q [kN/m ²]
K^*	3.00	20	2	120
$K_{0\max}$	0.43	20	2	17.2
$K_{0\min}$	0.29	20	2	11.6
$K_{\min}$	0.15	20	2	6



Example #1

Earth pressure

The screenshot displays the MIDAS/Civil software interface. The 'Load' menu is open, showing various load types. A red box labeled '2' points to the 'Hydrostatic Pressure' option under the 'Pressure Loads' sub-menu. A diagram of a retaining wall is shown in the main workspace, with a red box labeled '1' pointing to the vertical line representing the abutment. The 'Tree Menu' on the left shows the 'Load' type selected. The 'Message Window' at the bottom displays the following text:

```
-----SOLUTION TERMINATED
YOUR MIDAS JOB IS SUCCESSFULLY COMPLETED.....C:\Users\paa76924\Desktop\MIDAS\SSI\SSI_WIP
TOTAL SOLUTION TIME... 1.16 [SEC]
-----
```

Example #1

Earth pressure

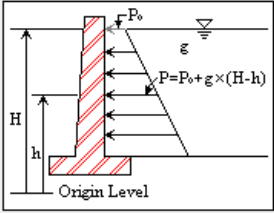
Hydrostatic Pressure Loads

1 Load Case Name: **K*** Select load case 'K*'

Load Group Name: Default

Options: ☒ Add ☐ Replace ☐ Delete

Load Type: Linear Loads



Element Types: Plate

2 Selection: ☐ Node ☒ Element Select 'Local z'

Direction: Local z

Projection: ☐ Yes ☒ No

Loads

Gradient Direction: Global (-Z)

Reference Level(h): 0 m

3 Constant Intensity: 0 kN/m² Assign -60 kN/m³

Gradient Intensity(g): -60 kN/m³

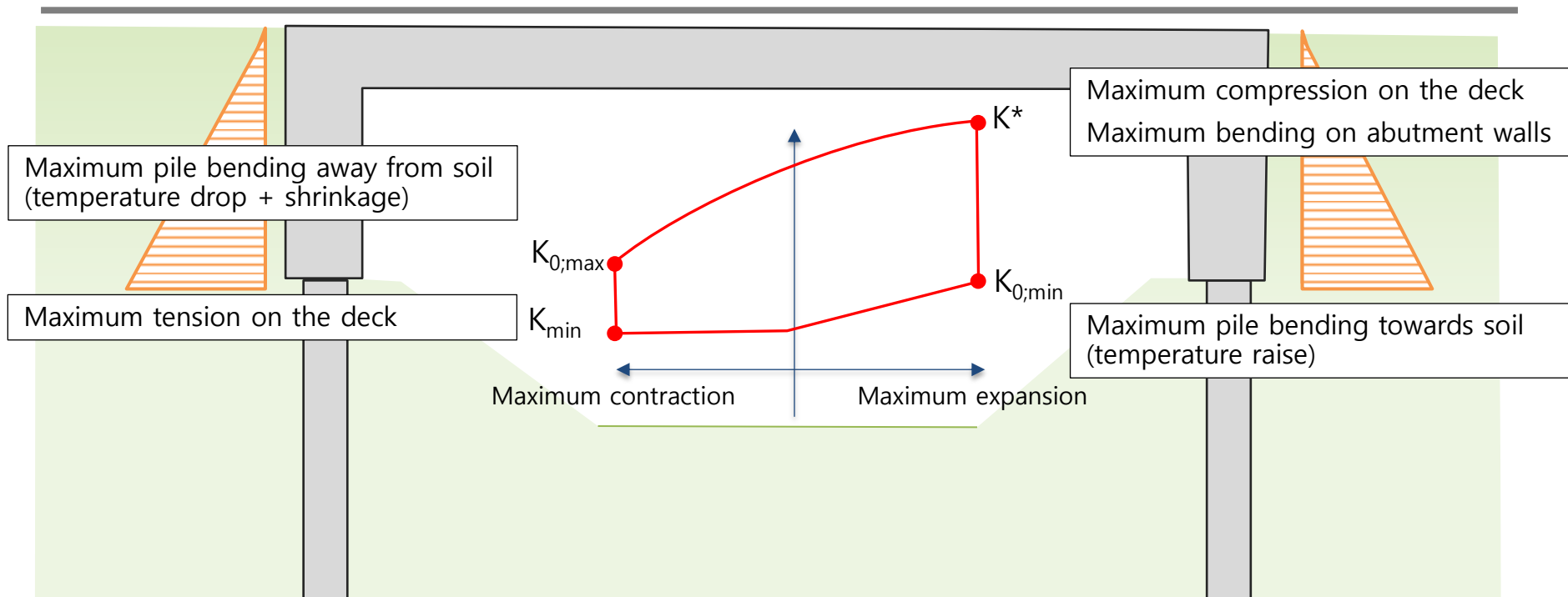
Apply Close

4

Repeat for load cases:
K0max: -8.6 kN/m³
K0,min: -5.8 kN/m³
Kmin: -3.0 kN/m³

Example #1

■ Envelopes



Example #1

Envelopes

1 Click 'Results' tab

2 Click 'Load Combination'

3 Create 4 combinations:
K* + T+
K0min + T+
K0max + T-
Kmin + T-

The screenshot shows the MIDAS software interface. The 'Results' tab is selected in the main menu. The 'Load Combination' button is highlighted in the 'Results' tab. The 'Load Combinations' dialog box is open, showing the 'Load Combination List' table. The table has columns: No, Name, Active, Type, and Description. It lists 4 combinations: 1. Exp_K* (Active, Add), 2. Exp_K0 (Active, Add), 3. Con_K0 (Active, Add), and 4. Con_Km (Active, Add). The 'Load Cases and Factors' table shows two load cases: T+(ST) and K*(ST), both with a factor of 1.0000. The 'Tree Menu' on the left shows the project structure. The 'Task Pane' at the bottom shows the file name: C:\Users\paa76924\Desktop\MIDAS\SSI\SSI_WIP.lcp.

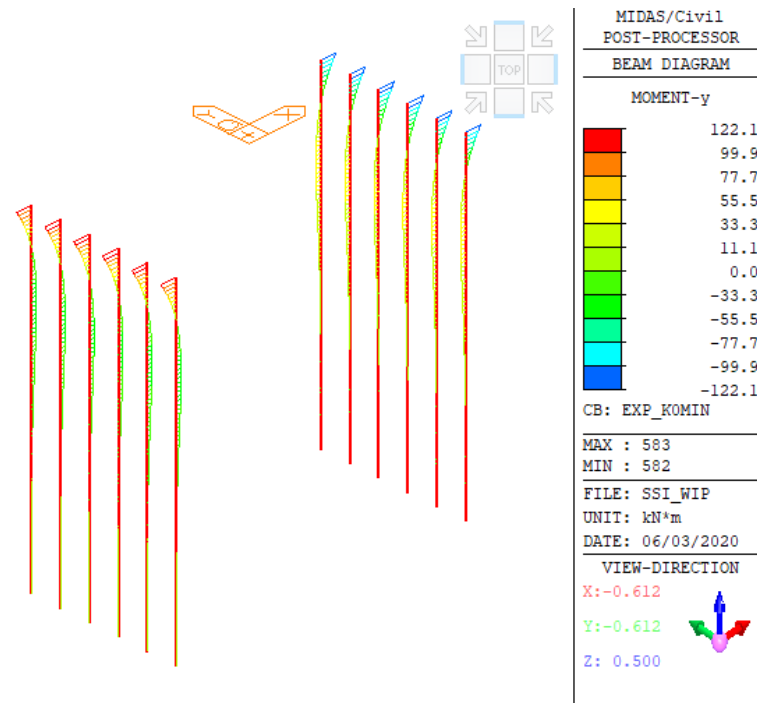
No	Name	Active	Type	Description
1	Exp_K*	Active	Add	
2	Exp_K0	Active	Add	
3	Con_K0	Active	Add	
4	Con_Km	Active	Add	

