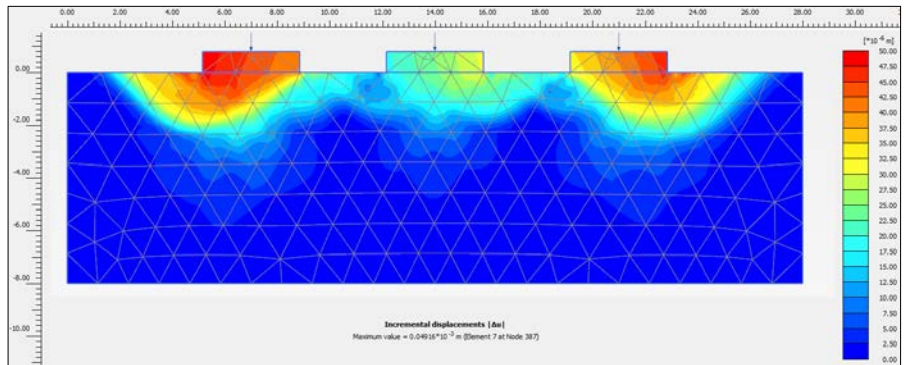
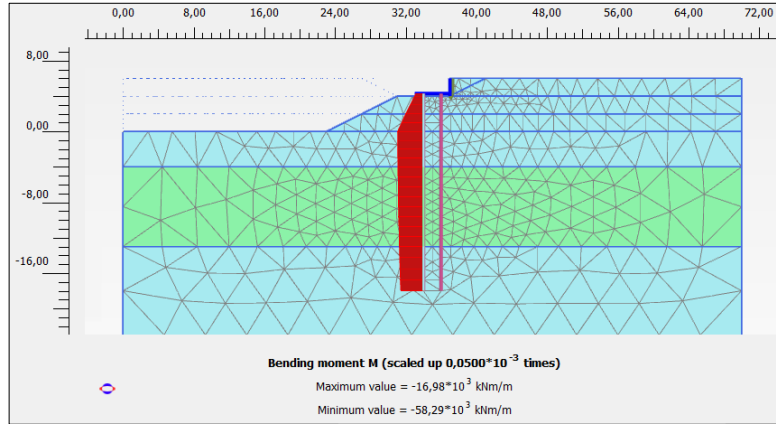




Escuela Técnica Superior de Ingeniería de Caminos y Puertos y de Ingeniería de Minas

RESOLUTION OF GEOTECHNICAL ENGINEERING PROBLEMS WITH PLAXIS 2D



Gonzalo García Ros
José Antonio Jiménez Valera
Iván Alhama Manteca
María Encarnación Martínez Moreno



Universidad Politécnica de Cartagena

RESOLUTION OF GEOTECHNICAL ENGINEERING PROBLEMS WITH PLAXIS 2D

Gonzalo García Ros

José Antonio Jiménez Valera

Iván Alhama Manteca

María Encarnación Martínez Moreno

Escuela Técnica Superior de Ingeniería de Caminos, Canales y Puertos y de Ingeniería de Minas

© 2022, Gonzalo García Ros
José Antonio Jiménez Valera
Iván Alhama Manteca
María encarnación Martínez Moreno

© 2022, Universidad Politécnica de Cartagena.

CRAI Biblioteca
Plaza del Hospital, 1
30202 Cartagena.
968325908
ediciones@upct.es



Primera edición, 2022

ISBN: 978-84-17853-60-0



Esta obra está bajo una licencia de Reconocimiento-NOcomercial-SinObraDerivada (by-nc-nd): no se permite el uso comercial de la obra original ni la generación de obras derivadas.

[http://es.creativecommons.org/blog/wp-content/uploads/2013/04/by-nc-nd.eu_petit .png](http://es.creativecommons.org/blog/wp-content/uploads/2013/04/by-nc-nd.eu_petit.png)

Preface

This text is a new effort of the Ground Engineering Area of Universidad Politécnica de Cartagena (UPCT) to develop teaching material that helps students of Bachelor and Master Courses of Civil Engineering to start using PLAXIS 2D. PLAXIS 2D is one of the most employed software to analyse and solve problems related to geotechnics. Nowadays, the use of commercial software (especially if it is as powerful as PLAXIS) must be an essential part of engineering study programmes, mainly because of two reasons. In first place, students have a practical learning experience that also helps them to reinforce the concepts they acquire in the theoretical lessons. In second place, they orientate and prepare themselves for their professional career when leaving university and entering the engineering labour market.

The aim of the problems that are solved in the present book is, exclusively, the initiation in the use of PLAXIS software. Despite the rigour from the geotechnical point of view, this text is focused on the guided creation from scratch of different PLAXIS 2D models that accurately show the geotechnical conditioning factors presented in each problem. By all means, the result collection and representation employing PLAXIS Output is of equal importance in the text.

PLAXIS 2D possibilities for both data introduction and model creation, and solution presentation with PLAXIS 2D, are varied. This has led us to carefully choose the exercises to solve in order to help the student to acquire the knowledge in a progressive and orderly manner.

In this way, this book consists of seven exercises, each of them easily identifiable and relatable to the different topics that are studied along the ground engineering study programme: safety analysis in embankments and excavations, soil settlements, flow nets under retaining structures, footings, sheet piles wall and deep foundations.

This book is thought as a mean to gradually learn to use the program, so a user without previous experience should follow the order in which the exercises are presented in

the text. Therefore, the first exercises are focus as a guide, carefully describing each of the options the student must select in the input interface. Here, also the presentation of results is thoroughly explained. As we go on reading, the text is less focused on being a guide and starts showing the versatility of the software: different types of calculation, constitutive models, structural elements, their combinations and, of course, the result interface.

This book and its content are only for educational purposes. Its elaboration, as well as the images from the software, has been authorized by PLAXIS, a property of Bentley Systems Company, to whom we are thankful for their collaboration. 'PLAXIS 2D Reference Manual 2019' and the online help at <https://www.plaxis.com/support>, have also been very helpful in the resolution of the problems, and the reader is encouraged to read those manuals for an advance experience.

The version that has been employed for the resolution of the problems has been 2016.01, owning the corresponding user's licence.

Do not hesitate to contact the authors for questions or suggestions. All comments are welcome.

The authors

Index

Exercise 01	
Long-term stability analysis of a cutting excavation in a homogeneous material	1
Exercise 02	
Stability analysis of an embankment on a clay layer	22
Exercise 03	
Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway	38
Exercise 04	
Flow net under concrete dam	72
Exercise 05	
Shallow foundations on clay soil	94
Exercise 06	
Excavation in heterogeneous soil between two diaphragm walls	134
Exercise 07	
Pile foundation under bridge deck support	191

Exercise 01

Long-term stability analysis of a cutting excavation in a homogeneous material

1.1. INTRODUCTION

The aim in the following exercise is to show a cutting excavated in a homogeneous soil, analysing different points of interest in this kind of works, especially the determination of the possible slip surface and the long-term global safety factor.

1.2. PARAMETERS OF THE PROBLEM

Let us consider a mass of homogeneous soil that is big enough (80-meter wide and 30-meter high). The cutting geometry (10 meters in the vertical direction with a 3H:2V slope) and the geotechnical parameters of the considered problem are presented in Figure 1.1 and Table 1.1, respectively.

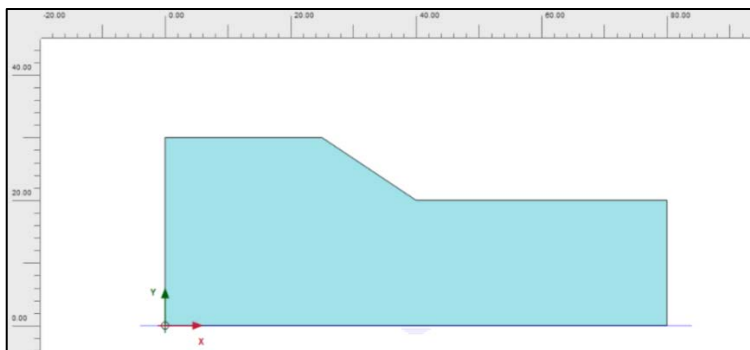


Figure 1. 1. Sketch of the cutting excavated in a homogeneous soil in PLAXIS 2D

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

Geotechnical parameters

Material	Clay
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	17.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	20 kN/m ³
Young's modulus (E')	15000 kPa
Poisson's ratio (ν')	0.33
Effective soil cohesion (c')	14 kPa
Effective friction angle (ϕ')	23.5°
Dilatancy angle (ψ)	0°

Geometrical parameters

Cutting height	10 m
Slope	3H:2V
Phreatic level	Half-way of the cutting (5 m)

Table 1. 1. Geotechnical and geometrical parameters of the cutting excavated in homogeneous soil

1.3. RESOLUTION OF CUTTING EXERCISE WITH PLAXIS 2 D

1.3.1 Introduction of the parameters of the problem

We start the finite element software PLAXIS 2D (PLAXIS input) and choose 'Start a new project' option in the emerging window 'Quick select'.

A new window now appears, which is called 'Project properties'. In this, which has three tabs, is where we name the project and we choose the units, the type of model and elements, some of the constant values and the measure of the borders among which the geometry of the soil and the structures are defined.

In 'Project' tab we name the project (Figure 1.2). Therefore, for example, in the title bar, we can write: 'Cutting excavated in homogeneous soil'.

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

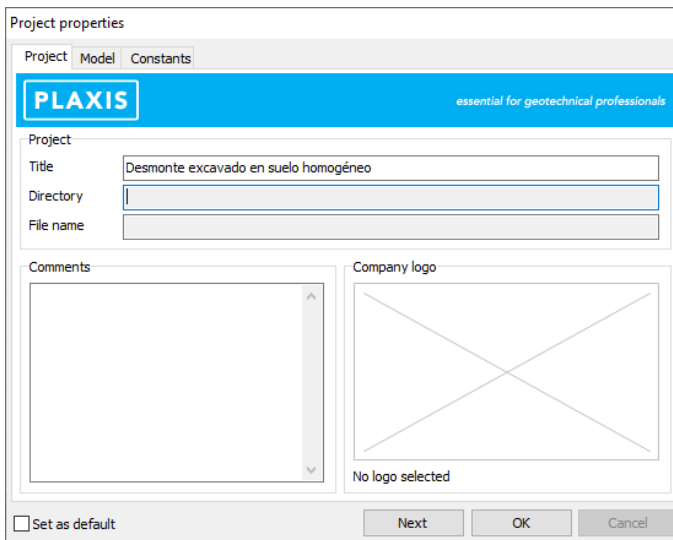


Figure 1. 2. 'Project properties' window. Basic data

Before clicking on 'OK' we go to 'Model' tab and we choose 'Plane strain' option for the model and '15-Noded' option for the elements. We keep the default units the software presents, as shown in Figure 1.3. Finally, in order to define the borders, we write the values in Table 1.2.

x_{\min}	0	m
x_{\max}	80	m
y_{\min}	0	m
y_{\max}	35	m

Table 1. 2. Coordinates for the Project borders

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

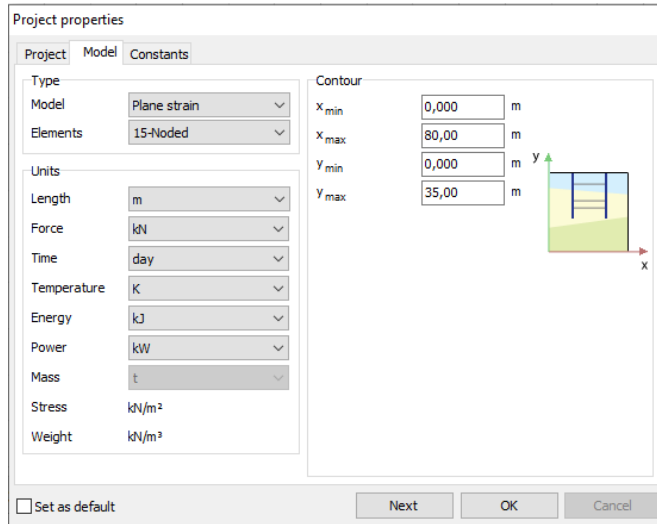


Figure 1. 3. 'Project properties' window. model, units and project borders

We keep the default values the software presents for the constants (Figure 1.4), and we click on 'OK' to save all the modifications that have been introduced in the 'Project properties' window.

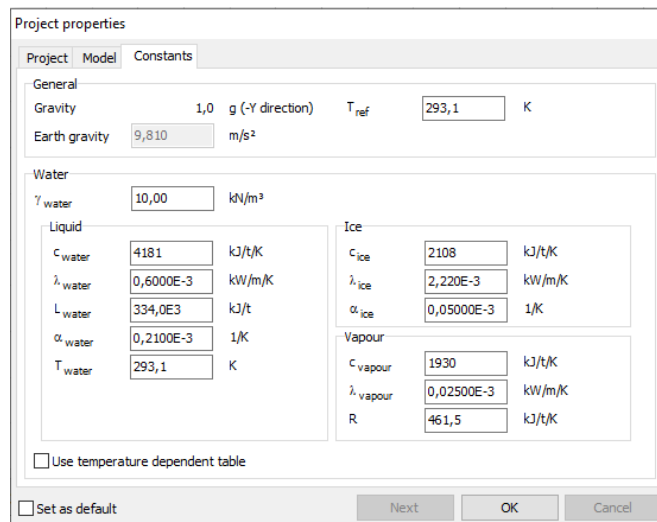


Figure 1. 4. 'Project properties' window. Project constants

1.3.2 Soil definition

In order to define the homogeneous material where the excavation has been carried out, we choose the blue tab, 'Soil'. We can find two blue tabs, which are the 'Geometry Modes', and three green ones, the 'Calculation Modes'.

If we click on the 'Create borehole' button, as shown in Figure 1.5, we can create the first necessary borehole to define the excavated material.

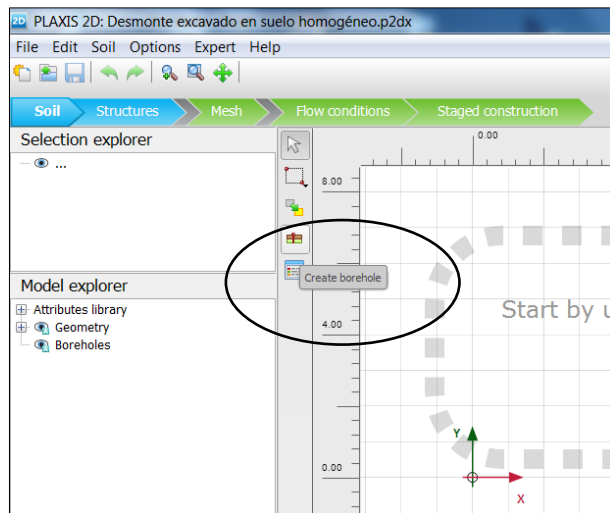


Figure 1. 5. 'Create borehole' button

Now we have to define the borehole. We first the point (0, 0), and an emerging window appears, 'Modify soil layers'. Here, we select 'Add', writing in 'Top' 30 and in 'Bottom' 0.

As the phreatic level is halfway the cutting, in 'Head', we write 25 (meters). Once all these modifications are done, we click on 'OK' to save the data of this first borehole, Figure 1.6.

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

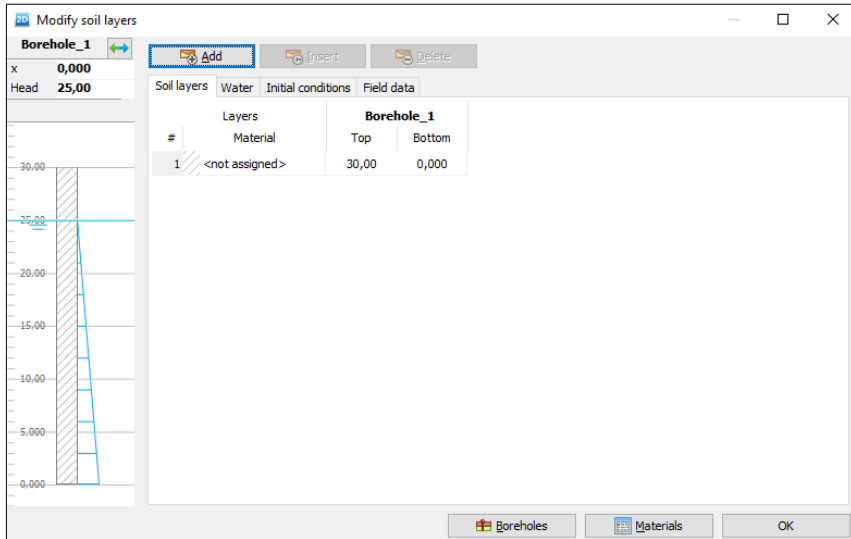


Figure 1. 6. First necessary borehole to define the cutting

The second borehole is placed in coordinates (25, 30). Again, we write 30 in ‘*Top*’ and 0 in ‘*Bottom*’. The value of ‘*Head*’ is 25.

The third borehole is located in coordinates (32.5, 25). In order to be able to place the borehole in coordinate $x=32.5$, it is necessary to have set the value 2 in ‘*Number of snap intervals*’, in ‘*Snapping options*’, Figure 1.7. This allows subdividing each interval of 1 meter in 2 when introducing data. After that, we write 25 in ‘*Top*’ and 0 in ‘*Bottom*’. ‘*Head*’ value is still 25.

The fourth and last borehole is located in coordinates (40, 20). In ‘*Top*’, we introduce 20 value and in ‘*Bottom*’, 0. As the phreatic level is now 20 m (ground surface), in ‘*Head*’ we write 20.

Once the geometry of the problem has been defined, we have to create the material to model the homogeneous soil. For this, we select the ‘*Show materials*’ button (Figure 1.8.). Here, we keep the default values (‘*Set type*’ in ‘*Soil interfaces*’) and click on ‘*New*’. Now we name the material (for example, ‘*Clay*’) and introduce the geotechnical parameters presented in Table 1.1. Figures 1.9 and 1.10 show the parameters once they are set in PLAXIS 2D.

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

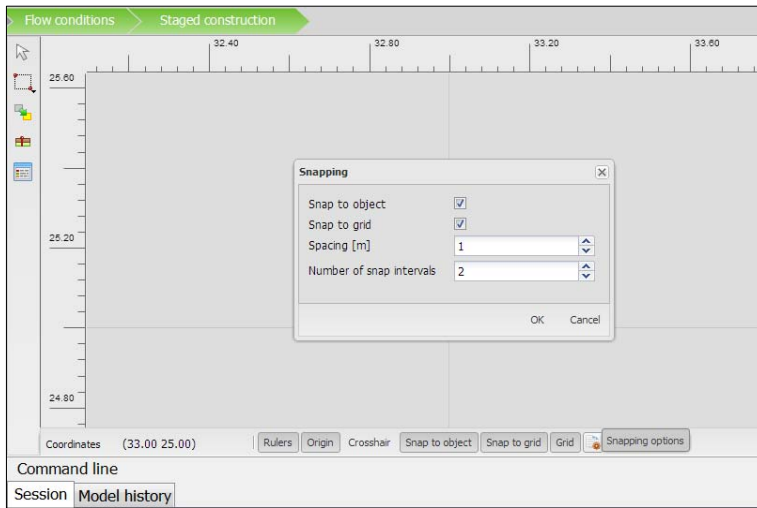


Figure 1. 7. Snapping options



Figure 1. 8. 'Show materials' button and cutting geometry

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

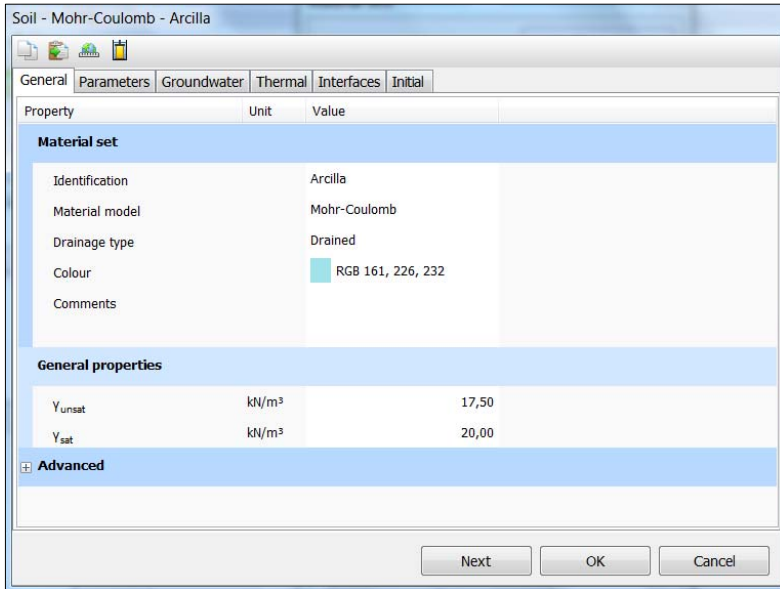


Figure 1. 9. General soil properties in our project

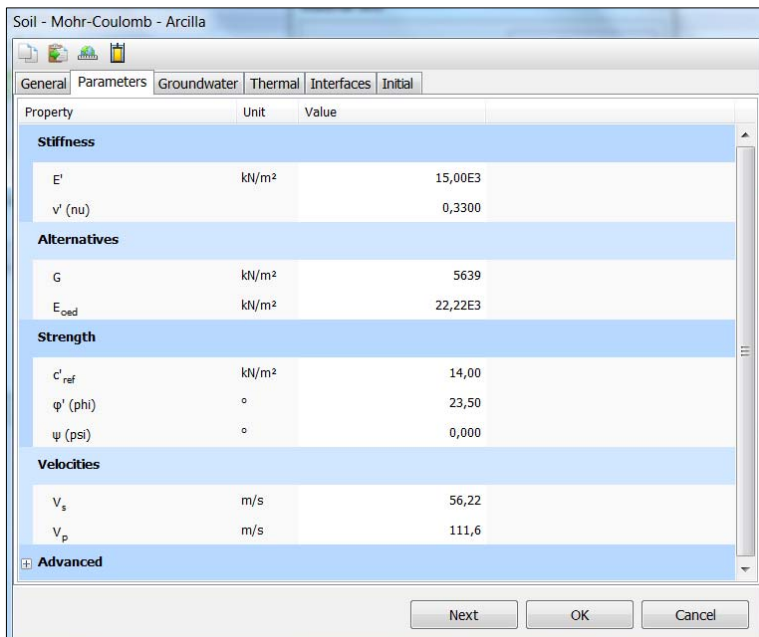


Figure 1. 10. Soil parameters in our project

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

In the geometry mode ‘*Soil*’, we select (in the workspace), with the right mouse button, the material that has been previously generated (select ‘*Set material*’, select ‘*Clay*’), Figure 1.11.

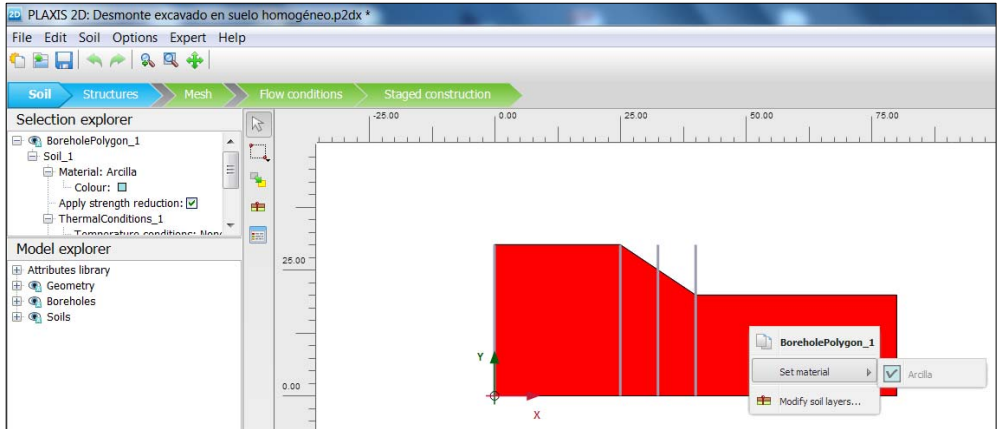


Figure 1. 11. Setting ‘Clay’ as the material to model the studied soil

1.3.3 Soil meshing

We select ‘*Mesh*’ calculation mode to mesh the soil. After that, we click on ‘*Generate mesh*’ button (Figure (1.12)) and we choose a medium element distribution (Figure 1.13).

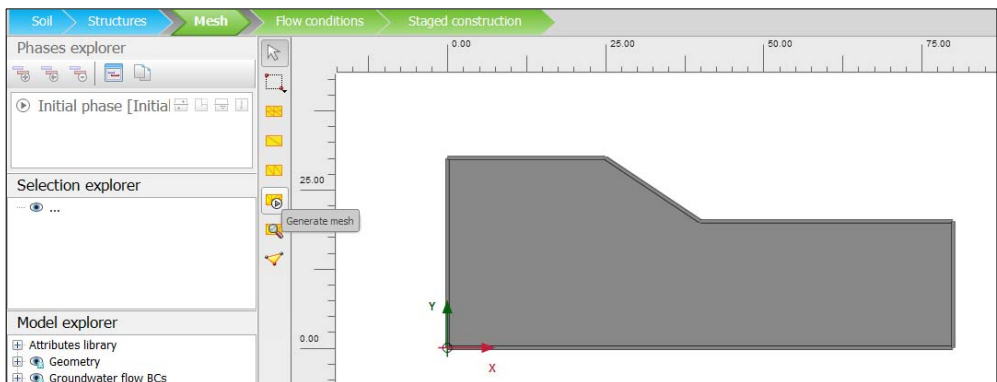


Figure 1. 12. Generating mesh in ‘*Mesh*’ calculation mode

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

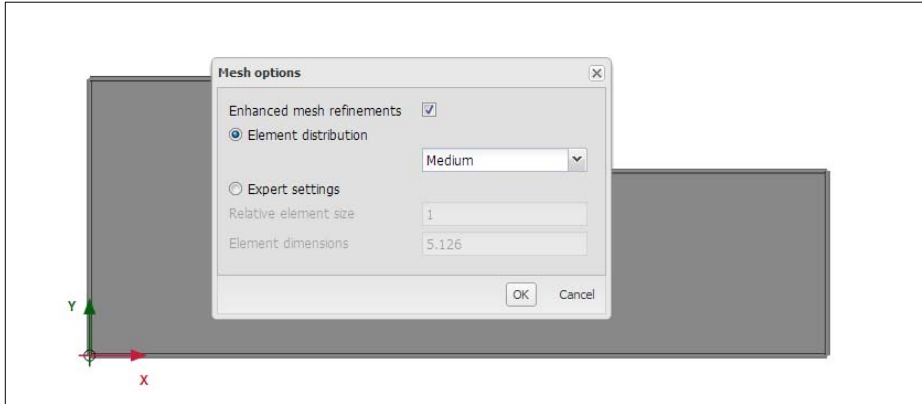


Figure 1. 13. Selecting a medium element distribution

In order to display the generated mesh, we select ‘*View mesh*’ button, which is right below ‘*Generate mesh*’ one. This opens PLAXIS Output program, allowing us to see the generated mesh (Figure 1.14).

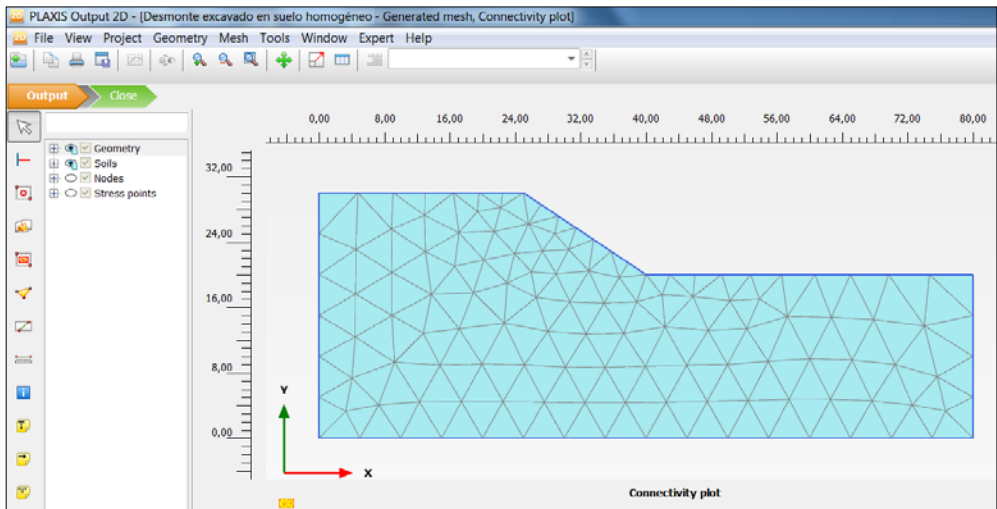


Figure 1. 14. Generated mesh, presented in PLAXIS Output

1.3.4 Display of the phreatic level in the ground

If we select ‘*Flow conditions*’ calculation mode, we can display the phreatic level created when defining the soil with the boreholes (Figure 1.15).

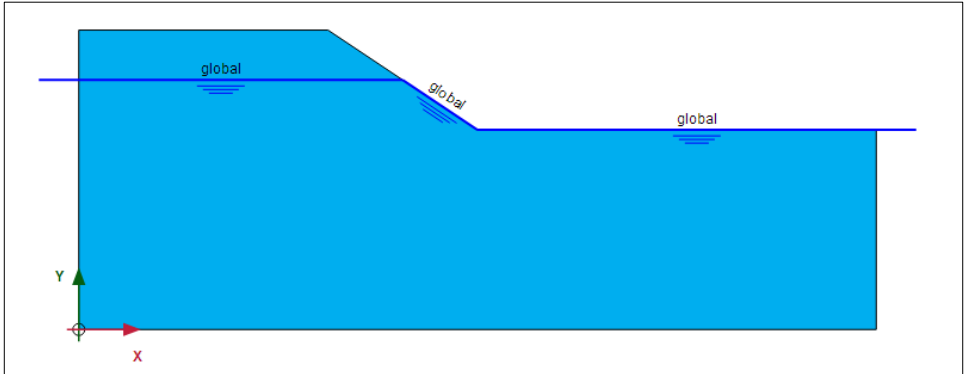


Figure 1. 15. Generated global phreatic level

1.3.5 Project phases

In this particular case, since we are studying the stability of a cutting, we just create two calculation phases. The first one is actually an Initial Phase in which the slope has already been created, while the second phase is the one that calculates the safety factor.

The first phase (*Initial Phase*) is already created. The default Calculation type is *'K0 procedure'*, but we choose *'Gravity loading'* instead (Figure 1.16). Both options determine initial stresses in the natural terrain, but they are obtained in a different way.

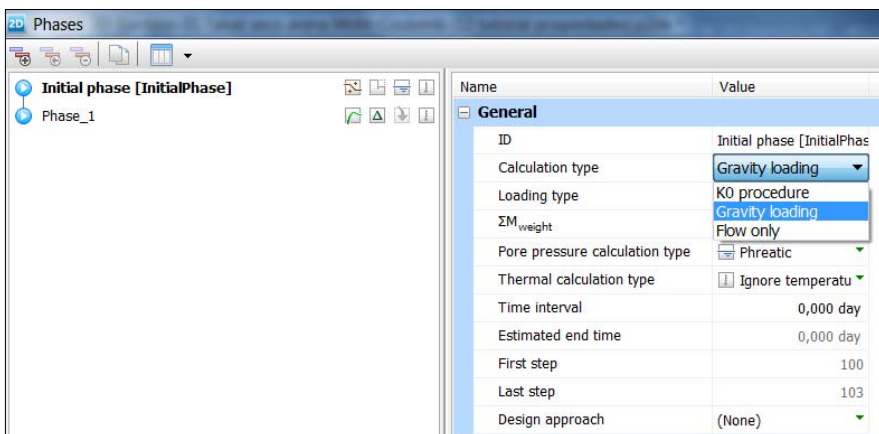


Figure 1. 16. Initial Phase definition. Selection of *'Gravity loading'* calculation type

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

In this way, when choosing ‘*K0 procedure*’, vertical stresses are calculated, and the horizontal stresses obtained from the vertical ones and the K_0 value that has been set. It is a calculation that does not involve finite elements (the governing equation are not solved), so the equilibrium is not guaranteed if we do not have horizontal soil layers.

Nevertheless, with ‘*Gravity loading*’ option, a plastic calculation is carried out, where initial stresses are obtained after applying the weight of the material. This option is, therefore, more appropriate when we do not have a horizontal stratigraphy. As explained later, this option does lead to displacements, so we must not consider those when starting the following phase.

In order to create the second phase (which allows obtaining the safety factor and the slip surface) we click on ‘*Add phase*’ button in ‘*Phase explorer*’ menu (Figure 1.17).

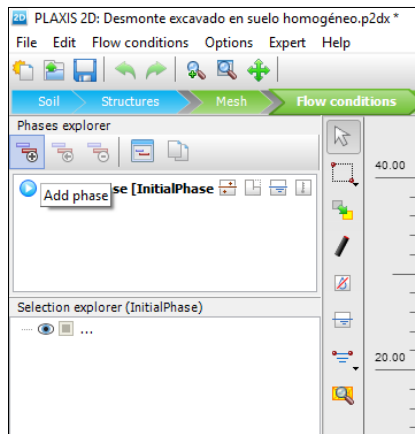


Figure 1. 17. Creation of the new phase in ‘*Phase explore*’ menu

Clicking twice on ‘*Phase_1*’ that has been created, an emerging window called ‘*Phase*’ appears. In ‘*General*’, we write a name for the phase next to ‘*ID*’, in order to recognize it. In this case, we name is as ‘*Safety*’. Observe that, despite having changed the name, the software keeps ‘*Phase_1*’ next to the given name.

Still in ‘*General*’, in ‘*Calculation type*’, we select ‘*safety*’, which allows calculating the safety factor of the cutting in the selected phase, in this case, the Initial Phase (‘*Start from Phase*’, ‘*Initial Phase*’), as presented in Figure 1.18. We must pay

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

attention and select ‘Reset displacements to zero’, while the rest of the calculation options are not changed. We click on ‘OK’ to accept.

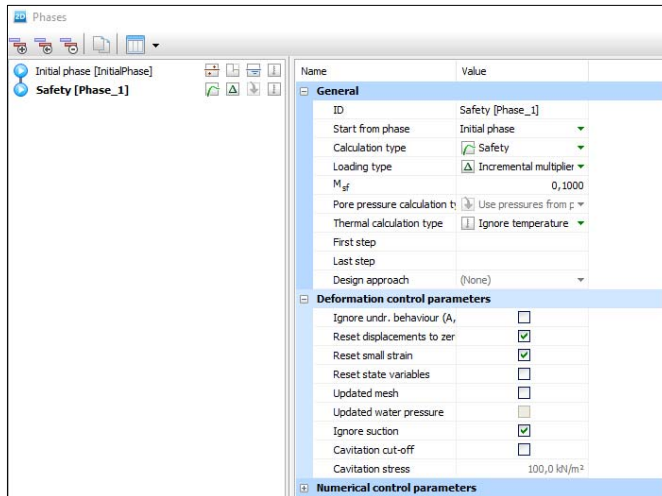


Figure 1. 18. Phase 1 ‘Safety’

Before starting to calculate it is possible to select those point that we considered interesting to obtain the advance graphical outputs in PLAXIS Output (in ‘*Staged construction*’ Calculation mode, ‘*Selecting points for curves*’). For example, we choose some points close to the possible slip surface.

It is also possible to calculate without introducing these points * and, once the slip surface is known, select the interesting ones. In this exercise we do as explained in this paragraph. Therefore, we click on ‘*Calculate*’ in the calculation mode ‘*Staged Construction*’ (Figure 1.19). A warning saying that no nodes have been chosen for the curves appears. We ignore it.

* **NOTE.** When the calculations are carried out without selecting nodes for advance graphical outputs, the results provided by PLAXIS Output are correct for both graphical representations shown in the workspace and information in tables. However, if we add nodes for obtaining these advance graphical outputs, the information that is obtained may be wrong in those graphics (although it is displayed correctly in the workspace and tables). For this reason, every time nodes for advance graphical output are added, we must re-calculate all the project phases.

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

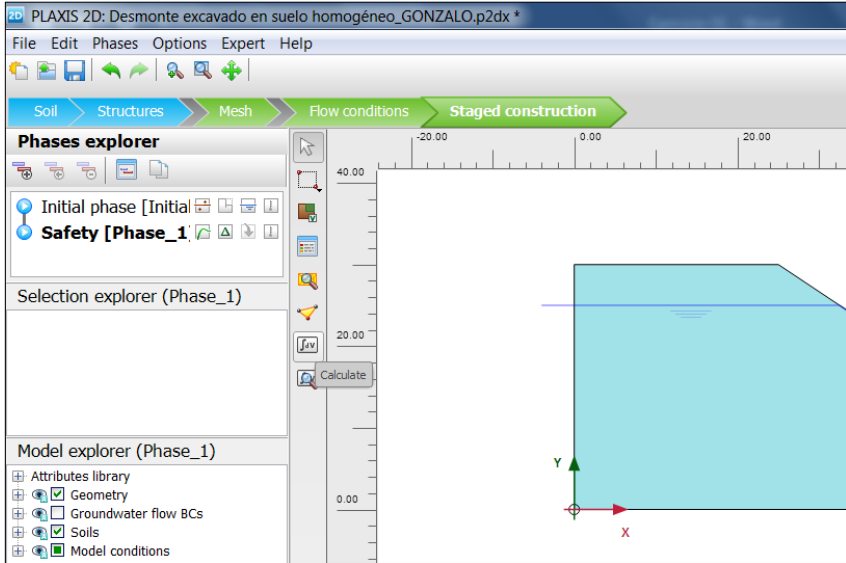


Figure 1. 19. 'Calculate' button in PLAXIS

If during the calculation no errors have appear, results can be displayed in PLAXIS Output clicking on 'View Calculation Results' button (Figure 1.20).

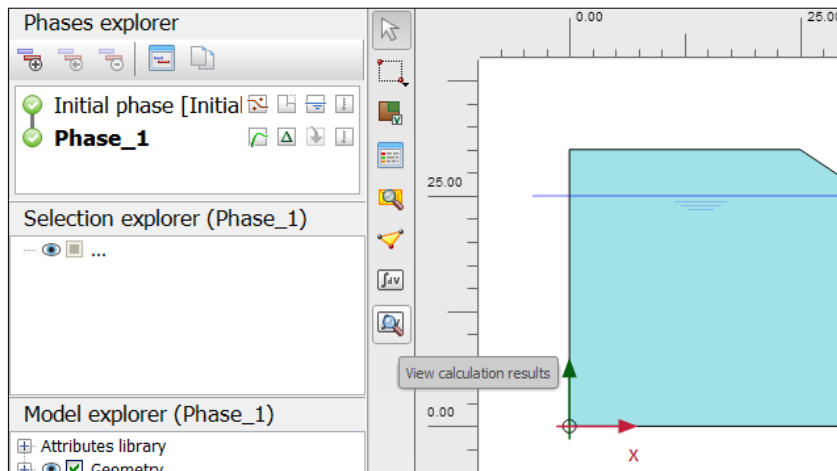


Figure 1. 20. Phases once they have been successfully calculated and 'View calculation results' button

1.3.6. Results in PLAXIS Output

In PLAXIS Output the deformed geometry of each of the calculated phases can be easily displayed. This graphic (*'Deformed mesh', $|u|$*) is the first output that appears in the workspace right after opening PLAXIS Output when clicking on *'View Calculation Results'* (Figures 1.21 and 1.22).

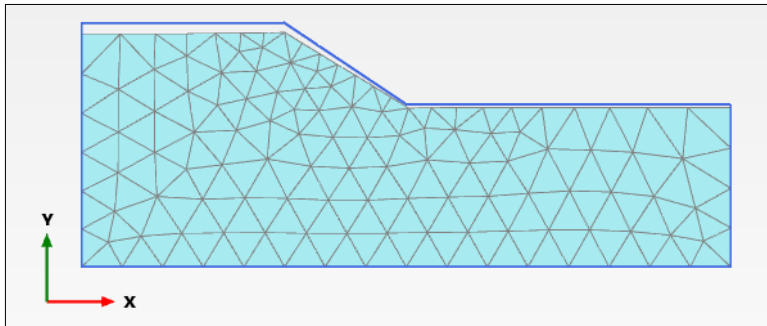


Figure 1. 21. Initial Phase deformed mesh

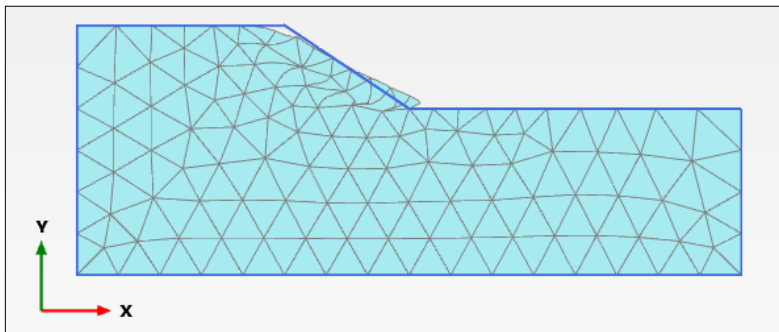


Figure 1. 22. 'Safety' Phase deformed mesh

In order to display the deformed mesh of a particular phase, we must select, in the bar above, that phase (Initial Phase or 'Safety' [Phase_1]). In this point, it is important to highlight that in *'Safety type'* phases, the numerical value of the displacements is not significant, since the computation that PLAXIS carries out in this type is focused on the calculation of the safety factor and, in any event, on the identification of the slip surface. However, it is not focused on the numerical value of the displacements. Talking about the displacements that represented in the Initial Phase (Figure 1.21), this are due to the procedure that has been chosen to obtain the

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

initial stresses in this phase, but they are not considered when calculating displacements in the following phases.

When the slip surface is required to be displayed, from the cutting deformation we can select ‘*Deformation*’, ‘*Total displacements*’, $|u|$, as shown in Figure 1.23.

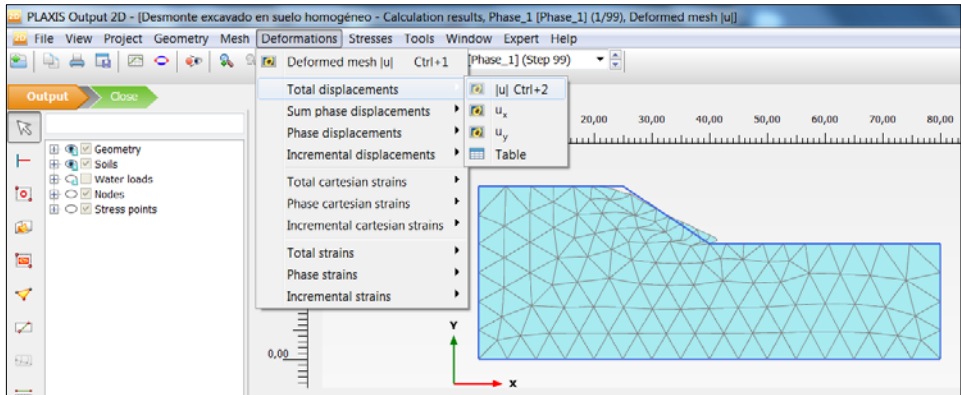


Figure 1. 23. Windows to select ‘*Total displacements*’ and obtain the possible slip surface

In Figure 1.24, the possible slip surface can be observed.

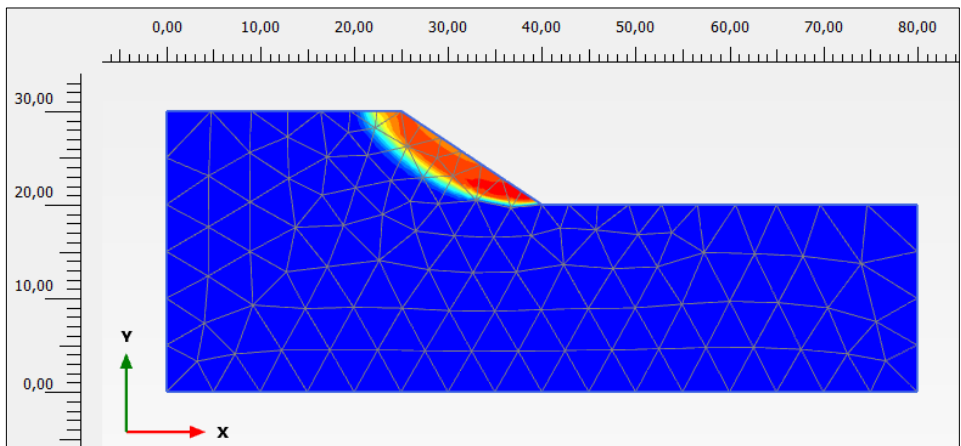


Figure 1. 24. Possible slip surface

There are different ways to obtain the global safety factor in PLAXIS Output, as presented below.

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

In the global safety factor analysis, PLAXIS employs ‘*phi-c strength reduction method*’, a routine that successively reduces ‘*tanφ*’ and ‘*c*’ (shear strength parameters of the soil), as well as its tensile strength, until failure occurs. ΣM_{sf} multiplier (which is the safety factor) is used to define the value of the shear strength parameters of the soil in a given analysis phase:

$$\Sigma M_{sf} = \frac{\tan\varphi}{\tan\varphi_{red}} = \frac{c}{c_{red}} = \frac{s_u}{s_{u,red}} = \frac{\text{tensile strength}}{\text{tensile strength}_{red}}$$

The easiest and most direct way to obtain the safety factor is with ‘*Calculation information*’ option, which can be found in ‘*Project*’ menu in the tool bar (Figure 1.25). In the emerging window, the exact safety factor value, ΣM_{sf} , is displayed (in this case, it is 1.133, Figure 1.26). For this, ‘*Safety*’ Phase must have been previously selected (drop-down menu right above workspace).

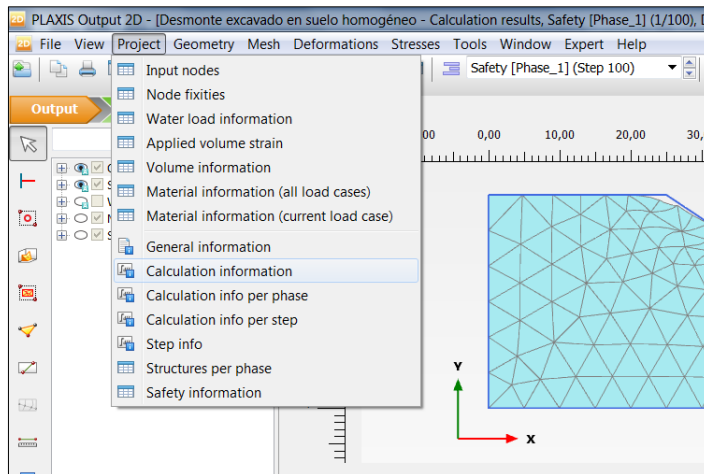


Figure 1. 25. ‘*Calculation information*’ selection to obtain the safety factor

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

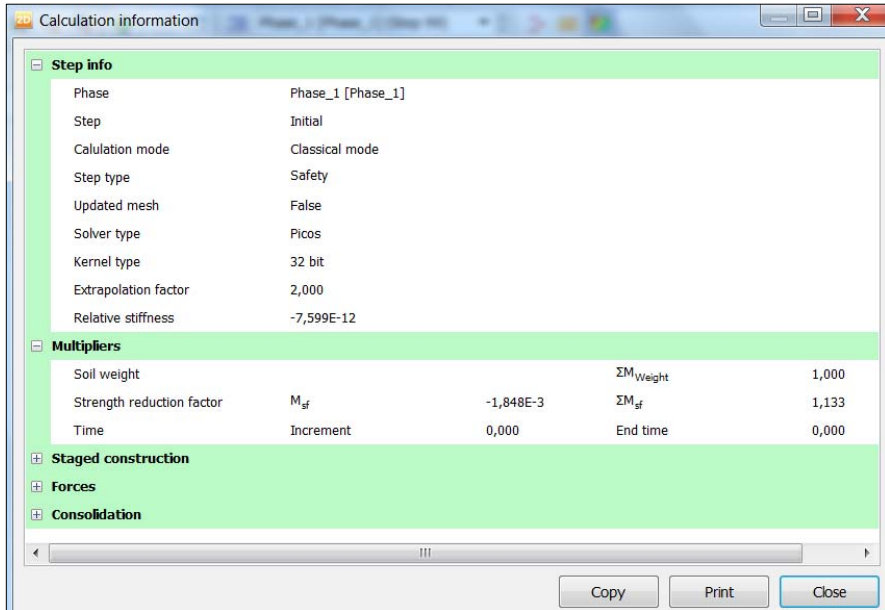


Figure 1. 26. Calculation information, where ΣM_{sf} is the safety factor

Another way to obtain the safety factor is plotting for any of the soil points any variable (for, example, displacements) versus the safety factor ΣM_{sf} :

However, for this it is necessary to select those points for which we want to obtain advance graphical outputs. For this, we go back to PLAXIS Input and in ‘*Staged construction*’ calculation mode, we choose ‘*Select points for curves*’ button (Figure 1.27). We can see that the program takes us back to PLAXIS Output.

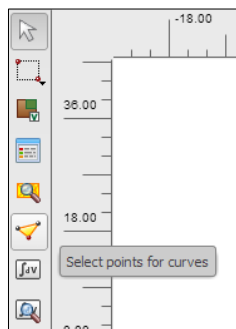


Figure 1. 27. ‘*Select points for curves*’ button

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

We select, for example, some points* that are close to the slip surface (see Figure 1.24) and a point on the foot of the slope (Figure 1.28). We select them by clicking on them in the workspace and, once we have finished the selection, we click on ‘Update’.

* **NOTE.** Two different kinds of points can be found: 1) ‘Nodes’ (in red), that are basically used when representing displacements; and 2) ‘Stress points’ (in purple), for essentially representing efforts and deformations.