LoadCase	Factor
T+(ST)	1.0000
K*(ST)	1.0000

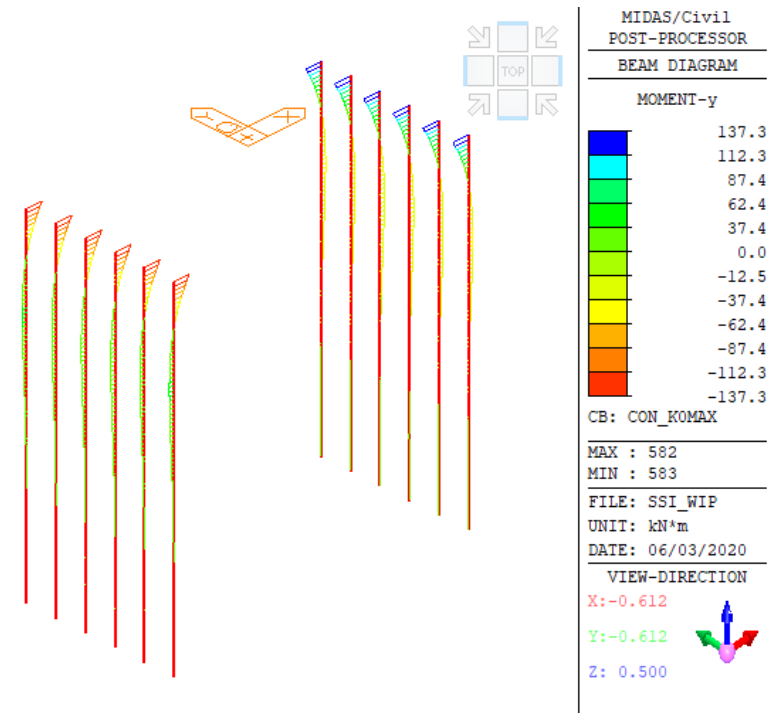
Example #1

■ Envelopes

$K_{0,min}$ + Expansion
maximum moment in piles in expansion

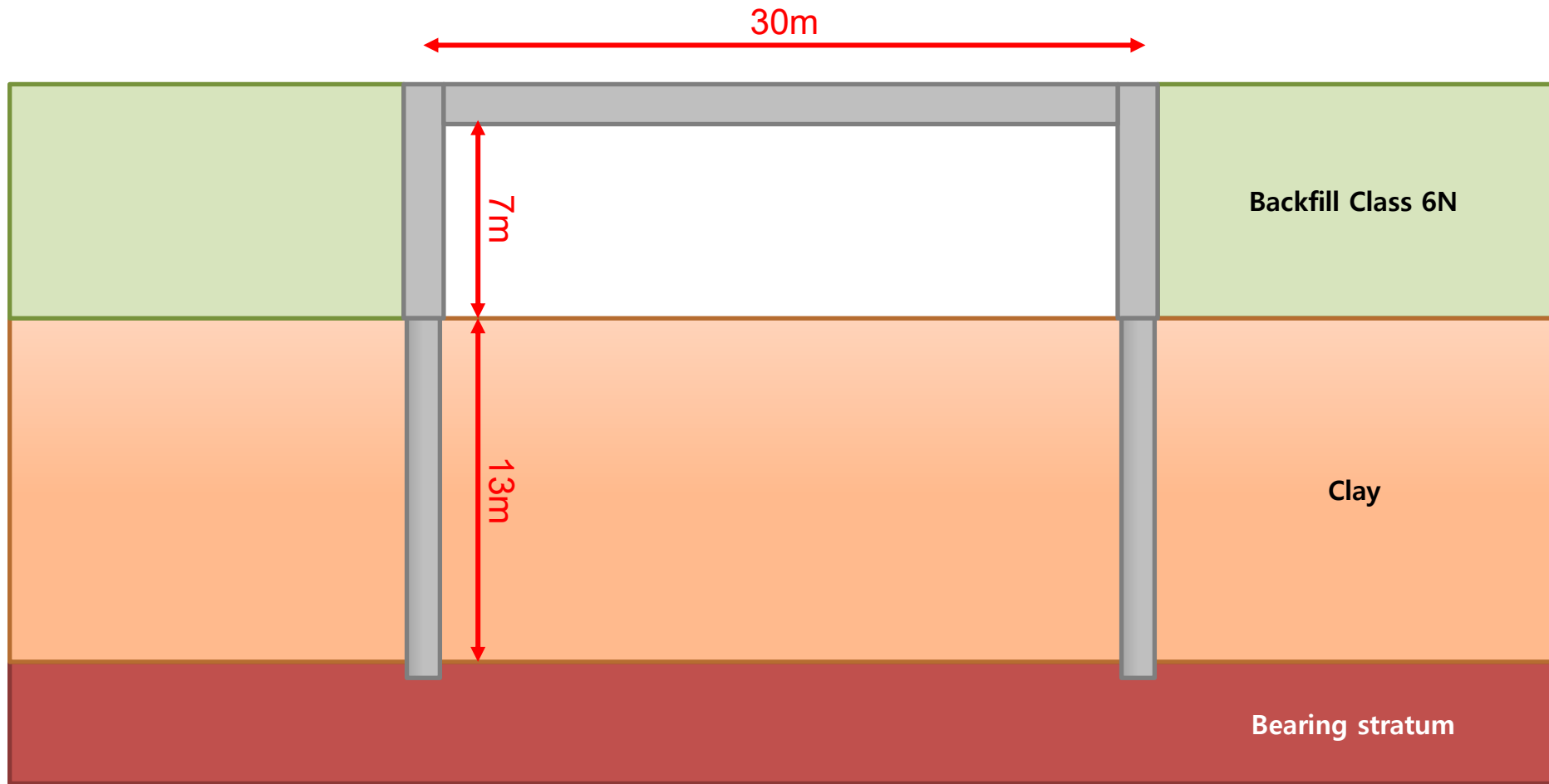


$K_{0,max}$ + Contraction
maximum moment in piles in contraction



Example #2

■ General arrangement



Example #2

■ Methods of analysis (according to PD 6694-1)

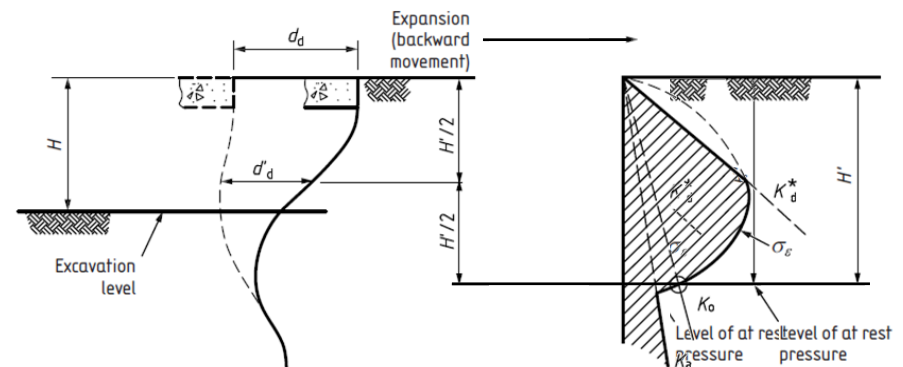
Limit equilibrium methods

- Depth of soil affected by abutment movement **can be identified** without soil-structure interaction analysis
- thermal movement ≤ 40 mm;
- skew $\leq 30^\circ$

Soil-structure interaction

- Depth of soil affected by abutment movement **cannot be easily identified** unless a soil-structure interaction analysis is performed
- thermal movement > 40 mm;
- skew $> 30^\circ$
- over-consolidated backfill material
- layered soils

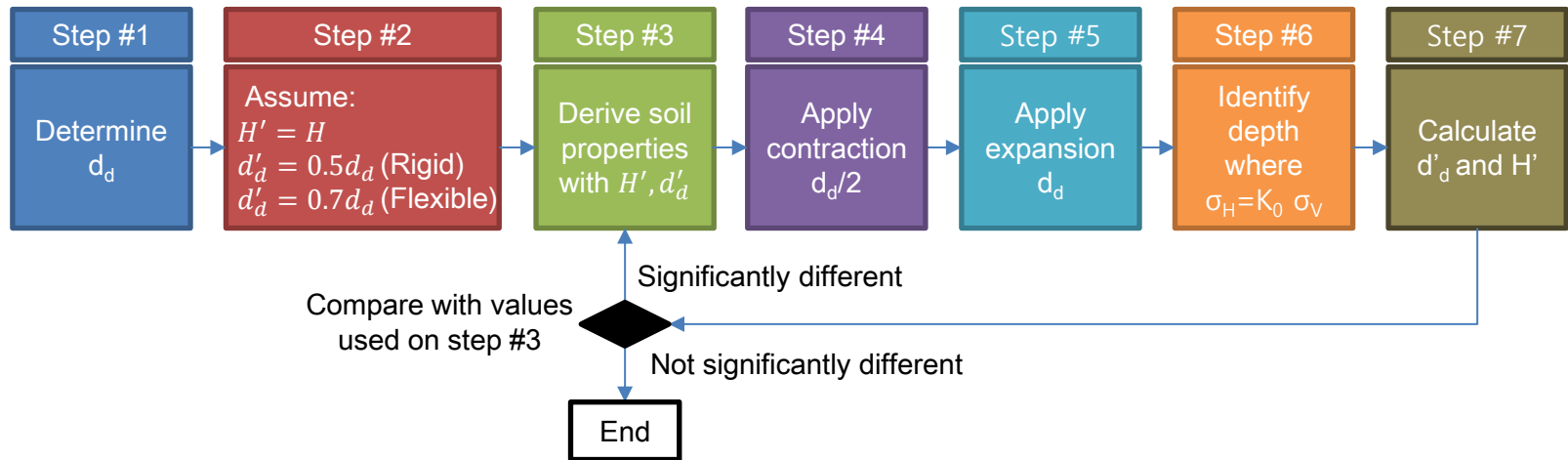
PD 6694-1:2011 9.4.5 Horizontal earth pressures on full height frame abutments on piles and embedded wall abutments (soil- structure interaction analysis)