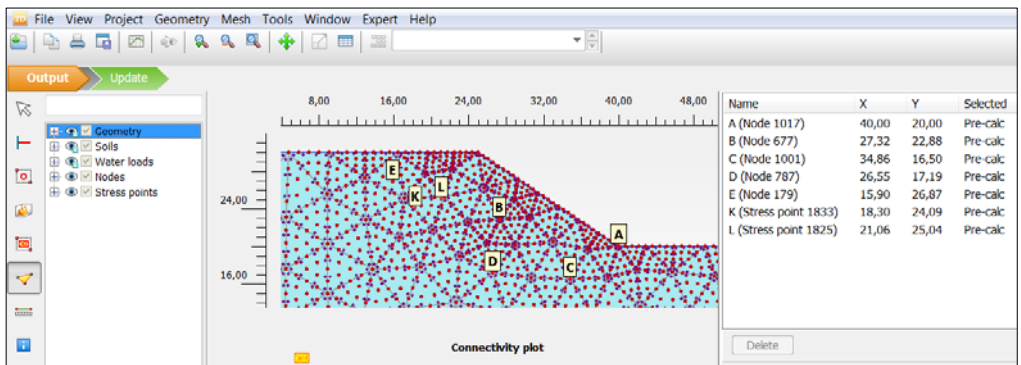


Figure 1. 28. Selected points for advanced graphical outputs

Now we re-calculate all the project phases: back in PLAXIS Input, we go to ‘Staged construction’ calculation mode and in the menu on the left, ‘Phase explorer’, with the right mouse button, we mark each phase for calculation (‘Mark for calculation’ option). When the calculation is done, in PLAXIS Output we click on ‘Curves manager’ button (Figure 1.29).

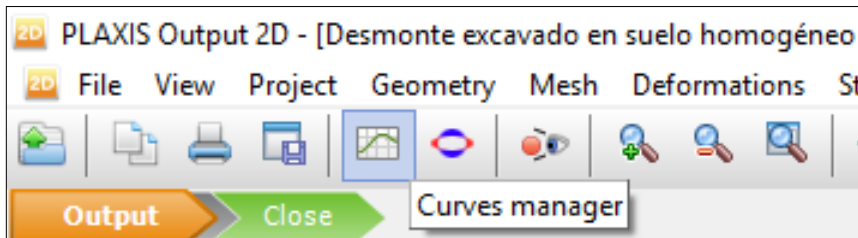


Figure 1. 29. ‘Curves manager’ button for creating advanced graphical outputs

Exercise 01. Long-term stability analysis of a cutting excavation in a homogeneous material

In ‘*Curves manager*’ window we mark ‘*Chart*’ tab and select ‘*New*’ (Figure 1.30) to create a new plot.

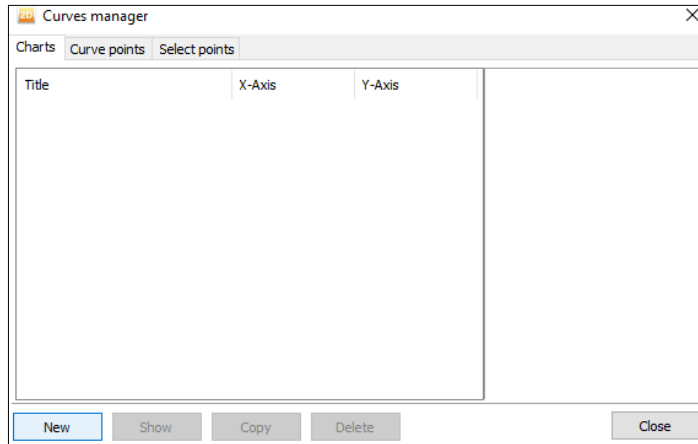


Figure 1. 30. Initial aspect of ‘*Curves manager*’ window

Figure 1.31 shows the parameters and options that must be chosen to plot, for point A, the total displacements versus the safety factors, so the relation between them can be observed (Figure 1.32).

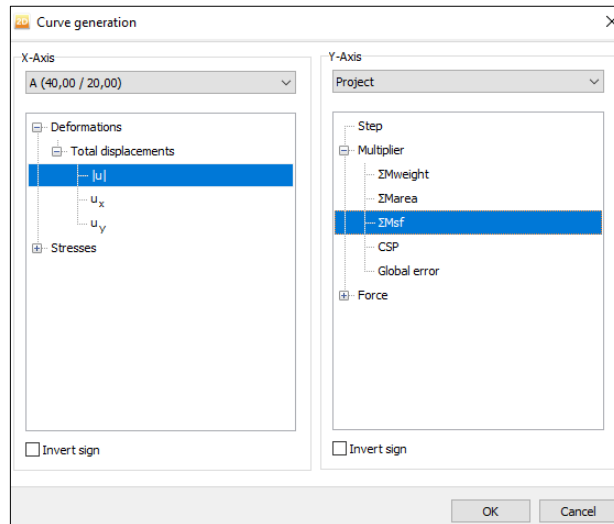


Figure 1. 31. Selection of the variables to plot in ‘*Curves manager*’

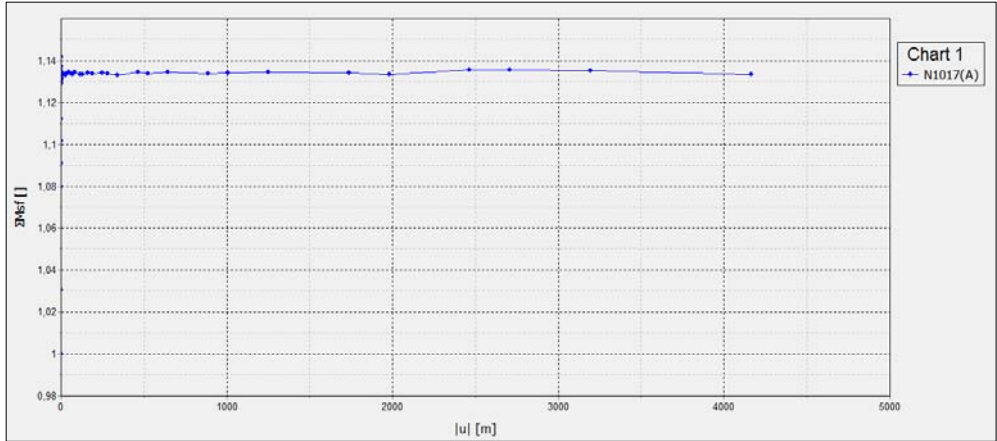


Figure 1. 32. Variation of the safety factor versus the total displacement (‘Safety’ calculation Phase) of point A

As we have previously commented, in ‘*Safety type*’ calculation phase, the numerical value of the displacements is not significant, since the computation carried out by PLAXIS when employing this mode only uses displacements to lead the process and indicate the possible failure mechanism. As observed in Figure 1.32, a safety factor of around 1.135 is rapidly obtained to keep constant from a hypothetical value of displacement. When closing this graphic, it is possible to save it and open it later in ‘*Curves manager*’.

Exercise 02

Stability analysis of an embankment on a clay layer

2.1 INTRODUCTION

On a thick clay layer, an embankment has been built, with a height of 5 meters and a crest length of 25 meters. The slope in both sides is 2H:1V. The aim is to obtain the global safety factor (embankment- foundation soil) when the phreatic level is at the foundation ground surface. Both geometrical and geotechnical parameters are presented in the following section.

2.2 GEOMETRICAL AND GEOTECHNICAL PARAMETERS

Figure 2.1 presents a sketch with the geometry of the problem.

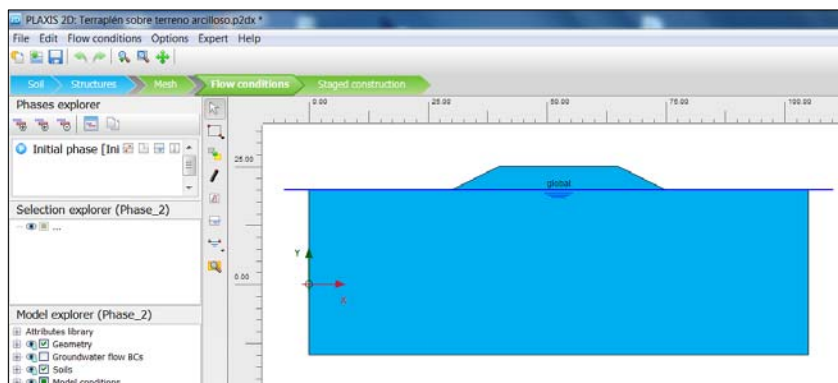


Figure 2. 1. Geometry of the problem

Exercise 02. Stability analysis of an embankment on a clay layer

Tables 2.1, 2.2 and 2.3 show the geotechnical parameters of the materials (foundation and embankment), as well as the geometrical ones for the embankment.

Geotechnical parameters of foundation soil

Foundation material	Clay
Constitutive model	Mohr-Coulomb
Material behaviour	Undrained (C)
Soil unit weight above phreatic level (γ_{unsat})	20 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	20 kN/m ³
Young's modulus (E')	14000 kPa
Poisson's ratio (ν')	0.495
Undrained shear strength ($S_{u,\text{ref}}$)	40 KPa

Table 2. 1. Geotechnical parameters of foundation soil

Geotechnical parameters of embankment

Embankment material	Embankment
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	20.7 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	23 kN/m ³
Young's modulus (E')	35000 kPa
Poisson's ratio (ν')	0.3
Effective soil cohesion (c')	15 kPa
Effective friction angle (ϕ')	28°
Dilatancy angle (ψ)	0°

Table 2. 2. Geotechnical parameters of embankment

Geometrical parameters

Embankment height	5 m
Embankment slope	2H:1V
Phreatic level	At the foundation ground surface

Table 2. 3. Geometrical parameters of embankment

2.3. RESOLUTION OF EMBANKMENT PROBLEM WITH PLAXIS 2D

2.3.1 Introduction of the parameters of the problem

We first name the new project in 'Project' tab in the emerging window 'Project properties'. In the bar title we write, for example: 'Embankment on a clay layer'.

In 'Model' tab, we check if 'Plain strain' option is selected for the model and we choose '15-Noded' for the elements.

In order to define the borders of the workspace, we use the values that appear in Table 2.4.

X _{min}	0	m
X _{máx}	105	m
Y _{min}	0	m
Y _{máx}	40	m

Table 2. 4. Coordinates for the project borders

We keep the default constants the program provides ('Constants' tab).

2.3.2 Soil definition (foundation and embankment)

We select 'Soil' geometry mode and click on 'Create borehole' button to create the first and only necessary borehole to define the foundation soil in this problem.

We mark point (0, 0) to define the borehole in 'Modify soil layers'. After that, we click on 'Add' and assign value 20 in 'Top' and -15 in 'Bottom'. A clay layer 35 meters thick is therefore considerate. As the phreatic level is located at the foundation ground surface, in 'Head' we write 20. Borehole data introduced in PLAXIS 2D appear in Figure 2.2.

Exercise 02. Stability analysis of an embankment on a clay layer

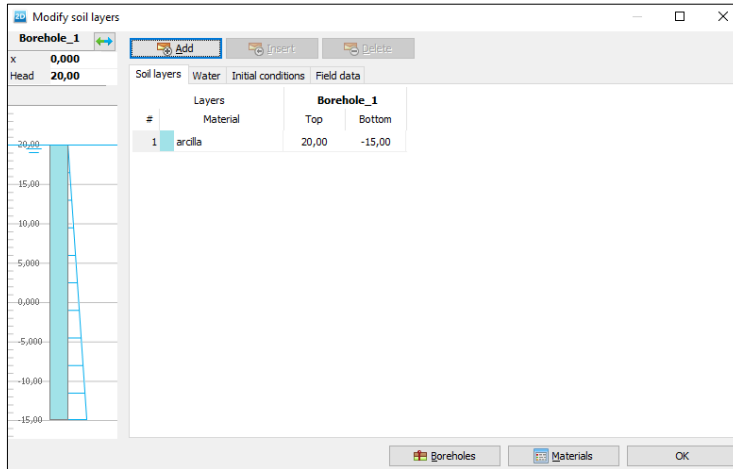


Figure 2. 2. First and only necessary borehole to define the foundation soil Geometry shown in Figure 2.3 appears.

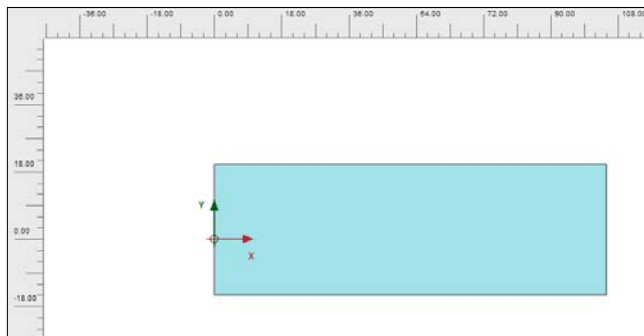


Figure 2. 3. Foundation soil geometry

Now we create the material to model the foundation soil. In order to define this material, in 'Soil' geometry model we click on 'Show materials' button. Keeping 'Set type' option in 'Soil and interfaces', we select 'New' and introduce the geotechnical parameters of the exercise for the clay (Table 2.1.) Figures 2.4 and 2.5 show the parameters once they have been introduced in the program. After creating this material, we can drag and drop from the material menu to the workspace, assigning to the foundation soil the material 'Clay'. We follow the same instructions for the definition of the embankment material (Table 2.2) and, when its geometry is defined, we will assign this new material.

Exercise 02. Stability analysis of an embankment on a clay layer

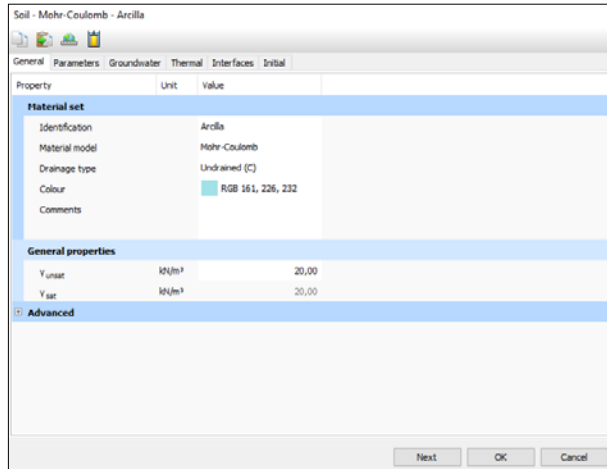


Figure 2. 4. General properties of foundation soil

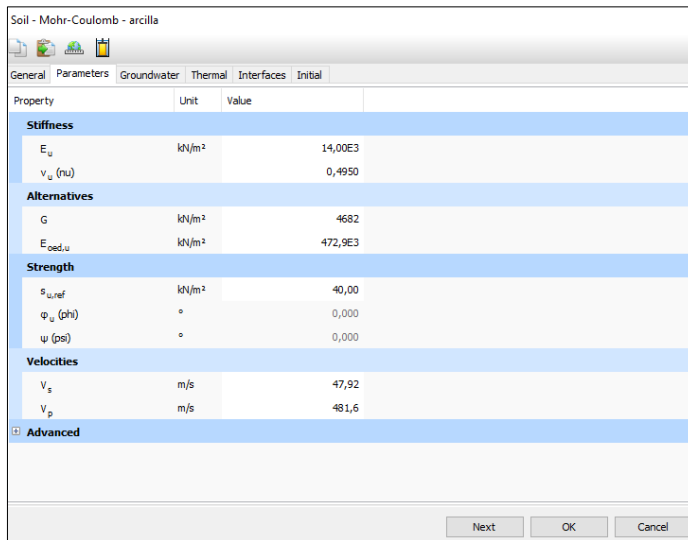


Figure 2. 5. Geotechnical parameters of foundation soil

2.3.3. Embankment geometry definition

In order to define the geometry of the embankment, we go to ‘Structures’ geometry mode and click on ‘Create soil polygon’ button (Figure 2.6).

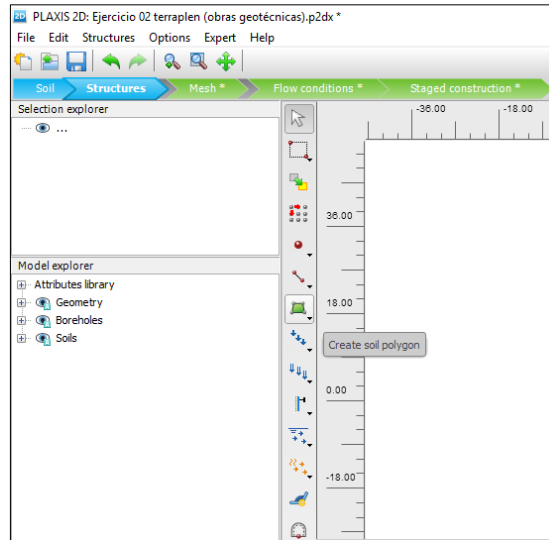


Figure 2. 6. ‘Create soil polygon’ button, in ‘Structures’ geometry mode

Now we create the embankment by clicking (on the workspace) on the following coordinates (Table 2.5):

Coordinates of the embankment contour points	
A	(30, 20)
B	(40, 25)
C	(65, 25)
D	(75, 20)

Table 2. 5. Coordinates of the embankment contour

The next step is to assign the material. In the workspace, we click on the drawn embankment with the right mouse button, marking ‘Soil’ option, then ‘Set material’ and finally we choose ‘Embankment’ material (Figure 2.7).

Exercise 02. Stability analysis of an embankment on a clay layer

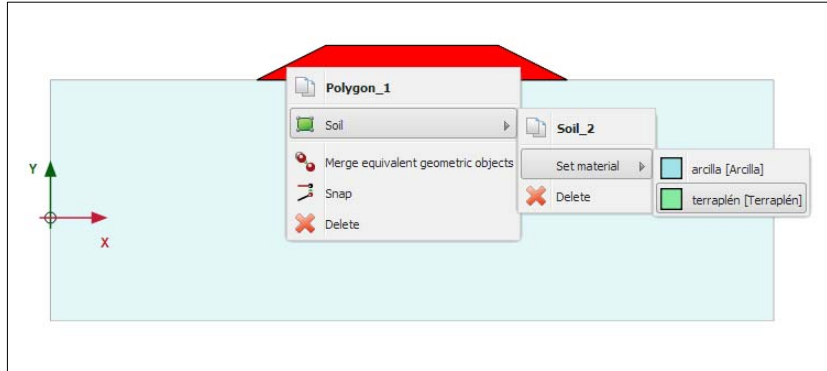


Figure 2. 7. Assignment of material to the created embankment polygon

2.3.4. Problem meshing

We select 'Mesh' calculation mode to mesh the soils, we click on 'Generate mesh' button and indicate a medium element distribution. We can display the generated mesh in PLAXIS Output by clicking on 'View mesh' button (Figure 2.8).

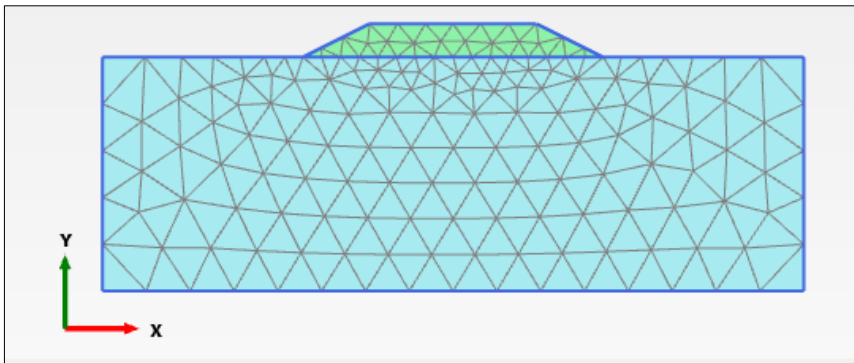


Figure 2. 8. Problem meshing in PLAXIS Output

2.3.5. Display of the phreatic level in the ground

If we select 'Flow conditions' calculation mode, we can display the phreatic level that has been created when defining soil in the first borehole (Figure 2.9).

Exercise 02. Stability analysis of an embankment on a clay layer

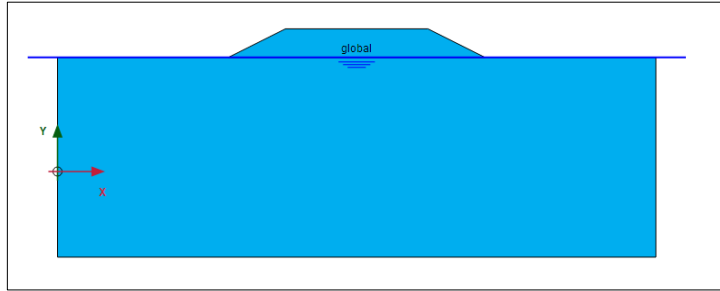


Figure 2. 9. Location of the phreatic level

2.3.6. Project phases

We define 3 different phases in our project:

- Initial Phase. In this phase we only considered the foundation soil. Therefore, we deactivate the embankment by clicking on the embankment with the right mouse button and selecting 'deactivate' (Figure 2.10). For this, we must have previously selected 'Staged construction' calculation mode. Once the embankment has been deactivated, the workspace looks as in Figure 2.11 for this Initial Phase.

For this exercise, unlike for the previous one, we keep the default calculation type, 'K0 procedure' (we go to 'Phase explorer' menu on the left and double click on 'Initial Phase'), since it is a horizontal soil layer.

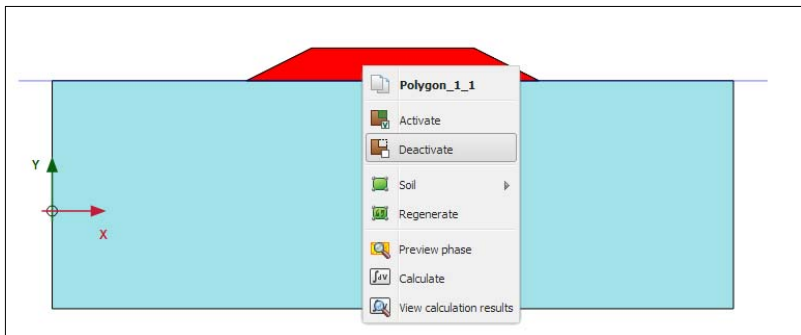


Figure 2. 10. Embankment deactivation for the Initial Phase

Exercise 02. Stability analysis of an embankment on a clay layer

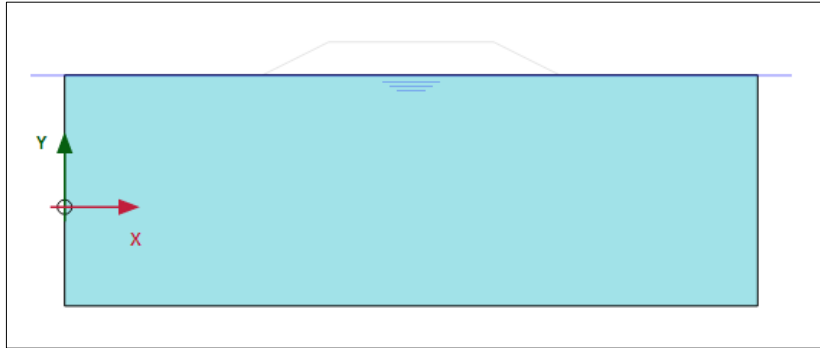


Figure 2. 11. Aspect of the workspace for the Initial Phase

- Phase 1 (Embankment). We create this new phase and activate the embankment (Figure 2.12).

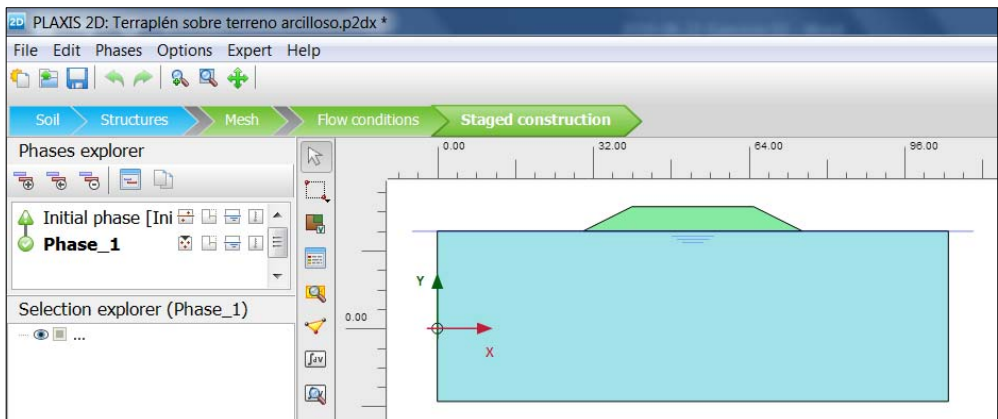


Figure 2. 12. Aspect of the workspace for Phase 1 (Embankment)

- Phase 2 (Safety). This phase is used to calculate the safety factor of the embankment on the foundation soil, as well as for graphically obtain the slip surface. Figure 2.13 shows how this new phase is defined.

Exercise 02. Stability analysis of an embankment on a clay layer

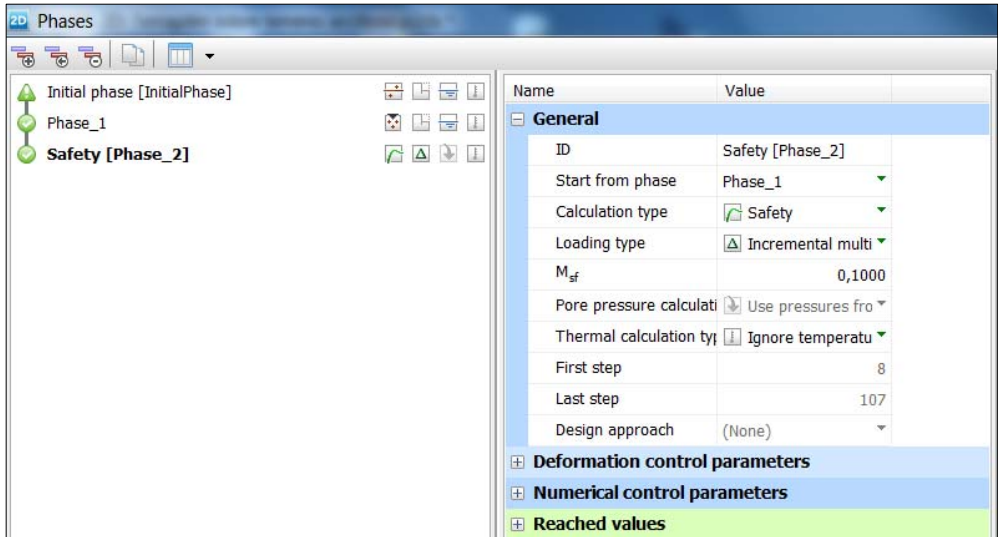


Figure 2. 13. Definition of Phase 2 (Safety)

Once all the phases have been defined, we click on the ‘*Calculate*’ button. If no node has been chosen yet to obtain the advanced graphical output later, a warning telling so will appear, although we can ignore it. Whether during the calculations PLAXIS does not find any mistake, we can display the results in PLAXIS Output, selecting ‘*View calculation results*’ button.

2.3.7. Results in PLAXIS Output

After the calculations, PLAXIS Output first displays the deformed mesh of the problem, which is available for each the defined phases (Figure 2.14 and 2.15).

Exercise 02. Stability analysis of an embankment on a clay layer

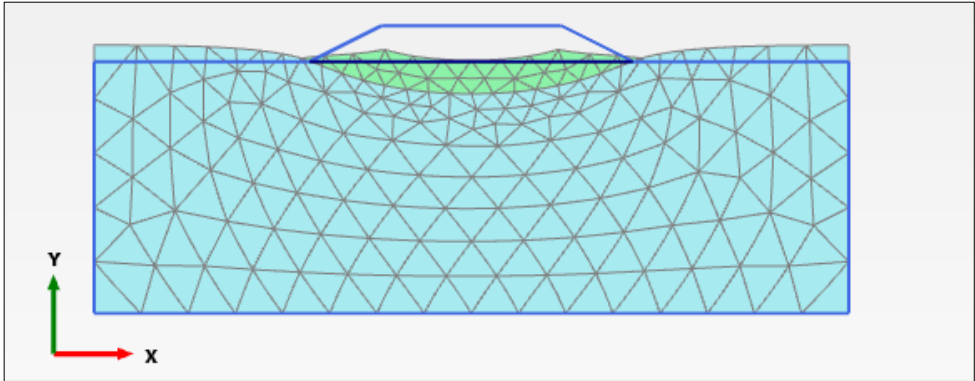


Figure 2. 14. Deformed mesh in Phase 1 (Embankment)

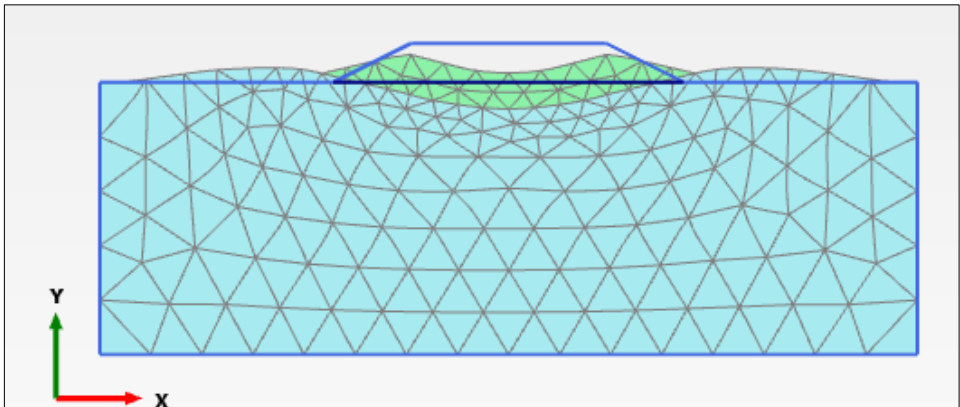


Figure 2. 15. Deformed mesh in Phase 2 (Safety)

In order to obtain the slip surface from the embankment deformations, we select 'Deformations', 'Total displacements', $|u|$. Total displacements, $|u|$, for Safety Phase are presented in Figure 2.16.

Exercise 02. Stability analysis of an embankment on a clay layer

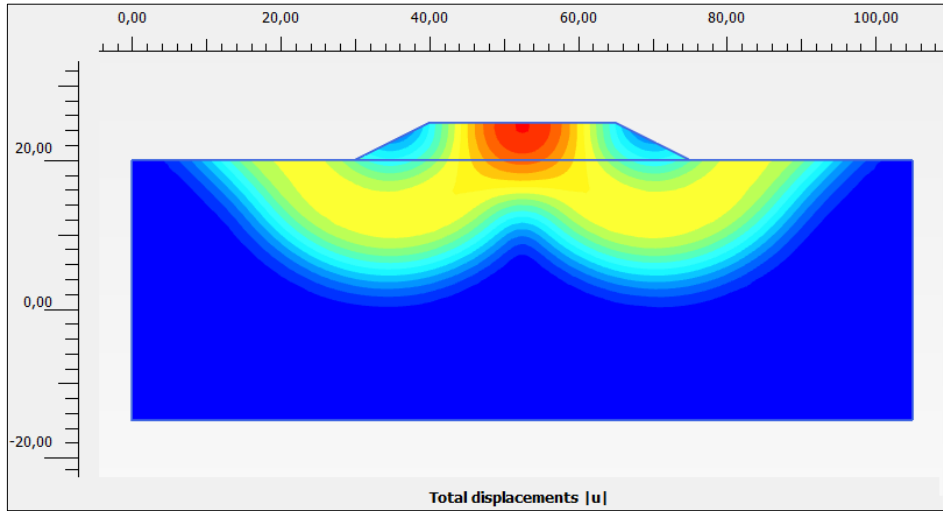


Figure 2. 16. Total displacements, $|u|$, representation (Safety Phase)

In this case, the slip surface is mainly located in the foundation soil, very deep. If we want to identify it more accurately, we must consider that the failure occurs in those points of the soil with higher deformation and, according to Figure 2.16, they are found in the yellow zone. Therefore, we can modify the legend configuration, restricting the represented displacements to a given range of values (clicking with the right mouse button on the graphic and selection 'Legend settings...' option), so the slip surface is displayed in a clearer way, Figure 2.17.

Exercise 02. Stability analysis of an embankment on a clay layer

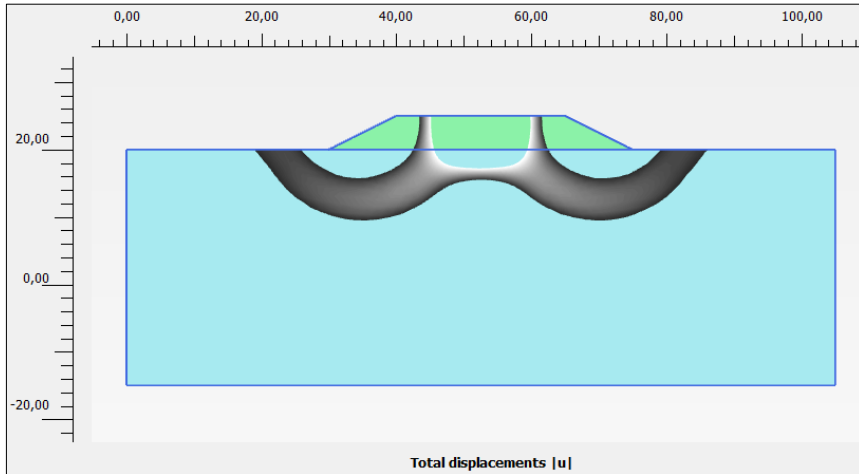


Figure 2. 17. Slip surface (Safety Phase)

Finally, we are going to obtain the safety factor. As commented in exercise 1, there are different ways to calculate it. As a first option, once we have selected Safety Phase in the drop-down menu that allows changing phase (Figure 2.18), we select in PLAXIS Output tool bar ‘*Calculation information*’ option in ‘*Project*’ tab. In the emerging window, we can the exact safety factor value, ΣM_{sf} (Figure 2.19), which, in this case, is 2.04

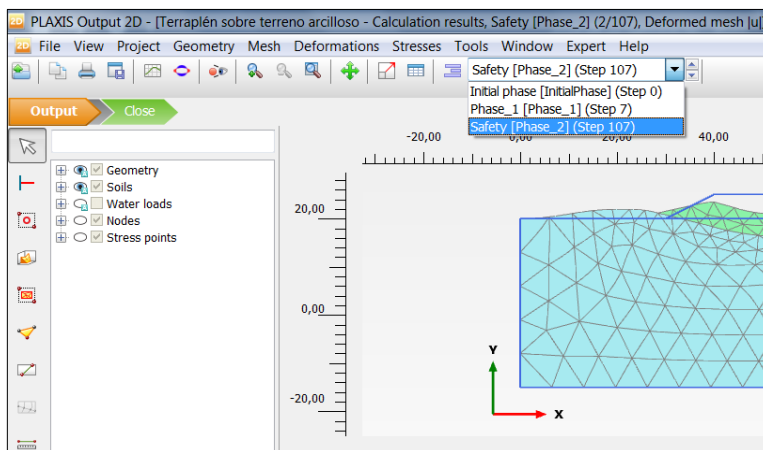


Figure 2. 18. Drop-down menu for phase selection

Exercise 02. Stability analysis of an embankment on a clay layer

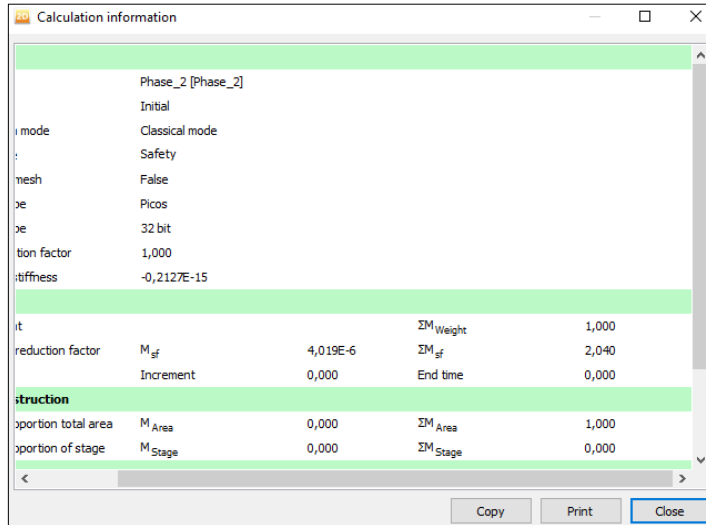


Figure 2. 19. Safety factor value

It is also possible to obtain the safety factor value by plotting in a foundation or embankment point any of the output variable versus safety factor, ΣM_{sf} . For this, it is necessary to select at least one point of the mesh to plot the evolution of the variables. Back to PLAXIS Input, and in 'Staged construction' calculation mode, we select 'Select points for curves' button.

In this case, the selected points can be the ones shown in Figures 2.20 and 2.21. Points can be selected clicking on those points.

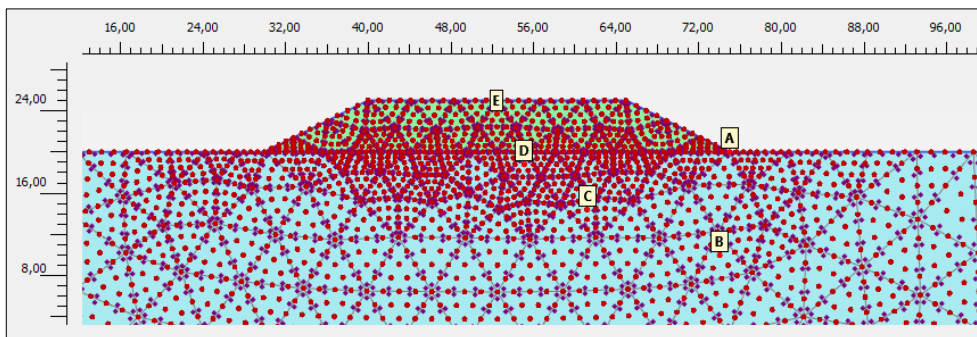


Figure 2. 20. Selected points of soil and embankment

Exercise 02. Stability analysis of an embankment on a clay layer

Name	X	Y	Selected
A (Node 539)	75,00	20,00	Pre-calc
B (Node 734)	74,07	10,01	Pre-calc
C (Node 1045)	61,22	14,41	Pre-calc
D (Node 1062)	55,15	18,79	Pre-calc
E (Node 908)	52,50	23,71	Pre-calc

Figure 2. 21. Coordinates of the selected points

Now we have to re-calculate all project phases, because if we do not do so, the advanced graphical output will show wrong values. Still in ‘*Staged construction*’ calculation mode, in ‘*Phase explorer*’ menu on the left, we mark each phase (‘*Mark for calculation*’) with the right mouse button, so they are ready to be calculated again.

Once calculations are over, we click on ‘*Curves manager*’ button in PLAXIS Output. In ‘*Charts*’ tab, we select ‘*New*’.

In Figure 2.22, we can observe the parameters that must be chosen in point A to plot total displacements, $|u|$, versus safety factor. As visible in Figure 2.23, with this procedure we obtain a similar value of safety factor. At this point, we must remember that the displacement that is represented in a safety phase is not significant.

When closing the graphic, it is possible to save it in order to open it again later in ‘*Curves manager*’.

Exercise 02. Stability analysis of an embankment on a clay layer

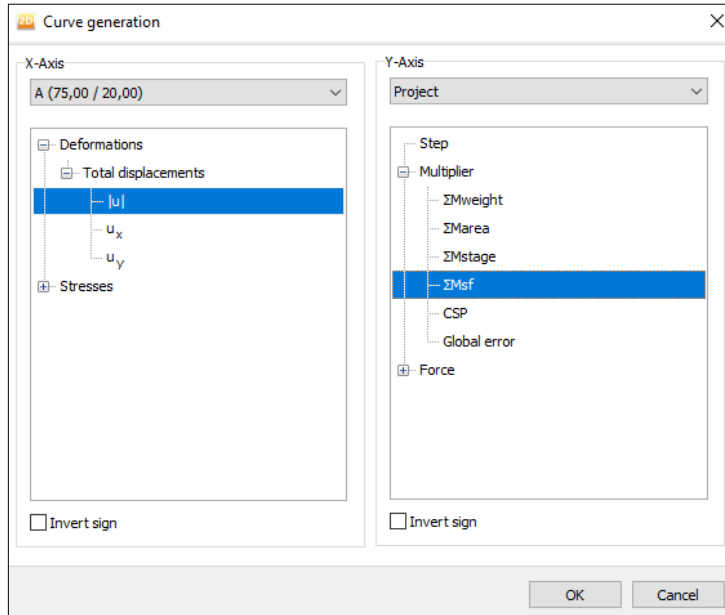


Figure 2. 22. Selection of the parameters to plot

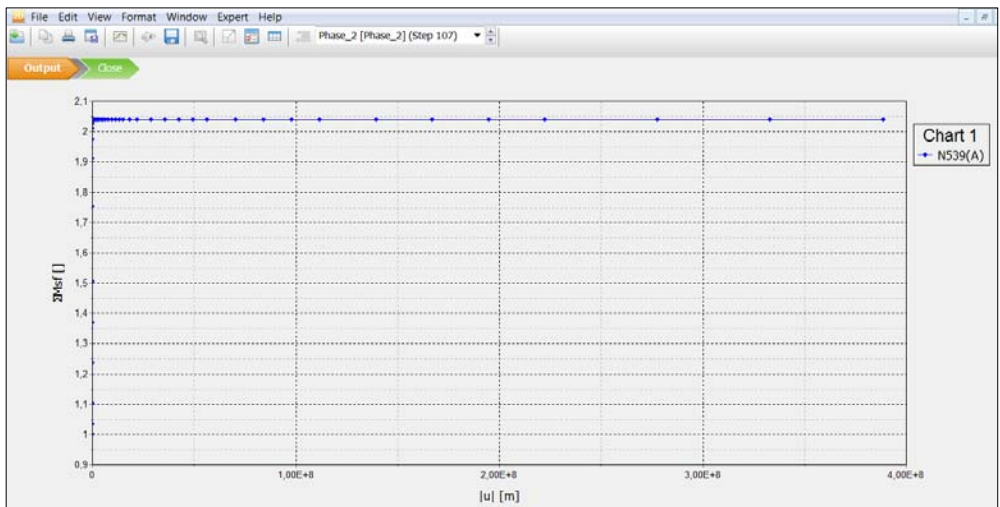


Figure 2. 23. Safety factor variation versus total displacement ('Safety' calculation Phase) in point A

Exercise 03

Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

3.1 INTRODUCTION AND DEFINITION OF THE PROBLEM

The aim of the exercise is to analyse the consolidation and study the stability of an embankment 10 meters high, 30 meters long in crest and 3H:2V slopes that is supposed to be built on a homogeneous layer 30 meters thick. The embankment is built with the same material, since this has been obtained from a cutting close to the area, but with geotechnical characteristics that are slightly different.

Four different cases are presented.

- First, the construction of the embankment in seven phases:
 - o 1st phase. Construction of the first 4 meters of embankment during a four-day period.
 - o 2nd phase. 10 days elapse, allowing soil consolidation.
 - o 3rd phase. The next 3 meters of embankment are built, during a 3-day period.
 - o 4th phase. 10 days elapse, allowing soil consolidation.
 - o 5th phase. The last 3 meters of embankment are built, during a 3-day period.
 - o 6th phase. 10 days elapse, allowing soil consolidation.
 - o 7th phase. A 23 kN/m² traffic load* is applied.
- Second, the embankment is also built in seven phases, but in the following way:
 - o 1st phase. Construction of the first 4 meters of embankment during a 4-day period.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

- 2nd phase. The consolidation of the soil is permitted until 70% degree of consolidation is reached.
- 3rd phase. The next 3 meters of embankment are built, during a 3-day period.
- 4th phase. The consolidation of the soil is permitted until 80% degree of consolidation is reached.
- 5th phase. The last 3 meters of embankment are built, during a 3-day period.
- 6th phase. The consolidation of the soil is permitted until 90% degree of consolidation is reached.
- 7th phase. A 23 kN/m² traffic load* is applied.

* For every case, for the study of traffic load application, we consider the two possible scenarios: the load application only on one half of the embankment (right side) and the load application on the whole embankment length.

3.2 PROBLEM PARAMETERS

3.2.1 Problem geometry

Figure 3.1 shows the general aspect of the problem geometry.

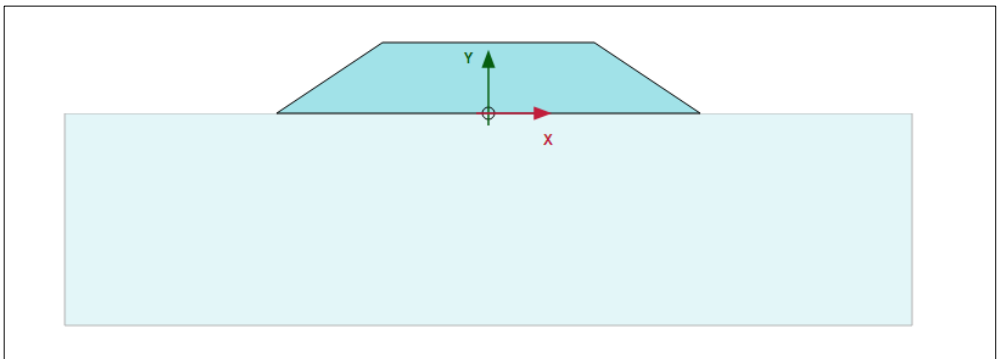


Figure 3. 1. General geometry of the problem

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

3.2.2 Soil parameters

Tables 3.1 and 3.2 show the properties for both foundation and embankment soil.

Foundation soil

Material	Natural clay (Previous soil)
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	20.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	22 kN/m ³
Young's modulus (E')	12500 kPa
Poisson's ratio (ν')	0.35
Effective soil cohesion (c')	7 kPa
Effective friction angle (ϕ')	32.5°
Dilatancy angle (ψ)	0°
Horizontal hydraulic conductivity (K_x)	0.02 m/day
Vertical hydraulic conductivity (K_y)	0.02 m/day

Table 3. 1. Geotechnical parameters of natural soil

Geotechnical parameters (embankment material)

Material	Compacted clay (embankment)
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	20.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	22 kN/m ³
Young's modulus (E')	28000 kPa
Poisson's ratio (ν')	0.35
Effective soil cohesion (c')	7 kPa
Effective friction angle (ϕ')	32.5°
Dilatancy angle (ψ)	0°
Horizontal hydraulic conductivity (K_x)	0.012 m/day
Vertical hydraulic conductivity (K_y)	0.012 m/day

Embankment geometrical parameters

Embankment height	10 m
Slope	3H:2V
Phreatic level	2 meters below natural soil

Table 3. 2. Geotechnical and geometrical parameters of embankment soil

3.3 RESOLUTION OF THE CONSOLIDATION PROBLEM WITH PLAXIS 2D

3.3.1 Introduction of the project parameters

We start PLAXIS 2D (PLAXIS Input) and choose ‘*Start a new project*’ option in ‘*Quick select*’ emerging window. In ‘*Project*’ tab we name the problem ‘*Consolidation and stability analysis of a highway embankment*’.

In ‘*Model*’ tab we choose ‘*Plain strain*’ option in model and ‘*15-Noded*’ in elements. We keep the default units, as presented in Figure 3.2. In order to define the project borders, we write the values in Figure 3.2. We keep the default constants (‘*Constants*’ tab).

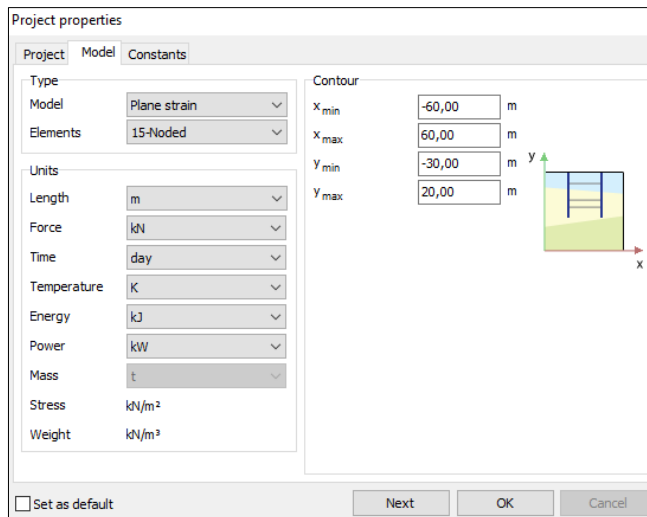


Figure 3. 2. Model type, units and project borders

3.3.2 Soil definition

In order to define the homogeneous layer on which the embankment is built, we start ‘*Soil*’ geometry mode. We click on ‘*Create borehole*’ button to define the foundation soil.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

We click on coordinate $(-60,0)$ and in ‘*Modify soil layers*’ emerging window we select ‘*Add*’ (with the pull-down marking ‘*Soil and interface*’ option). In ‘*Top*’ we write 0 and in ‘*Bottom*’, -30. Since the phreatic level is located 2 meters below the initial ground surface, we write -2 in ‘*Head*’. Figure 3.3 shows the borehole data. In this way, we create the geometry of the homogeneous foundation soil (Figure 3.4).

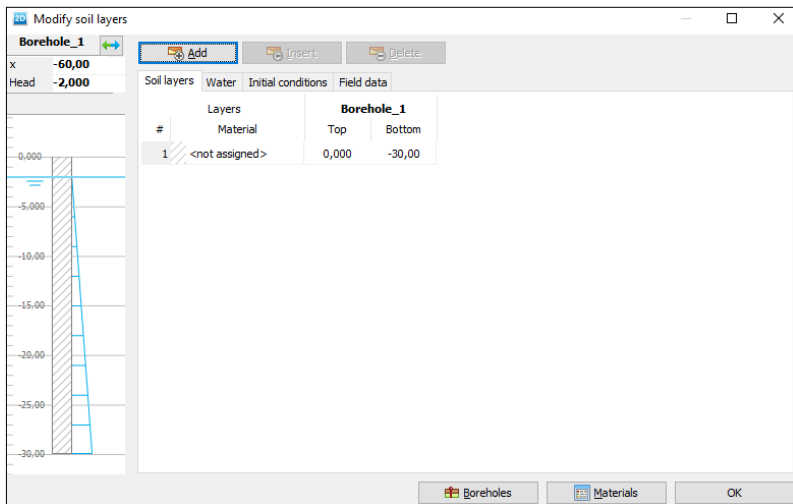


Figure 3. 3. Borehole for the foundation soil

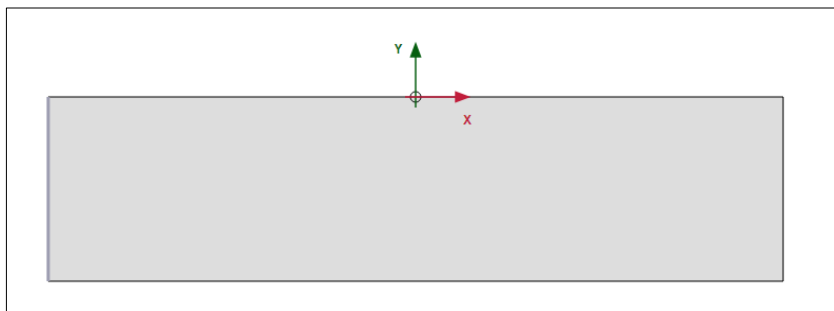


Figure 3. 4. Geometry of the homogeneous foundation soil

Now both materials, foundation soil and embankment, must be created by clicking on ‘*Show materials*’ button. We select ‘*New*’ and create the first material (‘*Previous soil*’) from data in Table 3.1. Then, clicking again on ‘*New*’, we create the second

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

material ('Embankment') with the geotechnical parameters in Table 3.2 (Figure 3.5, 3.6 and 3.7).