Example #2

■ Material properties (PD 6694-1:2011 Annex A)

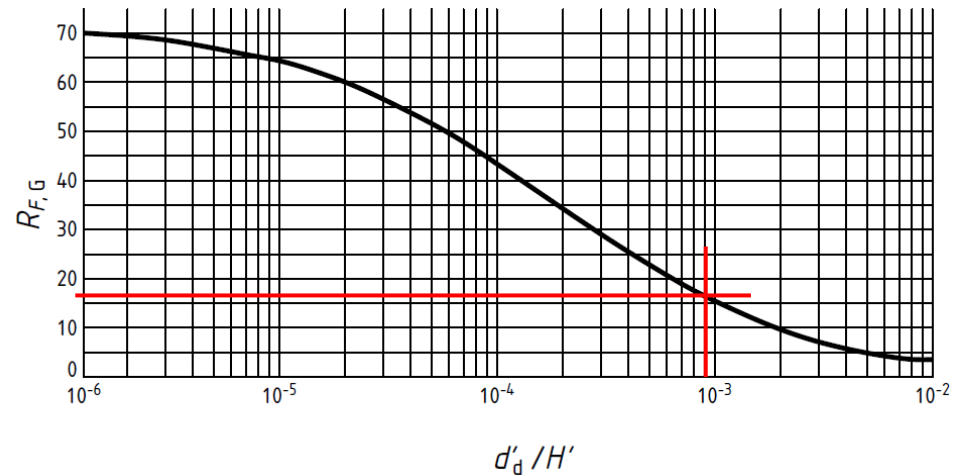
PD 6694-1:2011 Annex A



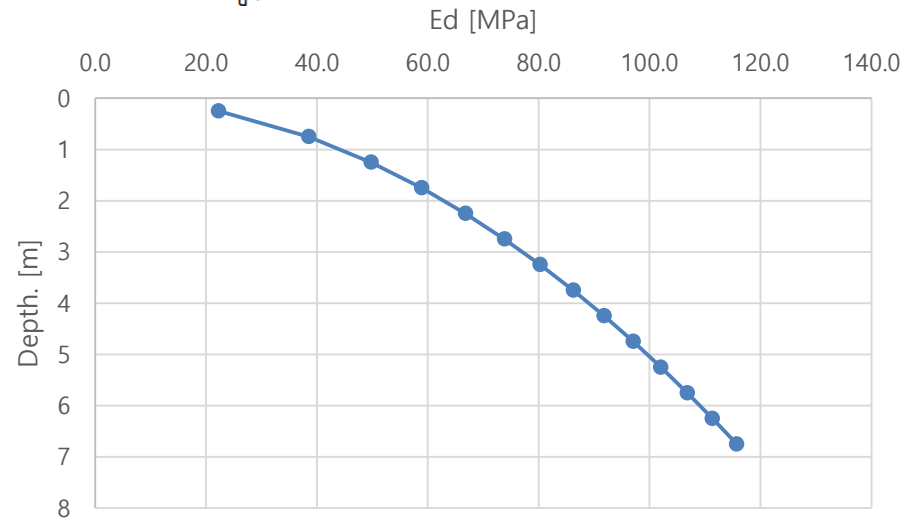
Example #2

Material properties (PD 6694-1:2011 Annex A)

φ	35
K_0	0.43
γ	20 kN/m ³
ν	0.2
d'	6.3 mm
H'	7 m
d'/H'	9.00E-04
$R_{F,G}$	16

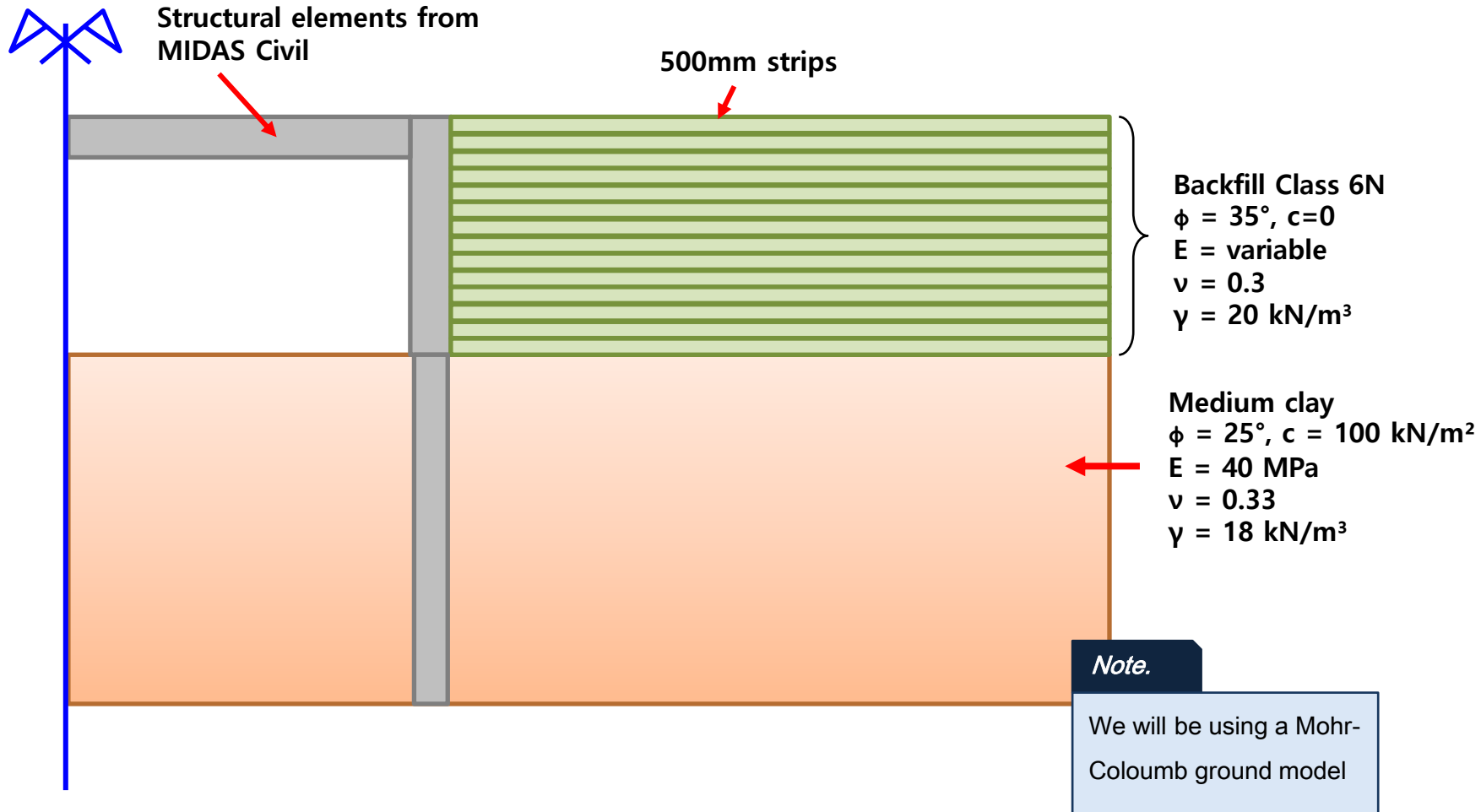


	z m	σ_v kN/m ²	σ_m kN/m ²	G kN/m ²	E kN/m ²	Ed kN/m ²	Ed MPa
1	0.25	5	3.088079	6185.671	14845.6	22268.42	22.3
2	0.75	15	9.264236	10713.9	25713.4	38570.03	38.6
3	1.25	25	15.44039	13831.58	33195.8	49793.69	49.8
4	1.75	35	21.61655	16365.75	39277.8	58916.69	58.9
5	2.25	45	27.79271	18557.01	44536.8	66805.25	66.8
6	2.75	55	33.96886	20515.55	49237.3	73855.98	73.9
7	3.25	65	40.14502	22302.75	53526.6	80289.92	80.3
8	3.75	75	46.32118	23957	57496.8	86245.21	86.2
9	4.25	85	52.49734	25504.18	61210.0	91815.03	91.8
10	4.75	95	58.67349	26962.72	64710.5	97065.78	97.1
11	5.25	105	64.84965	28346.31	68031.1	102046.7	102.0
12	5.75	115	71.02581	29665.44	71197.0	106795.6	106.8
13	6.25	125	77.20196	30928.36	74228.1	111342.1	111.3
14	6.75	135	83.37812	32141.69	77140.1	115710.1	115.7



Example #2

■ General arrangement



Example #2

General arrangement

The screenshot shows the MIDAS/Civil software interface. The 'Structure' tab is selected in the top menu bar. The 'Structure Type' dialog box is open, showing the 'X-Y Plane' option selected under 'Structure Type'. The 'Mass Control Parameter' section shows 'Lumped Mass' selected. The 'Gravity Acceleration' is set to 9.806 m/sec² and 'Initial Temperature' is set to 0 [C]. The 'Align Top of Beam Section with Center Line (X-Y Plane) for Display' and 'Align Top of Slab(Plate) Section with Center Line (X-Y Plane) for Display' options are unchecked. The 'OK' button is visible at the bottom right of the dialog box.