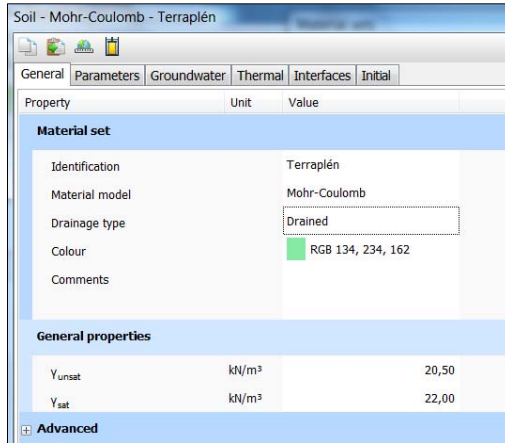


Figure 3. 5. General properties of embankment material

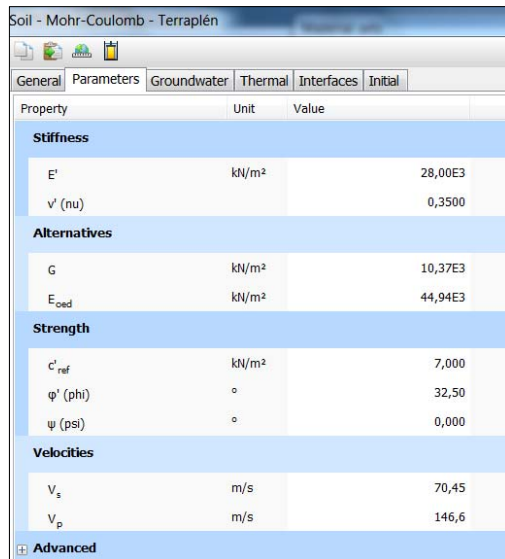


Figure 3. 6. Resistance parameters of embankment material

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

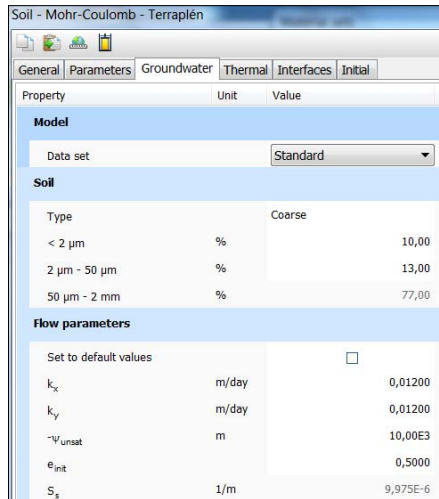


Figure 3. 7. Groundwater parameters of embankment material

We click with the right mouse button on the generated ground (Figure 3.4) and select (in 'Soil') 'Set material' and 'Previous soil' value.

3.3.3. Definition of the embankment geometry

In order to define the embankment, we go to 'Structures' geometry mode and select 'Create soil polygon' button.

Now, we can create the embankment. For this we introduce with the mouse in the workspace the coordinates in Table 3.3.

Coordinates of the embankment border points

A (-30, 0)
B (-15, 10)
C (15, 10)
D (30, 0)

Table 3. 3. Coordinates of the embankment borders

The next step is to assign to the created polygon the material that was generated in the previous point. We select 'Set material' and choose 'Embankment' material. Figure 3.1 shows the geometry of the embankment on the natural soil.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

As the embankment has been created to be divided into three construction phases (instead of been built in one phase), two lines must be created to separate the different layers.

In ‘Structures’ geometry mode, we click on ‘Create line’ button (Figure 3.8) and select the coordinates for each of them from Table 3.4. For this, we must write 2 in ‘Number of snap intervals’ option (in ‘snapping options’ menu, which is right below the workspace).

Line	First point coordinates	Second point coordinates
A	(-24, 4)	(24, 4)
B	(-19,5; 7)	(19,5; 7)

Table 3. 4. Coordinates for the lines that divide the embankment in three layers

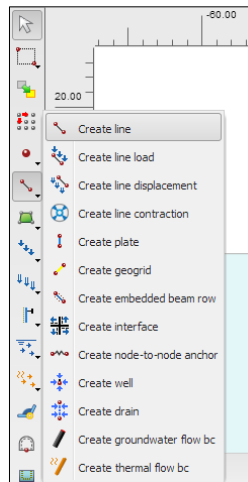


Figure 3. 8. ‘Create line button, inside ‘Structure’ geometry mode

3.3.4. Loads definition

Still in ‘Structure’ geometry mode, we click on ‘Create load’ button and select ‘Create line load’ (Figure 3.9).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

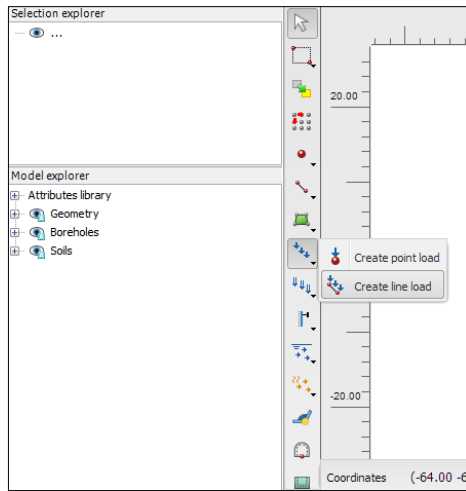


Figure 3. 9. 'Create line load' button, inside 'Structure' geometry mode

We create two symmetrical and identical loads of 23 kN/m^2 , each of them applied on a length of 15 meters. The load on the right is applied on coordinates (0, 10) to (15, 10), Figure 3.10. The one on the left goes from (-15, 10) to (0, 10), Figure 3.11.

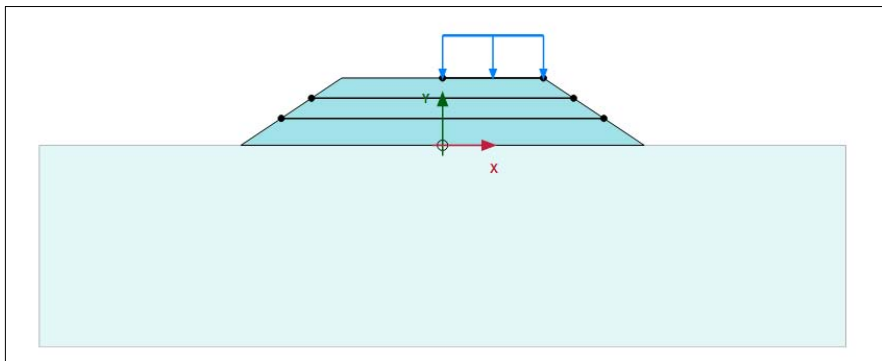


Figure 3. 10. Applied load on the right side of the embankment

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

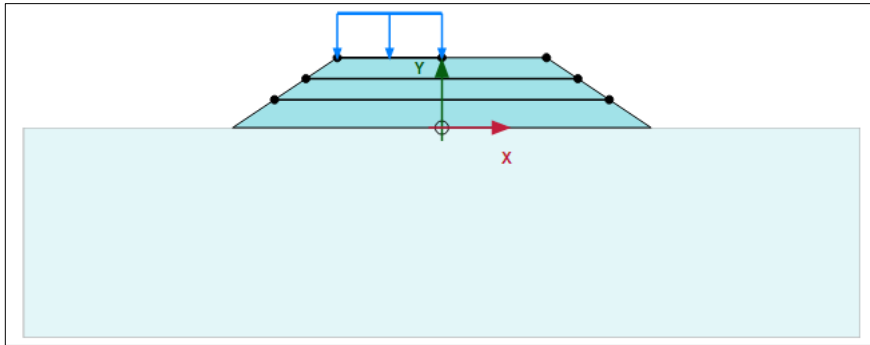


Figure 3. 11. Applied load on the left side of the embankment

Once the application line for both lines has been drawn (Figure 3.12), we define their value. For this, we select the load we want to give value to, go to 'Selection explorer' menu on the left and right -23 in $q_{y,start,ref}$ (the type of load is 'LineLoad'), Figure 3.13.

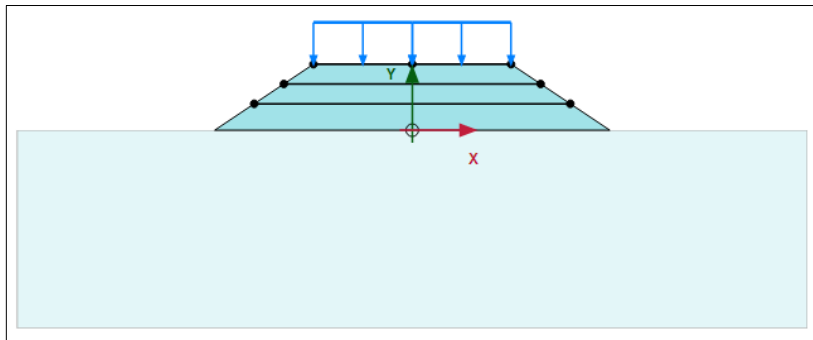


Figure 3. 12. Applied load on the whole embankment length

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

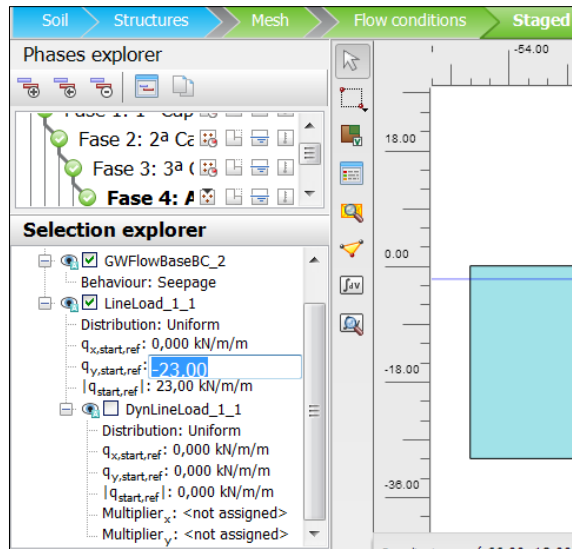


Figure 3. 13. Load value assignment in ‘Selection explorer’

3.3.5. Problem meshing

We select ‘Mesh’ calculation mode and click on ‘Generate mesh’ button, indicating medium element distribution. We can display the generated mesh in PLAXIS Output selecting ‘View mesh’ button (Figure 3.14).

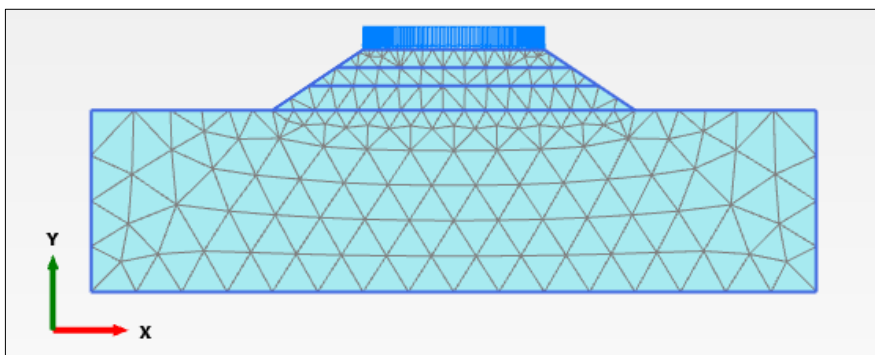


Figure 3. 14. Problem meshing

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

3.3.6. Display of the phreatic level in the ground

Selecting ‘*Flow conditions*’ calculation mode we can display the phreatic level that has been creating when defining the soil in the borehole (Figure 3.15).

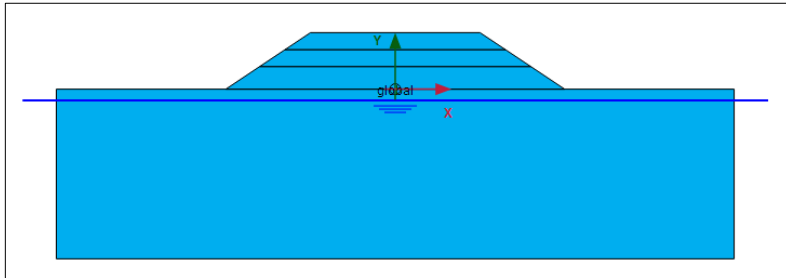


Figure 3. 15. Location of the phreatic level

3.3.7. Point selection for advanced graphical outputs

The selected points for plotting advanced graphical outputs are shown in Figures 3.16 and 3.17.

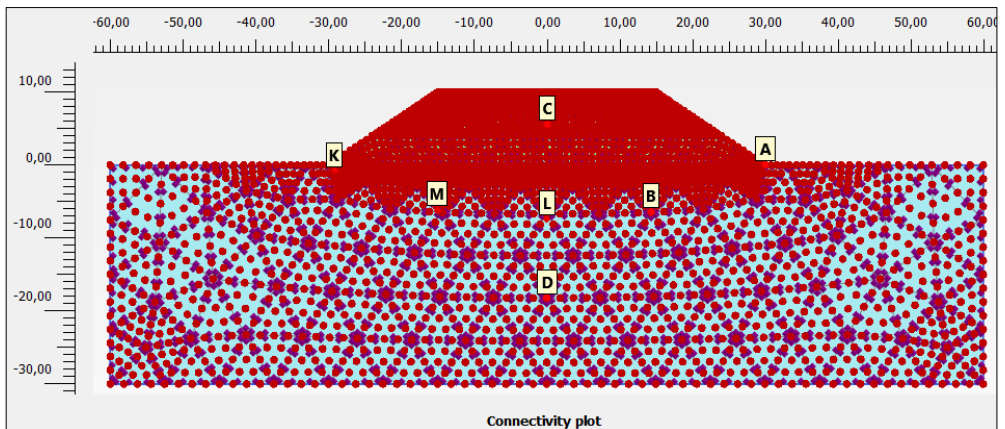


Figure 3. 16. Selected points in foundation and embankment for advanced graphical outputs

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

Name	X	Y	Selected
A (Node 300)	30,00	0,00	Pre-calc
B (Node 1245)	14,34	-6,47	Pre-calc
C (Node 1010)	0,04	5,50	Pre-calc
D (Node 1819)	0,00	-18,22	Pre-calc
K (Stress point 3440)	-29,17	-0,80	Pre-calc
L (Stress point 2701)	0,00	-7,40	Pre-calc
M (Stress point 2943)	-15,01	-6,21	Pre-calc

Figure 3. 17. Selected point coordinates in foundation and embankment for advanced graphical outputs

3.3.8. Project phase definition for the consolidation study

NOTE: It is recommendable to save the project with a clear name, since now we are defining the different phases involved in each of the exercise cases. Moreover, each of the cases should also be saved as an independent file once the phases have been defined and the calculations have been carried out).

First case: a) Embankment construction in seven phases employing consolidation times. The load is applied on the right side of the embankment.

8 phases must be defined:

- Initial Phase: just foundation soil.
- Phase 1: a 4-meter layer of embankment is built in a 4-day period.
- Phase 2: the consolidation of the soil due to the embankment load is allowed for 10 days.
- Phase 3: the next 3-meter layer of embankment is built in a 3-day period.
- Phase 4: the consolidation of the soil due to the embankment load is allowed for 10 days.
- Phase 5: the last 3-meter layer of embankment is built in a 3-day period.
- Phase 6: the consolidation of the soil due to the embankment load is allowed for 10 days.
- Phase 7: a 23 kN/m² traffic load is applied on the right side of the embankment.

In the Initial Phase we must deactivate the three embankment layers and the traffic load (just activate the foundation soil). We check that in 'Phase' window, the selected calculation type is 'K0 procedure', Figure 3.18.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

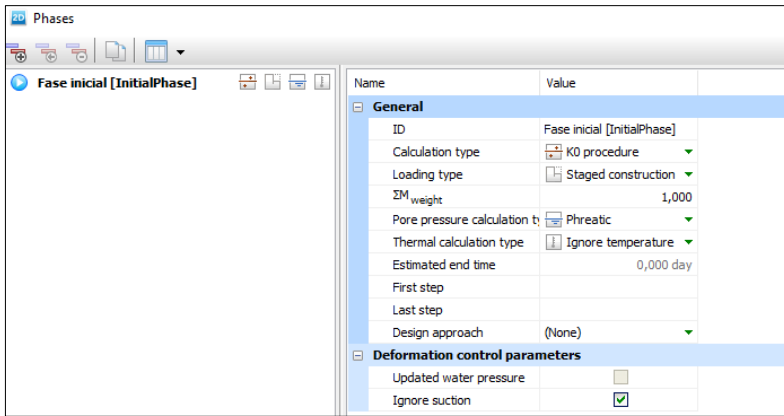


Figure 3. 18. Initial Phase, which corresponds to foundation soil

In Phase 1 we activate the first embankment layer and select ‘Consolidation’ calculation type. In the same window we must indicate a time interval of 4 days (Figure 3.19).

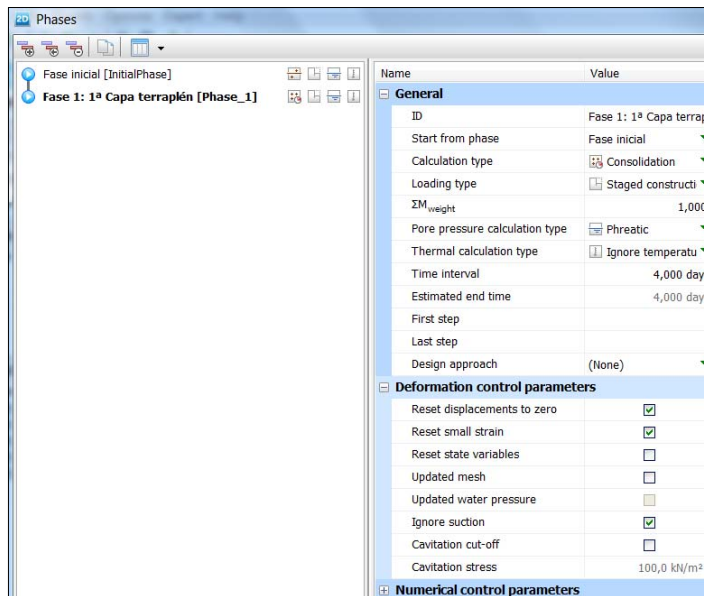


Figure 3. 19. Phase 1: first embankment layer construction (4 meters)

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

In Phase 2, we do not activate or deactivate anything. As calculation type, we select ‘*Consolidation*’ and as time interval, we write 10 days.

In Phase 3, we activate the second embankment layer and choose ‘*consolidation*’ as calculation type. In ‘*Phase*’ window we must write a 3-day time interval (Figure 3.20).

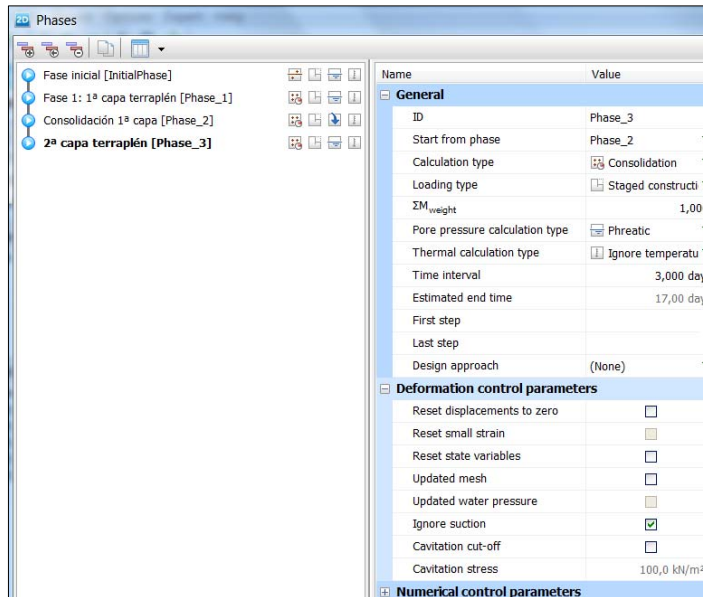


Figure 3. 20. Phase 3: second layer embankment construction (3 meters)

Phase 4 is very similar to Phase 2.

In Phase 5, we activate the third embankment layer, change the calculation type to ‘*consolidation*’ and indicate a time interval of 3 days (Figure 3.21).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

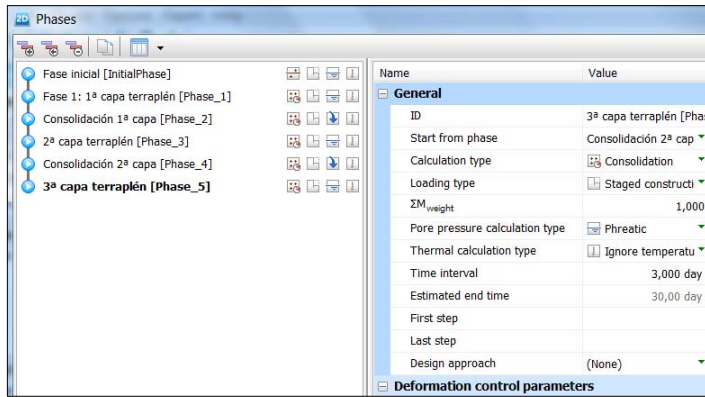


Figure 3. 21. Phase 5: third embankment layer construction (3 meters)

Phase 6 is again very similar to Phases 2 and 4.

In Phase 7, we activate the traffic load only on the right side of the embankment and we change the calculation type to 'Plastic' (Figure 3.22).

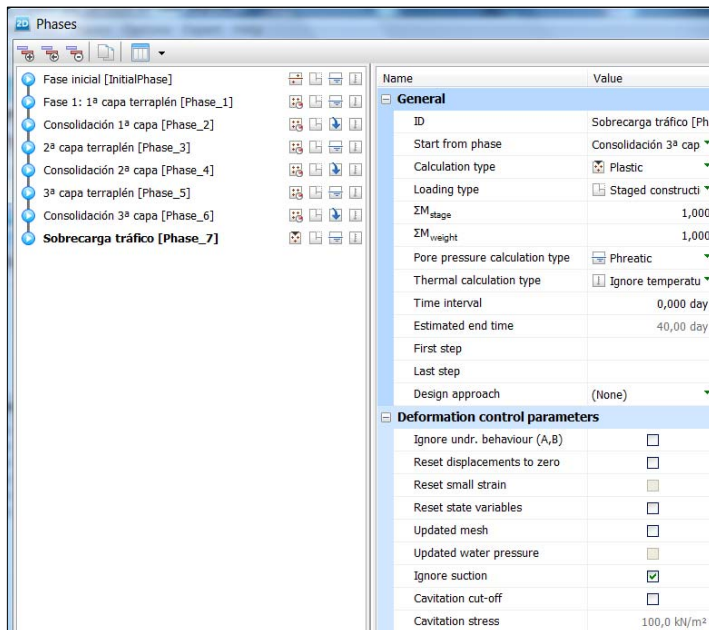


Figure 3. 22. Phase 7: application of the traffic load on the right side of the embankment

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

Figures 3.23 to 3.27 display the different phases of the embankment construction.

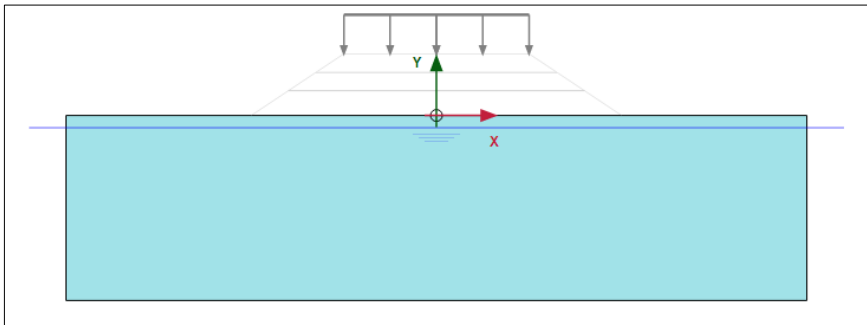


Figure 3. 23. Initial Phase: foundation soil

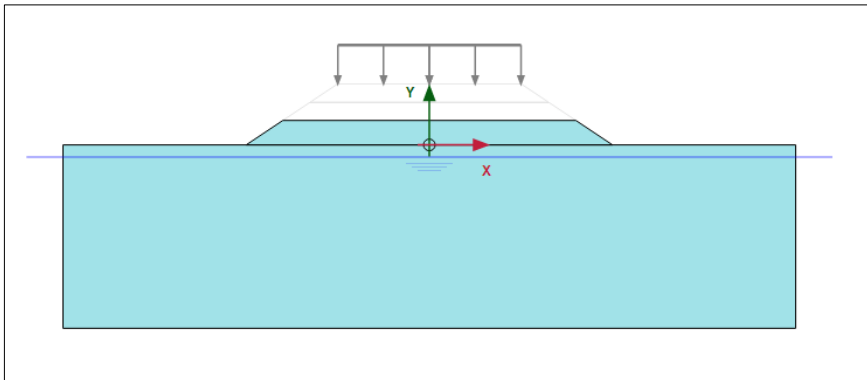


Figure 3. 24. Phase 1: first 4 meters of embankment

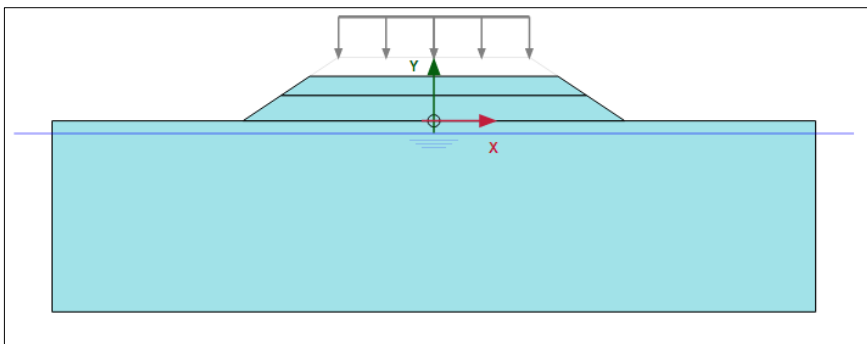


Figure 3. 25. Phase 3: second layer of embankment (3 meters)

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

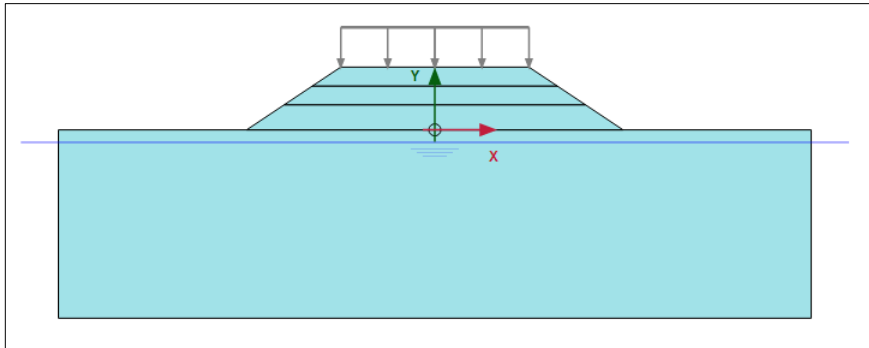


Figure 3. 26. Phase 5: last layer of embankment (3 meters)

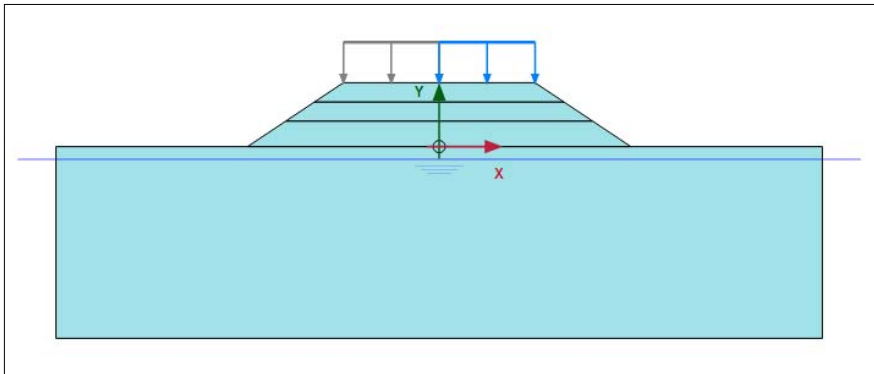


Figure 3. 27. Phase 7: traffic load applied on the right side of the embankment

NOTE. Once the phases have been defined, it is convenient to save the project with a clear name, since the next step is to define the rest of variants of the exercise (it is recommended to save each variant in a different file).

First case: b) Embankment construction in seven phases employing consolidation times. The traffic load is applied on the whole length of embankment.

7 phases must be defined:

Initial Phase, as well as Phases 1 to 6, is the same as in case a), so the previous saved file can be taken as a base. However, in Phase 7 the traffic load is applied on

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

the whole embankment length (Figure 3.28) 40 days after the beginning of the embankment construction.

Therefore, in order to define Phase 7, we activate the two traffic loads (right and left side of the embankment). The rest of the parameters are the same as in Phase 7 in First case b). We save the project with an appropriate name.

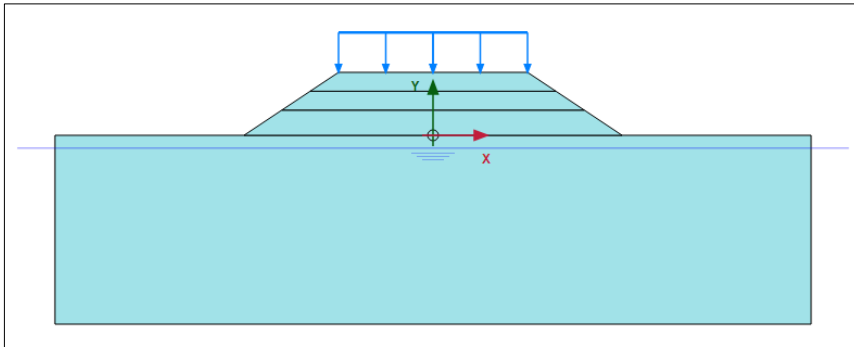


Figure 3. 28. Phase 4: traffic load applied on the whole length of the embankment, 90 days after the beginning

Second case: a) The embankment is built in seven phases but employing degrees of consolidation. The traffic load is applied on the right side of the embankment.

8 phases are defined:

- Initial Phase: only foundation soil.
- Phase 1: the first 4-meter layer of embankment is built in a 4-day period.
- Phase 2: the load is applied until 70% consolidation degree is reached.
- Phase 3: the next 3-meter layer of embankment is built in a 3-day period.
- Phase 4: the load is applied until 80% consolidation degree is reached.
- Phase 5: the last 3-meter layer of embankment is built in 3-day period.
- Phase 6: the load is applied until 90% degree of consolidation is reached.
- Phase 7: a 23 kN/m² traffic load is applied on the right side of the embankment.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

In the Initial Phase, we deactivate the three embankment layers and the traffic load (only activate the foundation soil). We check that in 'Phase' window the selected calculation type is 'K0 procedure'.

In Phase 1 we activate the first embankment layer and select 'consolidation' calculation type. In the same window we must write a time interval of 4 days.

Phase 2 is a consolidation phase linked to a degree of consolidation. In 'Phase' window we choose 'consolidation' calculation type, while in 'Loading type' we choose 'Degree of consolidation'. There we write 70% (Figure 3.29).

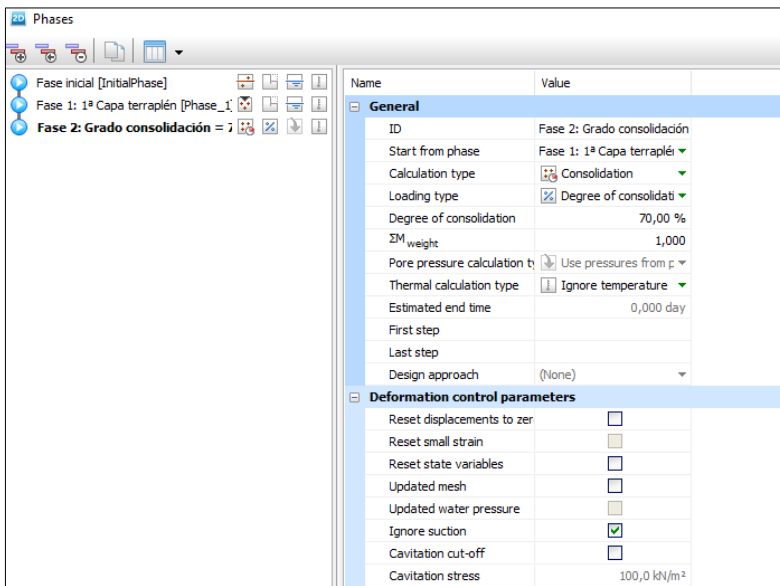


Figure 3. 29. Phase 2 linked to a degree of consolidation

In Phase 3, we activate the next embankment layer and select 'consolidation' calculation type, assigning a time interval of 3 days.

Phase 4 is another consolidation phase linked to a degree of consolidation. As done in Phase 2, in 'Degree of consolidation' we write 80% (Figure 3.30).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

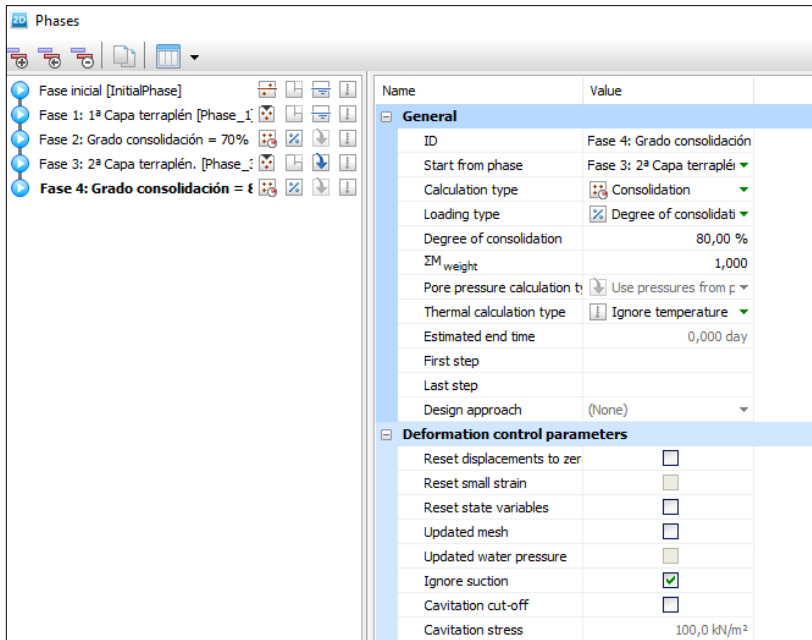


Figure 3. 30. Phase 4 linked to a degree of consolidation

In Phase 5 we activate the last embankment layer and select ‘consolidation’ calculation type, assigning a time interval of 3 days.

Phase 6 is another consolidation phase linked to a degree of consolidation. As in Phases 2 and 4, in ‘Degree of consolidation’ we write 90% (Figure 3.31).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

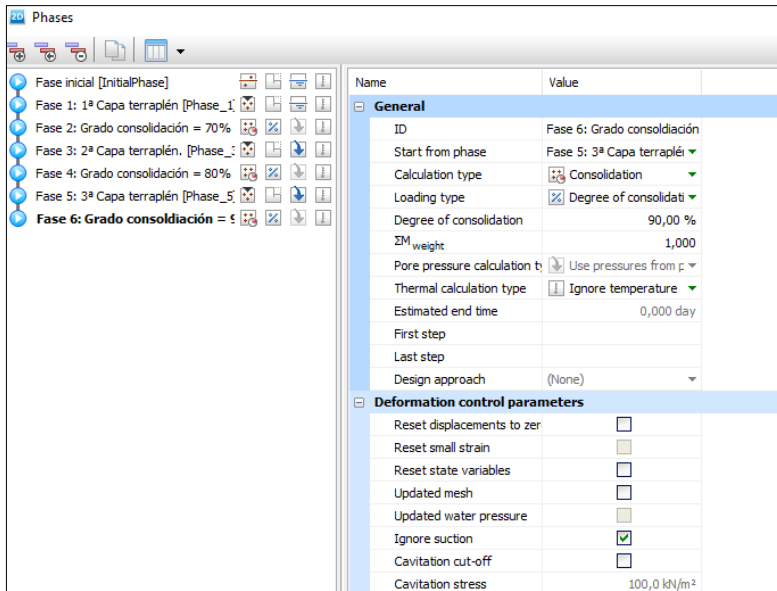


Figure 3. 31. Phase 6 linked to a degree of consolidation

In Phase 7, the traffic load is activated on the right side of the embankment, assigning 'Plastic' calculation type.

In Figure 3.32 the different created phases can be observed. We save the project with an appropriate name.

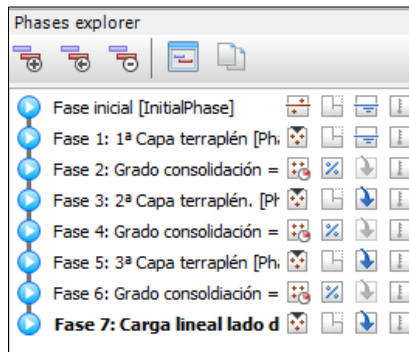


Figure 3. 32. Embankment phases linked to a degree of consolidation, with traffic load applied on the right side of the embankment

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

Second case: b) The embankment is built in seven phases but employing degrees of consolidation. The traffic load is applied on the whole length of the embankment.

8 phases must be defined:

The Initial Phase, as well as Phases 1 to 6, is the same as in Second case a). In Phase 7, the traffic load is applied on the whole length of the embankment after reaching a degree of consolidation of 90%.

Therefore, in order to define Phase 7, we activate the two traffic loads (on the right and the left side of the embankment). The rest of the parameters are the same as those in Phase 7 in Second case a).

We save the project in a file with an appropriate name.

3.3.9. Results in PLAXIS Output for consolidation study

In ‘*Staged construction*’ calculation mode, we click on ‘*Calculate*’ button. We can display the results in PLAXIS Output by selecting ‘*View calculation results*’ button.

First case: a) Embankment construction in seven phases employing consolidation times. The load is applied on the right side of the embankment.

In PLAXIS Output we can display deformation information in different ways. Sometimes it is interesting to show displacements represented by arrows. This can be obtained by selecting ‘*Deformations*’, ‘*Total displacements*’. Here we choose the total displacement option that we need: total displacements, $|u|$, horizontal total displacements, u_x , and vertical total displacements, u_y (Figure 3.33).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

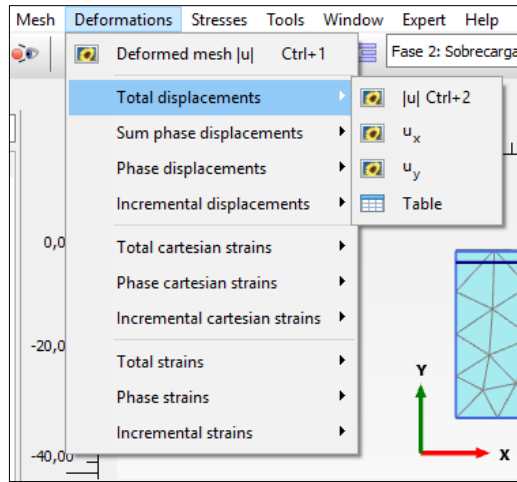


Figure 3. 33. Total displacements representation options

In order to represent displacements by arrows, we select 'Arrows' button (Figure 3.34).

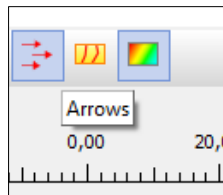


Figure 3. 34. 'Arrows' button to represent displacement

In this way, graphics in Figures 3.35, 3.36 and 3.37, which correspond to total displacements, $|u|$, u_x and u_y , in the last phase (Phase 7), are obtained.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

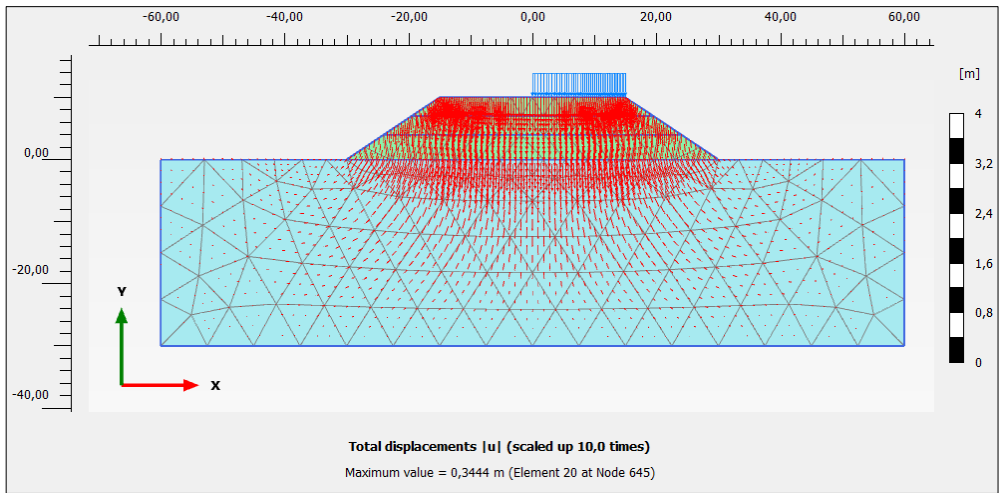


Figure 3. 35. Total displacements, $|u|$, in Phase 7

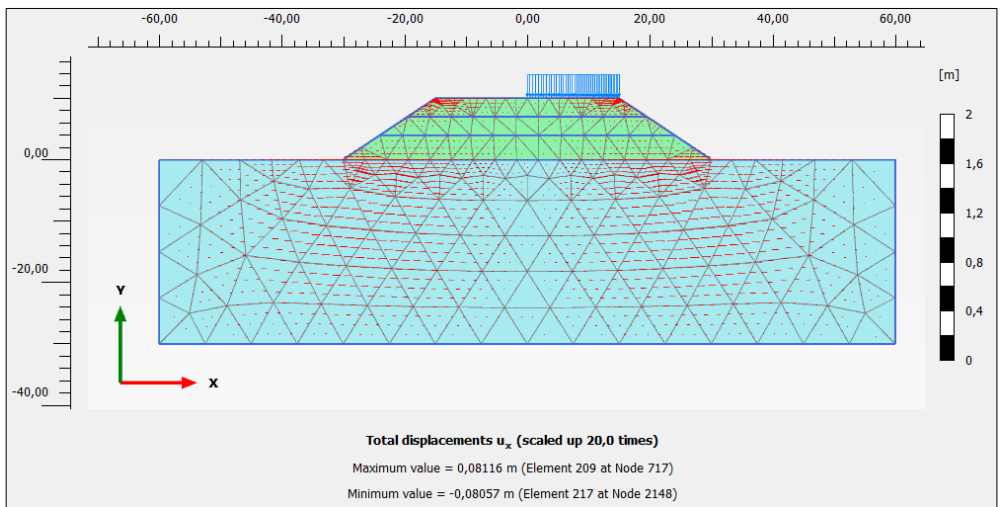


Figure 3. 36. Total horizontal displacements, u_x , in Phase 7

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

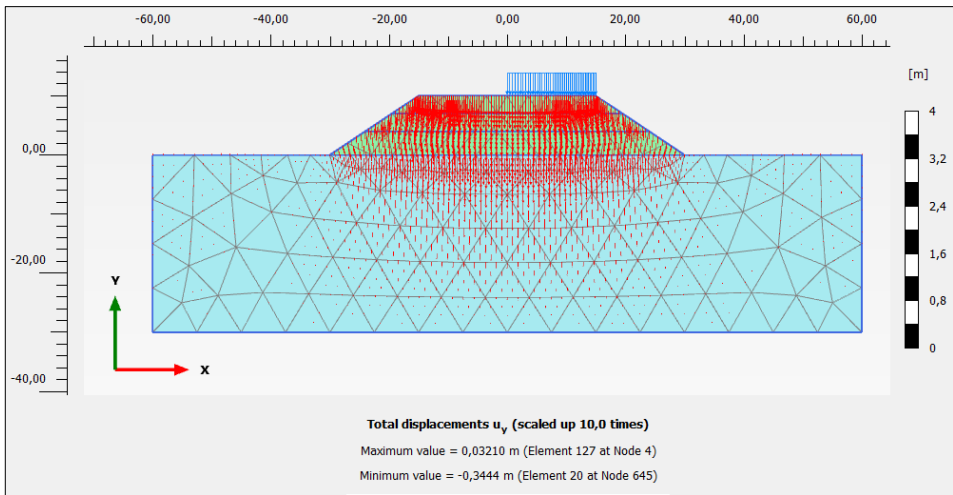


Figure 3. 37. Total vertical displacements, u_y , in Phase 7

In order to obtain total displacement advanced graphical outputs, and in the different nodes that have been previously selected (Figures 3.16 and 3.17), we click on 'Curves manager' and 'New' to create a new curve (activating 'Charts' tab). In 'X-axis', we select 'Project' in the pull-down menu, and 'Time' as the variable to plot. In 'Y-Axis' we choose the point (A, B, C or D) and select 'Deformations', 'Total displacements', and the displacements to plot, u_x , u_y , or, as we show here, $|u|$ (Figure 3.38).

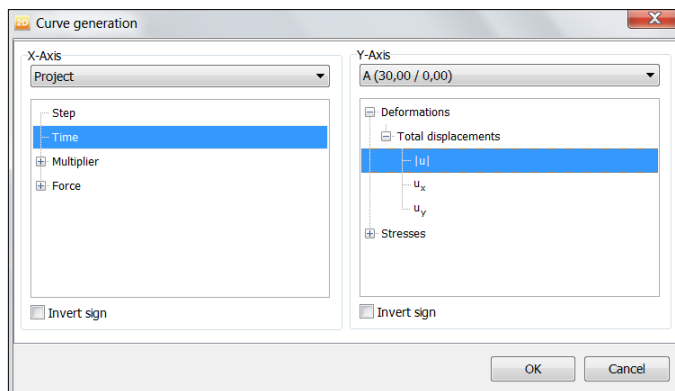


Figure 3. 38. Curve generator for total displacements in points

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

In the same graphic we can add other points, clicking on itself with the right mouse button and selecting 'Add curve', 'From current project...'. It is also possible to adjust different plotting options (axis limit, scale type, grid, etc.) selecting 'Settings'.

In Figure 3.39 total displacements, $|u|$, during the embankment construction in nodes A, B, C and D are shown.

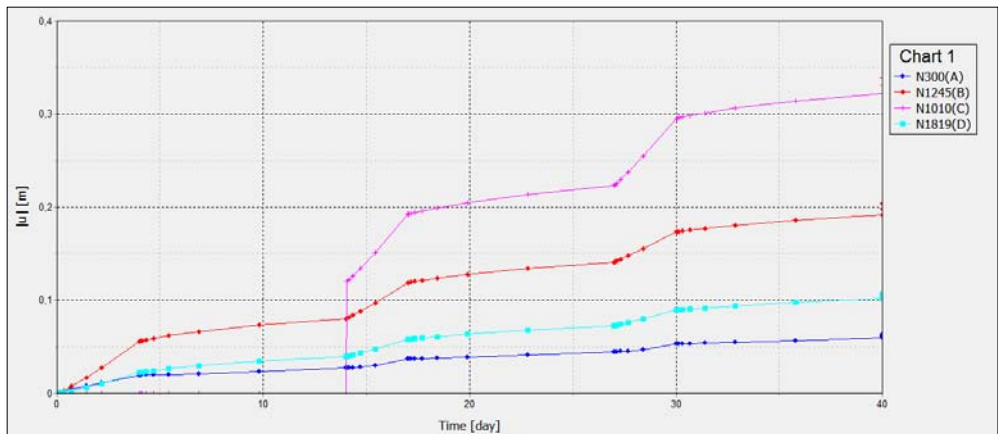


Figure 3. 39. Total displacements, $|u|$, during the embankment construction in nodes A, B, C and D

In the same way, it is also possible to plot the evolution of pore pressure (P_{excess} , Figures 3.40 y 3.41), or vertical effective stress (σ'_{yy} , Figures 3.42 y 3.43), in the 'Stress points' that were added to obtain this kind of graphics (Stress points K, L, and M in Figures 3.16 and 3.17).

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

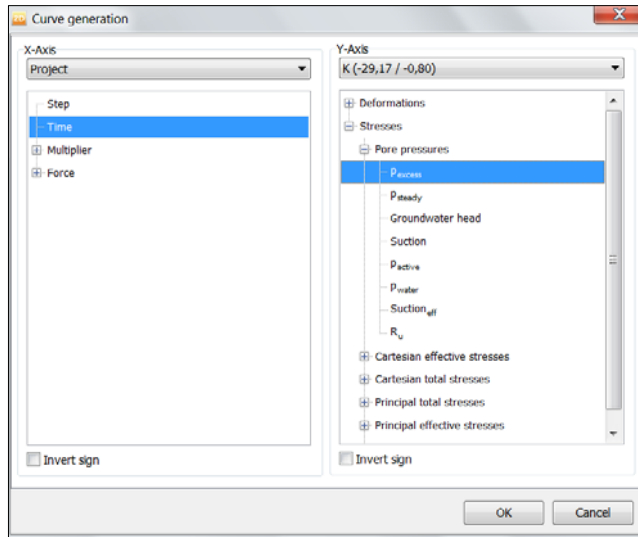


Figure 3. 40. Curve generator for pore pressure

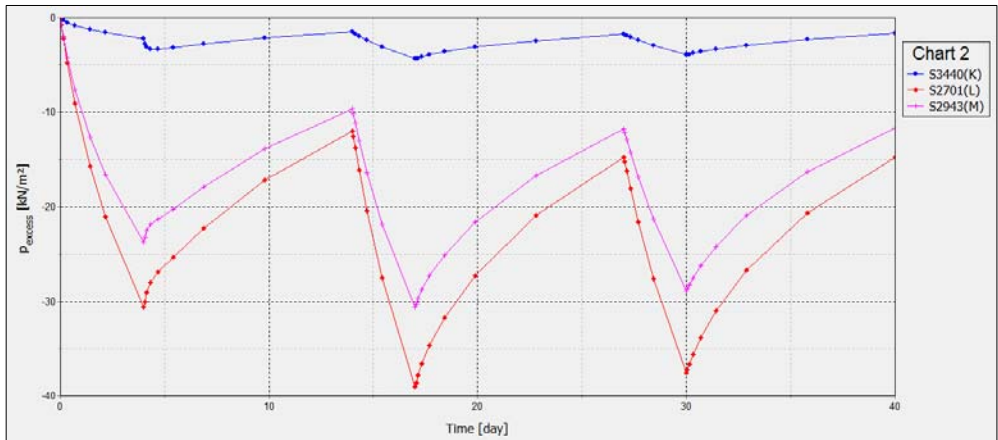


Figure 3. 41. Pore pressure during embankment construction in points K, L and M

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

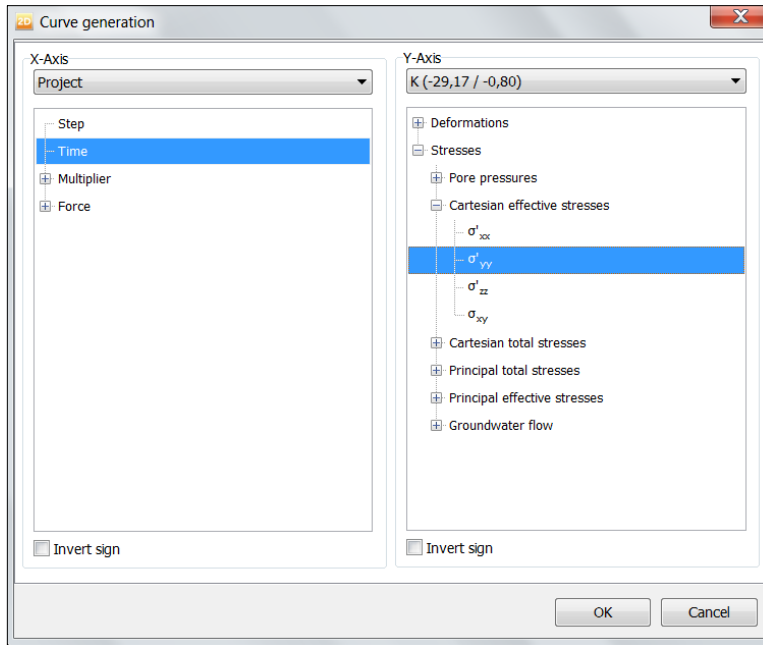


Figure 3. 42. Curve generator for vertical effective stress, σ'_{yy}

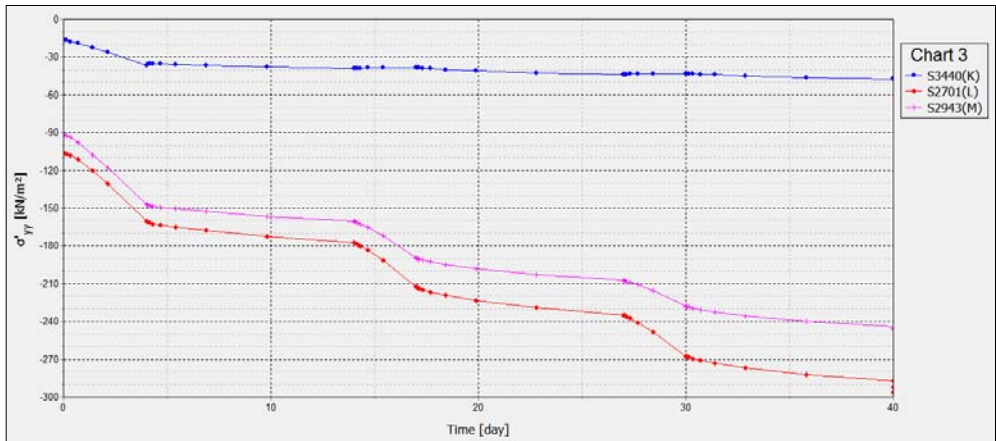


Figure 3. 43. Vertical effective stress, σ'_{yy} , during the embankment construction in points K, L and M

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

First case: b) Embankment construction in seven phases employing consolidation times. The traffic load is applied on the whole length of the embankment.

In Figures 3.44, 3.45 and 3.45 total displacements, $|u|$, pore pressure and vertical effective stress, σ'_{yy} , for Phase 7, are presented (traffic load already applied).

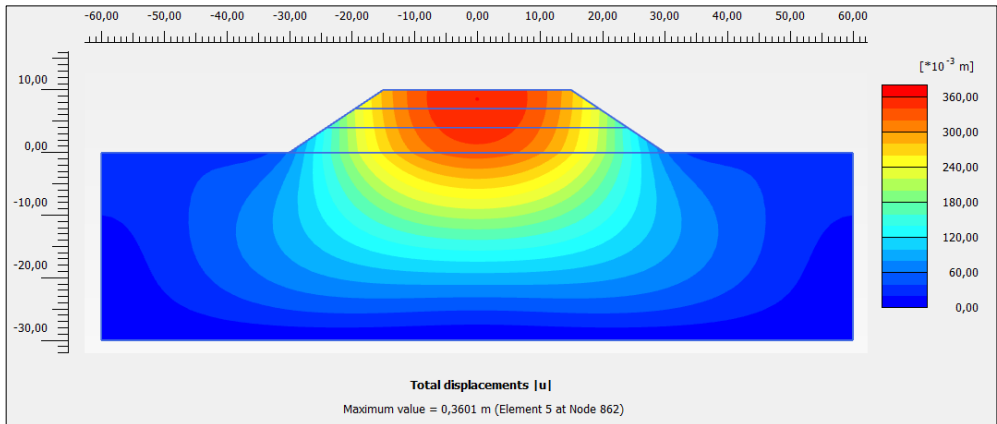


Figure 3. 44. Total displacements, $|u|$, in Phase 7

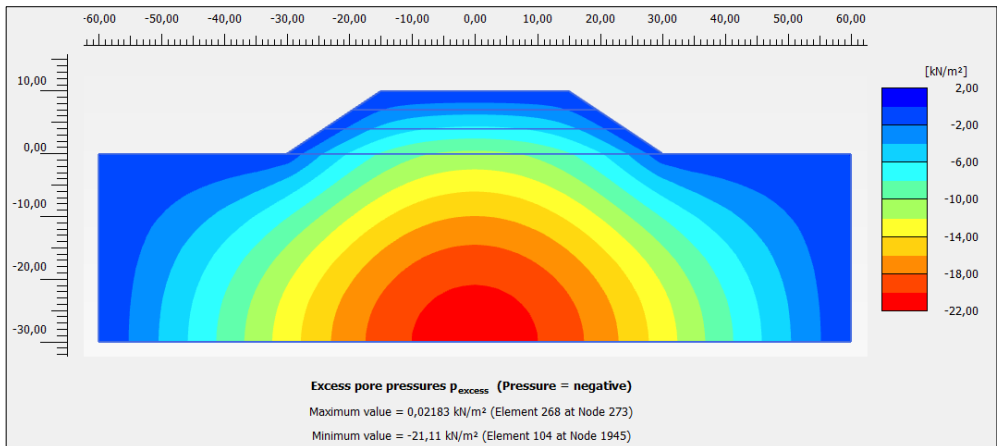


Figure 3. 45. Pore pressure in Phase 7

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

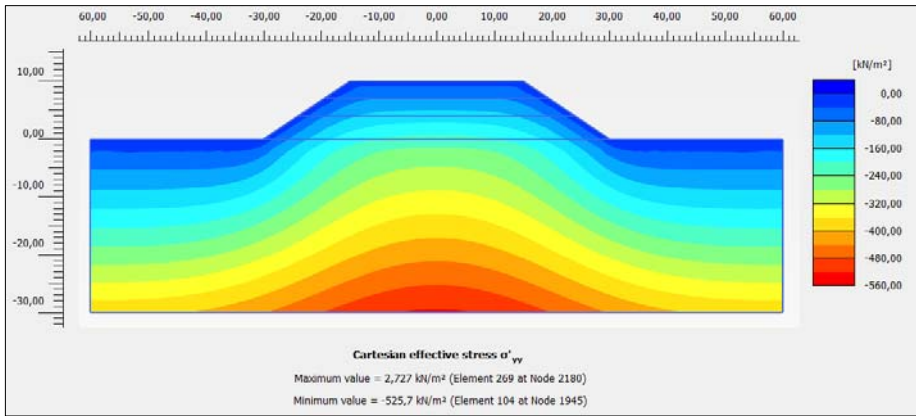


Figure 3. 46. Vertical effective stress, σ'_{yy} , in Phase 7

Second case: a) The embankment is built in seven phases but employing degrees of consolidation. The load is applied on the right side of the embankment

Figures 3.47, 3.48 and 3.49 represent, respectively, total displacement, $|u|$, pore pressure and vertical effective stress, σ'_{yy} , evolutions during the embankment construction.

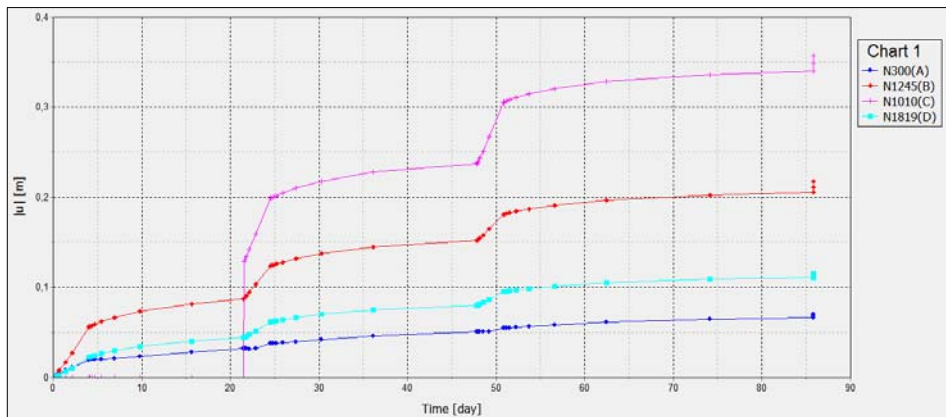


Figure 3. 47. Total displacements, $|u|$, during the embankment construction in nodes A, B, C and D

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

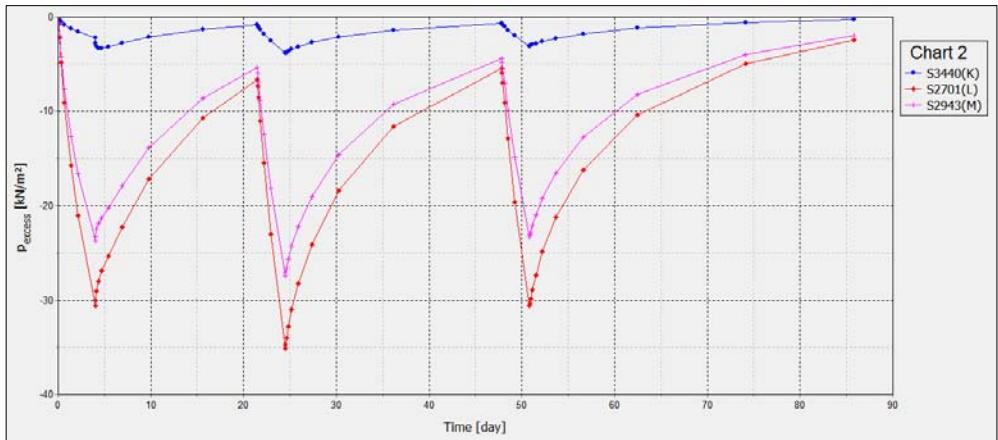


Figure 3. 48. Pore pressure during the embankment construction in points K, L and M

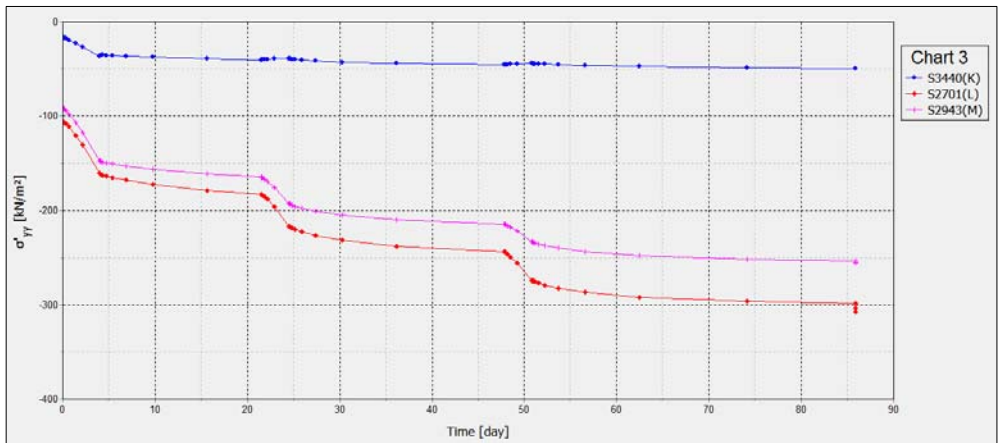


Figure 3. 49. Vertical effective stress during the embankment construction in points K, L and M

Second case: b) The embankment is built in seven phases but employing degrees of consolidation. The traffic load is applied on the whole length of the embankment.

Figures 3.50, 3.51 and 3.52 represent, respectively, total displacement, $|u|$, pore pressure and vertical effective stress, σ'_{yy} , evolutions during the embankment construction.

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

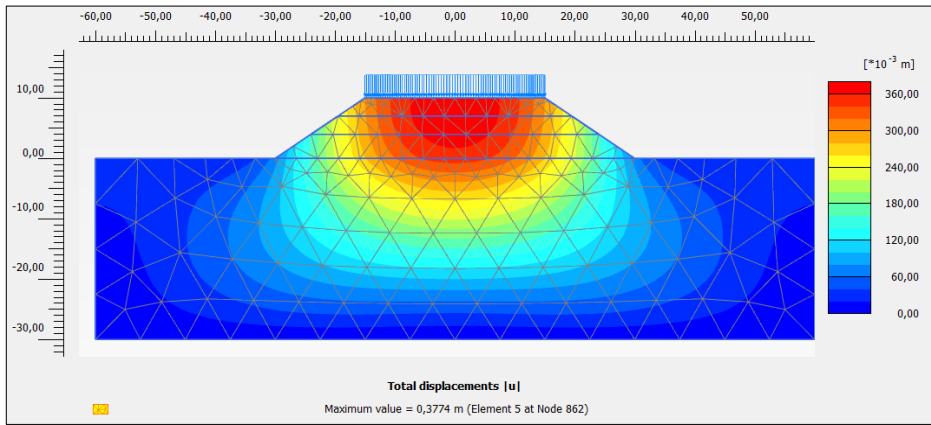


Figure 3. 50. Total displacements, $|u|$, in Phase 7

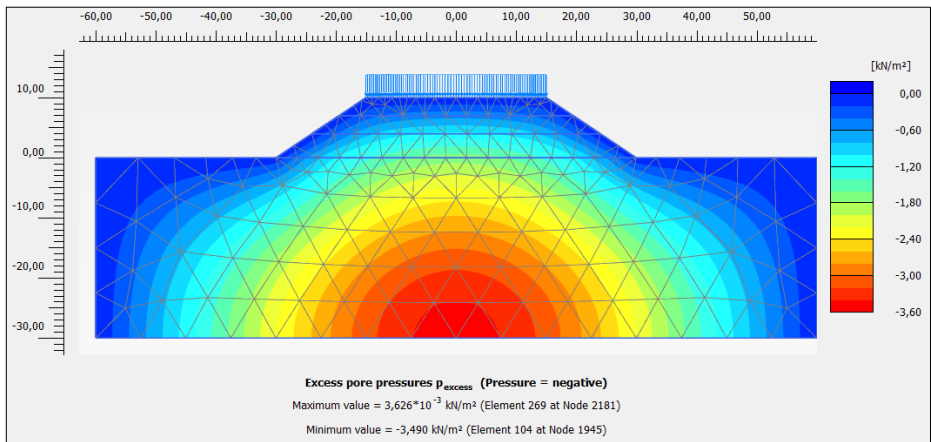


Figure 3. 51. Pore pressure in Phase 7

Exercise 03. Consolidation analysis of an embankment on a homogeneous soil for the creation of a highway

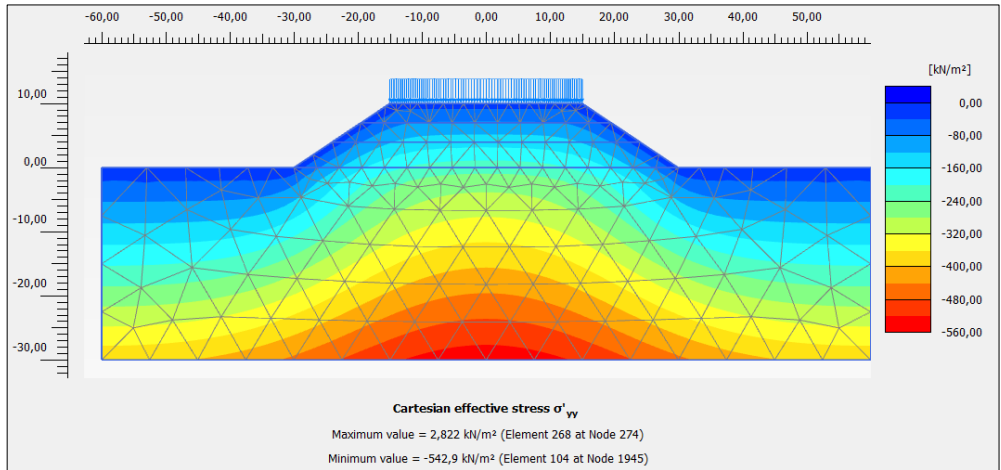


Figure 3. 52. Vertical effective stress, σ'_{yy} , in Phase 7.

Exercise 04

Flow net under concrete dam

4.1. INTRODUCTION

A concrete dam is built on a 10-meter thick sand-clay layer. The dam is 10 meters high, 10 meters wide in base and 5 meters wide in crest. The upstream length is 10 m, as it is downstream. The water level is 8 meters upstream and 3 meters downstream the dam. Both soil and concrete properties are presented below.

The aim of the exercise is to obtain the flow net (representation of equipotential lines and velocity vectors), as well as the infiltrated water flow under the structure.

4.2 GEOMETRICAL AND GEOTECHNICAL PARAMETERS

4.2.1 Problem geometry

Figure 4.1 shows the geometry of the problem.

Exercise 04. Flow net under concrete dam

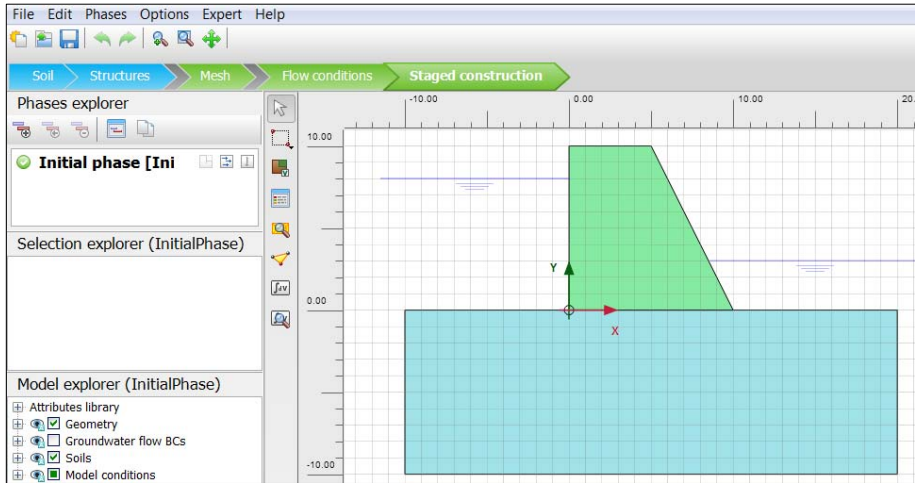


Figure 4. 1. Sketch of the dam on the sand-clay layer

4.2.2 Problem parameters

Tables 4.1 and 4.2 define parameters for foundation soil and concrete dam respectively.

Foundation soil

Material	Sand-clay
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	18.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	21.5 kN/m ³
Young's modulus (E')	22500 kPa
Poisson's ratio (ν')	0.32
Effective soil cohesion (c')	5 kPa
Effective friction angle (ϕ')	30°
Dilatancy angle (ψ)	0°
Horizontal hydraulic conductivity (K_x)	4 · 10 ⁻⁶ m/day
Vertical hydraulic conductivity (K_y)	10 ⁻⁶ m/day

Table 4. 1. Foundation soil parameters

Exercise 04. Flow net under concrete dam

Concrete dam

Material	Concrete
Constitutive model	Linear elastic
Material behaviour	Non-porous
Soil unit weight above phreatic level (γ_{unsat})	24 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	24 kN/m ³
Young's modulus (E')	200·10 ⁶ kPa
Poisson's ratio (ν')	0.15

Table 4. 2. Concrete dam parameters

4.3 RESOLUTION OF THE PROBLEM WITH PLAXIS 2D

4.3.1 Introduction of the project parameters

We start PLAXIS Input and select 'Start a new Project' option in 'Quick select' emerging window.

We name the problem in 'Project' tab. In the title bar we write: 'Flow net of a concrete dam on a sand-clay layer'.

In 'Model' tab we choose 'Plane strain' option in model and '15-Noded' in elements. We keep the default units (Figure 4.2).

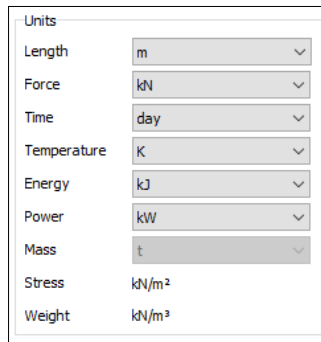


Figure 4. 2. Problem units

The border can be defined with the values presented in Figure 4.3.

Exercise 04. Flow net under concrete dam

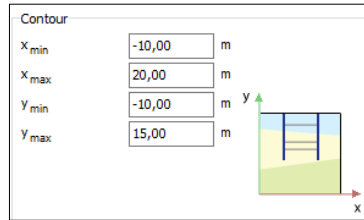


Figure 4. 3. Project borders

We keep the default constants ('Constants' tab).

4.3.2. Soil definition

In 'Soil' geometry mode we select 'Create borehole' button and define the foundation soil.

We click on coordinate (-10, 0) and in 'Modify soil layers' we select 'Add'. In 'Top' we write 0 and, in 'Bottom', -10.

As the phreatic level in this borehole is located 8 meters above the ground surface, we write 8 in 'Head'. Figure 4.4 shows the borehole data.

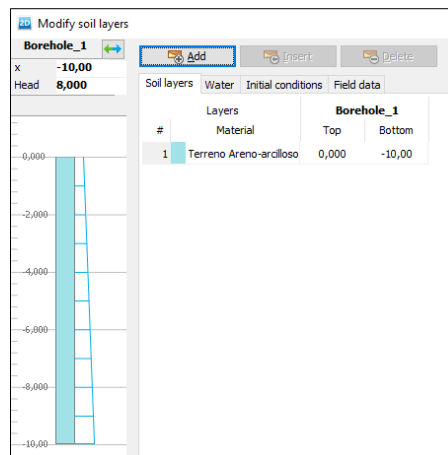


Figure 4. 4. First borehole definition

The second borehole is located in coordinate (0, 0), and we use the same data as in the first one (Figure 4.5).

Exercise 04. Flow net under concrete dam

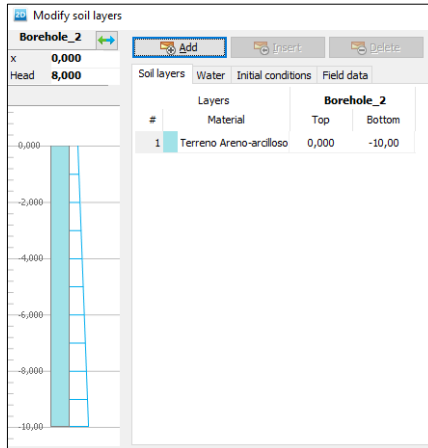


Figure 4. 5. Second borehole definition

The third borehole is located in coordinates (8.5, 0), so we first need to change 'Number of snap intervals' value to 2 in 'snapping options' menu. In 'Modify soil layers' window, we select 'Add'. In 'Top' we write 0 and -10 in 'Bottom'. As the phreatic level is located 3 meters above the ground surface, we write 3 in 'Head' (Figure 4.6).

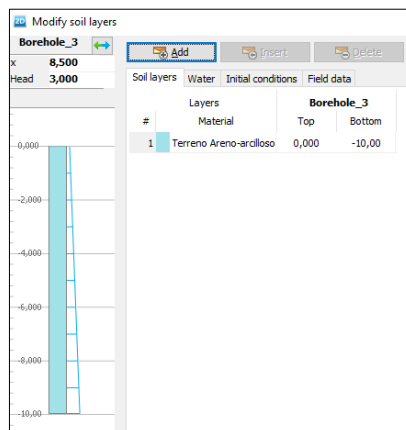


Figure 4. 6. Third borehole definition.

The fourth borehole is located in (10, 0). In 'Top', we write 0 and in 'Bottom' -10. Here, the phreatic level is also 3 meters above the ground surface, so we write 3 in

Exercise 04. Flow net under concrete dam

‘Head’. The last borehole is located in (20, 0), and the rest of the properties are the same as in the fourth one (Figure 4.7).

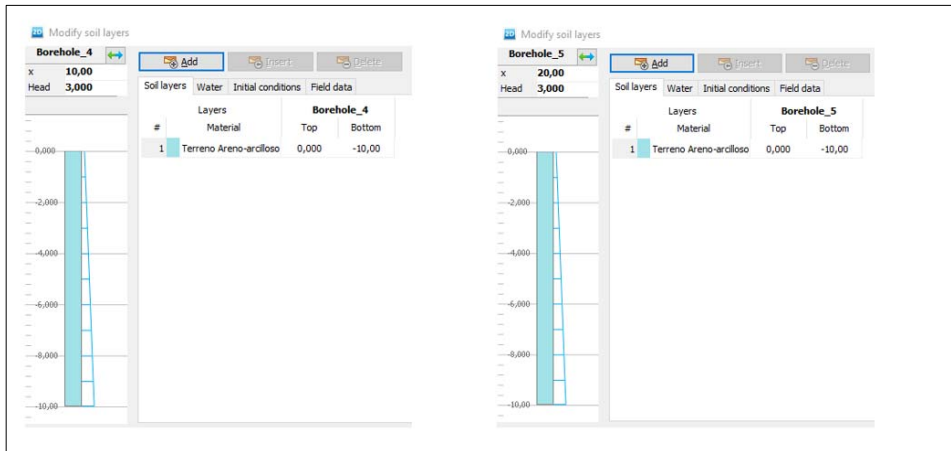


Figure 4. 7. Fourth (left) and fifth (right) borehole definition

In Figure 4.8 the foundation soil is presented.

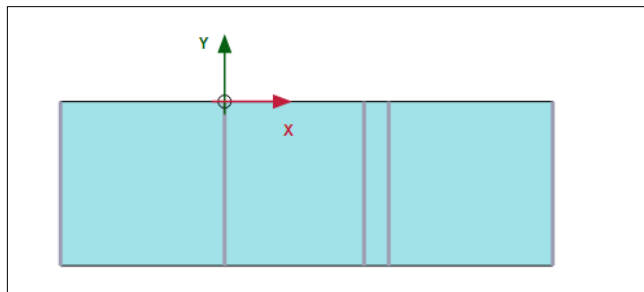


Figure 4. 8. Foundation soil

In order to create the foundation material, we select ‘*Show materials*’ button, we click on ‘*New*’ (once we are in ‘*Soil and interfaces*’) and we introduce the geotechnical parameters presented in 4.2.2 (Figures 4.9, 4.10 and 4.11).

Exercise 04. Flow net under concrete dam

Soil - Mohr-Coulomb - Terreno Arenoso-arcilloso

General Parameters Groundwater Thermal Interfaces Initial

Property	Unit	Value
Material set		
Identification		Terreno Arenoso-arcilloso
Material model		Mohr-Coulomb
Drainage type		Drained
Colour		RGB 161, 226, 232
Comments		
General properties		
γ_{unsat}	kN/m ³	18,50
γ_{sat}	kN/m ³	21,50

Advanced

Figure 4. 9. General properties of sand-clay soil

Soil - Mohr-Coulomb - Terreno Arenoso-arcilloso

General Parameters Groundwater Thermal Interfaces Initial

Property	Unit	Value
Stiffness		
E'	kN/m ²	22,50E3
ν' (ν_u)		0,3200
Alternatives		
G	kN/m ²	8523
E_{oed}	kN/m ²	32,20E3
Strength		
c'_{ref}	kN/m ²	5,000
ϕ' (ϕ_h)	°	30,00
ψ (ψ_i)	°	0,000
Velocities		
V_s	m/s	67,23
V_p	m/s	130,7

Figure 4. 10. Parameters of sand-clay soil

k_x	m/day	4,000E-6
k_y	m/day	1,000E-6

Figure 4. 11. Hydraulic conductivities of sand-clay soil

4.3.3. Dam definition

Concrete presents the properties shown in Figures 4.12 and 4.13.

Property	Unit	Value
Material set		
Identification		Hormigón
Material model		Linear elastic
Drainage type		Non-porous
Colour		RGB 134, 234, 162
Comments		
General properties		
Y_{unsat}	kN/m ³	24,00
Y_{sat}	kN/m ³	24,00

Figure 4. 12. General properties of concrete

Property	Unit	Value
Stiffness		
E	kN/m ²	200,0E6
ν (nu)		0,1500
Alternatives		
G	kN/m ²	86,96E6
E_{oed}	kN/m ²	211,2E6
Velocities		
V_s	m/s	5962
V_p	m/s	9291

Figure 4. 13. Elastic parameters of concrete

In order to create the dam, we go to 'Structures' geometry mode and select 'Create soil polygon'.

We select the points in Table 4.3 in the workspace.

Exercise 04. Flow net under concrete dam

Dam contour coordinates

A (0, 0)
B (0, 10)
C (5, 10)
D (10, 0)

Table 4. 3. Concrete dam contour coordinates

We click on the dam with the right mouse button and select ‘Concrete’ material, which has been created previously (Figure 4.14).

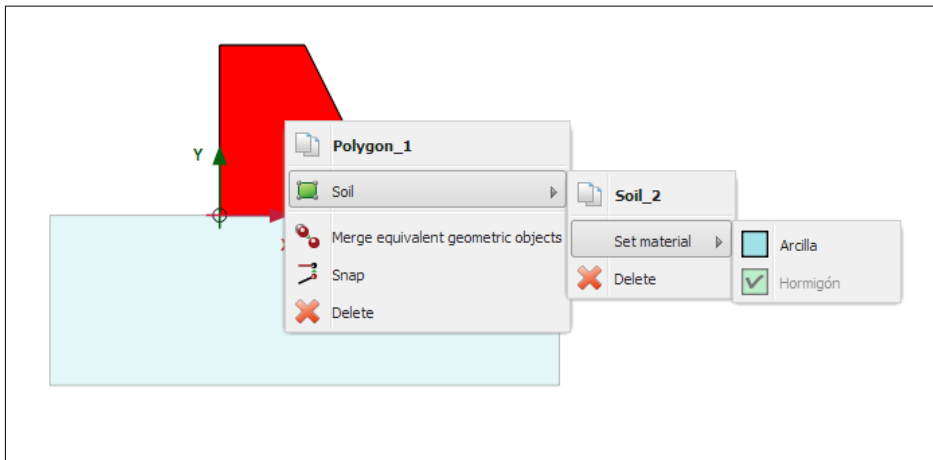


Figure 4. 14. Definition of ‘Concrete’ as dam material

The dam is defined as presented in Figure 4.15.

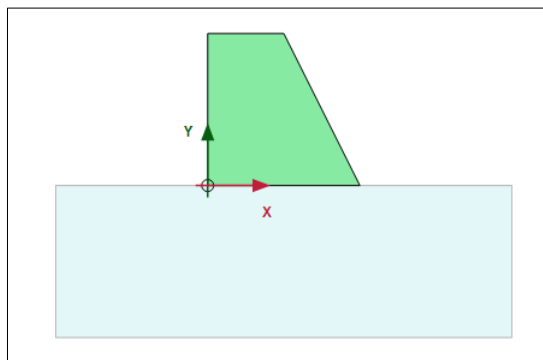


Figure 4. 15. Defined dam on sand-clay foundation soil

4.3.4. Problem meshing

We go to 'Mesh' calculation mode and click on 'Generate mesh' button, indicating a 'Fine' meshing. If we want to display the generated mesh in PLAXIS Output (Figure 4.16) we select 'View mesh' button.

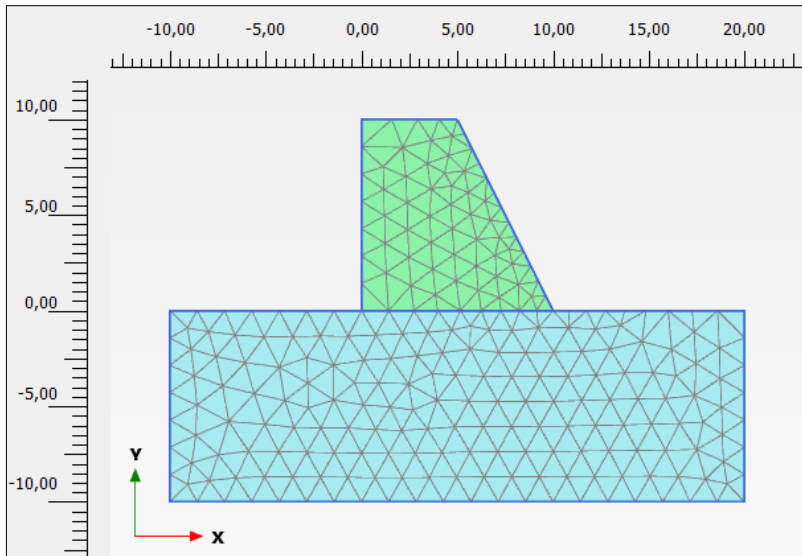


Figure 4. 16. Problem meshing

4.3.5. Project phases

In this case, we only need to create one phase ('Initial phase'). In 'Calculation type' we select 'Flow only' and in 'Pore pressure calculation type' we choose 'Steady state groundwater flow' (Figure 4.17).

Exercise 04. Flow net under concrete dam

Name	Value
General	
ID	Initial phase [InitialPhase]
Calculation type	Flow only
Pore pressure calculation type	Steady state groundwater flow
Thermal calculation type	Ignore temperature
Estimated end time	0,000 day
First step	
Last step	
Design approach	(None)
Numerical control parameters	
Solver type	Picos (multicore iterative)
Max cores to use	256
Max number of steps store	50
Flow control parameters	

Figure 4. 17. Initial phase configuration for the study of steady flow under the concrete dam

Finally, we click on ‘*Calculate*’ button to run the calculations. Once the phase is calculated, we go to PLAXIS Output to display the results clicking on ‘*View calculation results*’ button.

4.3.6. Results in PLAXIS Output

In order to see the equipotential lines, in ‘*Stresses*’ we select ‘*Pore pressure*’ and ‘*Groundwater head*’ (Figure 4.18). Figure 4.19 shows these equipotential lines.

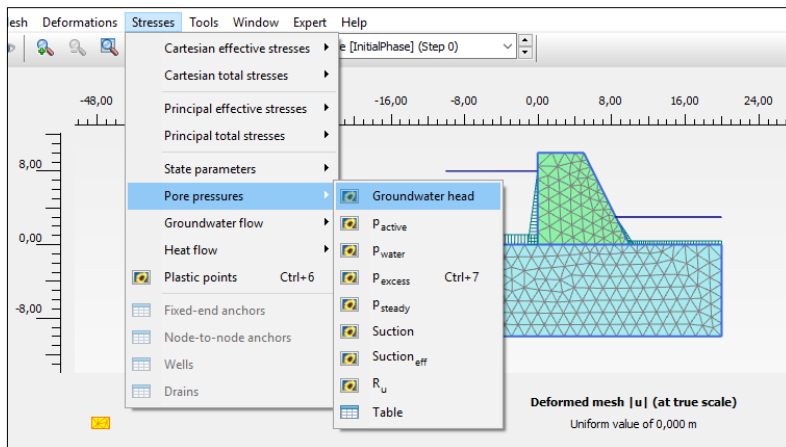


Figure 4. 18. Selection of equipotential lines representation

Exercise 04. Flow net under concrete dam

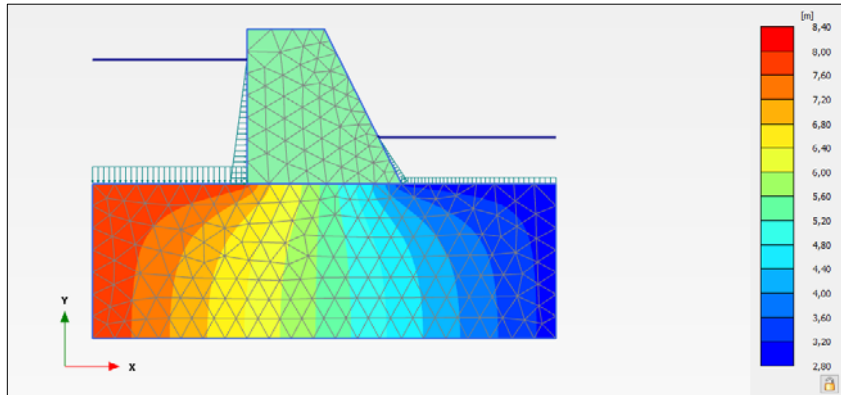


Figure 4. 19. Equipotential lines

We can also represent water velocity vectors if instead of selecting ‘*Groundwater head*’ we choose ‘*Groundwater flow*’ and $|q|$ (flux modulus), as explained in Figure 4.20. Figure 4.21 shows the representation of water velocity vectors.

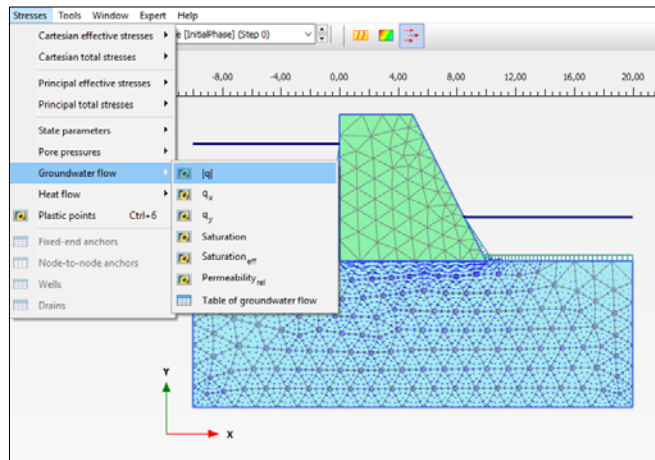


Figure 4. 20. Selection of water velocity vectors representation

Exercise 04. Flow net under concrete dam

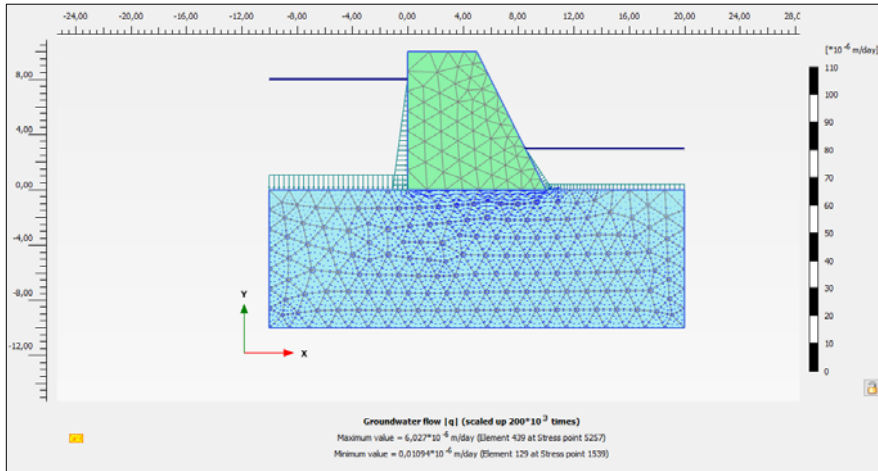


Figure 4. 21. Water velocity vectors

4.3.7. Flow net if lateral and bottom contour behave as impervious borders

Sometimes, we might be interested in blocking the water flow through the problem's contours. The aim of this is to obtain the flow net only considering the flow from upstream (line that goes from (-10, 0) to (0, 0)) to downstream (line that goes from (10, 0) to (20, 0)).

Therefore, we must 'close' the flow running through the contour lines of both dam (lines 1, 2, 3 and 5) and impervious borders (lines 7, 8 and 9), as shown in Table 4.4.

Dam contour	First point coordinates	Second point coordinates
Line 1	(0, 10)	(5, 10)
Line 2	(0, 0)	(0, 10)
Line 3	(5, 10)	(10, 0)
Line 5	(0, 0)	(10, 0)

Impervious contour	First point coordinates	Second point coordinates
Line 7	(-10, 0)	(-10, -10)
Line 8	(20, 0)	(20, -10)
Line 9	(-10, -10)	(20, -10)

Table 4. 4. Contour lines where the flow is blocked

In PLAXIS Input we go to 'Staged construction' calculation mode. We select, for example, Line 7 (Figure 4.22) and, in 'Selection explorer' menu, we choose 'Closed' in 'Behaviour' and activate it (green tick).

Exercise 04. Flow net under concrete dam

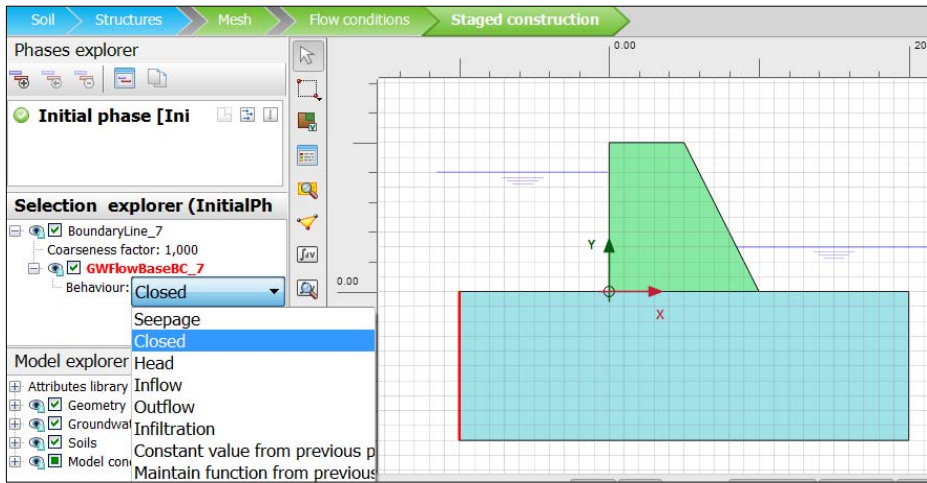


Figure 4. 22. 'Closed' option not to allow water flow through a line

We do the same for the rest of the lines and calculate again the phase. Figures 4.23 and 4.24 show equipotential lines and water velocity vectors respectively.

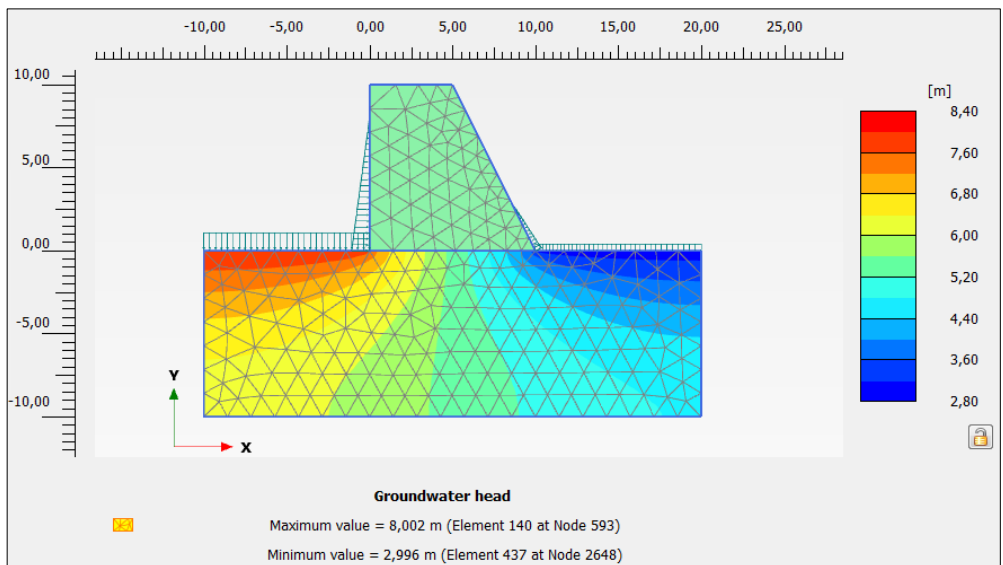


Figure 4. 23. Equipotential lines

Exercise 04. Flow net under concrete dam

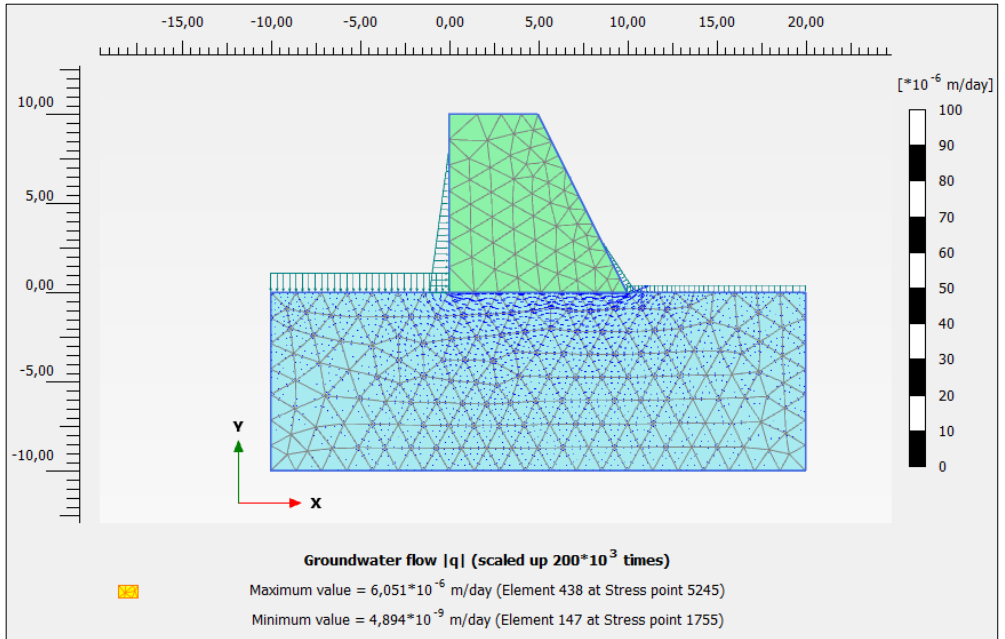


Figure 4. 24. Water velocity vectors

Now we obtain the inflow and outflow running under the dam. For this, and previously having represented $|q|$ ('Stresses', 'Groundwater flow') we select 'Cross section' button (Figure 4.25).

In the emerging table we write line 4 coordinates, from (-10, 0) to (10, 0), as shown in Figure 4.26, in order to obtain the flow running through that section (inflow). Points can also be selected straight on the workspace. As result, we obtain an inflow value ('Total discharge') of approximately $5,71 \cdot 10^{-6}$ m³/day/m (Figures 4.27 and 4.28).

Exercise 04. Flow net under concrete dam

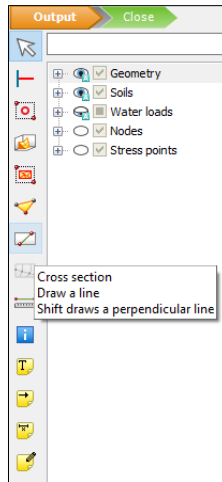


Figure 4. 25. 'Cross section' button

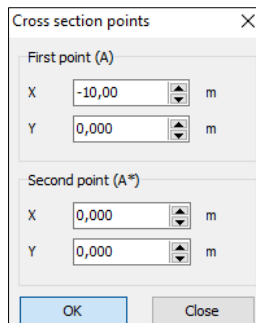


Figure 4. 26. Coordinates of the line (section) through which the inflow is running

Exercise 04. Flow net under concrete dam

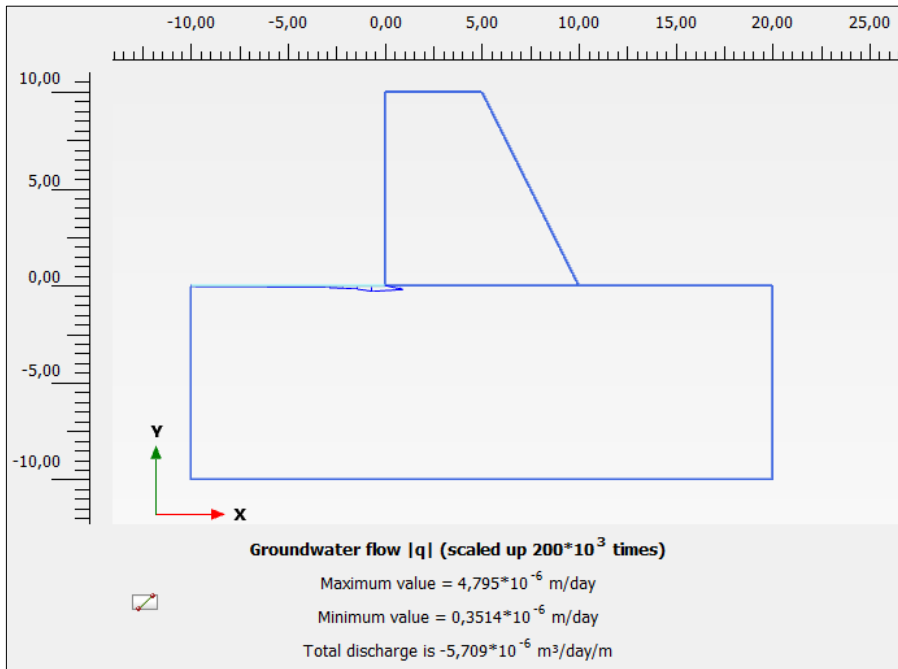


Figure 4. 27. Incoming flow (inflow) per meter of dam thickness

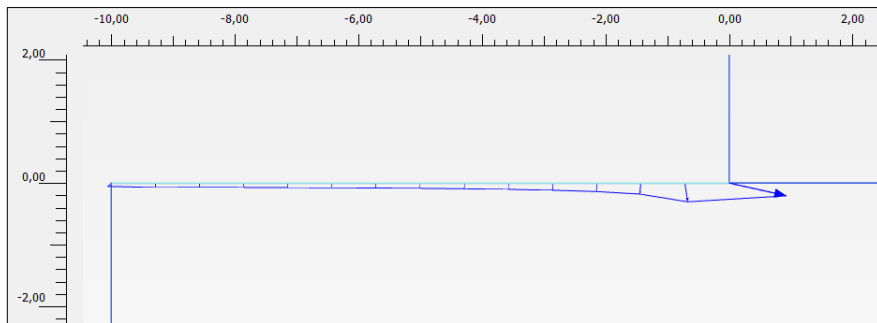


Figure 4. 28. Inflow per meter of dam thickness, zoomed image

In order to know the outflow through line 6, from (10, 0) to (20, 0), we apply a similar procedure, obtaining Figures 4.29 and 4.30. The outflow value ('Total discharge') is approximately $5,63 \cdot 10^{-6}$ m³/day/m, similar to inflow value.