1 Select 'Structure' tab

2 Select 'Structure type'

3 Select 'X-Y Plane'

Note.

This is a simplified example in 2D, but same approach is applicable to complex 3D problems

Example #2

Model

The screenshot displays the MIDAS/Civil software interface. The 'Node/Element' tab is selected in the top menu bar. The 'Create Nodes' dialog box is open on the left side of the screen. The dialog box contains the following fields and options:

- Start Node Number:** Set to 1.
- Coordinates (x,y,z):** Set to 0,0,0.
- Copy:** Number of Times: 0.
- Distances (dx,dy,dz):** Set to 0, 0, 0.
- Options:** ☒ Merge Duplicate Nodes, ☒ Intersect Frame Elements.
- Buttons:** Apply, Close.

Numbered annotations (1-5) point to the following elements:

- 1: Select 'Node/Element' tab
- 2: Select 'Create Nodes'
- 3: Input coordinates 0,0,0
- 4: Input distances 0,0,0
- 5: Apply

The bottom status bar shows the following information:

- Tree Menu Task Pane
- For Help, press F1
- None!
- U: 15, 0, -9.75
- G: 15, 0, -9.75
- kN
- m
- non
- 1
- 2

Example #2

Model

The screenshot displays the MIDAS/Civil software interface with the following components and annotations:

- Tree Menu:** Located on the left, showing a hierarchy with 'Nodes : 1'.
- Message Window:** At the bottom left, displaying the message: "The project will be saved by the auto-save feature."
- Command Message:** At the bottom center, showing coordinates: "U: 0, 0, 0" and "G: 0, 0, 0".
- Toolbar:** The top toolbar contains various modeling tools. The 'Extrude' tool is highlighted with a red box and labeled with a red circle '2' and the text "Select 'Extrude'".
- Model View:** The central workspace shows a single node represented by a square with an 'X'. It is labeled with a red circle '1' and the text "Select Node".
- Tree Menu (Right):** A secondary 'Tree Menu' window is open on the right, showing the 'Extrude Elements' dialog. It is labeled with a red circle '3' and the text "Select 'Node -> Line Element'". The dialog includes fields for 'Node Number' (2) and 'Element Number' (1).
- Extrude Type:** The 'Node -> Line Element' option is selected in the 'Extrude Type' section, labeled with a red circle '4' and the text "Input 15,0,0".
- Translation:** The 'Equal Distance' option is selected under the 'Translation' section.
- dx,dy,dz:** The input field for 'dx,dy,dz' contains the value "15, 0, 0" m, labeled with a red circle '5'.
- Buttons:** The 'Apply' button is highlighted with a red box at the bottom of the dialog.

Example #2

The image shows the MIDAS Gen software interface with the 'Extrude Elements' dialog box open. The dialog box has several tabs: 'Node', 'Element', 'Boundary', 'Mass', and 'Load'. The 'Node' tab is selected. The 'Extrude Elements' section shows 'Start Number' as 2 and 'Element Number' as 1. The 'Extrude Type' is set to 'Node -> Line Element'. The 'Element Attribute' section shows 'Element Type' as 'Beam', 'Material' as 1, 'Section' as 1, and 'Beta Angle' as 0. The 'Generation Type' is set to 'Translate', and 'Translation' is set to 'Equal Distance'. The 'dx,dy,dz' values are 0, -0.5, 0, and the 'Number of Times' is 40. The 'Merging Tolerance' is set to 0. The 'Apply' button is highlighted. The main workspace shows a 3D model of a structure with a node selected, indicated by a red arrow and the label 'Select Node'. The 'Extrude' button in the top toolbar is also highlighted with a red arrow and the label 'Select 'Extrude''. The bottom status bar shows 'Command Message' and 'Analysis Message' tabs, and the 'For Help, press F1' message.

Model

Tree Menu

Node Element Boundary Mass Load

Extrude Elements

Start Number

Node Number : 2

Element Number : 1

Extrude Type

Node -> Line Element

Source ☒ Remove ☐ Move

☐ Reverse I-J

Element Attribute

Element Type: Beam

Material : 1

Section : 1

Beta Angle : 0 [Deg]

Generation Type

☒ Translate ☐ Rotate ☐ Project

Translation

☒ Equal Distance ☐ Unequal Distance

dx,dy,dz: 0, -0.5, 0 m

Number of Times : 40

Merging Tolerance

Apply Close

2 Select 'Extrude'

3 Select 'Node -> Line Element'

1 Select Node

4 Input 0,-0.5,0

Input 40

6

Tree Menu Task Pane

Command Message Analysis Message

For Help, press F1

None! U: 0, 0, 0 G: 0, 0, 0 kN m none ? 0 / 2

Example #2

Midas Civil model

1500mm thk. deck

1000mm thk. abutment

750mm dia. piles

All elements C40/50

Note.
We are modelling a 1m strip of half of the structure along plane of symmetry.

Example #2

Midas Civil model

The screenshot displays the Midas Civil software interface. The 'File' menu is open, and the 'Export' option is highlighted. The 'Export' submenu is also open, showing various export options. The 'MXT File (for FEA/GTS)...' option is selected. A 'Save As' dialog box is open, showing the file name 'SSI_ContiguousFile.mxt' and the file type 'MXT Files (*.mxt)'. The 'Save' button is highlighted. The 'Save in' dropdown is set to 'SSI Contiguous File'. The 'File name' field is 'SSI_ContiguousFile.mxt'. The 'Save as type' is 'MXT Files (*.mxt)'. The 'Display comments' checkbox is checked, and the 'Save the Loads' checkbox is unchecked. The 'Save' button is highlighted.

1. Click on the 'File' menu icon.

2. Click on the 'Export' option in the 'File' menu.

3. Click on the 'MXT File (for FEA/GTS)...' option in the 'Export' submenu.

4. Click on the 'Save' button in the 'Save As' dialog box.

Note.
Most MIDAS Civil definitions can be exported using *.mxt files.

Example #2

Exchange file *.mxt

```
00001 ;-----
00002 ; MIDAS/Civil Text (MCT) File.
00003 ; Date : 2020/6/5
00004 ;-----
00005
00006 *VERSION
00007 8.1.5
00008
00009 *UNIT ; Unit System
00010 ; FORCE, LENGTH, HEAT, TEMPER
00011 KN , M, J, C
00012
00013 *STRUCTYPE ; Structure Type
00014 ; ISTYP, iMASS, iSMAS, bMASSOFFSET, bSELFWEIGHT, GRAV, TEMPER, bALIGNBEAM, bALIGNSLAB, bROTIRIGID
00015 3, 1, 1, NO, NO, 9.806, 0, NO, NO, NO
00016
00017 *REBAR-MATL-CODE ; Rebar Material Code
00018 ; CONC_CODE, CONC_MDB, SRC_CODE, SRC_MDE
00019 ASTM(RC), Grade 60, ASTM(RC), Grade 60
00020
00021 *NODE ; Nodes
00022 ; iNO, X, Y, Z
00023 1, 0, 0, 0
00024 2, 15, 0, 0
00025 3, 15, -0.5, 0
00026 4, 15, -1, 0
00027 5, 15, -1.5, 0
00028 6, 15, -2, 0
00029 7, 15, -2.5, 0
00030 8, 15, -3, 0
00031 9, 15, -3.5, 0
00032 10, 15, -4, 0
00033 11, 15, -4.5, 0
00034 12, 15, -5, 0
00035 13, 15, -5.5, 0
00036 14, 15, -6, 0
00037 15, 15, -6.5, 0
00038 16, 15, -7, 0
00039 17, 15, -7.5, 0
00040 18, 15, -8, 0
00041 19, 15, -8.5, 0
00042 20, 15, -9, 0
00043 21, 15, -9.5, 0
00044 22, 15, -10, 0
00045 23, 15, -10.5, 0
00046 24, 15, -11, 0
00047 25, 15, -11.5, 0
00048 26, 15, -12, 0
00049 27, 15, -12.5, 0
00050 28, 15, -13, 0
```

Note.

*.mxt files are text input files. You can modify or create files to manipulate your MIDAS GTS models

Ready Ln 0 / 354, Col 1 NUM

Example #2

Importing to MIDAS GTS NX

The screenshot illustrates the initial setup in MIDAS GTS NX. The 'New' menu is open, showing options like 'Create a new document', 'Open', 'Save', 'Import', 'Export', 'Close', and 'Close All'. A red circle with the number '1' highlights the 'New' menu icon, and a red circle with the number '2' highlights the 'Create a new document' option. A red arrow points from the 'Create a new document' option to the 'Analysis Setting' dialog box. In the dialog box, the 'Project Title' is 'SSI Example', 'Desc.' is empty, 'Model Type' is set to '2D' (indicated by a red circle with the number '3' and a red arrow), 'Unit System' is 'kN', 'm', 'J', 'sec', 'Initial Parameters' are 'Gravity Acceleration(g)' at '9.80665 m/sec²', 'Initial Temperature' at '0 [T]', and 'Plane Strain Thickness' at '1 m' (indicated by a red circle with the number '4'). The 'OK' button is highlighted with a red rectangle.

1 Select 'New'

2

3 Select '2D'

4

Analysis Setting

Project Title SSI Example Engineer

Desc.

Model Type

☐ 3D

☒ 2D

☐ Axisymmetric

Unit System

kN m J sec

Initial Parameters Water Parameters

Gravity Acceleration(g) 9.80665 m/sec²

Initial Temperature 0 [T]

Plane Strain Thickness 1 m

OK Cancel

Output

> GTS NX 2019 (v2.1) (64bit)

> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.

> License has been certified.

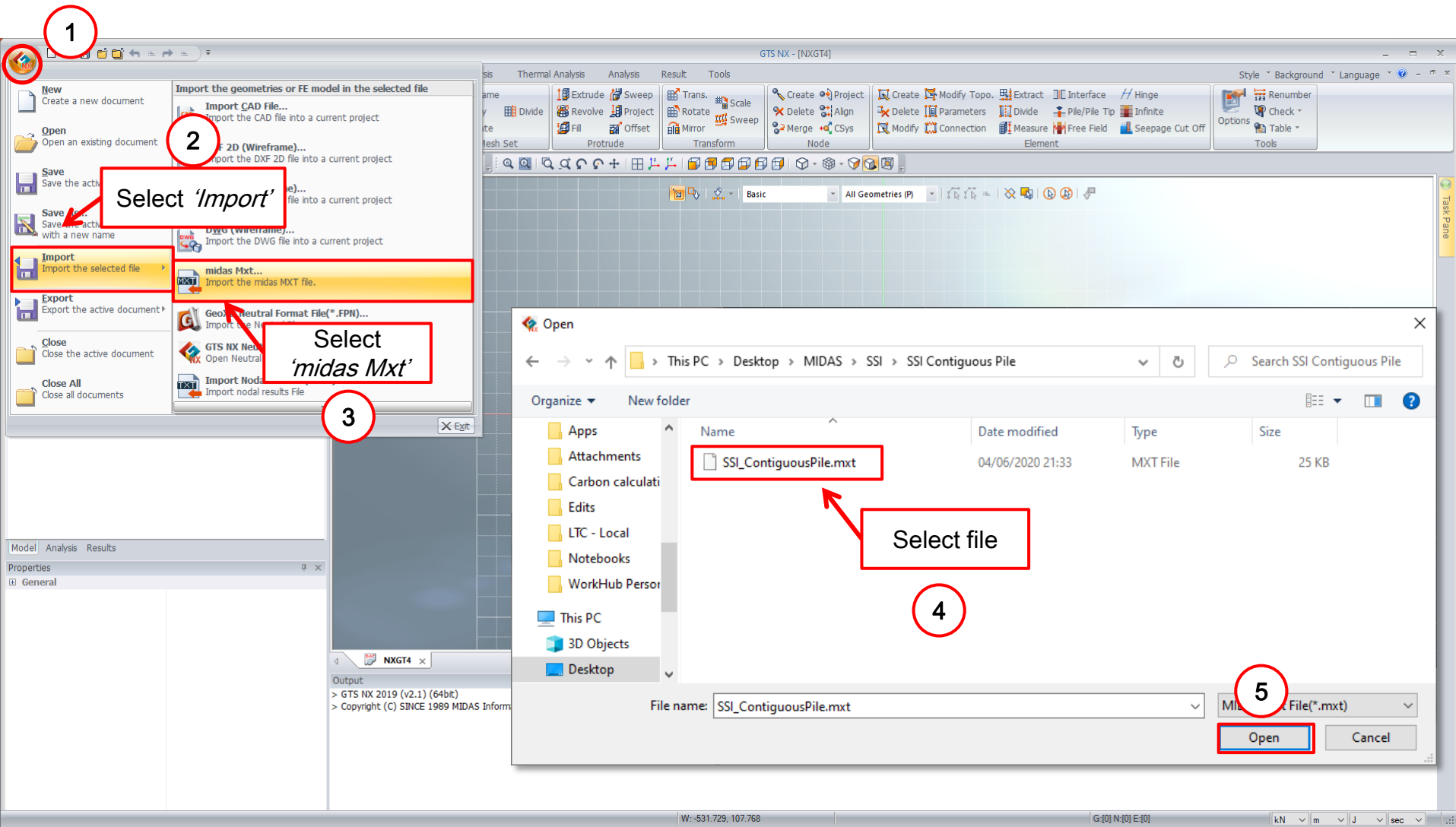
> Your maintenance period expires in 119 day(s).

> License has been certified.

> Your maintenance period expires in 119 day(s).

Example #2

■ Importing to MIDAS GTS NX



Example #2

Create FE mesh for surrounding soil

1 Select 'Rectangle'

2 Tick on 'Make face'

3 Create 2 surfaces

Backfill

Existing ground

15m 50m 7m 13m

Rectangle

2D

Method

Method ABS x, y

☐ Make Face

Geometry Set Geometry Set-1

OK Cancel Apply

W: 35.8363, -7.20584 X: 0°65 Y: -1°1 Z: -20°0 G: [5] N: [42] E: [41] kN m J sec

Example #2

Create FE mesh for surrounding soil

1 Select 'Mesh' tab

2 Select '2D'

3 Select 'Size' 0.5

4 Name the mesh

Generate mesh(Face)

Auto-Face Auto-Area Map-Face Map-Area

Selected 1 Object(s)

Size Method

☒ Size ☐ Division 0.5

Property

4 4: (Undefined)

Mesh Set Backfill

OK Cancel

Output

```
> GTS NX 2019 (v2.1) (64bit)
> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.
> Import MIDAS/Gen, MIDAS/Civil Text File <C:\Users\paa76924\Desktop\MIDAS\SSI\SSI Contiguous Pile\SSI_ContiguousPile.mxt>
> [Undo] Create Rectangle
> A mesh set[Auto-Mesh(2D)] which has 3537 nodes and 3380 elements has been created.
```

Example #2

■ Boundary conditions

1 Select 'Static/Slope tab'

2 Select 'Constraint'

3 Select 'Auto'

4 Apply

5 Select Nodes

6 Select 'Advanced'

7 Select 'Tx' and 'Ry'

8 Apply

Output

```
> GTS NX 2019 (v2.1) (64bit)
> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.
> Import MIDAS/Gen, MIDAS/Civil Text File <C:\Users\paa76924\Desktop\MIDAS\SSI\SSI Contiguous Pile\SSI_ContiguousPile.mxt>
> [Undo] Create Rectangle
> A mesh set[Auto-Mesh(2D)] which has 3537 nodes and 3380 elements has been created.
> A mesh set[Backfill] which has 1515 nodes and 1400 elements has been created.
> [Error] Cannot Delete Default Mesh Set.
> Work project is being saved by auto-save function.
```

Example #2

Materials

1

Select 'Mesh' tab

2

Select 'Material'

Add/Modify Material

No	Name	Type
1	C40/50	Isotropic-Elastic

Import from Excel

Select 'Import from Excel'

W: -9.8185, -1.65408 X: 0°65'Y: -1°1'Z: 20°0 G: [5] N: [4952] E: [4821] kN m J sec

> GTS NX 2019 (v2.1) (64bit)
> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.
> Import MIDAS/Gen, MIDAS/Civil Text File <C:\Users\paa76924\Desktop\MIDAS\SSI\SSI Contiguous Pile\SSI_ContiguousPile.mxt>
> [Undo] Create Rectangle
> A mesh set[Auto-Mesh(2D)] which has 3537 nodes and 3380 elements has been created.
> A mesh set[Backfill] which has 1515 nodes and 1400 elements has been created.
> [Error] Cannot Delete Default Mesh Set.
> Work project is being saved by auto-save function.