Exercise 04. Flow net under concrete dam

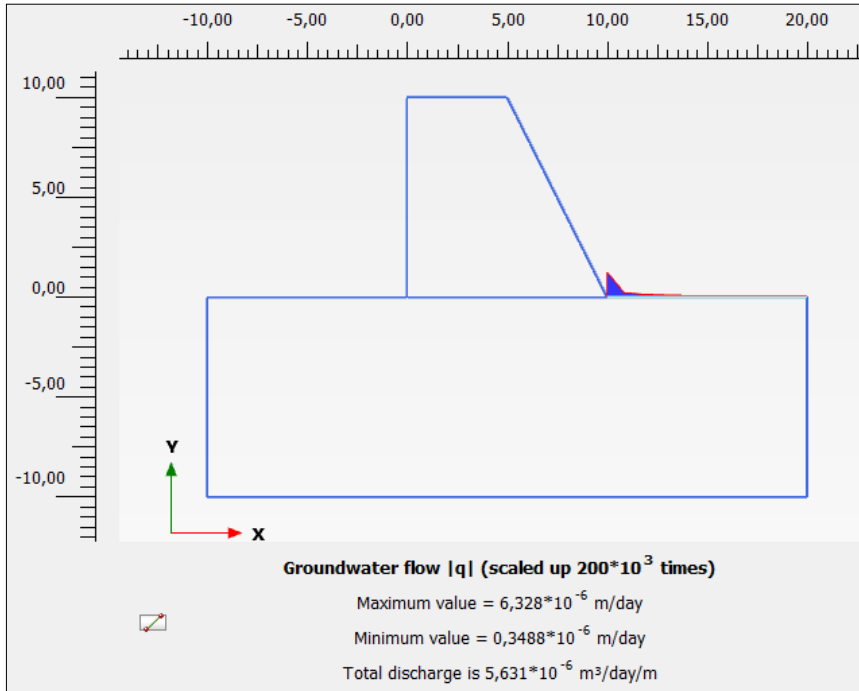


Figure 4. 29. Outcoming flow (outflow) per meter of dam thickness

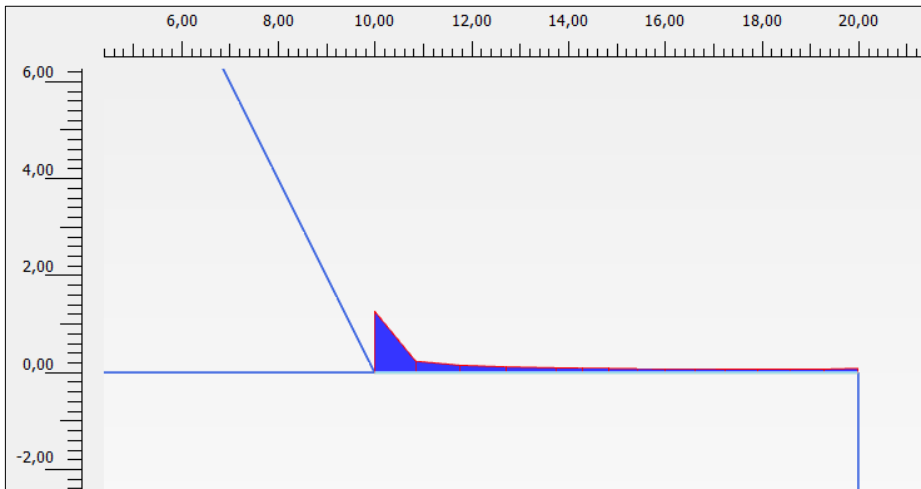


Figure 4. 30. Outflow per meter of dam thickness, zoomed image

4.4 FLOW NET UNDER THE DAM WITH SHEET PILE

4.4.1. Introduction of the sheet pile in PLAXIS Input

The geometry is the same as in the previous problem (we can use it as base and save it with a new name) but for the sheet pile built next to the dam toe. The sheet pile presents a length under the surface of 3 meters. Figure 4.31 presents the geometry of the problem and Table 4.5 the contour points of the sheet pile. As in the previous problem, lateral and bottom borders, the dam and the sheet pile are impervious.

Sheet pile contour coordinates.

A (8, 0)
B (8, -3)
C (9, -3)
D (9, 0)

Table 4. 5. Sheet pile contour coordinates

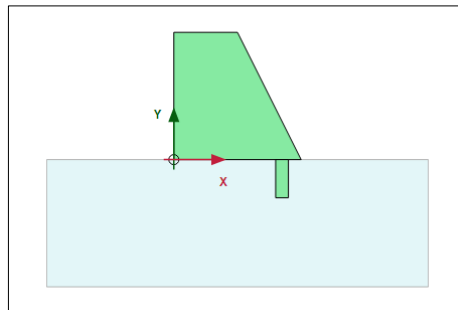


Figure 4. 31. Geometry of the problem with sheet pile

In order to create the sheet pile, we go to ‘Structures’ geometry mode and click on ‘Create soil polygon’. There, we select the sheet pile contour points presented in Table 4.5.

Once it is drawn, we assign it the material ‘Concrete’ (Figure 4.32).

Exercise 04. Flow net under concrete dam

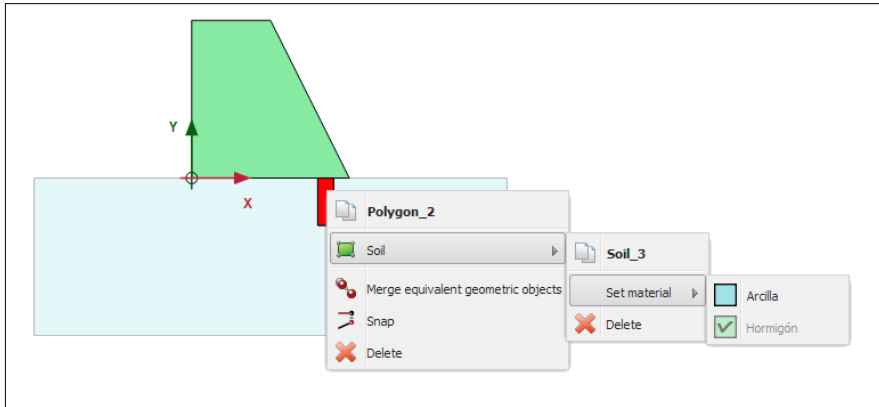


Figure 4. 32. 'Concrete' assignation to sheet pile

Now we go to 'Mesh' geometry mode. We click on 'Generate mesh' button, choosing a very fine mesh. We create a single phase as in the previous problem (Figure 4.17). In 'Calculation type', we select 'Flow only' and in 'Pore pressure calculation type', we choose, 'Steady state groundwater flow'. Now, we can calculate.

4.4.2. Flow net results with sheet pile

Figures 4.33 and 4.34 show both equipotential lines and water velocity vectors.

Exercise 04. Flow net under concrete dam

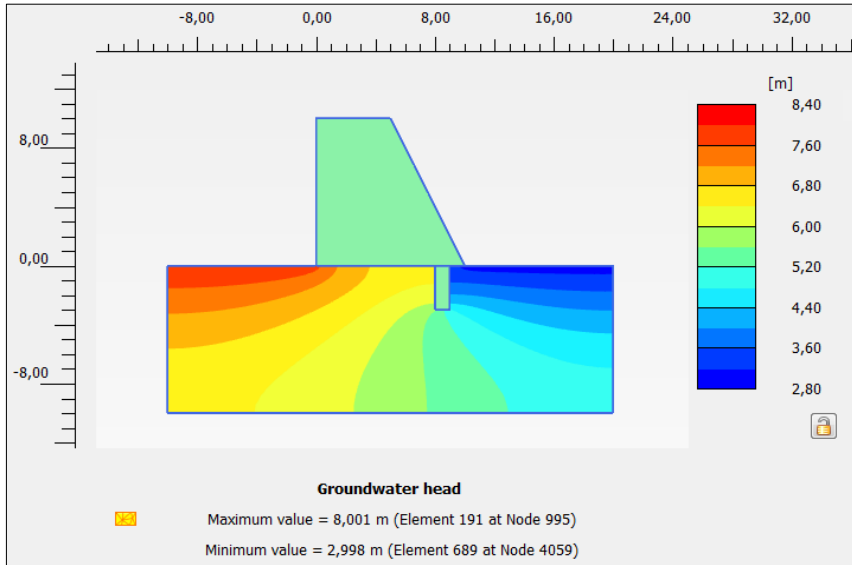


Figure 4. 33. Equipotential lines for dam with sheet piles

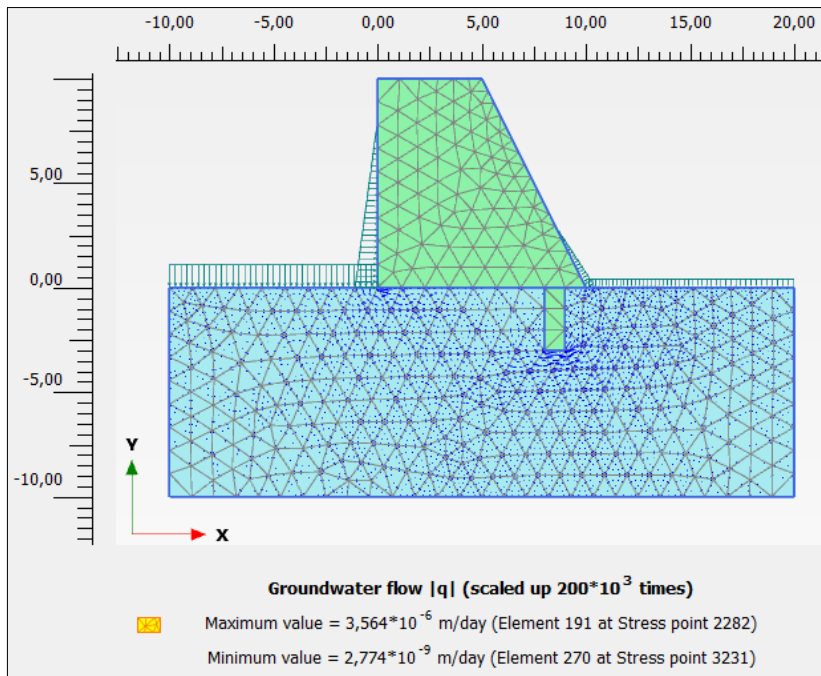


Figure 4. 34. Water velocity vectors under the dam with sheet piles

Exercise 04. Flow net under concrete dam

In this case, the outflow value ('Total discharge') is approximately $4,16 \cdot 10^{-6}$ m³/day/m (Figure 4.35). This means that the construction of the sheet piles considerably reduces the value of infiltrated flow.

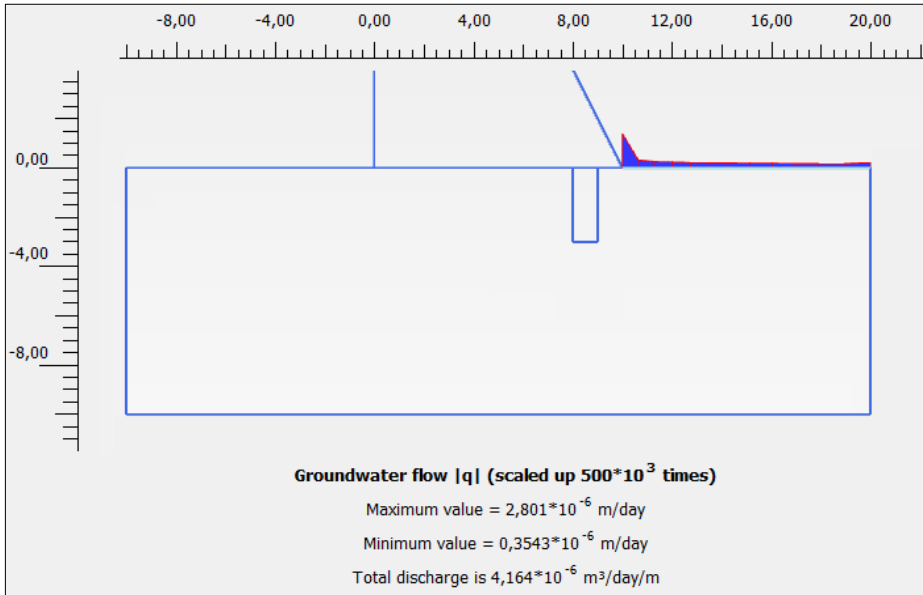


Figure 4. 35. Outflow per meter of dam thickness with sheet pile

Exercise 05

Shallow foundations on clay soil

5.1. INTRODUCTION

A five-floor building (ground floor + 3 middle slabs + attic) is built on an eight-meter thick clay layer. In order to know the load that is applied to each pile, and, therefore, to each footing, a study has been developed. Each footing row has three of them, so the simplified problem we are going to study has three footings. These, which are on natural soil, have 3.7 meters wide and 0.8 meters thick. The distance between the centres of the footings is 7 meters. It is considered that it is enough to study a ground length of 28 meters (Figure 5.1).

The ground floor and each of the slabs applies a load of 140 kN/m per footing (each floor), while the attic only applies 40 kN/m. in this way, once the construction is finished, the whole load each footing must bear is 600 kN/m.

The aim of this problem is to know if the soil can bear the load transmitted by the building (representing displacements and stresses, as well as safety factor). If it is not able, we will obtain its bearing capacity. The phreatic level is located on the ground surface.

5.2. PROBLEM PARAMETERS

The geometry of the problem is presented in Figure 5.1.

Exercise 05. Shallow foundations on clay soil

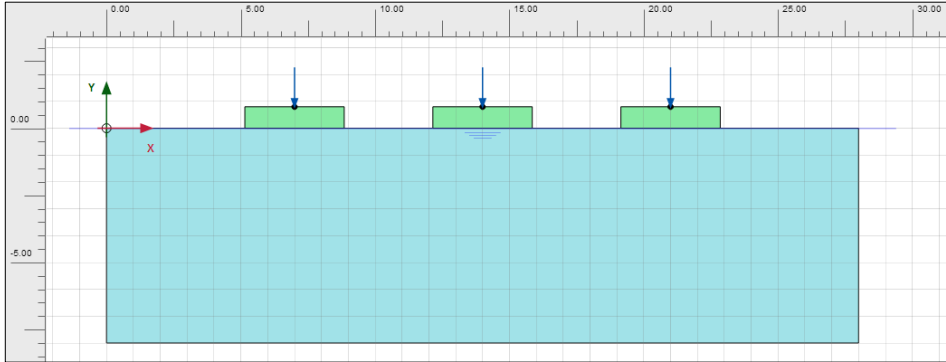


Figure 5. 1. Geometry of the problem in PLAXIS 2D

The geotechnical parameters of the soil where the building is placed are shown in Table 5.1.

Geotechnical parameters

Material	Clay
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	16.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	19 kN/m ³
Young's modulus (E')	47000 kPa
Poisson's ratio (ν')	0.33
Effective soil cohesion (c')	4.5 kPa
Effective friction angle (ϕ')	22°
Dilatancy angle (ψ)	0°

Table 5. 1. Soil geotechnical parameters

The properties of the concrete that is employed to build the footings are presented in Table 5.2.

Concrete footing	
Material	Concrete
Constitutive model	Linear elastic
Material behaviour	Non- porous
Soil unit weight above phreatic level (γ_{unsat})	24 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	24 kN/m ³
Young's modulus (E')	200·10 ⁶ kPa
Poisson's ratio (ν')	0.15

Table 5. 2. Parameters of footing concrete

5.3 RESOLUTION OF THE PROBLEM WITH PLAXIS 2D

5.3.1 Introduction of the problem parameters

Once we start PLAXIS Input, we choose ‘*Start a new project*’ option in ‘*Quick select*’ emerging window. In Title bar of ‘*Project*’ tab we write, for example: ‘*Footings on clay soil*’.

In ‘*Model*’ tab we select ‘*Plane strain*’ for model and ‘*15-Node*’ for elements. We keep the default units (Figure 5.2).



Figure 5. 2. Problem units

The project contour is defined as shown in Figure 5.3.

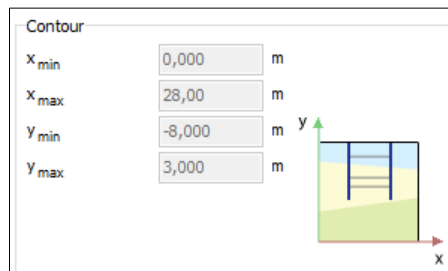


Figure 5. 3. Problem contours

We keep the default values of constants (‘*Constants*’ tab).

5.3.2. Soil definition

In order to define the soil below the building, we go to ‘Soil’ geometry mode and select ‘Create borehole’ button. We select point (0, 0) and click on ‘Add’. In ‘Top’ we write 0, and -8 in ‘Bottom’. The phreatic level is located at the ground surface, so in ‘Head’ we write 0 (Figure 5.4).

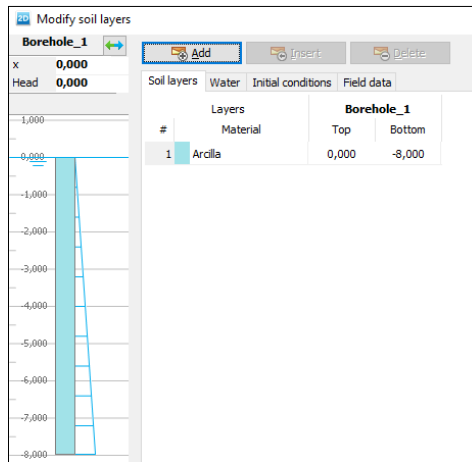


Figure 5. 4. First borehole

Now we create the material to model the soil the footings are built on. For this, we select ‘Show material’ button. We click on ‘New’ and introduce the geotechnical parameters presented in Table 5.1 (Figures 5.5 and 5.6).

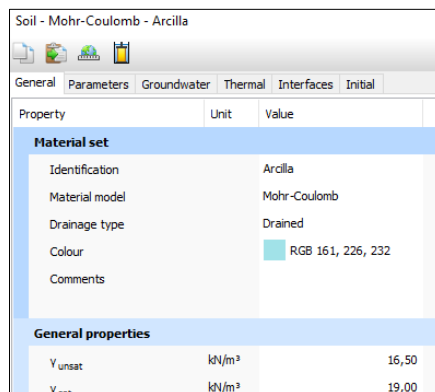


Figure 5. 5. General properties of clay soil

Exercise 05. Shallow foundations on clay soil

Soil - Mohr-Coulomb - Arcilla		
General Parameters Groundwater Thermal Interfaces Initial		
Property	Unit	Value
Stiffness		
E'	kN/m ²	47,00E3
ν' (nu)		0,3300
Alternatives		
G	kN/m ²	17,67E3
E_{oed}	kN/m ²	69,64E3
Strength		
c'_{ref}	kN/m ²	4,500
ϕ' (phi)	°	22,00
ψ (psi)	°	0,000
Velocities		
V_s	m/s	102,5
V_p	m/s	203,5

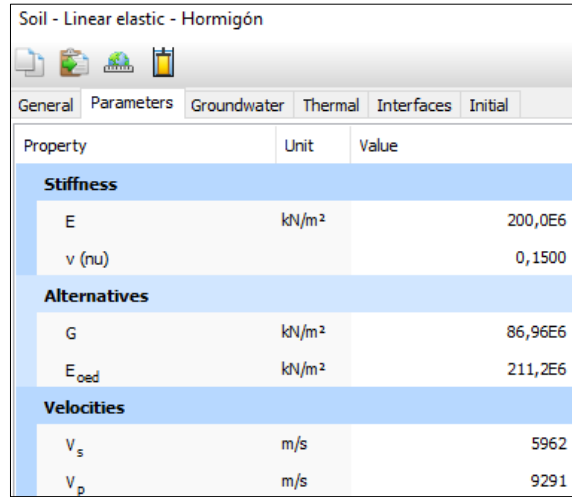
Figure 5. 6. Geotechnical parameters of clay soil

We select again 'New' and introduce concrete resistant parameters shown in Table 5.2, to model the footing material (Figure 5.7 and 5.8).

Soil - Linear elastic - Hormigón		
General Parameters Groundwater Thermal Interfaces Initial		
Property	Unit	Value
Material set		
Identification		Hormigón
Material model		Linear elastic
Drainage type		Non-porous
Colour		RGB 134, 234, 162
Comments		
General properties		
γ_{unsat}	kN/m ³	24,00
γ_{sat}	kN/m ³	24,00

Figure 5. 7. General properties of concrete

Exercise 05. Shallow foundations on clay soil



Property	Unit	Value
Stiffness		
E	kN/m ²	200,0E6
v (nu)		0,1500
Alternatives		
G	kN/m ²	86,96E6
E _{oed}	kN/m ²	211,2E6
Velocities		
V _s	m/s	5962
V _p	m/s	9291

Figure 5. 8. Resistant parameters of concrete.

Clicking with the right mouse button on the created soil, we select clay as the material to model it.

5.3.3. Definition of the three footings

We go to 'Structures' geometry mode, click on 'Create soil polygon' and then on 'Create soil rectangle' (Figure 5.9).

Exercise 05. Shallow foundations on clay soil

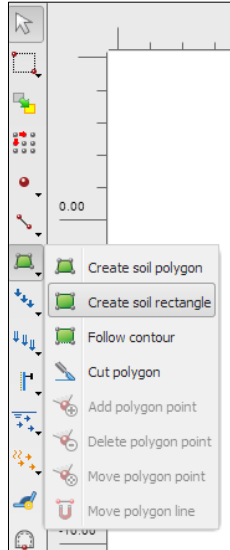


Figure 5. 9. 'Create soil rectangle' button to create a rectangle shaped structure

In the command bar we write (after '_polygon'): '(5.15 0) (8.85 0) (8.85 0.8) (5.15 0.8)' and press ENTER. The first footing is drawn according to these coordinates. Another option is to click on the workspace, first on point (5.15, 0) and then on point (8.85, 0.8). For this second option, we must indicate 20 in 'Number of snap interval' (from 'Snapping option' right below the workspace).

The second footing is created by writing '(12.15 0) (15.85 0) (15.85 0.8) (12.15 0.8)' in the command bar (after '_polygon'), and for the third one, we write '(19.15 0) (22.85 0) (22.85 0.8) (19.15 0.8)'.

This three footing we have just introduced are shown in Figure 5.10. Once they are created, we select concrete as the material to model them (Figure 5.11).

Exercise 05. Shallow foundations on clay soil

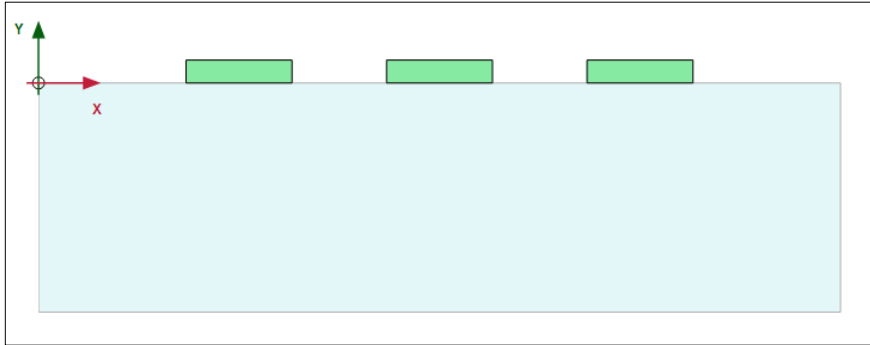


Figure 5. 10. Footings created on soil

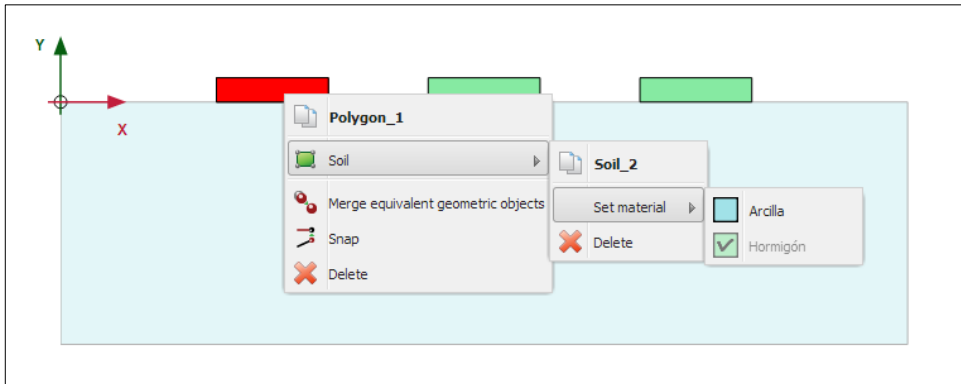


Figure 5. 11. Definition of concrete as footing material

5.3.4. Definition of the punctual loads applied to the footings

We select 'Create load' button and click on 'Create point load' (Figure 5.12).

Exercise 05. Shallow foundations on clay soil

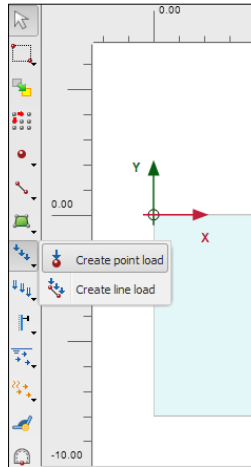


Figure 5. 12. Creation of punctual loads on footings

We click on coordinates (7; 0,8), (14; 0,8) and (21; 0,8), which are the centres of the footing. The three punctual loads are now created on each footing (Figure 5.13).

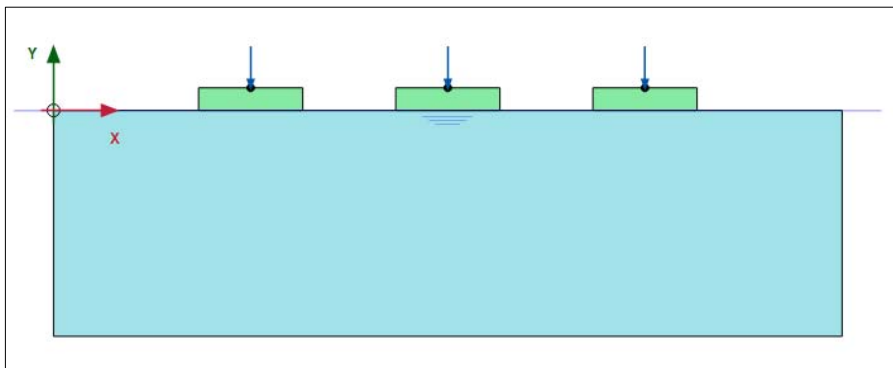


Figure 5. 13. Punctual loads on footings

5.3.5. Problem meshing

In 'Mesh' calculation mode, we click on 'Generate mesh' button, indicating a medium element distribution.

If we want to display the created mesh in PLAXIS Output (Figure 5.14) we select 'View mesh' button.

Exercise 05. Shallow foundations on clay soil

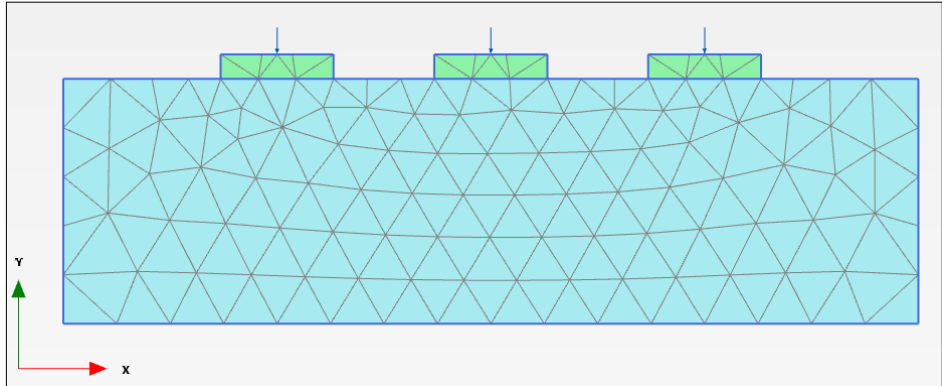


Figure 5. 14. Problem meshing.

5.3.6. Project phases

We define the following project phases:

- Initial Phase. Only clay soil.
- Phase 1. Footings are built.
- Phase 2. Ground floor slab is applied (first slab).
- Phase 3. Ground and first floor slabs are applied (second slab).
- Phase 4. Ground, first and second floor slab are applied (third slab).
- Phase 5. Ground, first, second and third floor slab are applied (fourth slab).
- Phase 6. All loads are applied, including attic.
- Phase 7. ‘Safety’ Phase to know the safety factor after the attic construction.

For the Initial Phase, in ‘*Staged construction*’ Calculation mode, we must deactivate the three footings, as well as the punctual loads applied on them.

We keep the characteristics that appear in Figure 5.15. The calculation type is the default one, ‘*K0 procedure*’, and the load type is also the default type, ‘*Staged construction*’.

Exercise 05. Shallow foundations on clay soil

General	
ID	Initial phase [InitialPhase]
Calculation type	K0 procedure
Loading type	Staged construction
ΣM weight	1,000
Pore pressure calculation type	Phreatic
Thermal calculation type	Ignore temperature
First step	0
Last step	0
Design approach	(None)

Figure 5. 15. Initial Phase characteristics

We keep 'Ignore temperature' selected in thermal calculation type and 'Phreatic' in pore pressure calculation type. Now, we create Phase 1 and activate the three footings by clicking on each of them with the right mouse button (Figure 5.16). Figure 5.17 shows the general characteristics in Phase 1 (with 'Plastic' as calculation type).

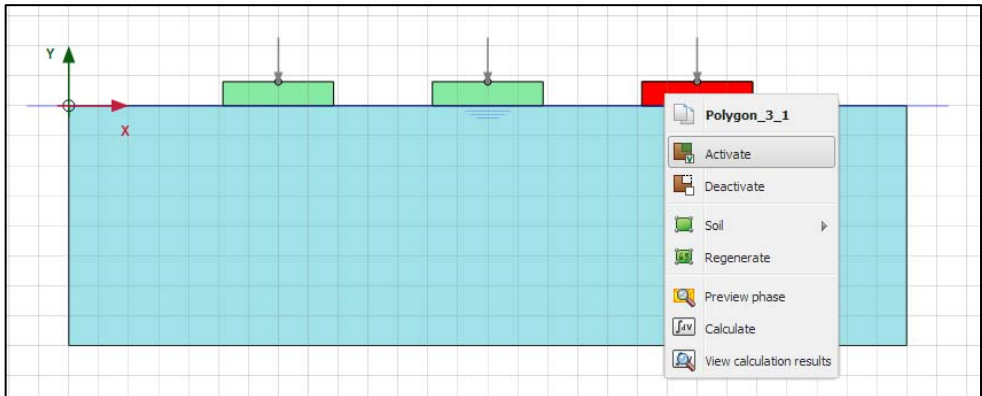


Figure 5. 16. Activation of third footing in Phase 1

Exercise 05. Shallow foundations on clay soil

General	
ID	Zapata [Phase_1]
Start from phase	Initial phase ▼
Calculation type	Plastic ▼
Loading type	Staged construction ▼
ΣM_{stage}	1,000
ΣM_{weight}	1,000
Pore pressure calculation type	Phreatic ▼
Thermal calculation type	Ignore temperature ▼
Time interval	0,000 day
First step	1
Last step	3
Design approach	(None) ▼

Figure 5. 17. Phase 1 ‘Footing’ characteristics

In Phase 2 (called, for example, ‘First slab’), we activate the three punctual loads (Figure 5.18). In Figure 5.19, the general characteristics for this phase (‘*Plastic*’ type) are presented.

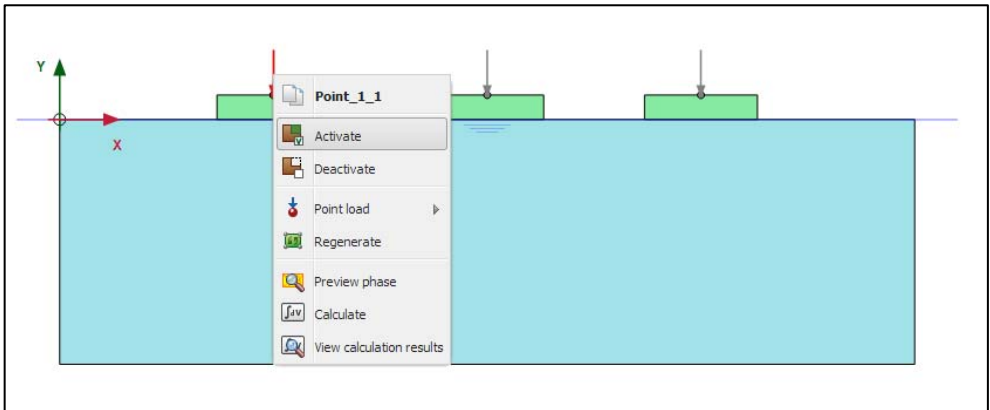


Figure 5. 18. First footing punctual load activation, in Phase 2 (first slab)

Exercise 05. Shallow foundations on clay soil

General	
ID	1er forjado [Phase_2]
Start from phase	Zapata
Calculation type	Plastic
Loading type	Staged construction
ΣM_{stage}	1,000
ΣM_{weight}	1,000
Pore pressure calculation type	Phreatic
Thermal calculation type	Ignore temperature
Time interval	0,000 day
First step	12
Last step	18
Design approach	(None)

Figure 5. 19. Phase 2 characteristics (first slab)

Finally, we have to assign a value to each of the loads. For this, we select the load in the workspace and in ‘*Selection explorer*’ menu on the left we write -140 (kN/m) in $F_{y,ref}$ (Figure 5.20). We can see here that the load is a punctual one (‘*PointLoad_1_1*’).

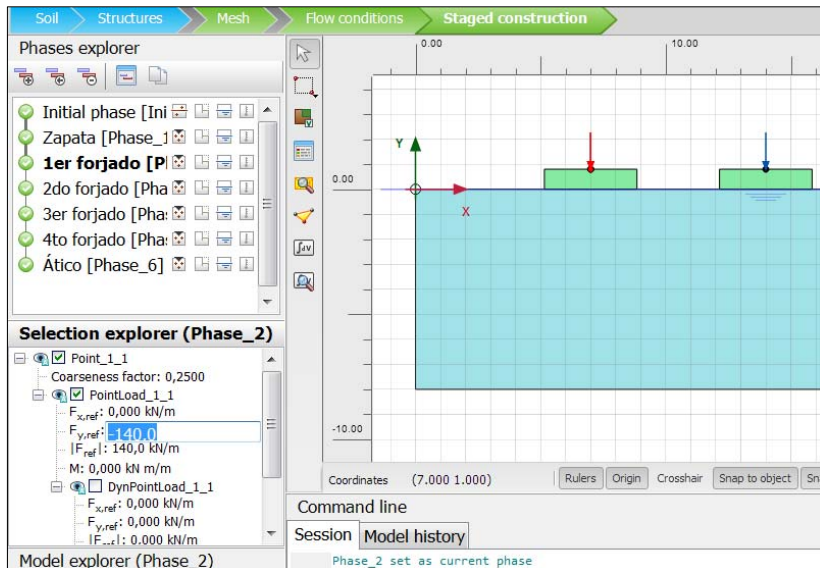


Figure 5. 20. First slab load assignation in ‘*Selection explorer*’

Exercise 05. Shallow foundations on clay soil

We click on ‘Add phase’ button to create Phase 3 (called ‘Second slab’). We activate the three punctual load and give them a value of -280 (kN/m), as presented in Figure 5.21. The characteristics are the same as in Phase 2.

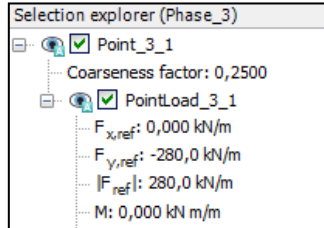


Figure 5. 21. Second slab load assignment

In order to create Phase 4 (called ‘Third slab’) we keep the loads activated and increase their value up to -420 kN/m. The characteristics of this phase are the same as in Phases 2 and 3.

We add Phase 5 (called ‘Fourth slab’), keeping the loads activated and increasing them up to -560 kN/m (Figure 5.22). This phase has the same characteristics as the previous one.

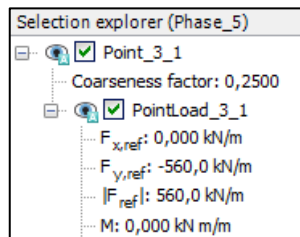


Figure 5. 22. Fourth slab load assignment

Now, we add Phase 6 (called ‘Attic’) and keep the three loads activated, giving them a value of -600 kN/m (Figure 5.23), and again the characteristics are the same as in the previous phase.

Exercise 05. Shallow foundations on clay soil

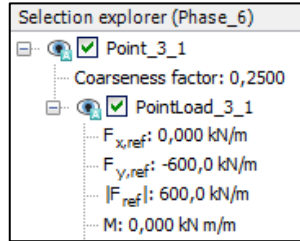


Figure 5. 23. Attic Phase load assignation

Finally, we can define ‘Safety’ Phase, right after ‘Attic’ one. For this, we select ‘Add phase’ button to create Phase 7 (called, ‘Safety’). Its characteristics appear in Figure 5.24 (‘Safety’ calculation type, with ‘Incremental multipliers’ loading type).

General	
ID	Safety [Phase_7]
Start from phase	Ático
Calculation type	Safety
Loading type	Incremental multipliers
M_{sf}	0,1000
Pore pressure calculation type	Use pressures from previous ph
Thermal calculation type	Ignore temperature
First step	
Last step	
Design approach	(None)
Deformation control parameters	
Ignore undr. behaviour (A,B)	<input type="checkbox"/>
Reset displacements to zero	<input type="checkbox"/>
Reset small strain	<input type="checkbox"/>
Reset state variables	<input type="checkbox"/>
Updated mesh	<input type="checkbox"/>
Updated water pressure	<input type="checkbox"/>
Ignore suction	<input checked="" type="checkbox"/>
Cavitation cut-off	<input type="checkbox"/>
Cavitation stress	100,0 kN/m ²
Numerical control parameters	
Max cores to use	256
Max number of steps stored	50
Use default iter parameters	<input checked="" type="checkbox"/>

Figure 5. 24. ‘Safety’ Phase characteristics

5.3.7. Selection of points for advanced graphical outputs and model calculation

Before calculating, we select a group of points under the footings to later plot advanced graphical outputs (Figure 5.25). Their coordinates are presented in Figure 5.26, where we can see that there are two different kinds of points: ‘Nodes’, that are used to plot displacements; and ‘Stress points’ that are used to represent stresses and deformations.

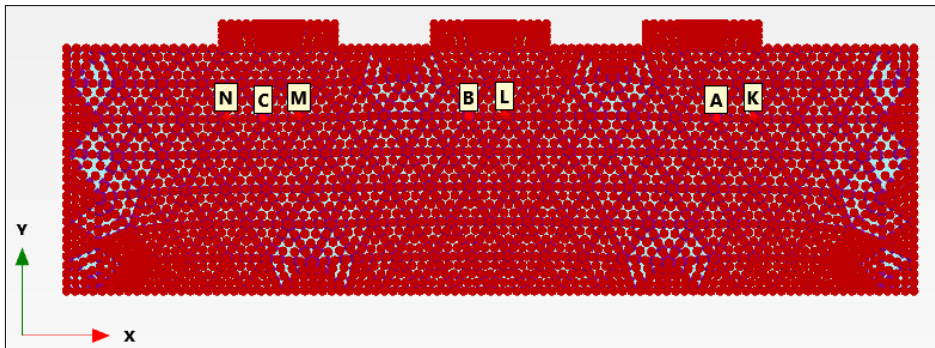


Figure 5. 25. Selection of points to plot advanced graphical outputs

Name	X	Y	Selected
A (Node 2229)	21,48	-2,30	Pre-calc
B (Node 1389)	13,28	-2,25	Pre-calc
C (Node 607)	6,48	-2,31	Pre-calc
K (Stress point 878)	22,69	-2,25	Pre-calc
L (Stress point 1694)	14,48	-2,18	Pre-calc
M (Stress point 1009)	7,67	-2,23	Pre-calc
N (Stress point 939)	5,29	-2,25	Pre-calc

Figure 5. 26. Coordinates of points for advanced graphical outputs

The last step to obtain results is clicking on ‘Calculate’ button and wait for the program to calculate each phase. We can display the results in PLAXIS Output by selecting ‘View calculation results’.

5.3.8. Results in PLAXIS Output

As we can see, the calculation of all phases is correct, so we can accept that all the loads applied during the building construction are compatible with the soil and the foundation solution that is proposed.

Exercise 05. Shallow foundations on clay soil

Another common way to check this is by plotting an advanced graphical output ('Curves manager') of ΣM_{stage} project variable. In this way, for example, Figure 5.27 show the evolution point A's vertical displacement (u_y) versus ΣM_{stage} .

For this, we select the advanced graphical output option as it appears in Figure 5.28. Once the graphic is plotted, it is very important to keep in mind that the default option in 'Curves manager' is to plot all project phases, including 'Safety' ones. Nevertheless, as we have already commented, displacement information in this type of phases is not valid, so we delete it from the graphic. We can do this by clicking on the graphic with the right mouse button and selecting 'Settings...'. After this, a window appears (Figure 5.29), where we mark point A tab ('N2229') and select 'Phases'. Finally, we unmark 'Safety' Phase (Figure 5.30), obtaining Figure 5.27 as presented in this document.

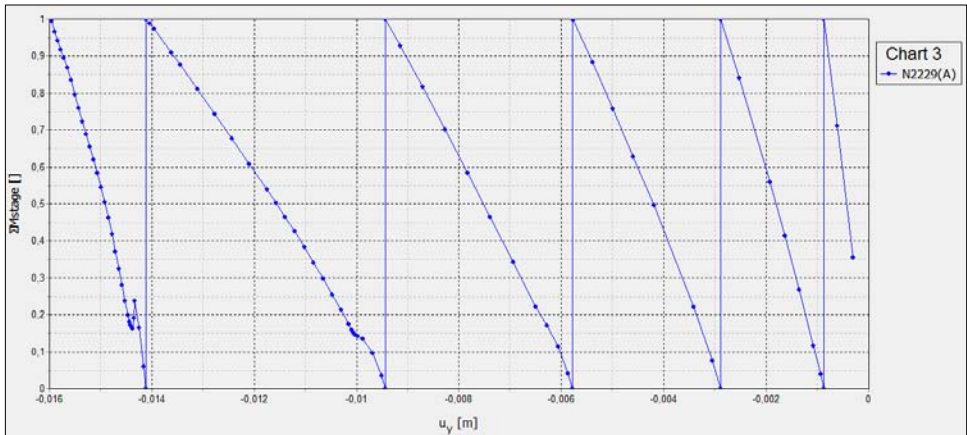


Figure 5. 27. Point A's vertical displacement versus ΣM_{stage}

Exercise 05. Shallow foundations on clay soil

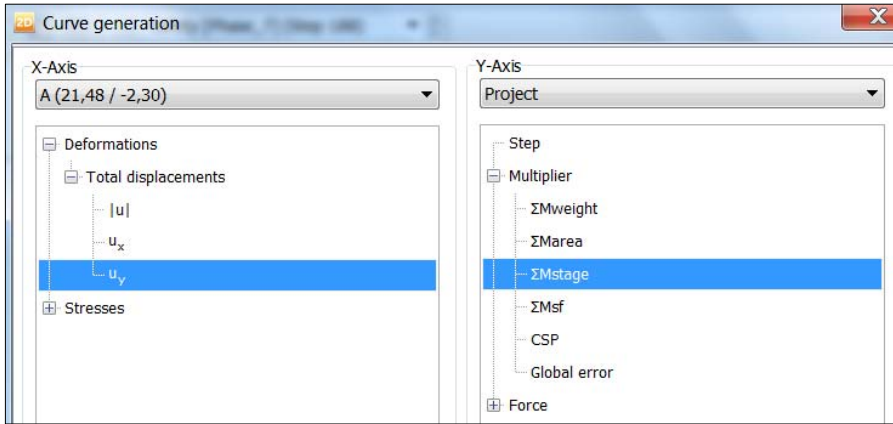


Figure 5. 28. Selection of options to plot point A's vertical displacement versus ΣM_{stage}

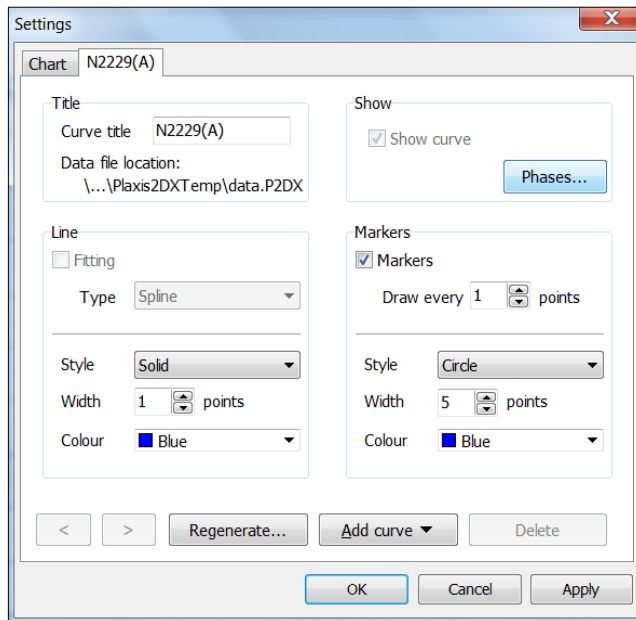


Figure 5. 29. Advanced graphical output settings window

Exercise 05. Shallow foundations on clay soil

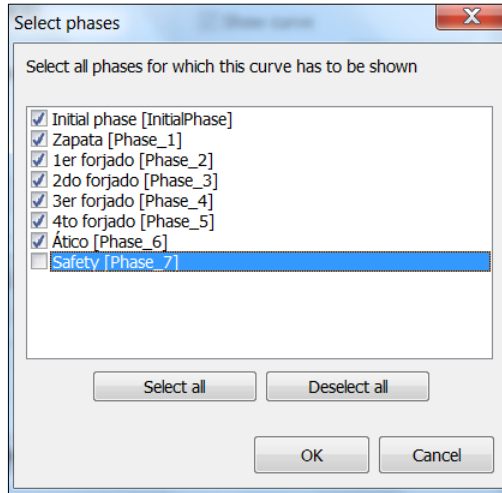


Figure 5. 30. Selection of phases to represent in the advanced graphical output

In Figure 5.27 we can observe how the program has been able to calculate 100% of all phases (ΣM_{stage} value is 1); this is, soil and footings bear all stresses and displacements of each of the phases. PLAXIS solves each phase by 'Steps', so ΣM_{stage} increases its value as the phase is calculated. For each of these steps, PLAXIS solves the governing equation and obtains the values for the output variables (that is the reason why we can see the displacement evolution). Once it reaches 100% of a phase (ΣM_{stage} takes a value of 1), the next phase is started (ΣM_{stage} value is 0). In Figure 5.27 we see how point A' vertical displacement, when Phase 6 'attic' is finished, takes a value of approximately 1.6 cm.

Now, we are going to plot ΣM_{stage} versus total vertical stresses, (σ_{yy}), for points K and L. For this, we open again 'Curves manager' and click on the options shown in Figure 5.31 (to plot point K).

Exercise 05. Shallow foundations on clay soil

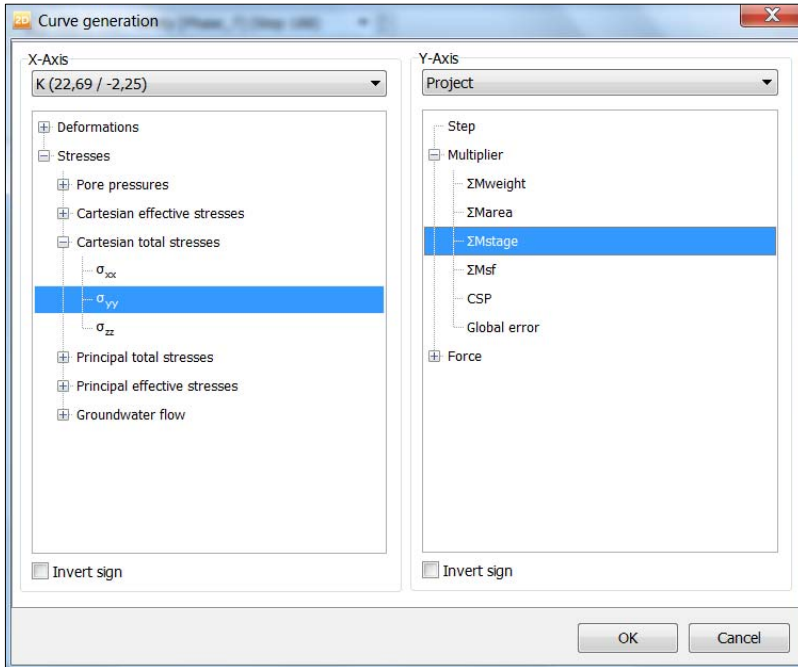


Figure 5. 31. Selection of points to plot total vertical stresses for point K versus ΣM_{stage}

With 'Settings' option we delete again 'Safety' Phase, modify other plotting options as well, such as line colour and markers (we change them into black), among others.

In order to add point L, we click on the graphic with the right mouse button, select 'Add curve' option, and afterwards, 'From current Project...'. Here, we select the same options as for point K. Once it is plotted, we also delete 'Safety' Phase for this point and, if we want, change line and mark colour to red, obtaining the graphic shown in Figure 5.32.

Exercise 05. Shallow foundations on clay soil

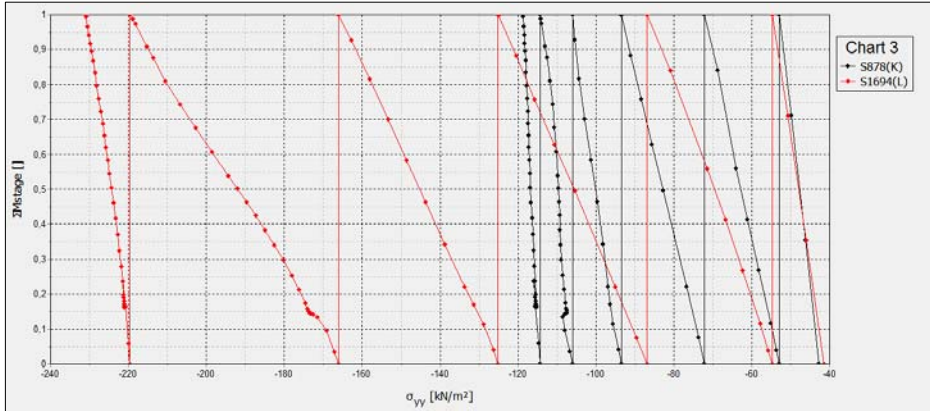


Figure 5.32. Total vertical stresses in points K and L versus ΣM_{stage}

As example, Figures 5.33, 5.34 and 5.35 show total displacements, $|u|$, in Phases 2, 4 and 6, respectively.

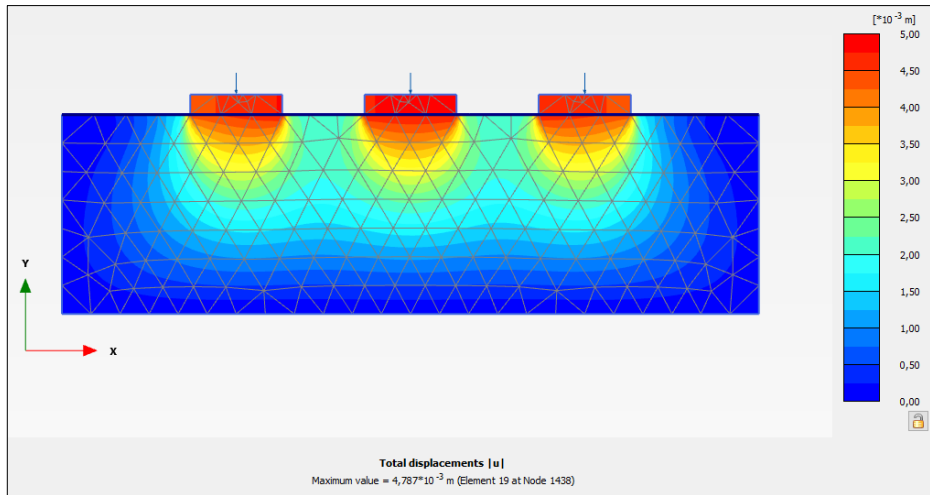


Figure 5.33. Total displacements, $|u|$, in Phase 2 (1st slab)

Exercise 05. Shallow foundations on clay soil

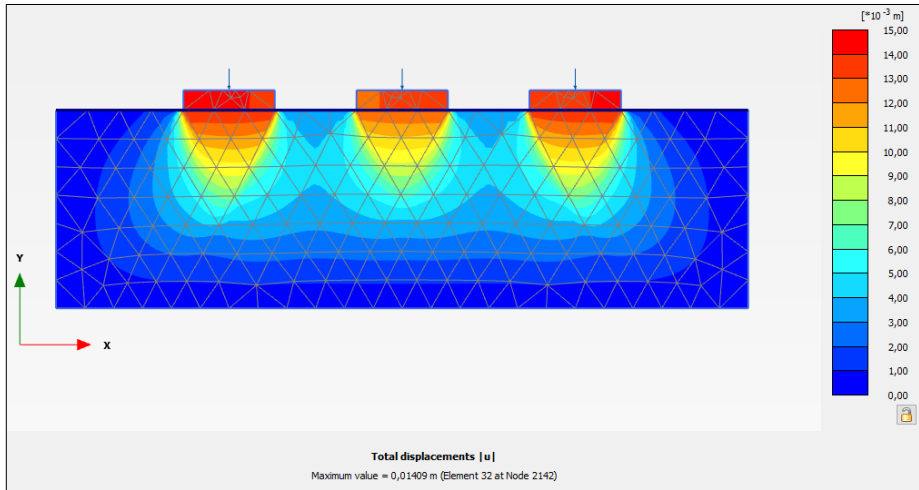


Figure 5. 34. Total vertical displacement, $|u|$, in Phase 4 (3rd slab)

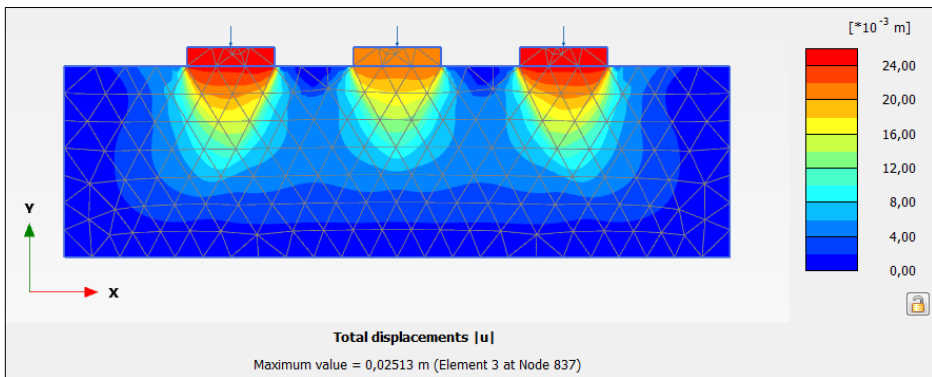


Figure 5. 35. Total displacements, $|u|$, in Phase 6 (attic)

It is visible that, when Phase 6 is finished, displacements under the central footing are lower than under the other two. It might be more representative to show the incremental displacements, $|\Delta u|$, in Phases 2, 4 and 6 (Figures 5.36, 5.37 and 5.38).

Exercise 05. Shallow foundations on clay soil

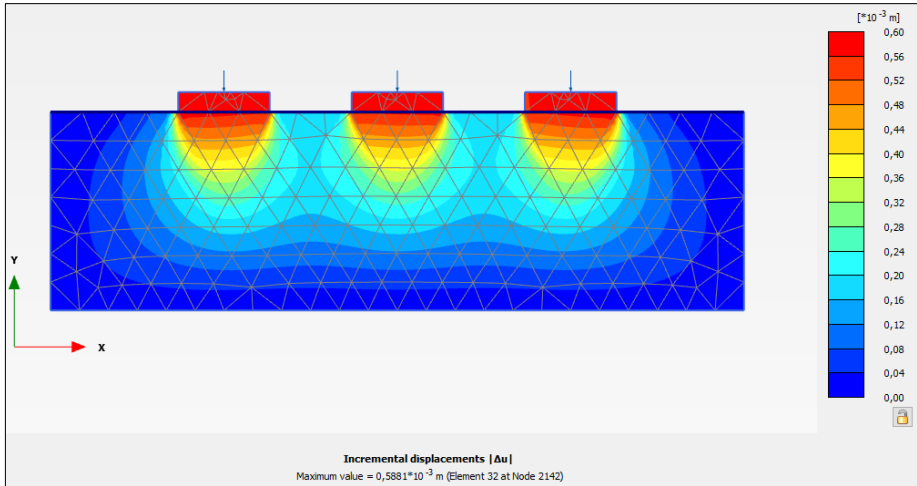


Figure 5. 36. Total incremental displacements, $|\Delta u|$, in Phase 2 (1st slab)

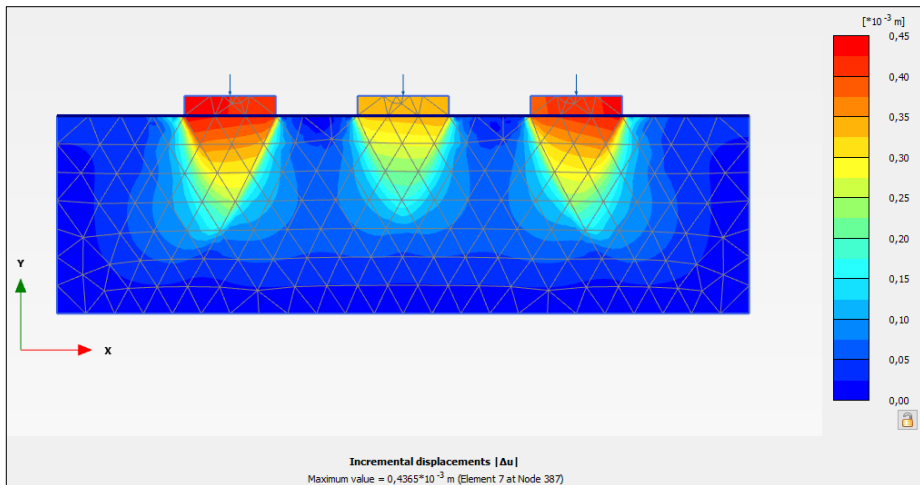


Figure 5. 37. Total incremental displacements, $|\Delta u|$, in Phase 4 (3rd slab)

Exercise 05. Shallow foundations on clay soil

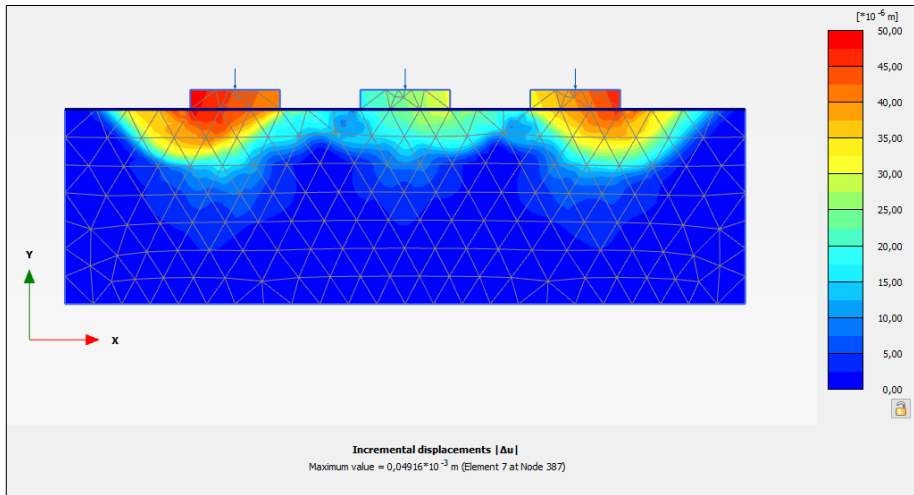


Figure 5.38. Total incremental displacements, $|\Delta u|$, in Phase 6 (attic)

Incremental displacements show the displacement difference that exists between a phase and the previous one. Therefore, for Phase 6 (Figure 5.38) it is visible the high difference of incremental displacement, $|\Delta u|$, under the central footing and the other ones. This is, between Phase 5 and Phase 6 the largest part of the deformation is produced under the outer footings (which means that these are closer to failure).

In this way, we look for the safety factor value that we have once the attic is built. For this, selecting 'Safety' Phase (after Phase 6 'Attic'), we click on 'Project' and then on 'Calculation information', obtaining a safety factor value of 1.019 (Figure 5.39).

NOTE. 'Calculation information' window always shows ΣM_{sf} value, no matter which phase is selected. However, this value is only valid when we are on a 'Safety' Phase, in which the program applies 'phi-c strength reduction method' to calculate it (see exercise one) once 100% of the previous phase has been reached. For the other phases (for example, 'Plastic' type ones), ΣM_{sf} value is never over 1, since the maximum possible value reached during the calculation is 100% of this phase.

Exercise 05. Shallow foundations on clay soil

Calculation information				
Step info				
Phase	Safety [Phase_7]			
Step	Initial			
Calculation mode	Classical mode			
Step type	Safety			
Updated mesh	False			
Solver type	Picos			
Kernel type	32 bit			
Extrapolation factor	1,000			
Relative stiffness	-0,02877E-6			
Multipliers				
Soil weight			ΣM_{Weight}	1,000
Strength reduction factor	M_{sf}	-0,04651E-3	ΣM_{sf}	1,019
Time	Increment	0,000	End time	0,000
Staged construction				
Active proportion total area	M_{Area}	0,000	ΣM_{Area}	1,000
Active proportion of stage	M_{Stage}	0,000	ΣM_{Stage}	0,000

Figure 5. 39. Safety factor ΣM_{sf} (for ‘attic’ Phase) in ‘Calculation information’ window

As the safety factor that is obtained is 1.019, a small load increase would suppose a failure in the soil underneath. If we plot total incremental displacements, $|\Delta u|$, for ‘Safety’ Phase, we can see, an approximation of the failure surface under the outer footings (Figure 5.40). The safety factor is so close to the unit that the incremental displacements under the central footings are negligible, and all this deformation is basically located under the footings that fail.

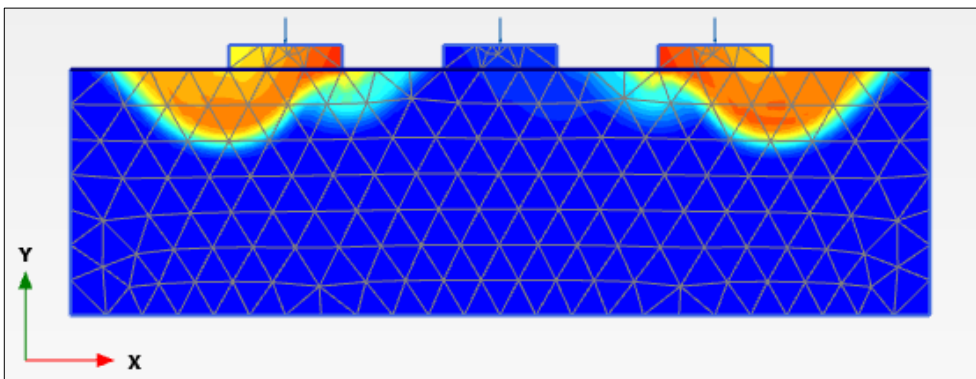


Figure 5. 40. Total incremental displacements, $|\Delta u|$, (approximate slip surface) under the outer footings for ‘Safety’ Phase.

The next point presents a variation of the problem: instead of an attic of 40 kN/m, a fifth slab of 140 kN/m load is built, so the soil will fail because of this load. In this way, we can obtain the bearing capacity. We save this project with an adequate name.

5.3.9. Construction of a fifth slab instead of an attic

We go to ‘*Staged construction*’ calculation mode, double click on Phase 6 (previously called ‘Attic’) and change its name in ‘*Phases*’ window (for example ‘Fifth slab’). We keep the same characteristics as in ‘Attic’ Phase (Figure 5.41.)

General	
ID	5to forjado [Phase_6]
Start from phase	4to forjado
Calculation type	Plastic
Loading type	Staged construction
ΣM_{stage}	1,000
ΣM_{weight}	1,000
Pore pressure calculation type	Phreatic
Thermal calculation type	Ignore temperature
Time interval	0,000 day
First step	57
Last step	88
Design approach	(None)

Figure 5. 41. General characteristics of the new Phase 6 (fifth slab)

In ‘*Selection explorer*’ menu, we keep the three punctual loads selected, but we change $F_{y,ref}$ value to -700 kN/m (Figure 5.42).

Selection explorer (Phase_6)	
<input checked="" type="checkbox"/>	Selection
	Coarseness factor: 0,2500
<input checked="" type="checkbox"/>	PointLoad
	$F_{x,ref}$: 0,000 kN/m
	$F_{y,ref}$: -700,0 kN/m
	$ F_{ref} $: 700,0 kN/m
	M: 0,000 kN m/m

Figure 5. 42. Punctual load value in the last ‘*Plastic*’ Phase ‘Fifth slab’

Exercise 05. Shallow foundations on clay soil

We delete the ‘Safety’ Phase we had, clicking with the right mouse button on the phase and select ‘Delete’. Now, we calculate. In Figure 5.42 we can appreciate that the soil failure has happened in this new phase.

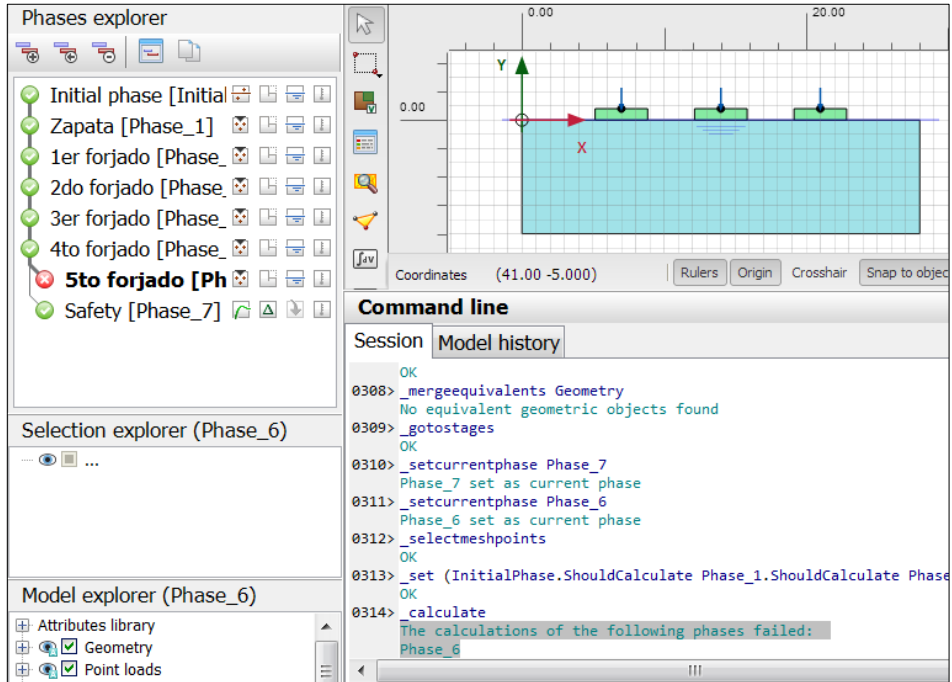


Figure 5. 43. Indication of calculation failure in Phase 6

If we want to know more about the soil failure in this new phase, we can represent ΣM_{stage} evolution versus vertical displacement, u_y , in point A. We can also obtain the safety factor after building the fourth slab.

For this, we create a ‘Safety’ Phase after Phase 5 (‘Fourth slab’). Clicking on Phase 5, we select ‘Add phase’ in the ‘Phases explorer’ menu on the left. Once it is created, we edit it as presented in Figure 5.44.

Exercise 05. Shallow foundations on clay soil

General	
ID	Safety [Phase_7]
Start from phase	4to forjado ▼
Calculation type	Safety ▼
Loading type	Incremental multipliers ▼
M _{sf}	0,1000
Pore pressure calculation type	Use pressures from previous ph ▼
Thermal calculation type	Ignore temperature ▼
First step	
Last step	
Design approach	(None) ▼
Deformation control parameters	
Ignore undr. behaviour (A,B)	<input type="checkbox"/>
Reset displacements to zero	<input type="checkbox"/>
Reset small strain	<input type="checkbox"/>
Reset state variables	<input type="checkbox"/>
Updated mesh	<input type="checkbox"/>
Updated water pressure	<input type="checkbox"/>
Ignore suction	<input checked="" type="checkbox"/>
Cavitation cut-off	<input type="checkbox"/>
Cavitation stress	100,0 kN/m ²
Numerical control parameters	
Max cores to use	256
Max number of steps stored	50

Figure 5. 44. Selection of options in ‘Safety’ Phase after the fourth slab

We recalculate all phases and go to PLAXIS Output. Figure 5.45 shows total incremental displacements, $|Δu|$, in Phase 6 (fifth slab). In this we can observe the failure mechanism of one of the outer footings (in this case, although it is a symmetric problem, the failure only appears in one the outer footings because of a calculation internal mechanism of the program).

Exercise 05. Shallow foundations on clay soil

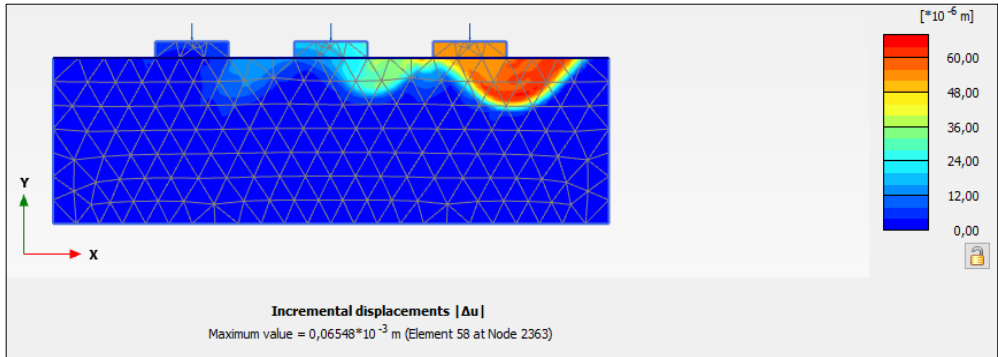


Figure 5. 45. Total incremental displacement, $|\Delta u|$, in Phase 6 (fifth slab)

In order to obtain safety factor in Phase 5 (fourth slab), we use the information in ‘Project’, ‘Calculation information’ (Figure 5.46).

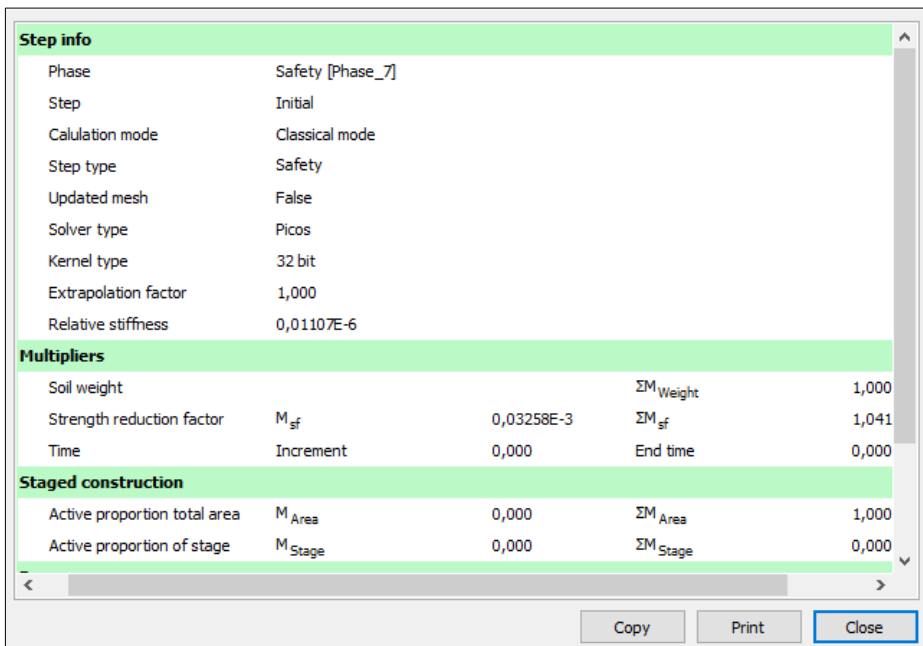


Figure 5. 46. Safety factor after the construction of the fourth slab

Figure 5.47 shows the value that ΣM_{stage} takes versus vertical displacements, u_y , in point A (delete ‘Safety’ representation in ‘Settings’). As we see below, ΣM_{stage} value is very useful to know soil bearing capacity.

Exercise 05. Shallow foundations on clay soil

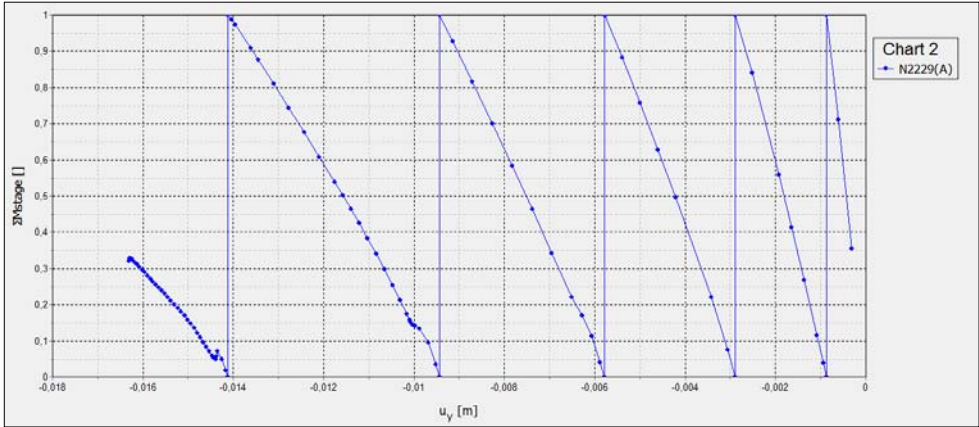


Figure 5. 47. ΣM_{stage} versus vertical displacements, u_y , for point A

As we can observe, the last phase is not complete, reaching 32% of the load due to the fifth slab.

It is also possible to obtain the exact value of ΣM_{stage} in a phase (in this case, Phase 6 ‘Fifth slab’). For this, we go to ‘Project’, ‘Calculation information’ (Figure 5.48).

Calculation information				
Step info				
Phase	Sto forjado [Phase_6]			
Step	Initial			
Calculation mode	Classical mode			
Step type	Plastic			
Updated mesh	False			
Solver type	Picos			
Kernel type	32 bit			
Extrapolation factor	1,000			
Relative stiffness	-1,897E-3			
Multipliers				
Soil weight			ΣM_{Weight}	1,000
Strength reduction factor	M_{sf}	0,000	ΣM_{sf}	1,000
Time	Increment	0,000	End time	0,000
Staged construction				
Active proportion total area	M_{Area}	0,000	ΣM_{Area}	1,000
Active proportion of stage	M_{stage}	-1,760E-3	ΣM_{Stage}	0,3193

Figure 5. 48. ΣM_{stage} value

Exercise 05. Shallow foundations on clay soil

We can see that the reached ΣM_{stage} value in Phase 6 is 0.3193. Finally, we can calculate the bearing capacity.

$$\frac{Q_u}{B} = \frac{F_{max}}{B} + (\gamma_{concrete} * d)$$

If 'B' is the footing width and 'd' its thickness, then F_{max} :

$$F_{max} = Q_{in} + \Sigma M_{stage} * (Q_{fin} - Q_{in})$$

where:

Q_{in} : load when the failure phase starts.

Q_{fin} : load that was supposed to be reached at the end of the failure phase.

Therefore:

$$F_{max} = 560 + 0,3193 * (700 - 560) = 604,7 \text{ kN/m}$$

$$\frac{Q_u}{B} = \left(\frac{604,7 \text{ kN/m}}{3.7} + 24 * 0,8 \right) = 182,63 \text{ kN/m}^2$$

5.3.10. Construction of a fifth slab instead of an attic and foundation slab

For this new project we can use the previous one as base. Since we are going to modify it, we save the project with a different name. In this problem, the foundation surface is increased, converting the three footings into a foundation slab.

We go to 'Structures' geometry mode, delete the three existing footings (Figure 5.49) and create the foundation slab. For this we must write in command line: 'polygon (5.15 0) (22.85 0) (22.85 0.8) (5.15 0.8)', as presented in Figure 50.

Exercise 05. Shallow foundations on clay soil

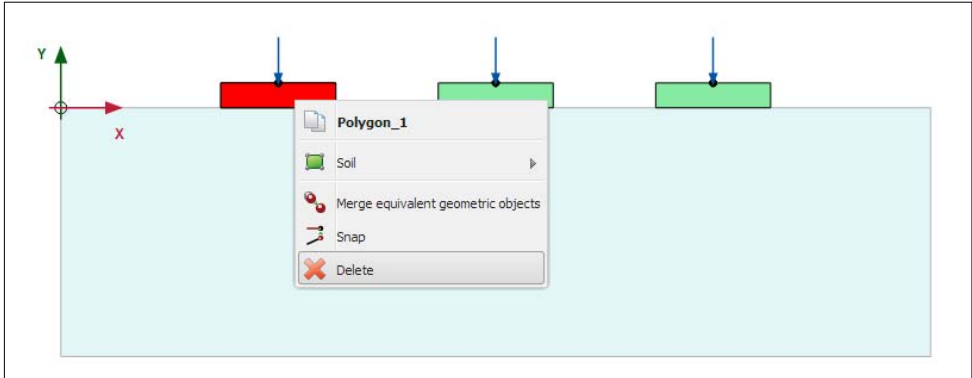


Figure 5. 49. Removal of the first footing, in 'Structure' geometry mode

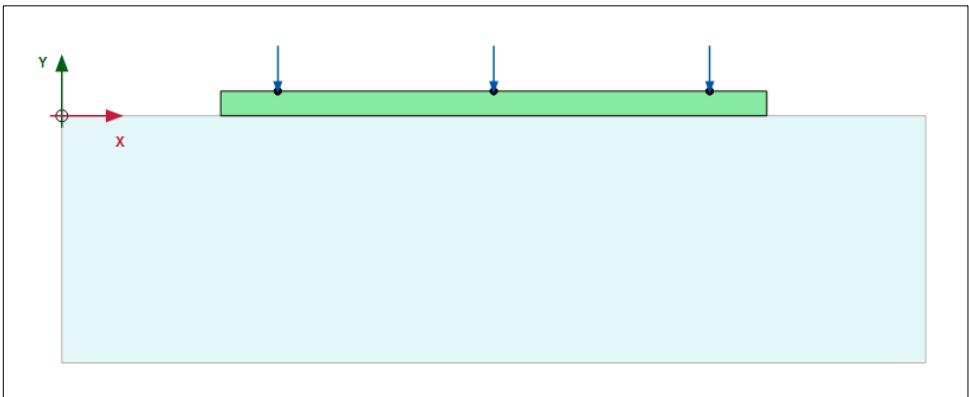


Figure 5. 50. New foundation slab

Once it has been created, we assign 'Concrete' as the material to model the foundation slab (Figure 5.51).

Exercise 05. Shallow foundations on clay soil

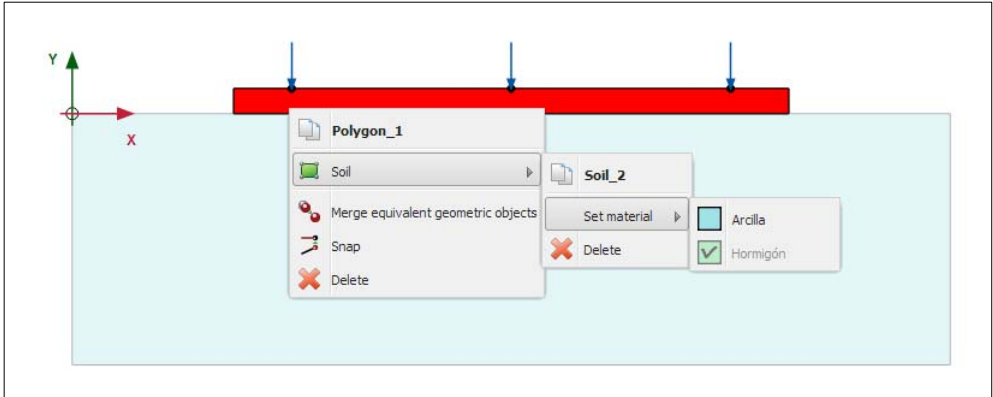


Figure 5. 51. Selection of 'Concrete' as the material to model the foundation slab

Since we have modified the geometry of the problem, we have to mesh the problem again. In 'Mesh' calculation mode we indicate a medium element distribution. We select 'View mesh' to display the created mesh (Figure 5.52).

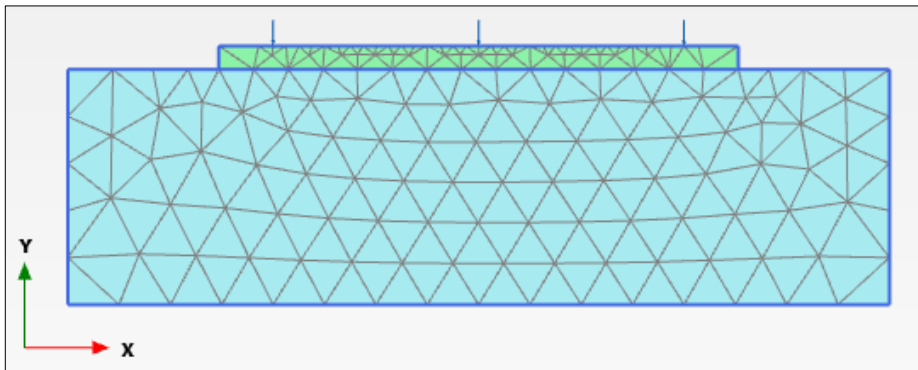


Figure 5. 52. Meshing of the problem after creating the foundation slab

The phases are the same as in the previous exercise, but instead of activating the three footings in Phase 1 (previously called 'Footing') and on, the foundation slab must be activated (we have to be sure that it is deactivated in the Initial Phase). If we want, we can call now Phase 1 'Foundation slab'.

Another small difference is that the safety factor study is done in a different phase. For this, we delete the previously 'Safety' Phase and create a new one after the last phase (Phase 6, 'Fifth slab'). This new 'Safety' Phase presents the same properties

as the ones created in previous points of the exercise. Figure 5.53 shows the different phases of the problem before calculating.

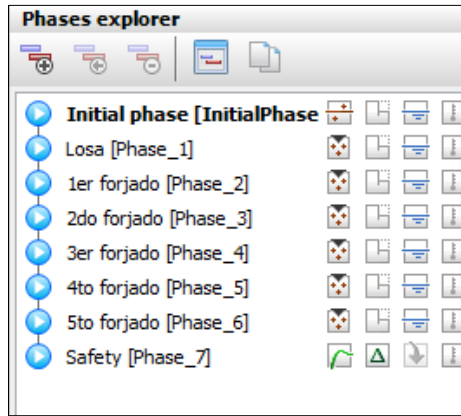


Figure 5. 53. Phases of the problem with foundation slab

Since we have a new mesh, we must select some points to represent advanced graphical outputs. The chosen points (Figure 5.54) are very close to those of the previous points, but they do not have the same coordinates because the generated mesh is not exactly the same. These coordinates are presented in Figure 5.55.

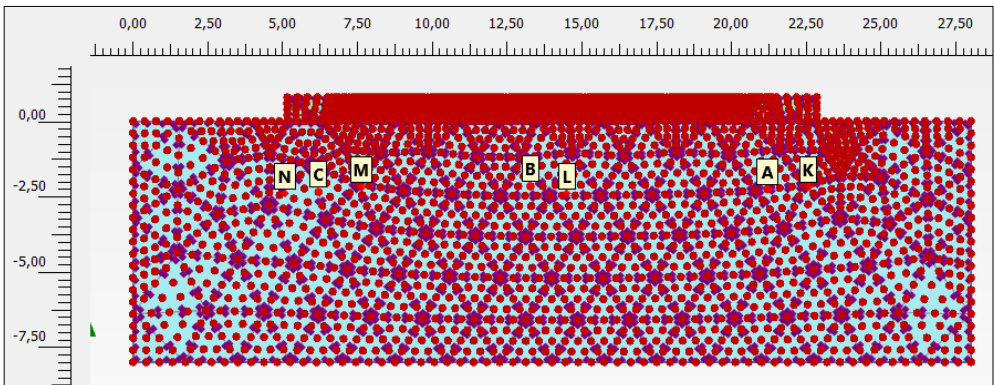


Figure 5. 54. Selection of points for advanced graphical outputs

Exercise 05. Shallow foundations on clay soil

Name	X	Y	Selected
A (Node 495)	21,20	-2,26	Pre-calc
B (Node 1463)	13,28	-2,16	Pre-calc
C (Node 2121)	6,19	-2,35	Pre-calc
K (Stress point 2546)	22,57	-2,18	Pre-calc
L (Stress point 2817)	14,48	-2,39	Pre-calc
M (Stress point 2594)	7,63	-2,19	Pre-calc
N (Stress point 2348)	5,08	-2,40	Pre-calc

Figure 5. 55. Coordinates of points to obtain advanced graphical outputs

Once the calculations are finished, we can see that all the phases are completely finished (Figure 5.56), which means that the soil will not fail despite the construction of the fifth slab. This is because we are employing a foundation slab instead of the three footings.

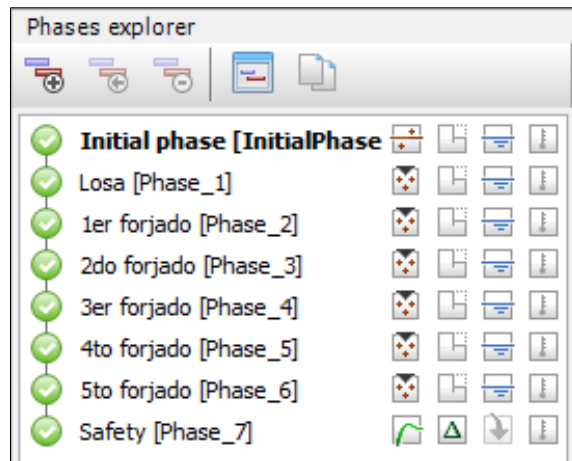


Figure 5. 56. Phases of the problem with foundation slab once the calculation is finished

Now we can represent some results obtained with PLAXIS Output. For example, Figure 5.57 displays the vertical displacements, u_v , when Phase 6 (with the fifth slab completely built).

Exercise 05. Shallow foundations on clay soil

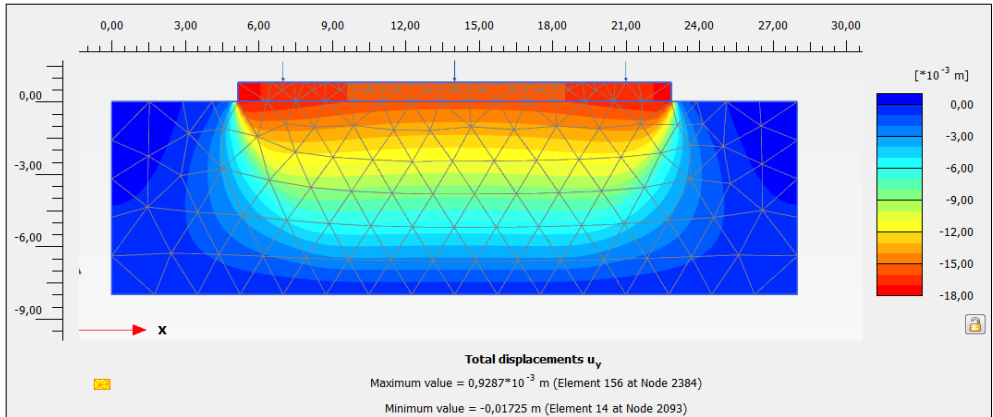


Figure 5. 57. Vertical displacements, u_y , in Phase 6 (fifth slab)

Figures 5.58 and 5.59 show the total incremental displacements, $|\Delta u|$, in Phase 6 and Safety Phase, respectively. In fact, Figure 5.59, Safety Phase shows a possible failure mechanism, if the load applied in Phase 6 was higher. We can obtain the safety factor in Phase 6 from the information in ‘Project’, ‘Calculation information’ (we have to select Safety Phase before, which is defined right after Phase 6). In this way, in Figure 5.60 we can see that ΣM_{sf} value is 1.631.

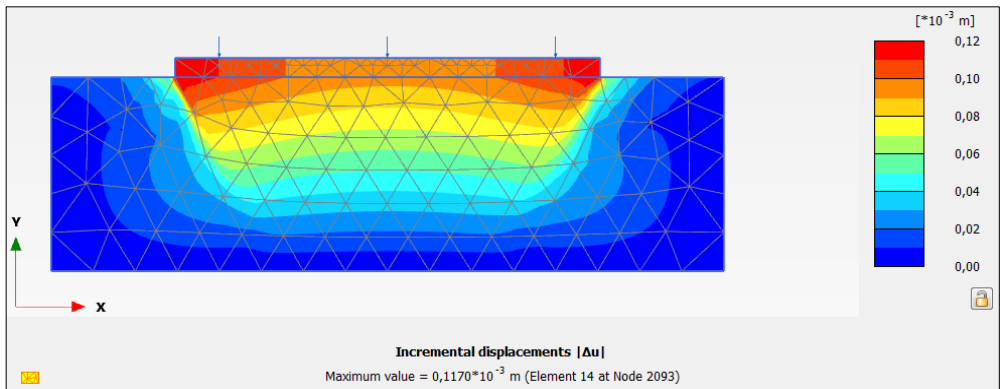


Figure 5. 58. Total incremental displacement, $|\Delta u|$, in Phase 6 (fifth slab)

Exercise 05. Shallow foundations on clay soil

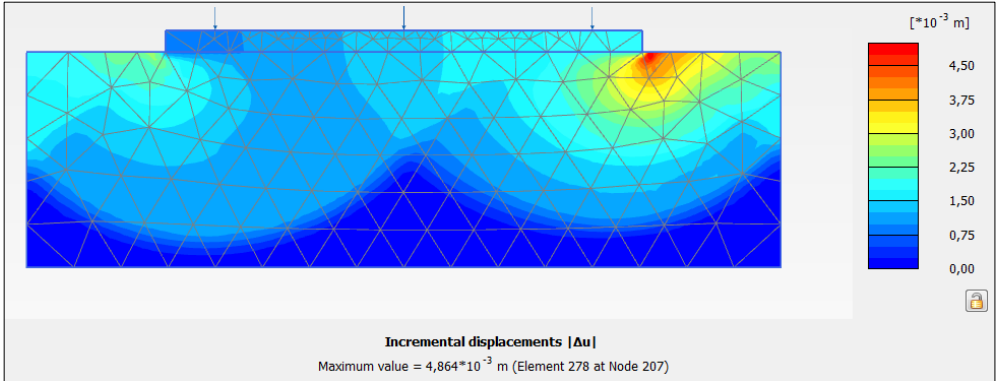


Figure 5. 59. Total incremental displacements, $|\Delta u|$, in Safety Phase (after fifth slab)

2D Calculation information				
Step info				
Phase	Safety [Phase_7]			
Step	Initial			
Calculation mode	Classical mode			
Step type	Safety			
Updated mesh	False			
Solver type	Picos			
Kernel type	32 bit			
Extrapolation factor	1,000			
Relative stiffness	0,1461E-3			
Multipliers				
Soil weight			ΣM_{Weight}	1,000
Strength reduction factor	M_{sf}	0,3131E-3	ΣM_{sf}	1,631
Time	Increment	0,000	End time	0,000
Staged construction				
Active proportion total area	M_{Area}	0,000	ΣM_{Area}	1,000
Active proportion of stage	M_{Stage}	0,000	ΣM_{Stage}	0,000

Figure 5. 60. Safety factor in Phase 6 (after building all slabs)

We save the project.

5.3.11. Construction of four slabs and an attic with a foundation slab

We take the previous file as base project and save it with a different name. The only difference is that the applied load is now -600 kN/m. The last phase is now called 'Attic' instead of 'Fifth slab'. We delete the existing safety phase and create a new one after the attic construction.

We can check that the displacements are lower here than in the previous point (five slabs and a foundation slab). Figure 5.61 displays vertical displacements, u_y , when Phase 6 is finished (once the attic has been built).

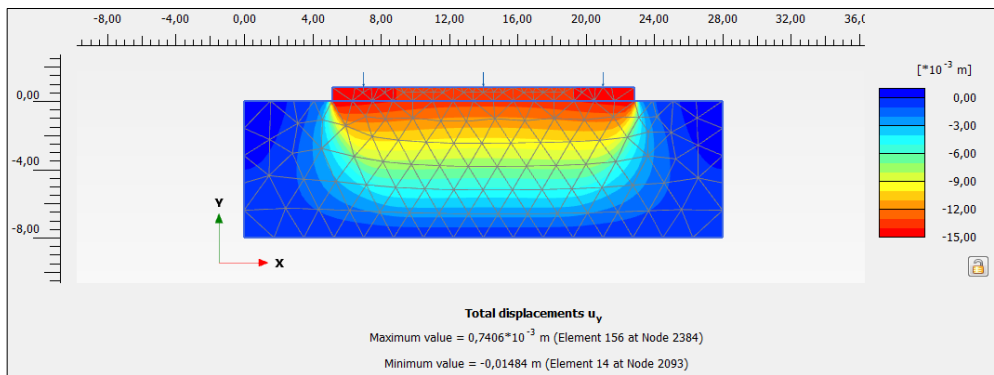


Figure 5. 61. Vertical displacements, u_y , in Phase 6 (attic)

Figures 5.62 and 5.63 show the total incremental displacements, $|\Delta u|$, in Phase 6 and Safety Phase, respectively. Figure 5.63 (Safety Phase) displays a possible failure mechanism, if the applied load in Phase 6 was higher.

Exercise 05. Shallow foundations on clay soil

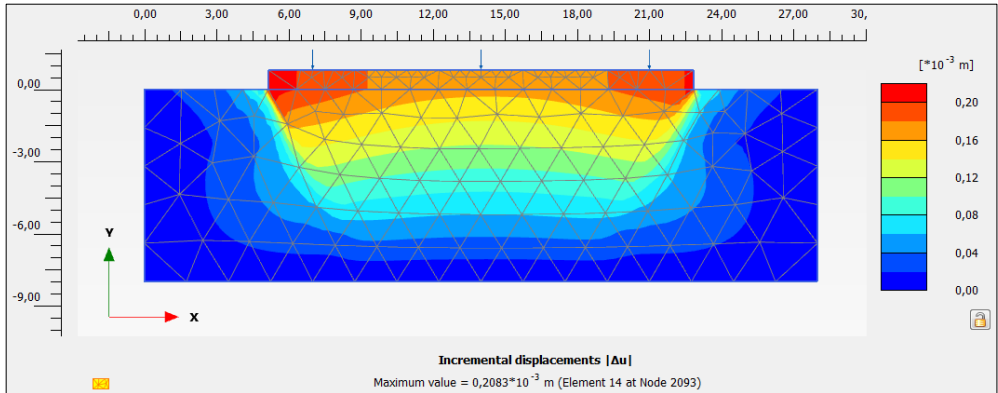


Figure 5. 62. Total incremental displacements, $|\Delta u|$, in Phase 6.

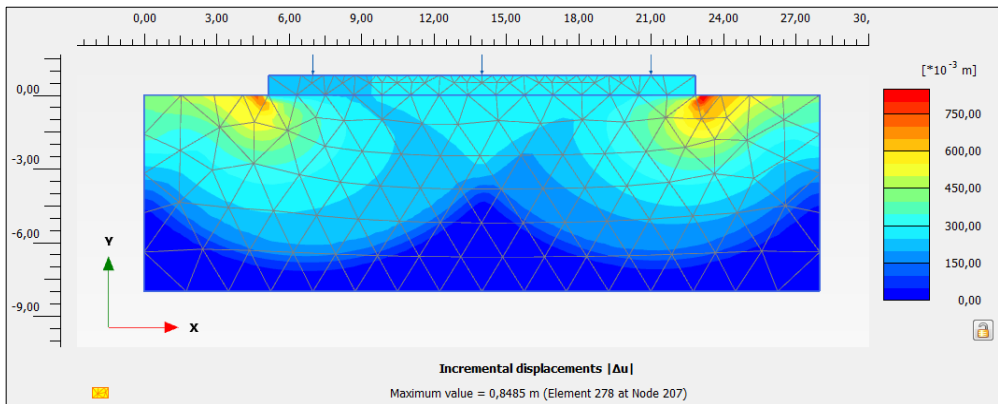


Figure 5. 63. Total incremental displacements, $|\Delta u|$, in Safety Phase (after building the attic)

In 'Project', 'Calculation information' we can obtain the safety factor in Phase 6 (we must select Safety Phase previously, which is right after Phase 6). In this way, we see in Figure 5.64 that ΣM_{sf} is 1.736, a bit higher than in the previous point (five slabs and a foundation slab).

Exercise 05. Shallow foundations on clay soil

Calculation information				
Mode	Classical mode			
	Safety			
Mesh	False			
Order	Picos			
Order	32 bit			
Iteration factor	1,000			
Stiffness	-2,785E-9			
Iteration				
Iteration			ΣM_{Weight}	1,000
Reduction factor	M_{sf}	-4,514E-6	ΣM_{sf}	1,736
	Increment	0,000	End time	0,000
Iteration				
Proportion total area	M_{Area}	0,000	ΣM_{Area}	1,000
Proportion of stage	M_{Stage}	0,000	ΣM_{Stage}	0,000
		0,000 kN/m		
		0,000 kN/m		

Figure 5. 64. Safety factor in Phase 6 (after building the attic)

Exercise 06

Excavation in heterogeneous soil between two diaphragm walls

6.1. INTRODUCTION

An excavation is dug. It is 8-meter-deep in a 21-meter stratum of heterogeneous soil. The stratum presents 3 layers: the first one, which involves the first 2 meters that are closer to the surface, and they are clay; the second layer (9 meters) is formed of sands; and the third one, the last 10 meters, is silt.

The excavation is carried out with diaphragm walls, and it will have a final depth of 8 meters. Anchors will be used to reinforce the walls.

The phreatic level is located at 5.5 meters below the soil surface. Moreover, a non-constant- value distributed load is applied on the soil surface.

The details of the whole geometry of the problems, the load values, the layers characteristics and the excavation, walls and anchors procedures will be defined in the following sections.

For all the phases, the variables to obtain are bending moments on the walls, total displacements and horizontal effective stress on the soil. In addition, for the last phase, the flow net is studied: iso-potential lines, pore pressure, velocity vectors and infiltrated water flow.

6.2 GEOMETRY AND GEOTECHNICAL PARAMETERS OF THE PROBLEM

A scheme of the geometry is shown in Figure 6.1:

- Soil width to consider: 75 meters.
- Excavation width: 15 meters (centred in the middle of the 75 meter of soil width).
- Trapezoidal load.
 - First end (10.5 meters on the right of the excavation axis): 12.2 kN/m/m.
 - Primer extremo a 10,5 metros a la derecha del eje de la excavación y de valor 12 kN/m/m.
 - Second end (25.5 meters on the right of the excavation axis): 4 kN/m/m.
- Wall depth: 12 meters.
- Excavation phases:
 - 2 meters are removed, and the first anchors are placed (upper anchors).
 - 3.5 meters are removed, and the second anchors are placed (lower anchors).
 - The last 2.5 meters of soil are removed.
- Anchors have a length of 17 meters (having to add 4 meters of anchorage zone) and an angle of 30° to the horizontal. Materials and anchors characteristics are defined later.