Example #2

Property assignment

The screenshot displays the GTS NX software interface with three numbered callouts illustrating the property assignment process:

- 1 Select 'Property'**: Points to the 'Property' icon in the 'Geometry' tab of the top toolbar.
- 2 Create properties for all imported soil materials**: Points to the 'Add/Modify Property' dialog box. This dialog contains a table with the following data:

No	Name	Type	Material
5	Weathered Soil	2D	Plane Strain
7	Backfill_1	2D	Plane Strain
8	Backfill_2	2D	Plane Strain
9	Backfill_3	2D	Plane Strain
10	Backfill_4	2D	Plane Strain
11	Backfill_5	2D	Plane Strain
12	Backfill_6	2D	Plane Strain
13	Backfill_7	2D	Plane Strain
14	Backfill_8	2D	Plane Strain
15	Backfill_9	2D	Plane Strain
16	Backfill_10	2D	Plane Strain
17	Backfill_11	2D	Plane Strain
18	Backfill_12	2D	Plane Strain
- 3 Use plane strain elements**: Points to the 'Create/Modify 2D Property' dialog box. In this dialog, the 'Plane Strain' option is selected under the 'Gauging Shell' tab. The 'Material' is set to '1: C40/50' and 'Material CSys' is set to 'Global Rectangular'.

The background shows a 3D model of a bridge pier with a mesh and a 2D cross-section view. The bottom status bar indicates coordinates: W: -9.54674, -4.98319; X: 0°65 Y: -1°1 Z: 20°0; G: [5] N: [4952] E: [482].

Example #2

Property assignment

The screenshot displays the GTS NX software interface for a project named "SSL_ContiguousPile.gts". The main window shows a 2D mesh model of a pile and backfill. A red circle with the number "1" highlights the "Parameters" button in the top toolbar. A red arrow points from a text box to the "Add/Modify Mesh Parameter" dialog box, which is open. The dialog box has tabs for "Node", "1D", "2D", "3D", and "Other", with "2D" selected. It contains a "Change Property" dropdown, a "Select Object(s)" button, and a list of 2D properties. The "Property" dropdown is set to "20: Backfill_14". The "Output" window at the bottom shows the following text:

```
Output
> GTS NX 2019 (v2.1) (64bit)
> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.
> Import MIDAS/Gen, MIDAS/Civil Text File <C:\Users\paa76924\Desktop\MIDAS\SSL\SSL Contiguous Pile\SSL_ContiguousPile.mxt>
> [Undo] Create Rectangle
> A mesh set[Auto-Mesh(2D)] which has 3537 nodes and 3380 elements has been created.
> A mesh set[Backfill] which has 1515 nodes and 1400 elements has been created.
> [Error] Cannot Delete Default Mesh Set.
> Work project is being saved by auto-save function.
> Work project is being saved by auto-save function.
```

The status bar at the bottom indicates the current coordinates and units: W: 31.3539, 13.361; X: 0°65 Y: -1°1 Z: -20°0; G [S] N [4952] E [4821]; kN, m, J, sec.

Example #2

Self-weight

The screenshot displays the GTS NX software interface for a project named 'GTS NX - [SSI_ContiguousPile.gts]'. The 'Static/Slope Analysis' tab is selected in the top menu bar. A red box labeled '1' points to this tab with the text 'Select 'Static/Slope tab''. The 'Tools' menu is open, and the 'Self Weight' option is highlighted with a red box labeled '2' and the text 'Select 'Self-weight''. The 'Gravity' dialog box is open, showing the 'Gravity' tab. The 'Name' field is 'Gravity', the 'Reference Object' is 'Coordinate', and the 'Ref. CSys' is 'Global Rectangular'. The 'Components' section shows 'Gx' as 0, 'Gy' as 0, and 'Gz' as -1. A red box labeled '3' points to the 'Gz' field with the text 'Gravity in Y direction, we are working on the X-Y plane'. The 'Load Set' dropdown is set to 'Selfweight'. The 'OK', 'Cancel', and 'Apply' buttons are at the bottom of the dialog. The left sidebar shows a tree view of the model, including 'Coordinate System', 'View Point', 'Work Plane', 'Datum', 'Material', 'Property', 'Geometry', 'Bedding Plane', 'Export Shape', 'Mesh Control', 'Mesh', 'Default Mesh Set', 'Ground', 'Backfill', 'Structure', 'Contact', 'Element Parameters', and 'Hinge'. The bottom status bar shows coordinates and units: 'W: 13.0354, 3.24613 X: 0°65 Y: -1°1 Z: 20°0 G: [5] N [4952] E [4821] kN m J sec'.

Select 'Static/Slope tab'

1

Select 'Self-weight'

2

Gravity

Gravity

Name Gravity

Reference Object

Type Coordinate

Ref. CSys Global Rectangular

Components

Gx 0

Gy 0

Gz -1

Spatial Distribution

Base Function None

Load Set Selfweight

OK Cancel Apply

Gravity in Y direction, we are working on the X-Y plane

Output

```
> GTS NX 2019 (v2.1) (64bit)
> Copyright (C) SINCE 1989 MIDAS Information Technology Co., Ltd. ALL RIGHTS RESERVED.
> Import MIDAS/Gen, MIDAS/Civil Text File <C:\Users\paa76924\Desktop\MIDAS\SSI\SSI Contiguous Pile\SSI_ContiguousPile.mxt>
> [Undo] Create Rectangle
> A mesh set[Auto-Mesh(2D)] which has 3537 nodes and 3380 elements has been created.
> A mesh set[Backfill] which has 1515 nodes and 1400 elements has been created.
> [Error] Cannot Delete Default Mesh Set.
> Work project is being saved by auto-save function.
> Work project is being saved by auto-save function.
> [Error] Select Element
```

W: 13.0354, 3.24613 X: 0°65 Y: -1°1 Z: 20°0 G: [5] N [4952] E [4821] kN m J sec

Example #2

Thermal loads

The screenshot displays the GTS NX software interface with various toolbars and a 3D model of a bridge deck. Two 'Temperature' dialog boxes are open, one for 'Contraction' and one for 'Expansion'. Red callouts with numbers 1 through 9 provide a step-by-step guide for applying thermal loads.

- 1**: Select 'Element temperature' in the 'Thermal Analysis' toolbar.
- 2**: Select the deck element in the 3D model.
- 3**: In the 'Temperature' dialog, type 'Contraction' in the 'Name' field.
- 4**: Type -30 in the 'Temperature' field.
- 5**: Type 'Contraction' in the 'Load Set' dropdown.
- 6**: Click the 'Apply' button.
- 7**: In the 'Temperature' dialog, type 'Expansion' in the 'Name' field.
- 8**: Type 60 in the 'Temperature' field.
- 9**: Type 'Expansion' in the 'Load Set' dropdown.

The software interface includes toolbars for Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, Thermal Analysis, Analysis, Result, and Tools. The 3D model shows a bridge deck with a mesh of elements. The 'Temperature' dialog boxes have tabs for 'Nodal Temperature' and 'Element Temperature', and fields for 'Name', 'Temperature', and 'Load Set'.

Example #2

Construction stages

Select 'Stage Set'

1

2

Type name

3

Click 'Add'

Construction Stage Set

Name: Stages

Stage Type: Stress

Add

Modify

Copy

Delete

Define CS...

4

Click 'Define CS...'

Close

> [Error] Cannot Delete Default Mesh Set.
> Work project is being saved by auto-save function.
> Work project is being saved by auto-save function.
> [Error] Select Element
> Work project is being saved by auto-save function.

W: 2.54795, 7.82805 X: 0°65 Y: -1°1 Z: 20°0 G: [5] N[4952] E[4821] kN m J sec

Example #2

■ Construction stages

Define Construction Stage

Construction Stage Set Name: Stages

Stage ID: 1: Initial

Stage Name: Initial

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Set Data

- Mesh
 - Backfill
 - Default Mesh Set
 - Ground
 - Structure
- Boundary Condition
 - Boundary Set-1
- Static Load
 - Contraction
 - Expansion
 - Selfweight
- Contact

Activated Data

- Mesh
 - Ground
 - Boundary Condition
 - Boundary Set-1
 - Static Load
 - Selfweight
- Contact

Deactivated Data

- Mesh
 - Boundary Condition
 - Static Load
- Contact

Initial Condition

☐ Define Water Level For Global

0 m None

☐ Define Water Level For Mesh Set

Input Water Level...

☐ LDF...

☐ Clear Displacement

☐ Slope Stability(SRM)

Sort By: Name

Show Data: All

Buttons: Save, Close

Example #2

■ Construction stages

Define Construction Stage

Construction Stage Set Name: Stages

Stage ID: 2: Construction Stage-2

Stage Name: Stage 2

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Set Data:

- Mesh
 - Backfill
 - Default Mesh Set
 - Ground
 - Structure
- Boundary Condition
 - Boundary Set-1
- Static Load
 - Contraction
 - Expansion
 - Selfweight
- Contact

Activated Data:

- Mesh
 - Structure
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Initial Condition:

- ☐ Define Water Level For Global
 - 0 m
 - None
- ☐ Define Water Level For Mesh Set
 - Input Water Level...

Buttons: Analysis Control..., Output Control..., LDF..., Clear Displacement, Slope Stability(SRM), Save, Close

Sort By: Name

Show Data: All

Example #2

■ Construction stages

Define Construction Stage

Construction Stage Set Name: Stages

Stage ID: 3: Backfill
Stage Name: Backfill
Stage Type: Stress

Move to Previous Move to Next
New Insert Delete

Set Data Activated Data Deactivated Data

Set Data: Mesh, Backfill, Default Mesh Set, Ground, Structure, Boundary Condition, Boundary Set-1, Static Load, Contraction, Expansion, Selfweight, Contact

Activated Data: Mesh, Backfill, Boundary Condition, Static Load, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Contact

Initial Condition:
☐ Define Water Level For Global
0 m None
☐ Define Water Level For Mesh Set
Input Water Level...
☐ LDF...
☐ Clear Displacement
☐ Slope Stability(SRM)
☐ Slope Stability(SAM)

Sort By: Name Show Data: All

Save Close

Example #2

■ Construction stages

Define Construction Stage

Construction Stage Set Name: Stages

Stage ID: 4: Contraction
Stage Name: Contraction
Stage Type: Stress

Move to Previous Move to Next
New Insert Delete

Set Data Activated Data Deactivated Data

Set Data: Mesh, Backfill, Default Mesh Set, Ground, Structure, Boundary Condition, Boundary Set-1, Static Load, Contraction, Expansion, Selfweight, Contact

Activated Data: Mesh, Boundary Condition, Static Load, **Contraction**, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Contact

Initial Condition:
☐ Define Water Level For Global
0 m None ...
☐ Define Water Level For Mesh Set
Input Water Level...
☐ LDF...
☒ Clear Displacement
☐ Slope Stability(SAM)
☐ Slope Stability(SAM)

Tick 'Clear Displacement'

Sort By: Name Show Data: All Save Close

Example #2

■ Construction stages

Define Construction Stage

Construction Stage Set Name: Stages

Stage ID: 5: Construction Stage-1

Stage Name: Expansion

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Set Data:

- Mesh
 - Backfill
 - Default Mesh Set
 - Ground
 - Structure
- Boundary Condition
 - Boundary Set-1
- Static Load
 - Contraction
 - Expansion
 - Selfweight
- Contact

Activated Data:

- Mesh
- Boundary Condition
- Static Load
- Expansion
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Initial Condition:

- ☐ Define Water Level For Global
 - 0 m
 - None
- ☐ Define Water Level For Mesh Set
 - Input Water Level...

Buttons: Analysis Control..., Output Control..., LDF..., Clear Displacement, Slope Stability(SRM)

Sort By: Name

Show Data: All

Buttons: Save, Close

Example #2

Analysis Case

1 Select 'Analysis' tab

2 Select 'General'

3 Type name

4 Select 'Construction Stage'

5 Select Construction Stage Name

The screenshot displays the MIDAS NX software interface. The 'Add/Modify Analysis Case' dialog box is open, showing the 'Analysis' tab. The 'General' sub-tab is selected. The 'Case Setting' section includes a 'Name' field with 'CS', a 'Type' dropdown menu set to 'Construction Stage', and a 'Construction Stage Set' dropdown menu set to 'Stages'. The 'Analysis Case Model' section is empty. The 'Active Sets' section is also empty. The 'Output' section at the bottom shows analysis results for 'NXGT4' and 'SSI_ContiguousPile.gts'. The 'Solve Each Load Set Independently' checkbox is unchecked. The 'OK' button is highlighted.

Example #2

Set up analysis

The screenshot displays the MIDAS NX software interface for a project named 'GTS NX - [SSI_ContiguousPile.gts:1]'. The 'Analysis' tab is active in the top menu bar. Within the 'Analysis' toolbar, the 'Perform' button (represented by a running person icon) is highlighted with a red box. A red arrow points from a circled number '1' to this button, with a text box stating 'Select 'Perform''. The main workspace shows a 3D model of a pile foundation with a grid of colored layers. The bottom status bar displays system information and coordinates.

Model Analysis Results

Properties General

Output

```
> ANALYSIS COMPLETED
>
> [SYSTEM INFO]
> NUMBER OF THREADS : 1
> MAXIMUM MEMORY USAGE : 162 MB
> AVAILABLE MEMORY : 25974 MB
> TOTAL CPU TIME : 32.375 sec
> WALL CLOCK TIME : 32.482 sec
> TOTAL WARNINGS : 0
```

W: -1.50711, -0.43547 X: 0°65 Y: -20°0 Z: 1°1 G:[5] N:[4952] E:[4821] kN m J sec

Example #2

Results

The screenshot displays the GTS NX software interface for the file 'GTS NX - [SSI_ContiguousPile_test1.gts]'. The 'Results' tab is selected in the top menu bar. The left-hand 'Results' tree shows a list of analysis results, with 'Expansion' and 'S-XX TOTAL' highlighted. The main window shows a 3D stress contour plot of a pile structure. A color scale on the right indicates stress values in kN/m^2 , ranging from -1.83853e+002 (blue) to +6.86009e+001 (red). The plot shows a vertical pile with a horizontal cross-section, with stress contours indicating the distribution of S-XX TOTAL stress. The bottom status bar shows the work project is being saved by the auto-save function.

1 Select 'Results' tab

2 Select 'Expansion'

3 Select 'S-XX TOTAL'

Example #2

Results

1 Select 'Result' tab

2 Select 'Cutting Diag.'

3 Type in name

4 Type in coordinates

5 Select (+)X Dir.

6 Type in 40

7

The screenshot displays the MIDAS NX software interface. The 'Result' tab is selected in the top menu. The 'Cutting Diag.' option is highlighted in the 'Advanced' section. The 'On-Curve Diagram' dialog box is open, showing the 'Cutting Diagram Mode' set to 'Cutting Line'. Under 'Define Positions', the 'Type' is '2-Points Line'. Point 1 coordinates are (15.1, 0, 0) and Point 2 coordinates are (15.1, -20, 0). The 'Section' is set to '(+)X Dir.' and the 'Division' is set to 40. The 'Apply' button is highlighted. The background shows a 3D model of a pile and a displacement legend on the right side.

DISPLACEMENT TX, m	Value
0.0%	+8.95231e-003
0.1%	+8.20629e-003
0.2%	+7.46026e-003
0.4%	+6.71424e-003
0.5%	+5.96821e-003
0.8%	+5.22218e-003
1.5%	+4.47616e-003
3.1%	+3.73013e-003
5.8%	+2.98410e-003
10.9%	+2.23808e-003
21.8%	+1.49205e-003
54.9%	+7.46026e-004
	+0.00000e+000