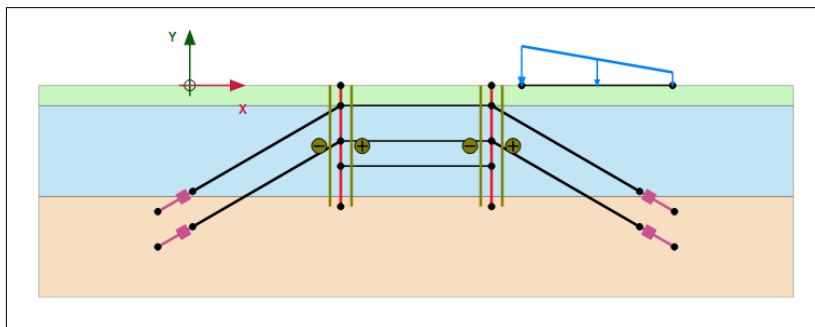


Figure 6. 1. Geometry of the problem in PLAXIS-2D

In Tables 6.1, 6.2 and 6.3 the geotechnical parameters of each soil layer are presented.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Geotechnical parameters of the first layer (clay)

Material	Clay
Constitutive model	Hardening-Soil
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	17.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	20 kN/m ³
Initial void ratio (e_{init})	0.5
Stiffness	
Secant (E_{50}^{ref})	21000 kN/m ²
Tangent ($E_{\text{oad}}^{\text{ref}}$)	21000 kN/m ²
Unloading/ reloading ($E_{\text{ur}}^{\text{ref}}$)	55000 kN/m ²
Power for stress-level dependency of stiffness (m)	0.5
Strength	
Reference effective cohesion (c'_{ref})	7 kPa
Effective friction angle (ϕ')	35°
Dilatancy angle (ψ)	0°
Advanced stiffness	
K_0 in a normally consolidated soil (K_0^{nc})	0.4264
Soil	
< 2 μm	46%
2 μm – 50 μm	26%
50 μm – 2 mm	28%
Flow parameters	
Horizontal hydraulic conductivity k_x	0.08 m/day
Vertical hydraulic conductivity k_y	0.08 m/day
Interfaces	
Strength type	Rigid
Strength ratio (R_{inter})	1
Initial	
k_0 determination	Automatic
Over-consolidation	
Over-consolidation ratio (OCR) ($\sigma'_{v,\text{max}} / \sigma'_{v,\text{actual}}$)	1
Pre-overburden ratio (POP) ($\sigma'_{v,\text{max}} - \sigma'_{v,\text{actual}}$)	15 kN/m ²

Table 6. 1. Geotechnical parameters of the first layer

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Geotechnical parameters of the second layer (sand)

Material	Sand
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	18 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	21 kN/m ³
Stiffness	
Young's modulus (E')	34000 kPa
Poisson's coefficient (ν')	0.3
Strength	
Effective cohesion (c'_{ref})	0 kPa
Effective friction angle (φ')	28°
Dilatancy angle (ψ)	0°
Soil	
< 2 μm	10%
2 μm – 50 μm	13%
50 μm – 2 mm	77%
Flow parameters	
Horizontal hydraulic conductivity k_x	4.3 m/day
Vertical hydraulic conductivity k_y	4.3 m/day
Interfaces	
Strength type	Manual
Strength ratio (R_{inter})	0.8

Table 6. 2. Geotechnical parameters of the second layer (sand)

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Geotechnical parameters of the third layer (silt)

Material	Silt
Constitutive model	Hardening-Soil
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	16.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	19.5 kN/m ³
Initial void ratio (e_{init})	0.5
Stiffness	
Secant (E_{50}^{ref})	17000 kN/m ²
Tangent ($E_{\text{oed}}^{\text{ref}}$)	13000 kN/m ²
Unloading/ reloading ($E_{\text{ur}}^{\text{ref}}$)	41000 kN/m ²
Power for stress-level dependency of stiffness (m)	0.5
Strength	
Reference effective cohesion (c'_{ref})	5 kPa
Effective friction angle (φ')	30°
Dilatancy angle (ψ)	0°
Advanced stiffness	
K_0 in a normally consolidated soil (K_0^{nc})	0.485
Soil	
< 2 μm	19%
2 μm – 50 μm	41%
50 μm – 2 mm	40%
Flow parameters	
Horizontal hydraulic conductivity k_x	0.15 m/day
Vertical hydraulic conductivity k_y	0.15 m/day
Interfaces	
Strength type	Manual
Strength ratio (R_{inter})	0.7
Initial	
k_0 determination	Automatic
Over-consolidation	
Over-consolidation ratio (OCR) ($\sigma'_{v,\text{max}} / \sigma'_{v,\text{actual}}$)	1
Pre-overburden ratio (POP) ($\sigma'_{v,\text{max}} - \sigma'_{v,\text{actual}}$)	5 kN/m ²

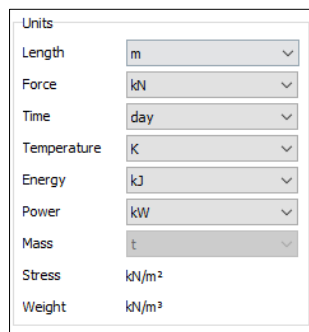
Table 6. 3. Geotechnical parameters of the third layer (silt)

6.3 RESOLUTION OF THE PROBLEM WITH PLAXIS 2D

6.3.1. Introduction of the project parameters

We start PLAXIS 2D (PLAXIS input) and choose ‘*Start a new Project*’ in ‘*Quick select*’ emerging window. In ‘*Project*’ tab, we give a name to the problem, for example, ‘*Excavation in heterogeneous soil*’.

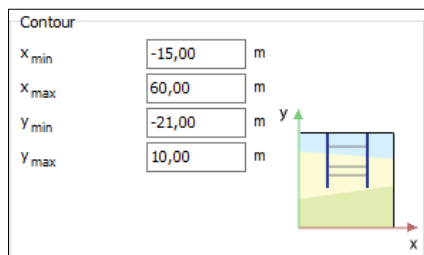
We choose ‘*Plain stain*’ option in ‘*Model*’ tab, and ‘*15-Noded*’ elements. We keep the default units, as shown in Figure 6.2.



Units	
Length	m
Force	kN
Time	day
Temperature	K
Energy	kJ
Power	kW
Mass	t
Stress	kN/m ²
Weight	kN/m ³

Figure 6. 2. Problem units in PLAXIS

We define the contour writing the values displayed in Figure 6.3.



Contour	
x _{min}	-15,00 m
x _{max}	60,00 m
y _{min}	-21,00 m
y _{max}	10,00 m

Figure 6. 3. Definition of the problem contour

We keep the default constants in ‘*Constant*’ tab.

6.3.2 Material and layer definition

In order to create the three materials that formed the heterogeneous soil, we select ‘Show materials’ button. We select ‘New’ (set type ‘Soil and interfaces’) and introduce the geotechnical parameters for the first layer (‘Clay’, Table 6.1).

Figures 6.4-6.8 show ‘Clay’ properties.

Property	Unit	Value
Stiffness		
E_{50}^{ref}	kN/m ²	21,00E3
E_{oed}^{ref}	kN/m ²	21,00E3
E_{ur}^{ref}	kN/m ²	55,00E3
power (m)		0,5000
Alternatives		
Use alternatives		<input type="checkbox"/>
C_c		0,01643
C_s		5,645E-3
e_{init}		0,5000
Strength		
c'_{ref}	kN/m ²	7,000
ϕ' (phi)	°	35,00
ψ (psi)	°	0,000
<input type="checkbox"/> Advanced		
Set to default values		<input checked="" type="checkbox"/>
Stiffness		
v'_{ur}		0,2000
p_{ref}	kN/m ²	100,0
K_0^{nc}		0,4264

Figure 6. 4. ‘Clay’ parameters

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

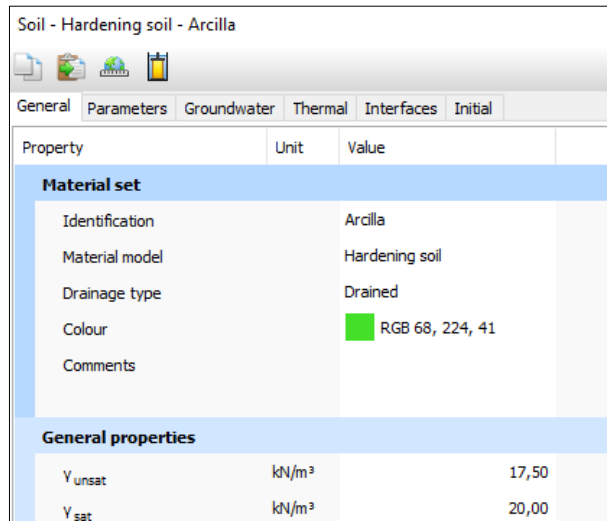


Figure 6. 5. 'Clay' general properties

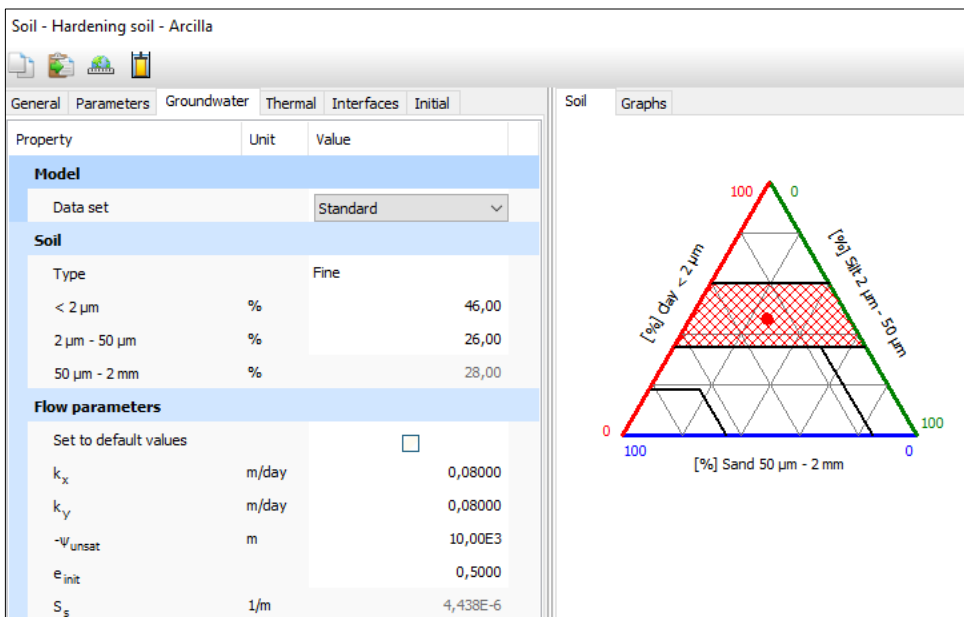


Figure 6. 6. 'Clay' groundwater parameters

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Property	Unit	Value
Strength		
Strength		Rigid
R_{inter}		1,000
Consider gap closure		<input checked="" type="checkbox"/>
Real interface thickness		
δ_{inter}		0,000
Heat transfer		
R	$m^2 K/kW$	0,000

Figure 6. 7. ‘Clay’ interface configuration

Property	Unit	Value
K0 settings		
K_0 determination		Automatic
Overconsolidation		
OCR		1,000
POP	kN/m^2	15,00

Figure 6. 8. ‘Clay’ determination of K_0 and over-consolidation parameters

In the same way we create the second layer (‘Sand’, Table 6.2). Figures 6.9-6.12 show the introduced properties for this material.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Soil - Mohr-Coulomb - Arena		
General Parameters Groundwater Thermal Interfaces Initial		
Property	Unit	Value
Material set		
Identification		Arena
Material model		Mohr-Coulomb
Drainage type		Drained
Colour		RGB 50, 169, 220
Comments		
General properties		
γ_{unsat}	kN/m ³	18,00
γ_{sat}	kN/m ³	21,00

Figure 6. 9. 'Sand' general properties

Soil - Mohr-Coulomb - Arena		
General Parameters Groundwater Thermal Interfaces Initial		
Property	Unit	Value
Stiffness		
E'	kN/m ²	34,00E3
ν' (nu)		0,3000
Alternatives		
G	kN/m ²	13,08E3
E_{oed}	kN/m ²	45,77E3
Strength		
c'_{ref}	kN/m ²	0,000
φ' (phi)	°	28,00
ψ (psi)	°	0,000
Velocities		
V_s	m/s	84,42
V_p	m/s	157,9

Figure 6. 10. 'Sand' parameters

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

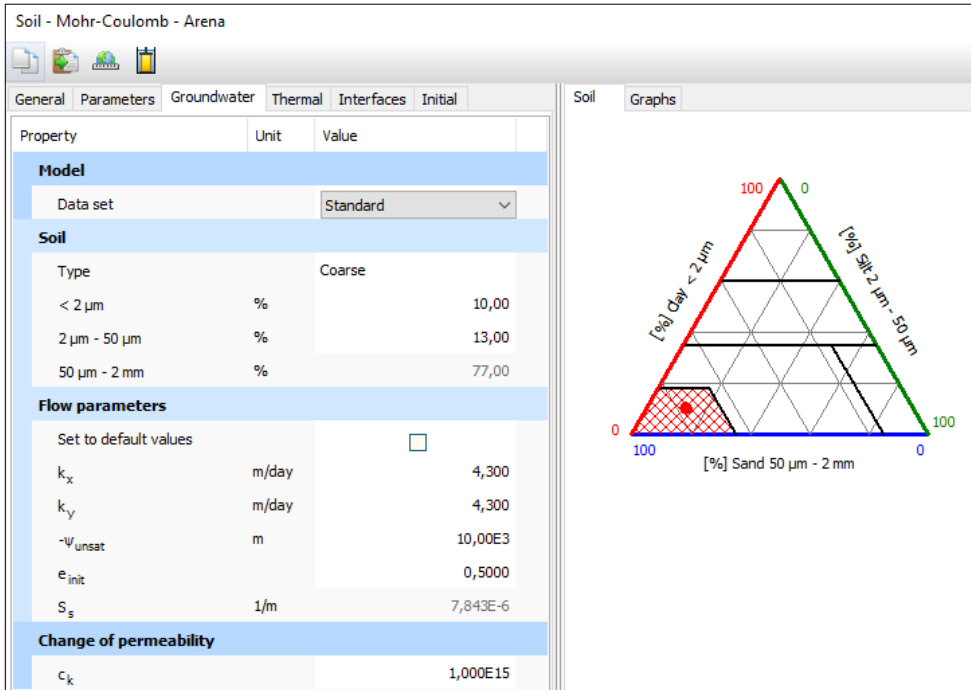


Figure 6. 11. 'Sand' groundwater parameters

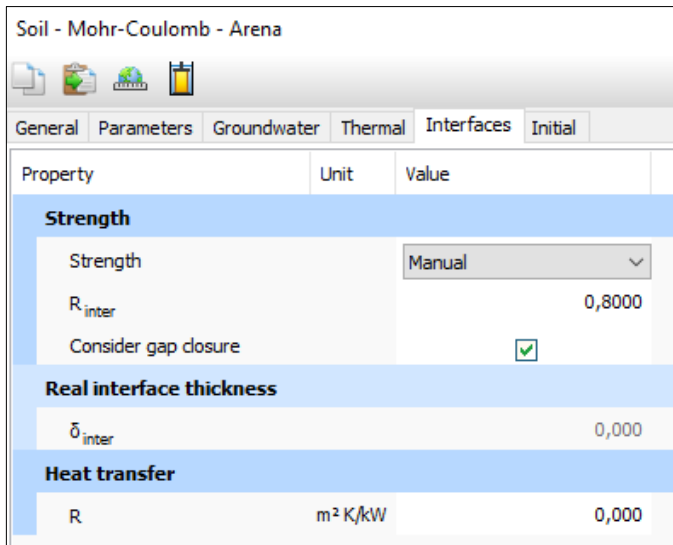


Figure 6. 12. 'Sand' interface configuration

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Finally, we create the third layer ('Silt', Table 6.3). It is a material with the same constitutive model (Drained hardening soil) as 'Clay', so the introduction of the parameters is similar.

In order to define the heterogeneous soil where the excavation is going to be dug, we select 'Soil' geometry mode. Then we click on 'Create borehole' button and select (0, 0) coordinate.

We select 'Add' to create the upper layer. In 'Top', we write 0 and in 'Bottom' -2. In 'Material' pull-down menu, we select 'Clay'.

Selecting again 'Add', we create the middle layer. In 'Top', we write -2 and in 'Bottom' -11. In 'Material' pull-down menu, we select 'Sand'.

Finally, we create the last layer selecting 'Add'. We write -11 in 'Top' and -21 in 'Bottom'. In 'Material' pull-down menu, we select 'Silt'.

Since the phreatic level is located 5.5 meters below the surface, so in 'Head' we write -5.5, Figure 6.13.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

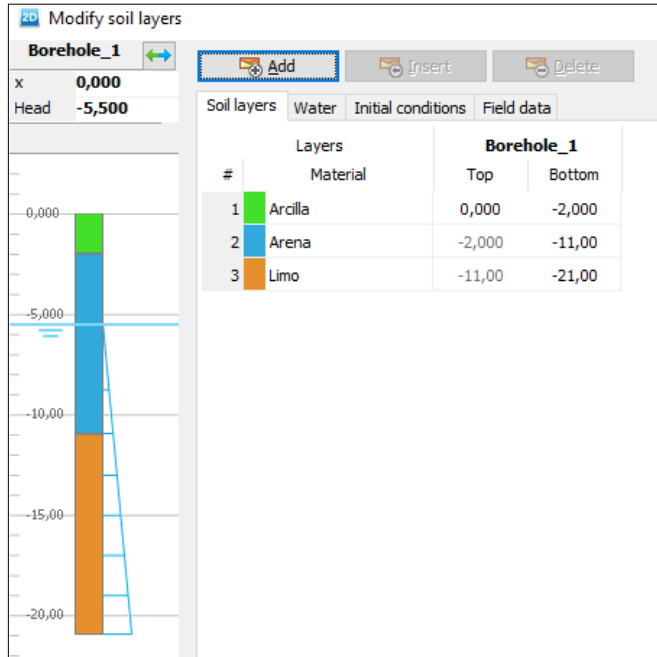


Figure 6. 13. Soil stratigraphy definition in 'Modify soil layers' window

Figure 6.14 shows the resulting geometry.

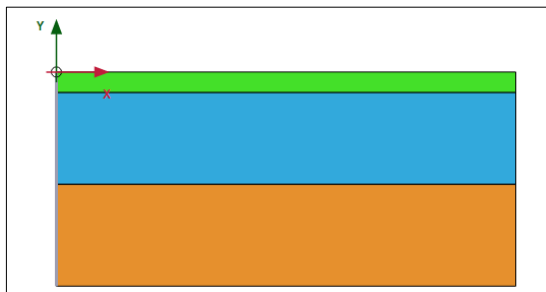


Figure 6. 14. Geometry of the three layers where the excavation will be dug

6.3.3 Definition of the structural walls

We go 'Structures' geometry mode and draw the two diaphragm walls as plates ('Create line', 'Create plate', Figure 6.15). The coordinates of each plate are defined in Table 6.4.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

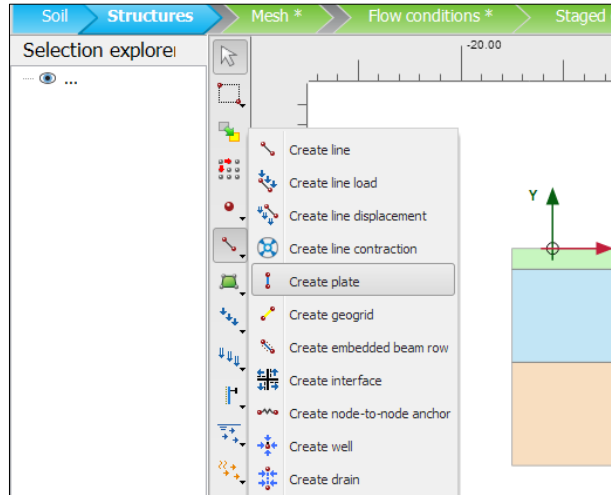


Figure 6. 15. 'Create plate' button

Plate	Initial coordinate	Final coordinate
Left	(15, 0)	(15, -12)
Right	(30, 0)	(30, -12)

Table 6. 4. Diaphragm wall coordinates

Figure 6.16 shows the two plates once they have been created.

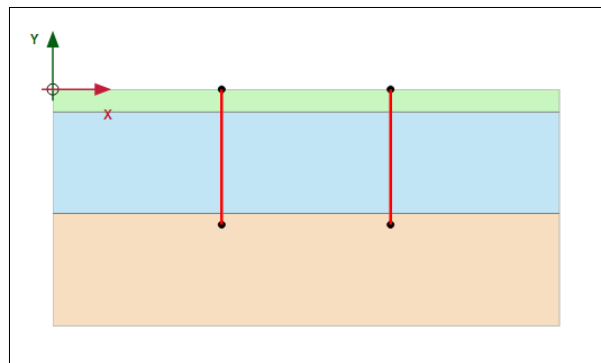


Figure 6. 16. Plates created in the soil

We must now create and assign a new material for plates. For this, we select both plates at the same time (pressing CTRL) and, in 'Selection explorer', in 'Plate' option, we select 'New' in 'Material' sub-option (Figure 6.17).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

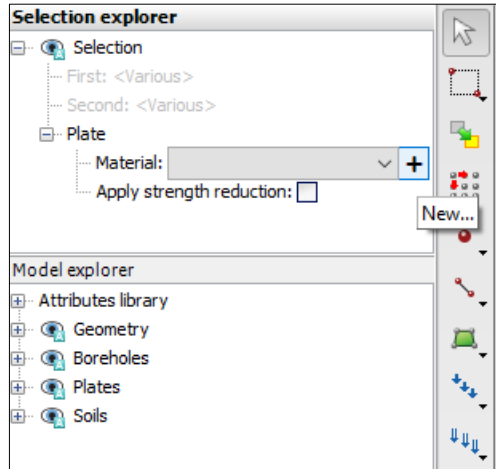
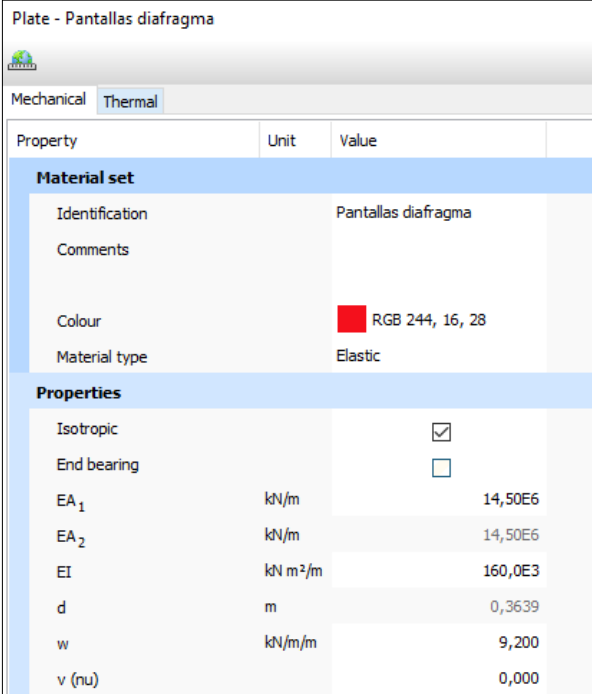


Figure 6. 17. Creation of a new material for the plates

After assigning a name to this new material ('Diaphragm walls'), we write the mechanical properties of the plate material: material type, axial stiffness (EA), bending stiffness (EI), weight (w), and Poisson's coefficient (ν), Figure 6.18.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls




Property	Unit	Value
Material set		
Identification		Pantallas diafragma
Comments		
Colour		 RGB 244, 16, 28
Material type		Elastic
Properties		
Isotropic		<input checked="" type="checkbox"/>
End bearing		<input type="checkbox"/>
EA ₁	kN/m	14,50E6
EA ₂	kN/m	14,50E6
EI	kN m ² /m	160,0E3
d	m	0,3639
w	kN/m/m	9,200
v (nu)		0,000

Figure 6. 18. Mechanical properties of the plate material

Finally, in ‘*Selection explorer*’ menu this new material will appear (Figure 6.19).

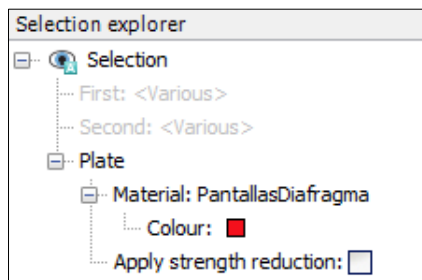


Figure 6. 19. ‘Diaphragm wall’ material in ‘*Selection explorer*’ menu

If we want to edit the properties of this material, it is possible to do this clicking with the right mouse button on the material and select ‘*Edit*’ option (Figure 6.20).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

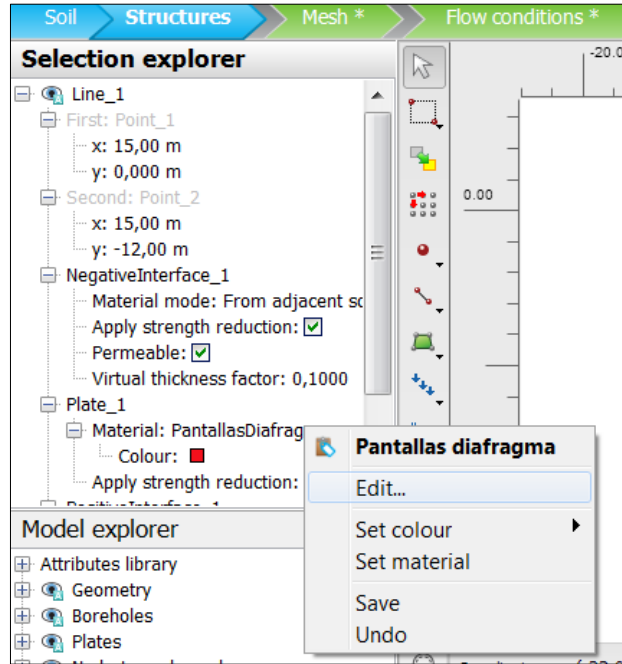


Figure 6. 20. Selection for the edition of plate material

The last step for modelling the walls is creating their positive and negative interfaces. With this we can simulate the interaction between the soil and the plate employing the interface strength ratio that has been designed for the soil material. For this, we select, for example, the left plate with the right mouse button. We choose ‘Create’ option, and then we click on ‘Positive interface’. We do the same again, but now selecting ‘Negative interface’. The two interfaces for the left plate have been created (Figures 6.21-6.23).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

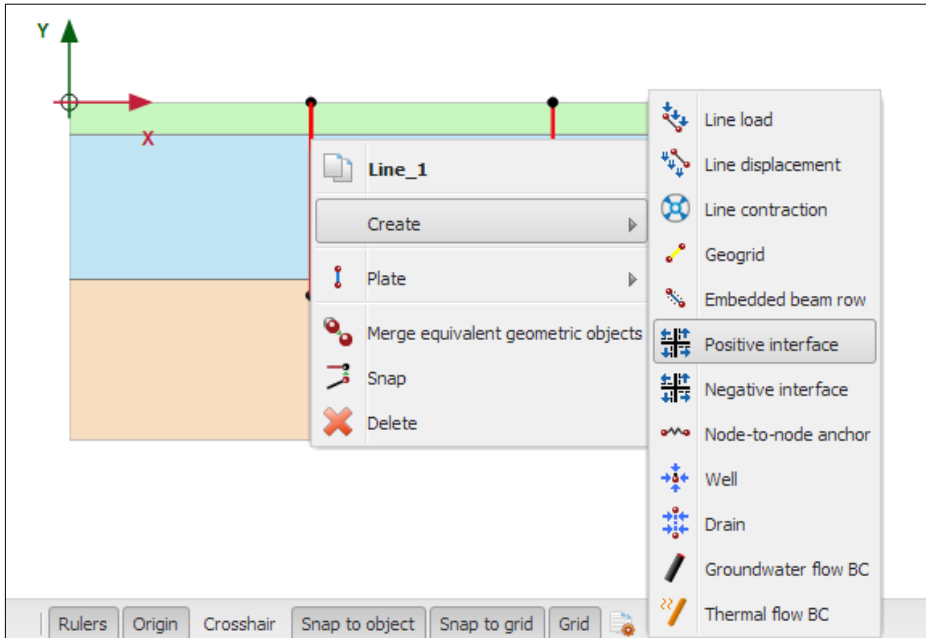


Figure 6. 21. Positive interface creation

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

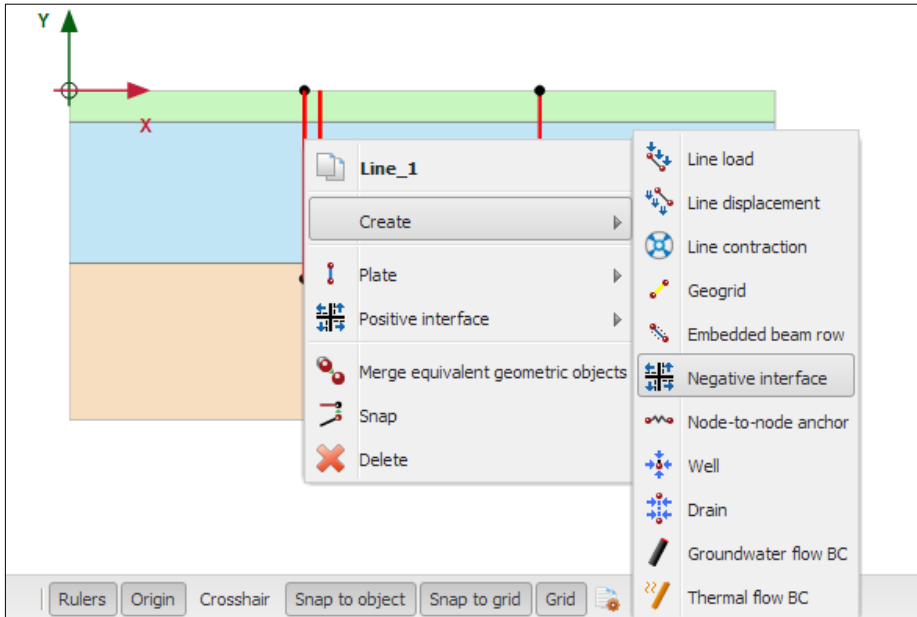


Figure 6. 22. Negative interface creation

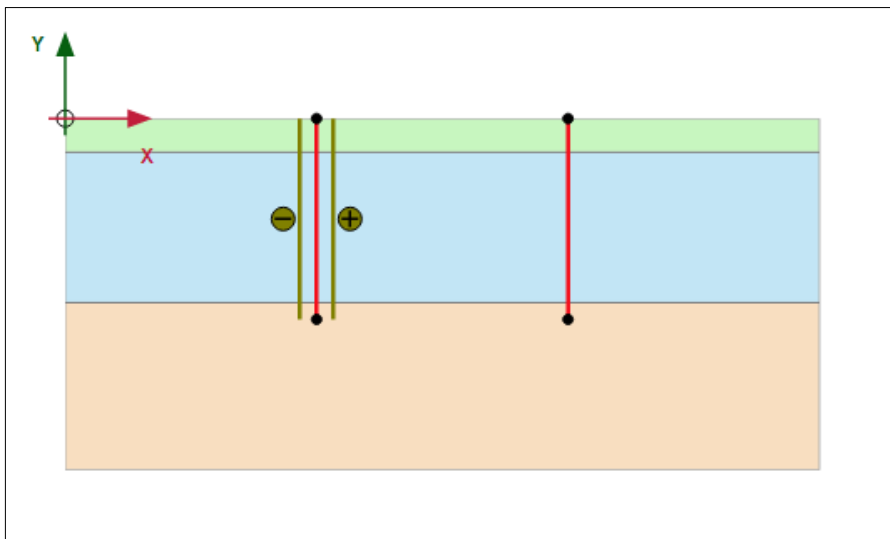


Figure 6. 23. Positive and negative interface created for the left plate

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Following the same steps for the right plates, we finally have created the interfaces for both plates (Figure 6.24):

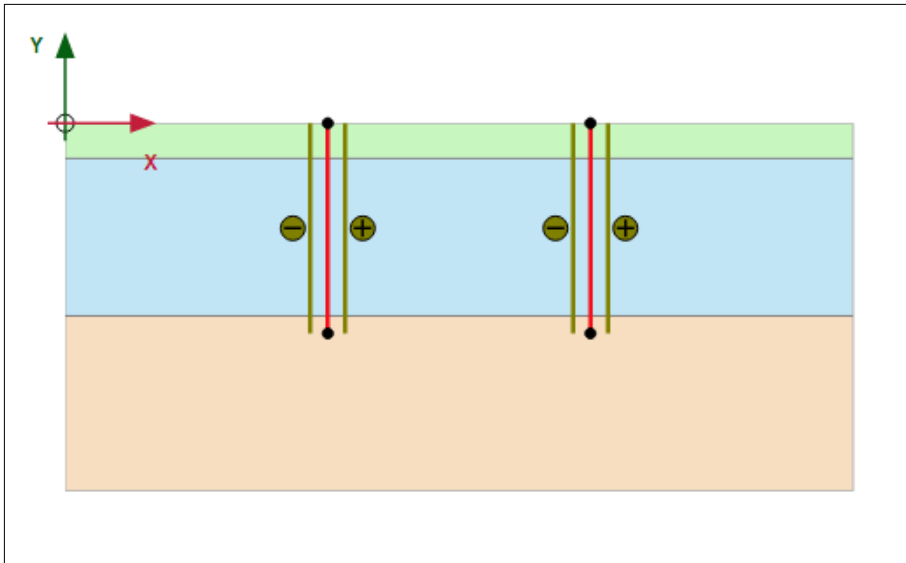


Figure 6. 24. Positive and negative interfaces created for both plates

In order to define the different excavation levels (Table 6.5) we must create three lines, one for each of them.

Excavation level	Initial coordinate	Final coordinate
1	(15, -2)	(30, -2)
2	(15; -5,5)	(30; -5,5)
3	(15, -8)	(30, -8)

Table 6. 5. Excavation levels

For this, we click on 'Create line' button ('Create line' option). After this, we can directly draw them in the workspace, or we can write them in the command line:

```
_line (15 -2) (30 -2)  
_line (15 -5.5) (30 -5.5)  
_line (15 -8) (30 -8)
```

6.3.4. Load definition

On the right of the excavation, at three meters from it (10.5 meters on the right of the excavation axis), the soil is applied a trapezoidal load as follows:

First end: 10.5 meters on the right of the excavation axis, with a value of 12 kN/m/m.

Second end: 25.5 meters on the right of the excavation axis, with a value of 4 kN/m/m.

Therefore, the load has a total length of 15 meters.

In this way, we create a load which goes from the coordinate (33, 0) to (48, 0). On its left border, it will have a value of -12 kN/m/m, and on its right border -4 kN/m/m.

First, we create the load from the point (33, 0) to (48, 0) with ‘*Create line*’, ‘*Create line load*’.

Now (keeping selected the load) we go ‘*Selection explorer*’ menu, and in ‘*Distribution*’ option, we choose ‘*Linear*’. Finally, in $q_{y,start,ref}$, we write -12 kN/m/m, and in $q_{y,end,ref}$ -4 kN/m/m (Figure 6.25).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

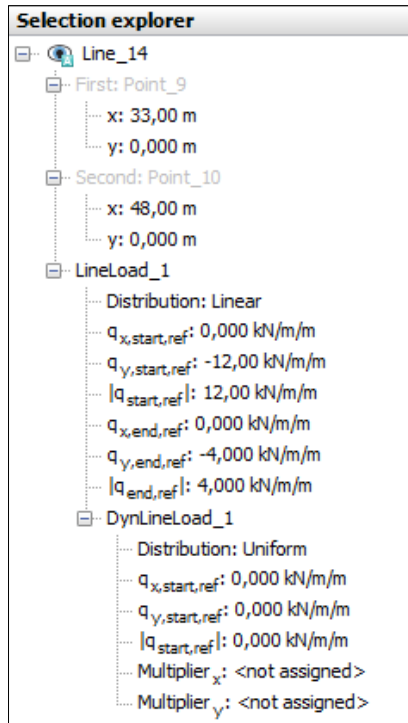


Figure 6. 25. Linear distributed load definition

6.3.5 Anchor definition

In order to create the anchors, we click on 'Create structure' button and select 'Create node-to-node anchor' option (Figure 6.26). Anchors have a length of 17 meters and an angle to the horizontal of 30°.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

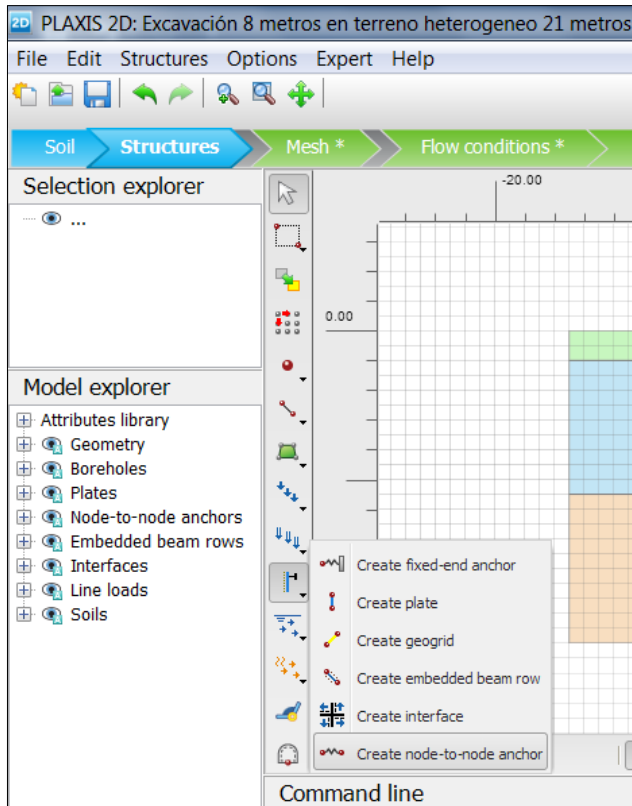


Figure 6. 26. 'Create node-to-node anchor' button

The coordinates of the initial and final nodes for each anchor are displayed in Table 6.6.

Anchor	First point	Second point
1	(15, -2)	(0,28; -10,5)
2	(15; -5,5)	(0,28; -14)
3	(30, -2)	(44,72; -10,5)
4	(30; -5,5)	(44,72; -14)

Table 6. 6. Anchor nodes

If these coordinates have not been correctly introduced, we can modify them in 'Selection explorer' menu (Figure 6.27).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

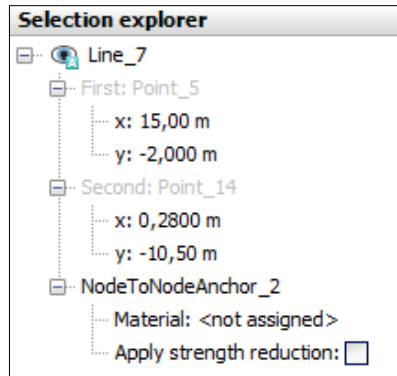


Figure 6. 27. Initial and final coordinates of the anchor nodes

In Figure 6.28 the four anchors are shown.

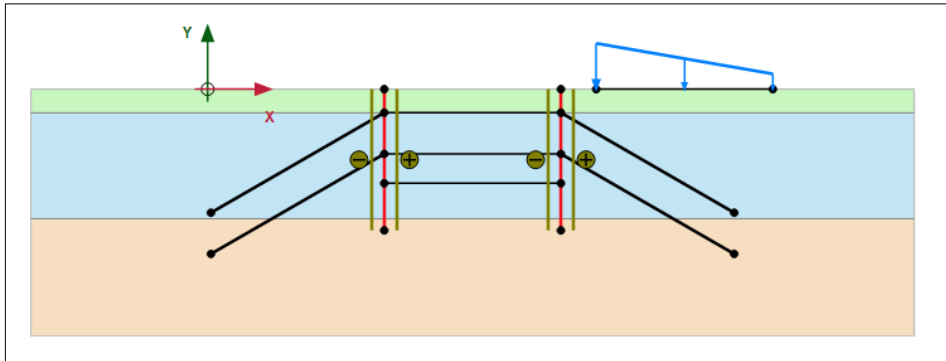


Figure 6. 28. Anchors between the soil and the walls

Now, we have to create the anchor material. We click on 'Show Materials'. In 'Set type' we choose 'Anchors' (Figure 6.29). We click on 'New' and introduce the material type, axial stiffness (EA) and the space between them, Figure 6.30.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

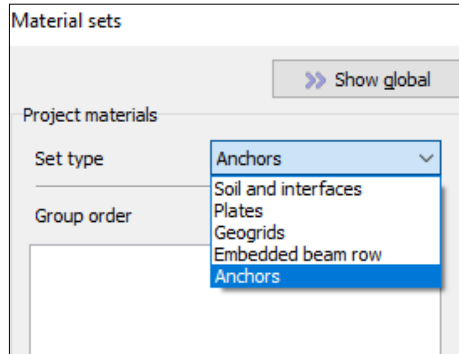


Figure 6. 29. Choosing 'Anchors' to create the anchor material

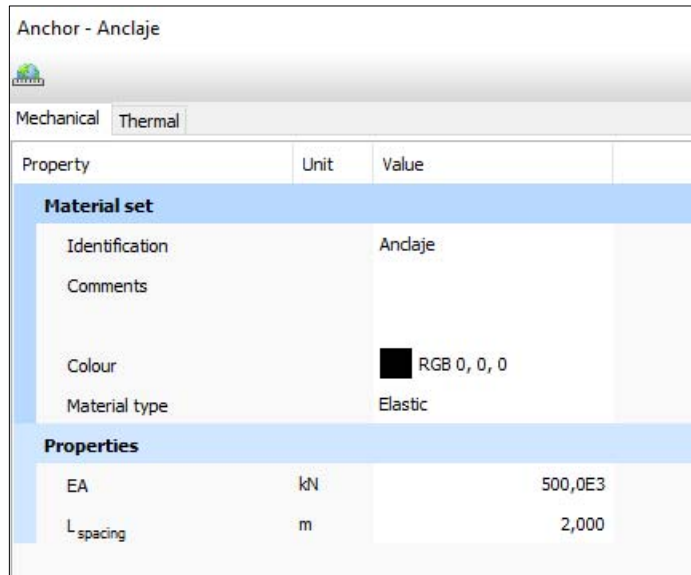


Figure 6. 30. Properties of the anchor material and the space between them

Now we select the four anchors at the same time and click on the right mouse button. We select 'Node-to-node anchor', 'Set material' and finally we assign 'Anchor' material (Figure 6.31).

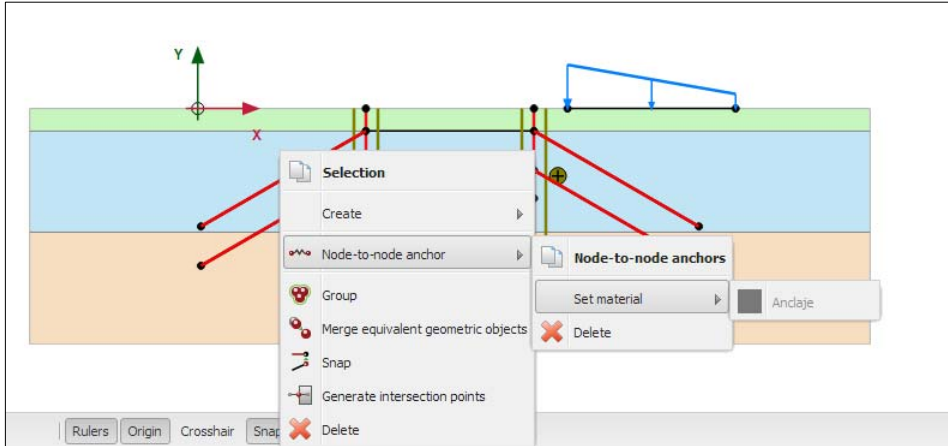


Figure 6. 31. Material assignment for the anchors

6.3.6 Definition for the anchor zone material and its introduction in the model

First, we define the anchor zone material ('Grout-body'). For this, we click on 'Show Material' menu. In 'Set type', we choose 'Embedded beam row' (Figure 6.32). We select 'New' and introduce the parameter values: stiffness (E), unit weight (γ), pile type, predefined pile type, diameter, spacing, axial skin resistance (T_{skin}), base resistance (F_{max}), and interface stiffness factor, Figures 6.33 and 6.34.

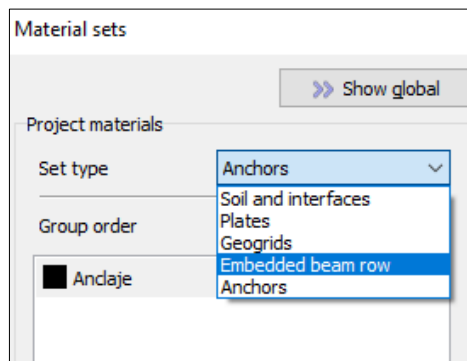


Figure 6. 32. 'Embedded beam row' selection for creating the anchor zone material

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls


Material set		
Identification	Grout body	
Comments		
Colour	 RGB 199, 82, 143	
Material type	Elastic	
Properties		
E	kN/m ²	8,400E6
γ	kN/m ³	0,000
Pile type	Predefined	
Predefined pile type	Massive circular pile	
Diameter	m	0,3000
A	m ²	0,07069
I	m ⁴	0,3976E-3
L _{spacing}	m	2,000
Rayleigh α	0,000	
Rayleigh β	0,000	

Figure 6. 33. Anchor zone material and spacing

Axial skin resistance		
Axial skin resistance	Linear	
T _{skin, start, max}	kN/m	500,0
T _{skin, end, max}	kN/m	500,0
Lateral skin resistance		
Lateral skin resistance	Unlimited	
Base resistance		
F _{max}	kN	0,000
Interface stiffness factor		
Default values	<input checked="" type="checkbox"/>	
Axial stiffness factor	0,6026	
Lateral stiffness factor	0,6026	
Base stiffness factor	6,026	

Figure 6. 34. Properties of the anchor zone materials

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Now, in order to create the anchors, we select ‘*Create structure*’ button and then ‘*Create embedded beam row*’ (Figure 6.35).

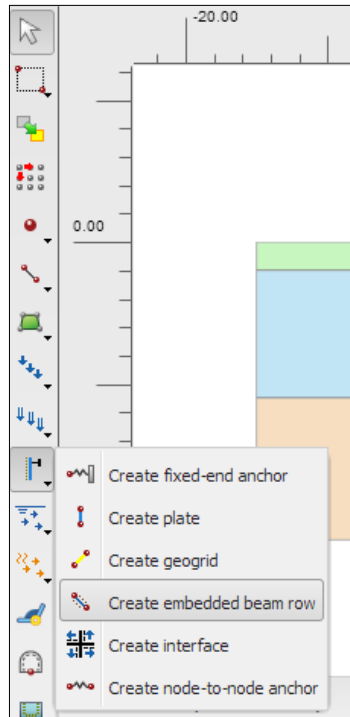


Figure 6. 35. Creation of the structural element for the anchor zone

Initial and final coordinates of each of the anchor zone nodes are shown in Table 6.7.

Anchor	First coordinate	Second coordinate
1	(0,28; -10,5)	(-3,184; -12,5)
2	(0,28; -14)	(-3,184; -16)
3	(44,72; -10,5)	(48,184; -12,5)
4	(44,72; -14)	(48,184; -16)

Table 6. 7. Initial and final nodes of the anchor zone

Once they have been created, we assigned them their material. For this, we click with the right mouse button on each of them, assigning them their material (‘*Grout body*’), Figure 6.36.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

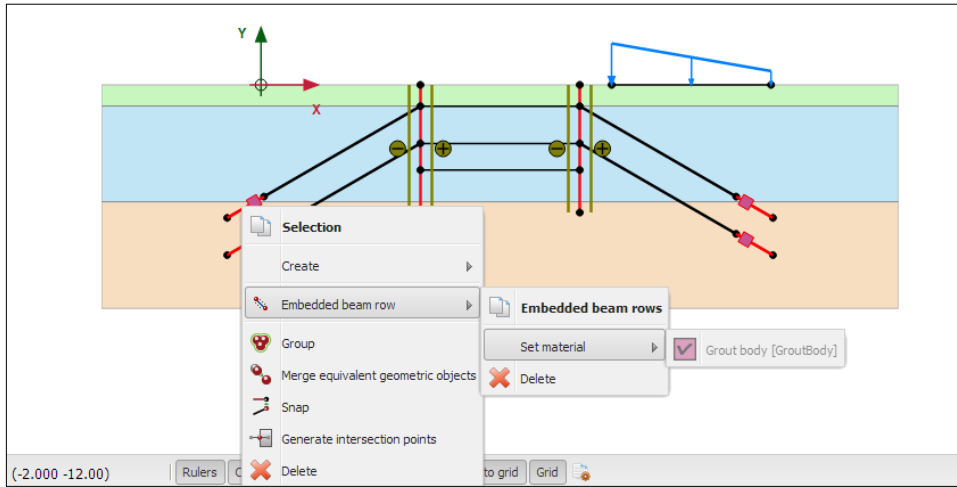


Figure 6. 36. 'Grout body' election as anchor zone material

Finally, the excavation scheme, before meshing and defining the phases, is the following (Figure 6.37):

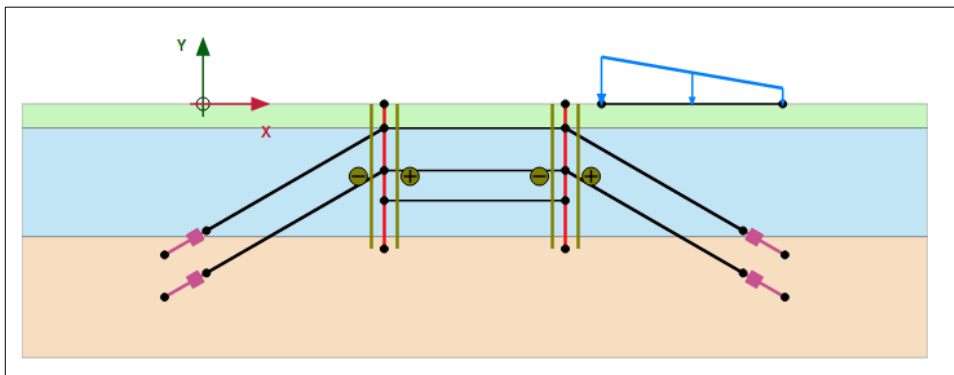


Figure 6. 37. Excavation scheme before meshing

6.3.7. Problem meshing

We select 'Mesh' geometry mode to mesh the soil and click on 'Generate mesh' button. We choose medium element distribution.