Example #2

Results

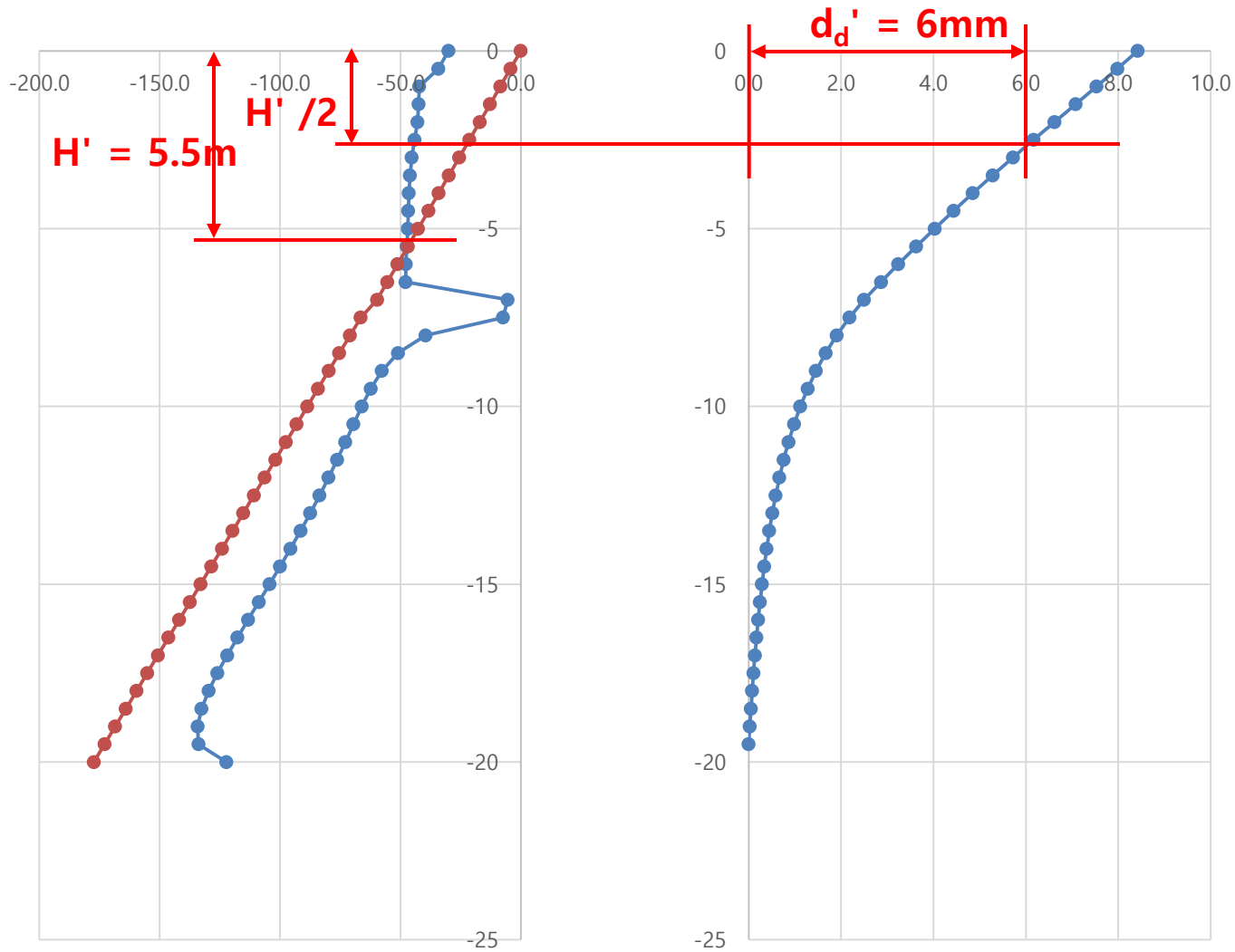
The screenshot shows the GTS NX software interface with the Results tab active. A table of values is displayed, and red annotations highlight the steps to view the table:

- 1** Right-click 'Back' On-curve Diagram
- 2** Select 'Show Table'
- 3** Select values

ID	X (m)	Y (m)	Z (m)	Value
1	15.1	0	0	-3.002948e+001
2	15.1	-0.5	0	-3.419955e+001
3	15.1	-1	0	-4.213879e+001
4	15.1	-1.5	0	-4.242853e+001
5	15.1	-2	0	-4.289170e+001
6	15.1	-2.5	0	-4.416166e+001
7	15.1	-3	0	-4.526085e+001
8	15.1	-3.5	0	-4.599934e+001
9	15.1	-4	0	-4.645812e+001
10	15.1	-4.5	0	-4.671980e+001
11	15.1	-5	0	-4.690530e+001
12	15.1	-5.5	0	-4.730461e+001
13	15.1	-6	0	-4.776598e+001
14	15.1	-6.5	0	-4.783240e+001
15	15.1	-7	0	-5.403166e+000
16	15.1	-7.5	0	-7.393581e+000
17	15.1	-8	0	-3.952780e+001
18	15.1	-8.5	0	-5.102869e+001
19	15.1	-9	0	-5.771347e+001
20	15.1	-9.5	0	-6.233514e+001
21	15.1	-10	0	-6.607732e+001
22	15.1	-10.5	0	-6.949107e+001
23	15.1	-11	0	-7.284282e+001
24	15.1	-11.5	0	-7.626851e+001
25	15.1	-12	0	-7.983145e+001
26	15.1	-12.5	0	-8.355596e+001
27	15.1	-13	0	-8.744424e+001
28	15.1	-13.5	0	-9.148611e+001
29	15.1	-14	0	-9.566438e+001
30	15.1	-14.5	0	-9.995713e+001
31	15.1	-15	0	-1.043382e+002
32	15.1	-15.5	0	-1.087758e+002

Example #2

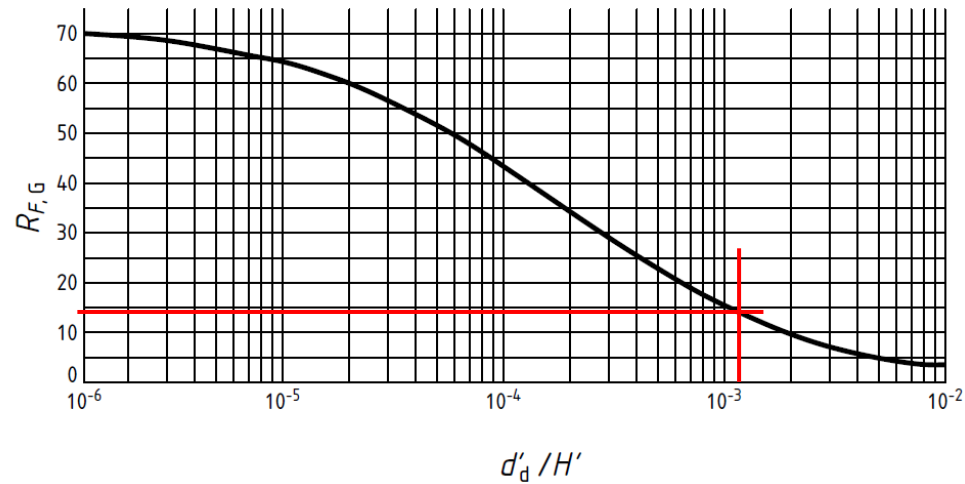
■ Extract results



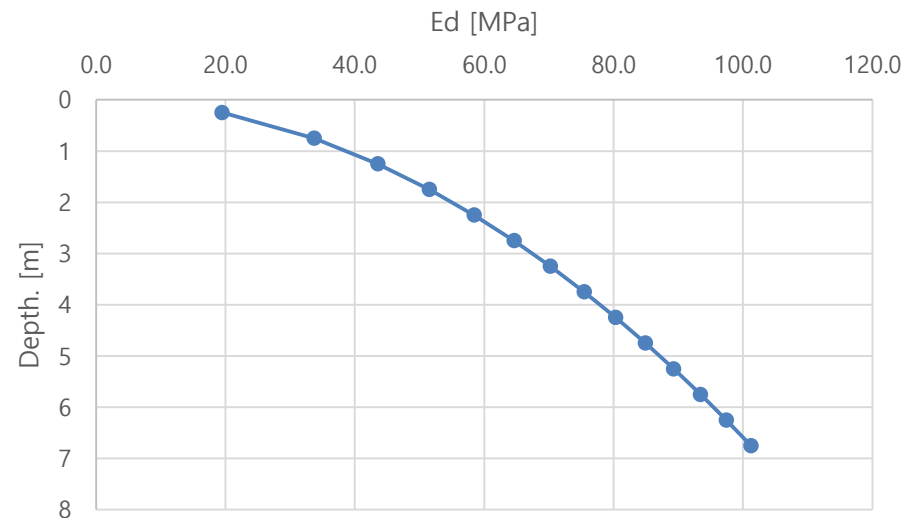
Example #2

Recalculate material properties

d' 6.4 mm
 H' 5.5 m
 d'/H' 1.16E-03
 $R_{F,G}$ 14



	z m	σ_v kN/m ²	σ_m kN/m ²	G kN/m ²	E kN/m ²	Ed kN/m ²	Ed MPa
1	0.25	5	3.088079	5412.462	12989.9	19484.86	19.5
2	0.75	15	9.264236	9374.66	22499.2	33748.78	33.7
3	1.25	25	15.44039	12102.63	29046.3	43569.48	43.6
4	1.75	35	21.61655	14320.03	34368.1	51552.11	51.6
5	2.25	45	27.79271	16237.39	38969.7	58454.59	58.5
6	2.75	55	33.96886	17951.11	43082.7	64623.98	64.6
7	3.25	65	40.14502	19514.91	46835.8	70253.68	70.3
8	3.75	75	46.32118	20962.38	50309.7	75464.56	75.5
9	4.25	85	52.49734	22316.15	53558.8	80338.15	80.3
10	4.75	95	58.67349	23592.38	56621.7	84932.55	84.9
11	5.25	105	64.84965	24803.02	59527.2	89290.87	89.3
12	5.75	115	71.02581	25957.26	62297.4	93446.13	93.4
13	6.25	125	77.20196	27062.31	64949.5	97424.32	97.4
14	6.75	135	83.37812	28123.98	67497.6	101246.3	101.2



Example #2

- Update model, recalculate and extract results