The generated mesh is displayed in Figure 6.38.

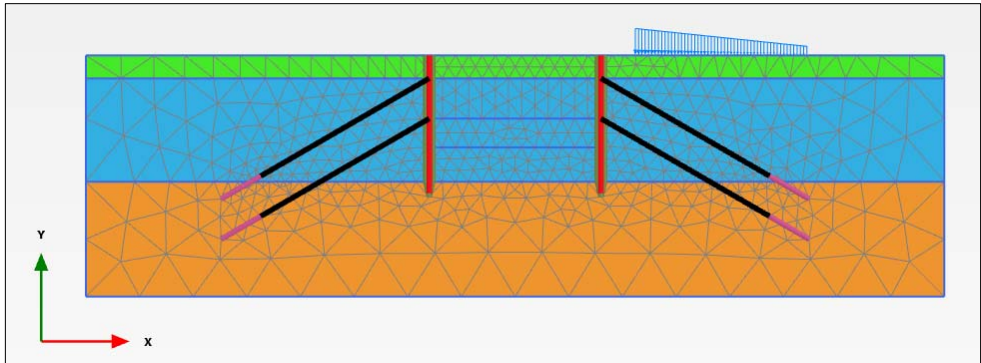


Figure 6. 38. Mesh for the studied problem

6.3.8. Project phases

The excavation has seven phases:

- Initial Phase. Only the ‘natural’ soil is considered.
- Phase 1. The two walls, their interfaces and the load are activated.
- Phase 2. The two first meters of soil (clay) are dug.
- Phase 3. The upper anchors are activated
- Phase 4. The next 3.5 meters of soil (sand) are dug.
- Phase 5. The lower anchors are activated.
- Phase 6. Final excavation of the last 2.5 meters.

In order to define the Initial Phase, in ‘*Phase explorer*’ menu we double click on ‘*Initial Phase*’, In ‘*Calculation type*’ we choose ‘*K₀ procedure*’. In ‘*Loading type*’ we keep ‘*Staged construction*’. Finally, ‘*Phreatic*’ is the selected option for ‘*Pore pressure calculation type*’ (Figure 6.39).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Name	Value
General	
ID	Initial phase [InitialPhase]
Calculation type	K0 procedure
Loading type	Staged construction
ΣM weight	1,000
Pore pressure calculation type	Phreatic
Thermal calculation type	Ignore temperature
Estimated end time	0,000 day
First step	
Last step	
Design approach	(None)
Deformation control parameters	
Updated water pressure	<input type="checkbox"/>
Ignore suction	<input checked="" type="checkbox"/>

Figure 6. 39. Initial Phase configuration

We click on ‘Add’ to create Phase 1 (‘Plastic’ type). The name of this phase can be: ‘Walls and interfaces’. Here, we must activate the two diaphragm walls, their interfaces (watch that once the soil has been divided by some lines for the different excavation phases, walls and interfaces have also been divided), Figure 6.40. We also activate the load.

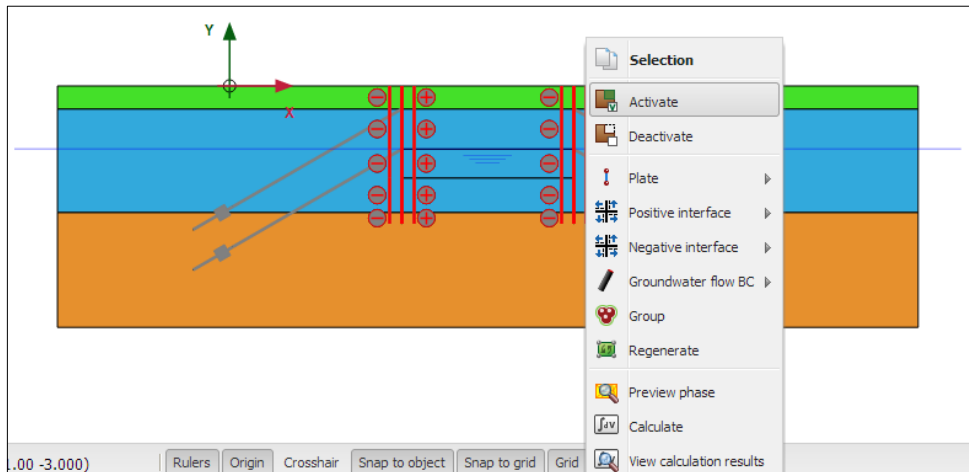


Figure 6. 40. Activation of diaphragm walls and interfaces

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

We click on ‘Add’ to create Phase 2 (‘Plastic’ type). Here, we deactivate the first soil layer (Figure 6.41):

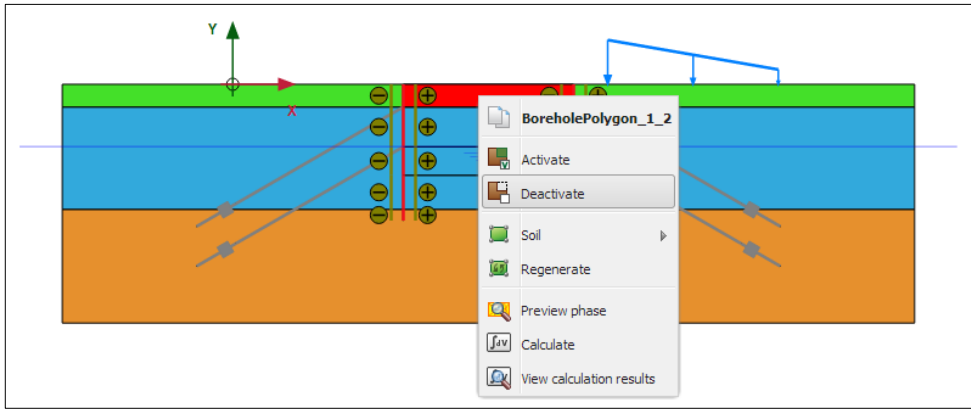


Figure 6. 41. Deactivation of the first layer (clay)

In Phase 3 (‘Plastic’ type), we activate the two upper anchors (Figure 6.42).

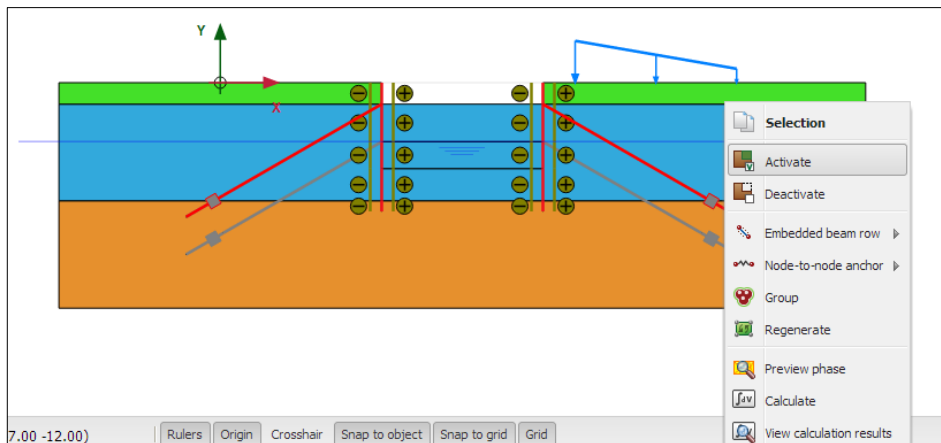


Figure 6. 42. Activation of the upper anchors

Now we are going to apply the prestress force* to each of the anchors. For this, in ‘Selection explorer’ menu, we click on ‘Adjust Prestress’, and in $F_{prestress}$ we write 550 kN (Figure 6.43).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

* **NOTE.** This prestress force value is only applied in the phase where it is defined. For the following phases, this value is increased and decreased according to the force and stress on the surroundings.

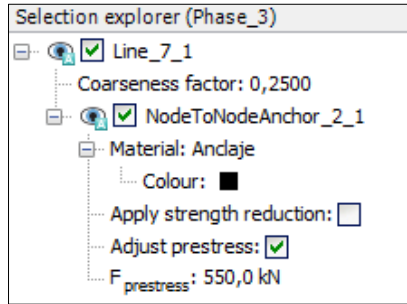


Figure 6. 43. Prestress force adjustment in an upper anchor

In Phase 4 ('Plastic' type), we deactivate the sand layer (3.5-meter depth), Figure 6.44.

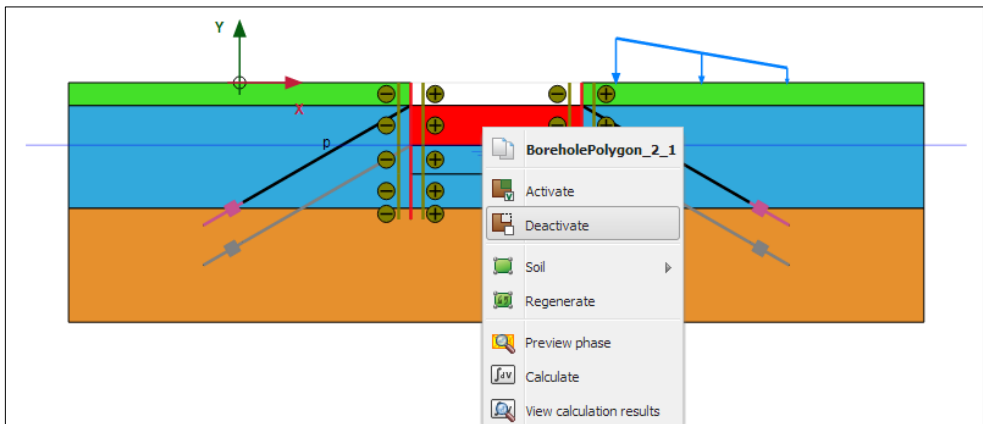


Figure 6. 44. Deactivation of the second layer (3.5 meters of sand)

In Phase 5 ('Plastic' type), we activate the two lower anchors (Figure 6.45) and in 'Selection explorer' menu we mark 'Adjust Prestress'. Finally, we assign a value of 1100 kN to $F_{prestress}$ to each of the lower anchors, Figure 6.46.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

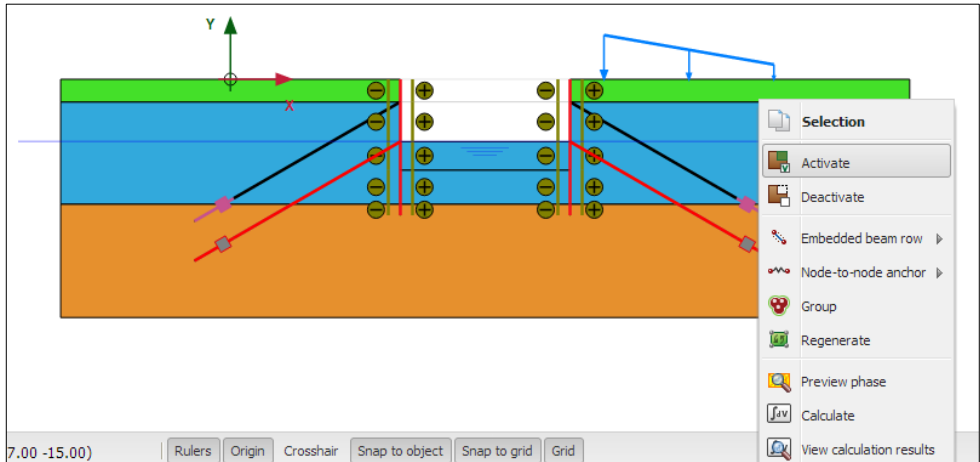


Figure 6. 45. Activation of the lower anchors

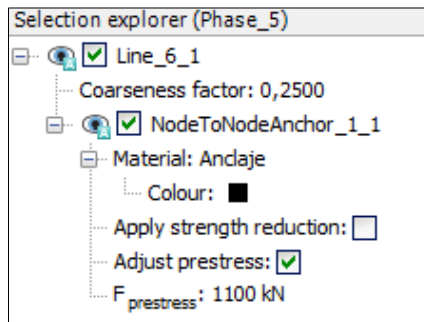


Figure 6. 46. Prestress force adjustment of a lower anchor

Finally, in Phase 6 ('Plastic' type) we must change 'Pore pressure calculation type' to 'Steady state groundwater flow' (Figure 6.47), since, in this last excavation, the initial phreatic level is reached.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

General	
ID	Excavación final [Phase_6]
Start from phase	Andajes inferiores
Calculation type	Plastic
Loading type	Staged construction
ΣM_{stage}	1,000
ΣM_{weight}	1,000
Pore pressure calculation type	Steady state groundwater flow
Thermal calculation type	Ignore temperature
Time interval	0,000 day
Estimated end time	0,000 day
First step	
Last step	
Design approach	(None)

Figure 6. 47. Phase 6 configuration (final excavation)

Now we deactivate the last 2.5 meters (Figure 6.48):

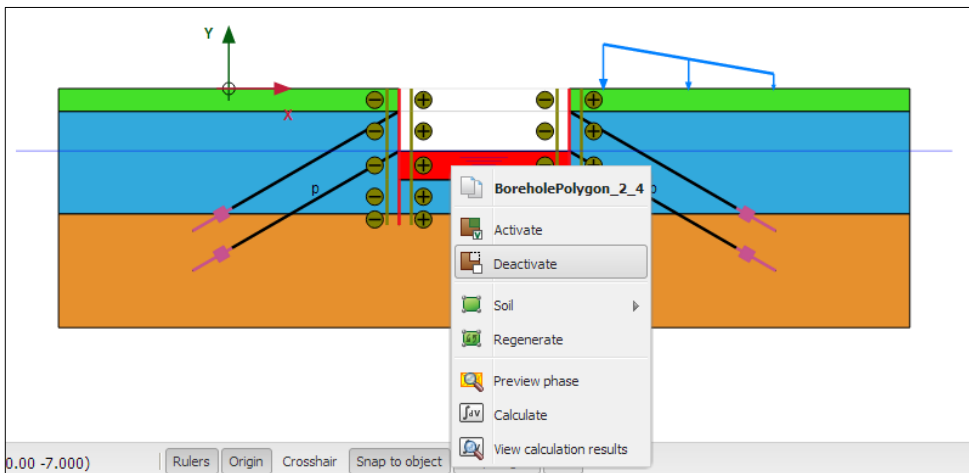


Figure 6. 48. Deactivation of the third layer (2.5 meters of sand)

We go to 'Flow condition' calculation mode. In 'Model explorer' menu we choose 'Attributes library' and then 'Water levels'.

We select 'Create water lever' in the vertical tool bar (Figure 6.49):

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

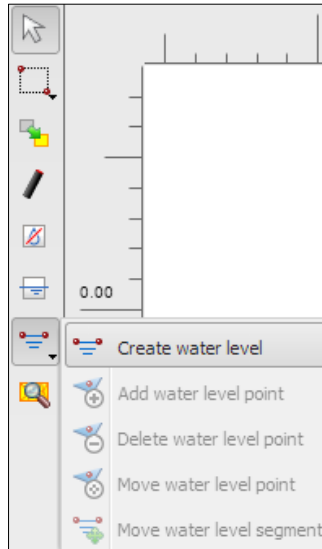


Figure 6. 49. Phreatic level creation

The new phreatic level goes from coordinate (-15, -5.5) to (60, -5.5), passing through (15, -8) and (30, -8), as in Figure 6.50.

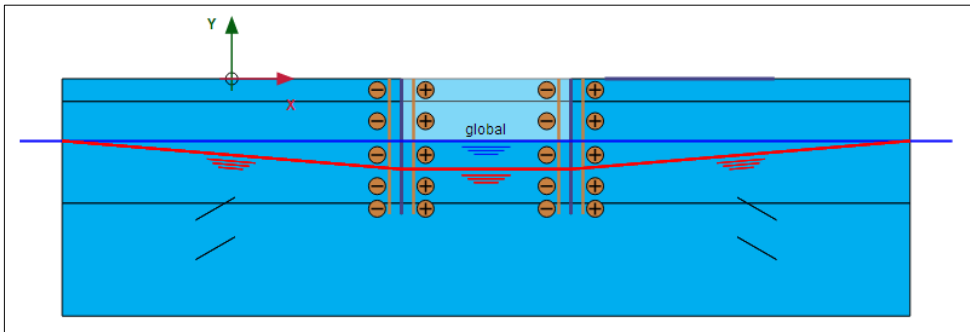


Figure 6. 50. New phreatic level.

We open 'Water levels' menu and 'User water levels'. We double click on 'UserWaterLevel_1' and rename it as 'Lower_phreatic_level' (Figures 6.51 and 6.52).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

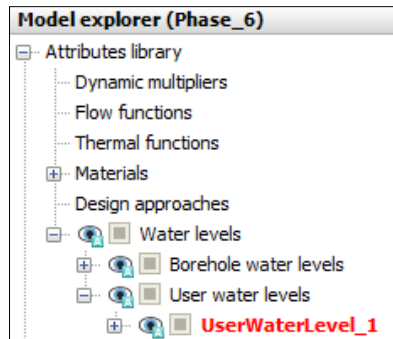


Figure 6. 51. User water level selection

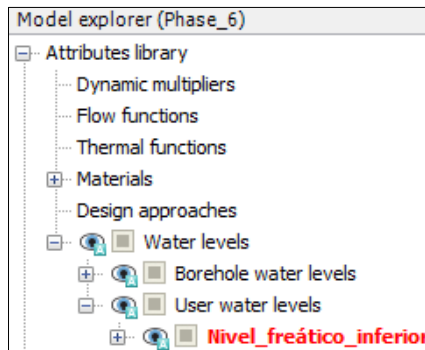


Figure 6. 52. User water level name change

Finally, in 'Model explorer' menu, we choose 'Model conditions' and then 'Water'. We change 'GlobalWaterLevel' value to 'Lower_phreatic_level' (Figure 6.53).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

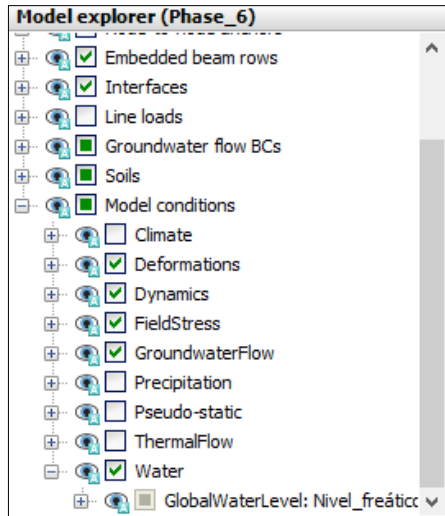


Figure 6. 53. Definition of the created water level ('Lower_phreatic_level') as 'GlobalWaterLevel'

The model and defined water level for Phase 6 are shown in Figure 6.54.

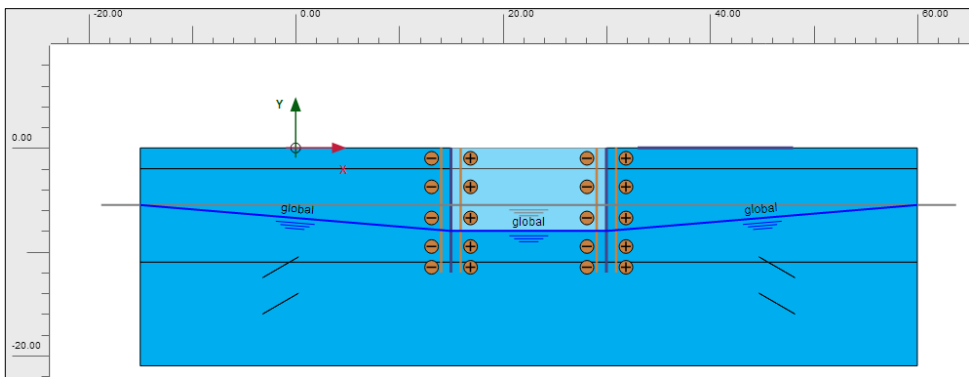


Figure 6. 54. Phase 6 configuration in 'Flow conditions' calculation mode

Before starting the calculation, we can choose some points (Figures 6.55 and 6.56) to display, if wanted, advance graphical outputs.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

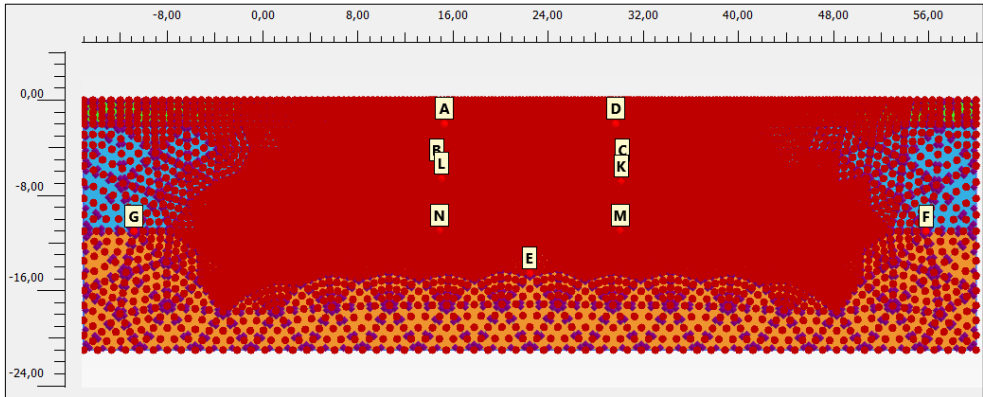


Figure 6. 55. Chosen points for the advance graphical outputs

Name	X	Y	Selected
A (Node 3104)	15,00	-2,00	Pre-calc
B (Node 2850)	15,00	-5,50	Pre-calc
C (Node 8267)	30,00	-5,50	Pre-calc
D (Node 8428)	30,00	-2,00	Pre-calc
E (Node 4170)	22,50	-14,56	Pre-calc
F (Node 5620)	55,78	-11,00	Pre-calc
G (Node 1357)	-10,78	-11,00	Pre-calc
K (Stress point 6...)	30,15	-6,83	Pre-calc
L (Stress point 7...)	15,07	-6,59	Pre-calc
M (Stress point 6...)	30,08	-10,94	Pre-calc
N (Stress point 4...)	14,92	-10,94	Pre-calc

Figure 6. 56. Coordinates for the chosen points for the advance graphical outputs

Once all the phases have been defined, we click on ‘*Calculate*’ button to start the calculations. When these have finished, clicking on ‘*View calculation results*’, we go to PLAXIS Output to display the results.

6.3.9. Problem results

First, we are going to show the bending moments on each of the walls for each phase.

For this, we select ‘*Structural forces in volumes*’ button (Figure 6.57), and with the mouse we select the right-side wall initial (30, 0) and final (30, -12) points. It is also

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

possible to introduce those points in ‘*Centre-line points*’ window (Figure 6.58). If doing so, the steps are:

“*Point #1*”: 30 for X and 0 for Y; we click on ‘*Add*’.

“*Point #2*”: 30 for X and -12 for Y; we click on ‘*Add*’.

We click on ‘*OK*’ (without adding the last point).

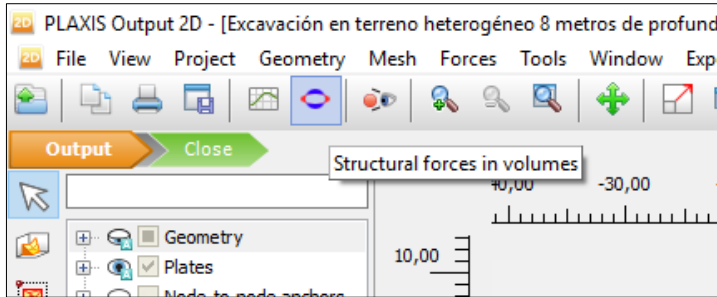


Figure 6. 57. ‘*Structural forces in volumes*’ option selection

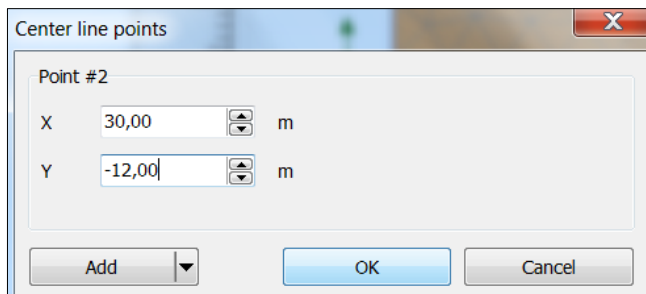


Figure 6. 58. Definition of the second point of the right-side wall for displaying the bending moments

In Figures 6.59-6.64 we can see the bending moments, M , applied on the right-side wall for each phase. It is also possible to display axial forces, N , and shear force, Q , in ‘*Forces*’ option, Figure 6.65.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

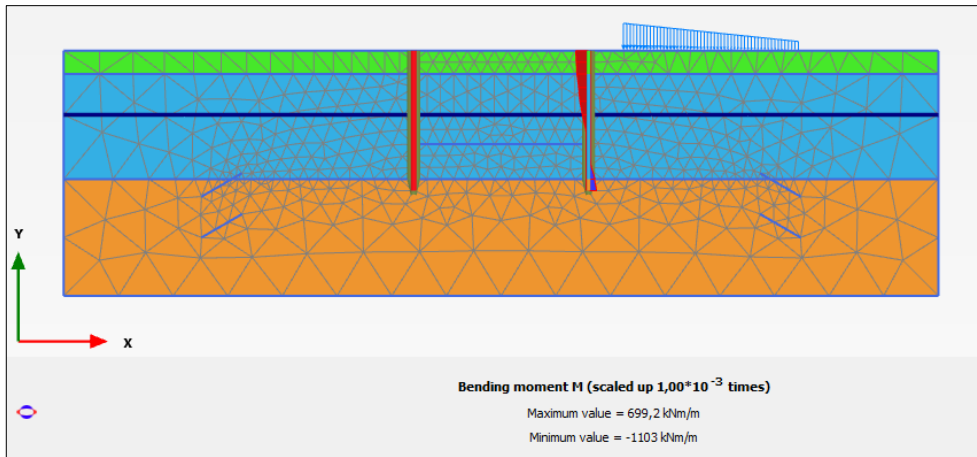


Figure 6. 59. Bending moments on the right-side wall in Phase 1

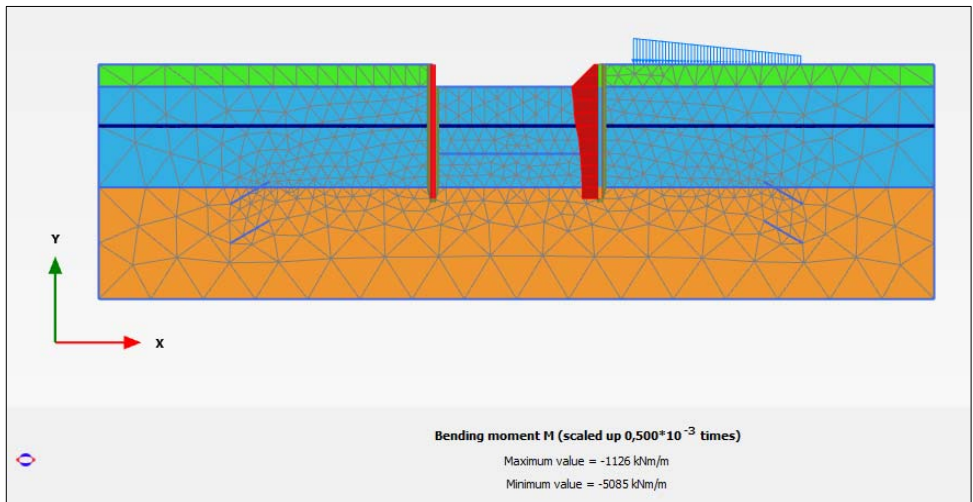


Figure 6. 60. Bending moments on the right-side wall in Phase 2

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

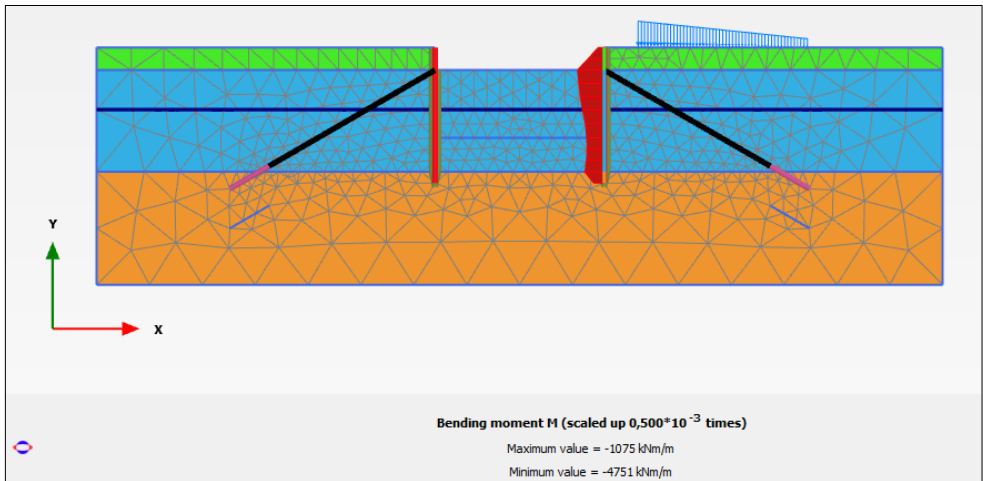


Figure 6. 61. Bending moments on the right-side wall in Phase 3

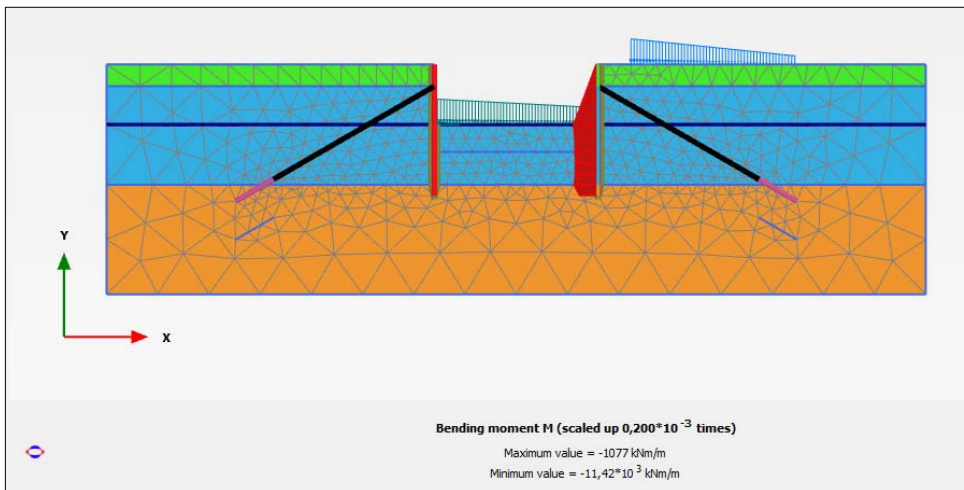


Figure 6. 62. Bending moments on the right-side wall in Phase 4

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

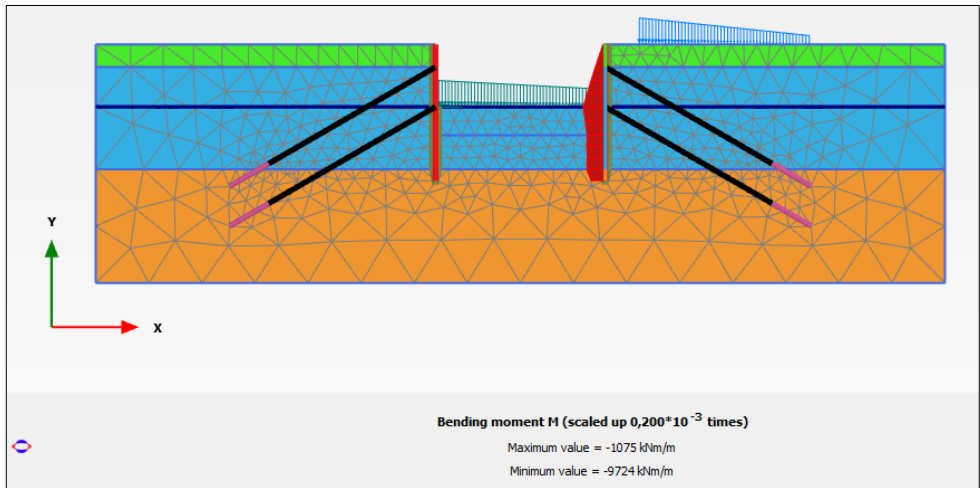


Figure 6. 63. Bending moments on the right-side wall in Phase 5

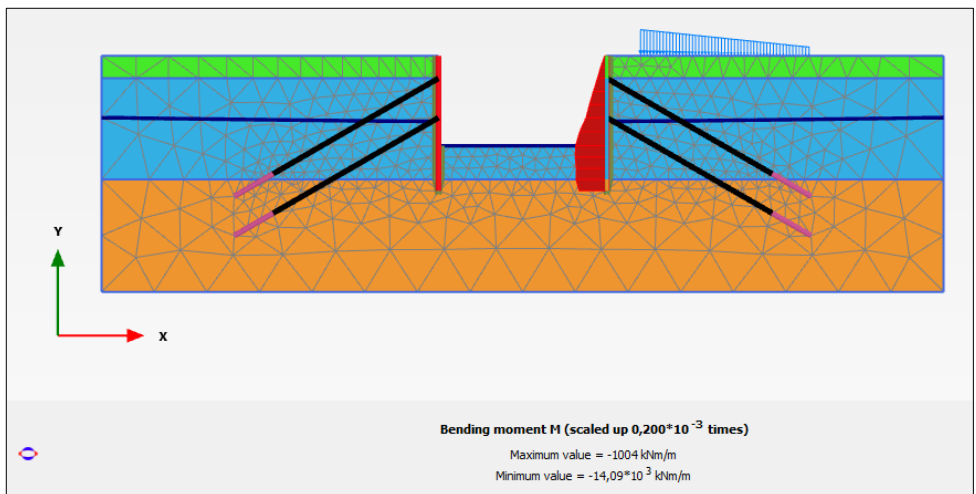


Figure 6. 64. Bending moments on the right-side wall in Phase 6

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

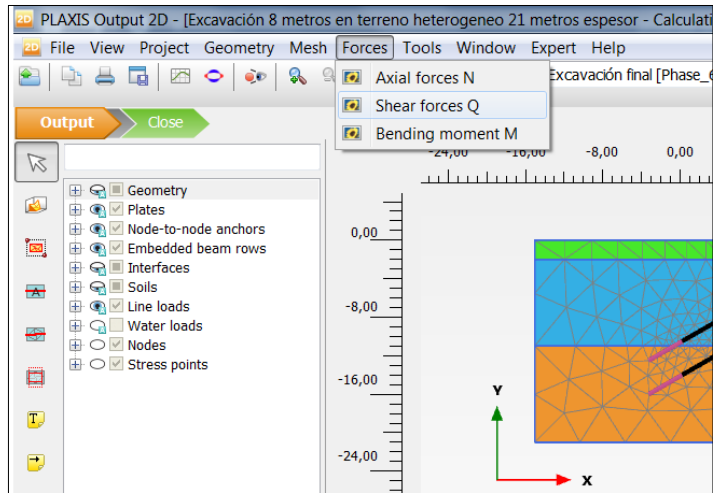


Figure 6. 65. Selection of axial forces, N, shear forces, Q or bending moments, M as displayed forces

In Figures 6.66-6.71 the bending moments on the left-side wall for each phase are displayed.

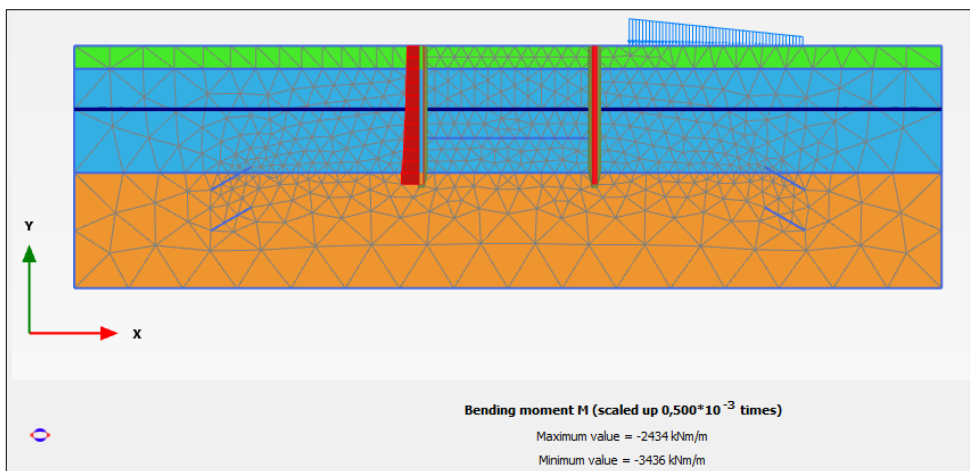


Figure 6. 66. Bending moments on the left-side wall in Phase 1

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

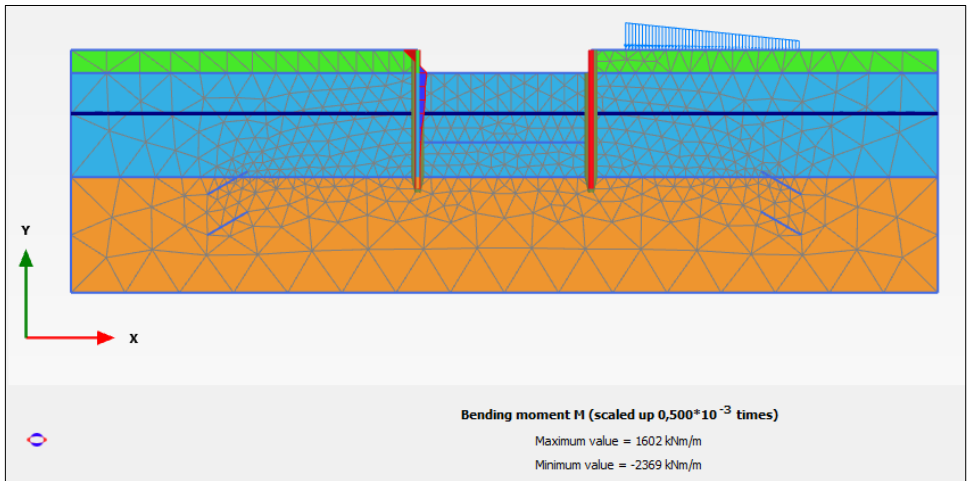


Figure 6. 67. Bending moments on the left-side wall in Phase 2

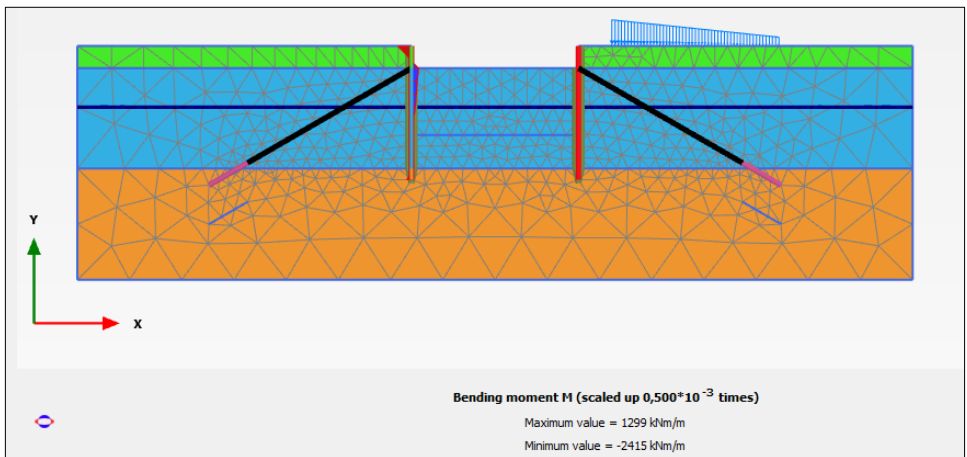


Figure 6. 68. Bending moments on the left-side wall in Phase 3

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

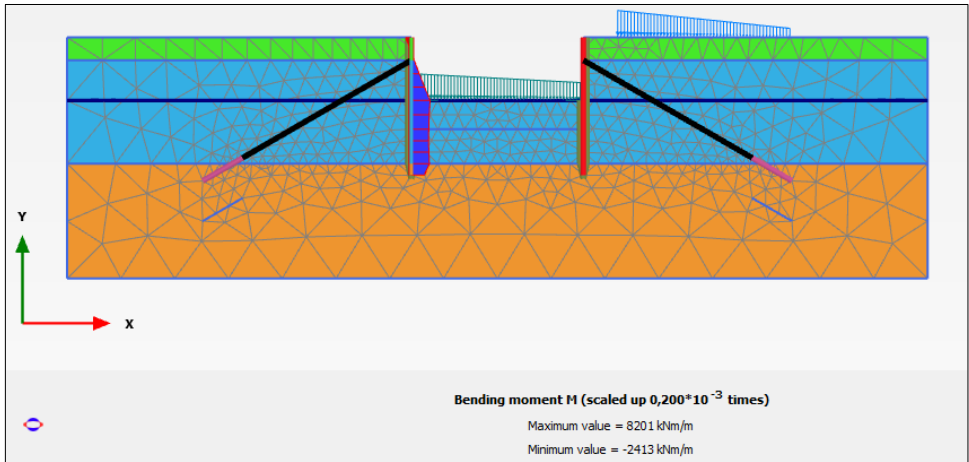


Figure 6. 69. Bending moments on the left-side wall in Phase 4

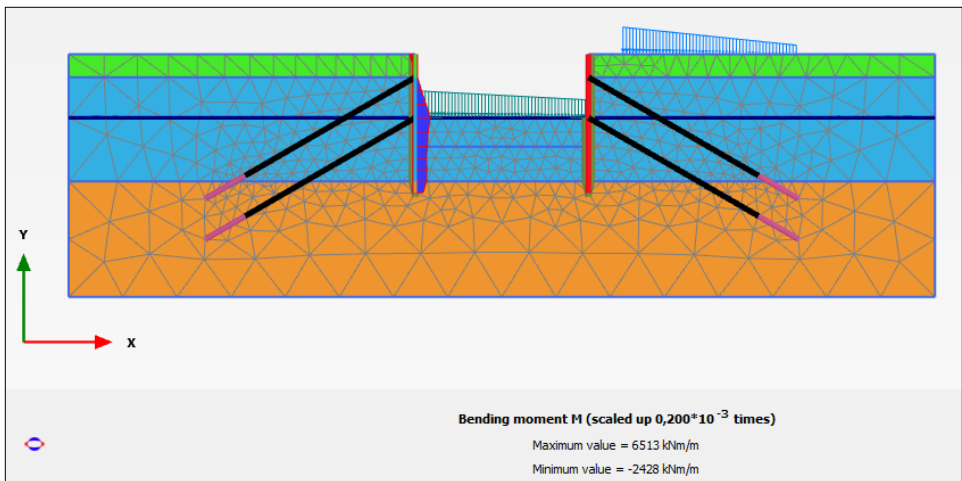


Figure 6. 70. Bending moments on the left-side wall in Phase 5

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

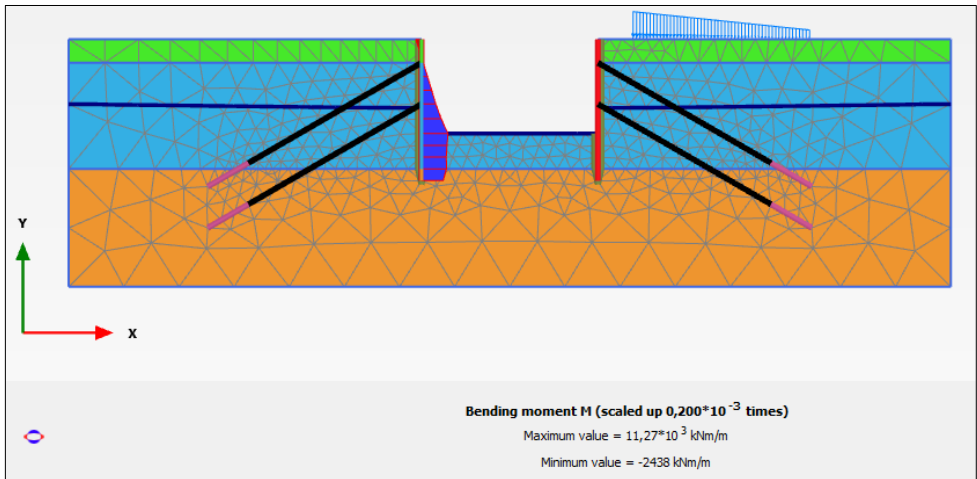


Figure 6. 71. Bending moments on the left-side wall in Phase 6

In Figures 6.72-6.74. total displacements in soil, $|u|$, in Phases 1, 3 and 6 are displayed.

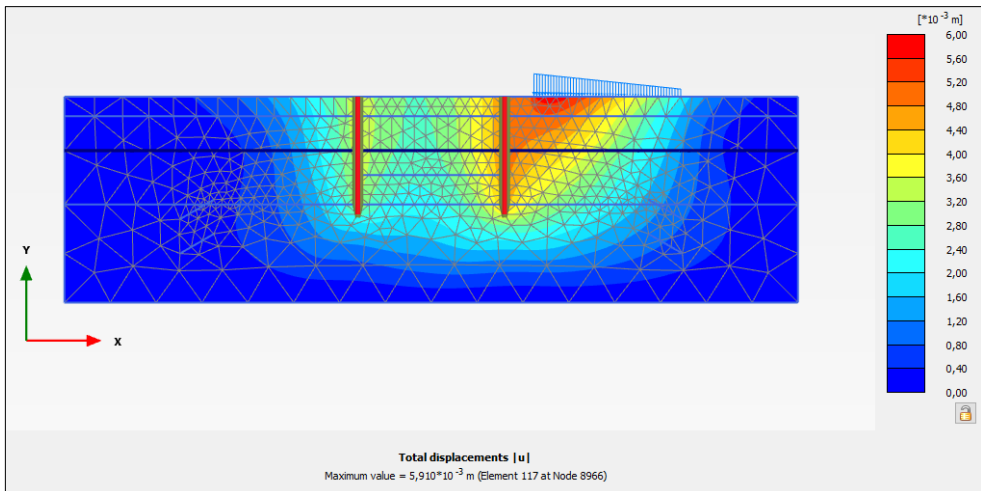


Figure 6. 72. Total displacements, $|u|$, in Phase 1

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

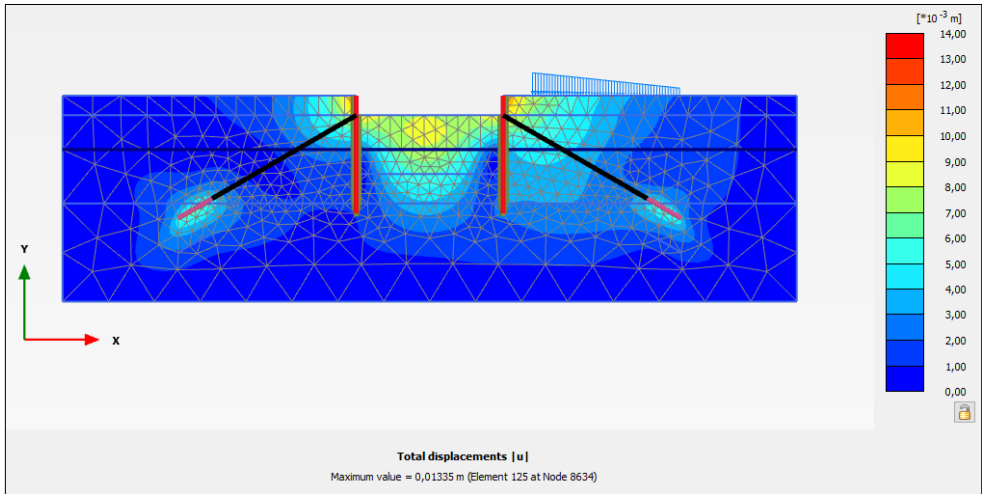


Figure 6. 73. Total displacements, $|u|$, in Phase 3

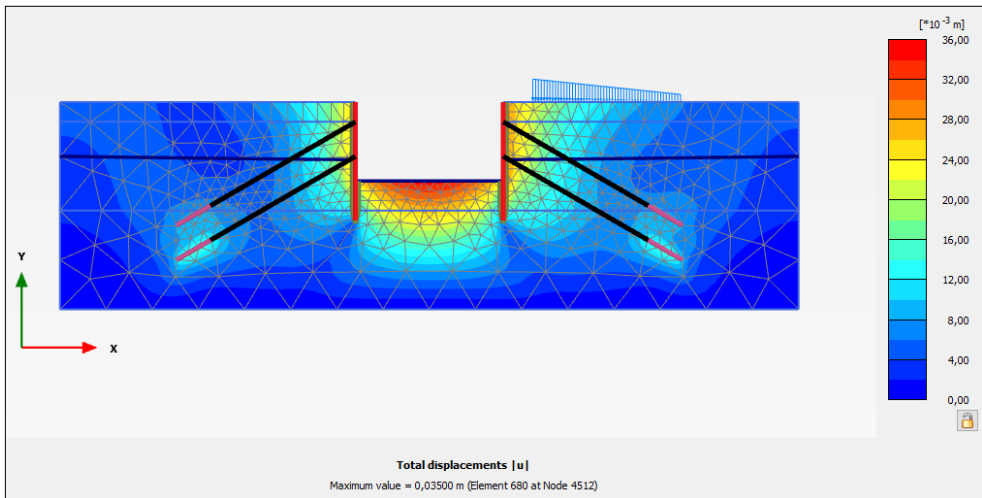


Figure 6. 74. Total displacements, $|u|$, in Phase 6

In order to show the horizontal effective stresses, σ'_{xx} , in 'Stresses' button we choose 'Cartesian effective stress' and finally ' σ'_{xx} ' (Figure 6.75).

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

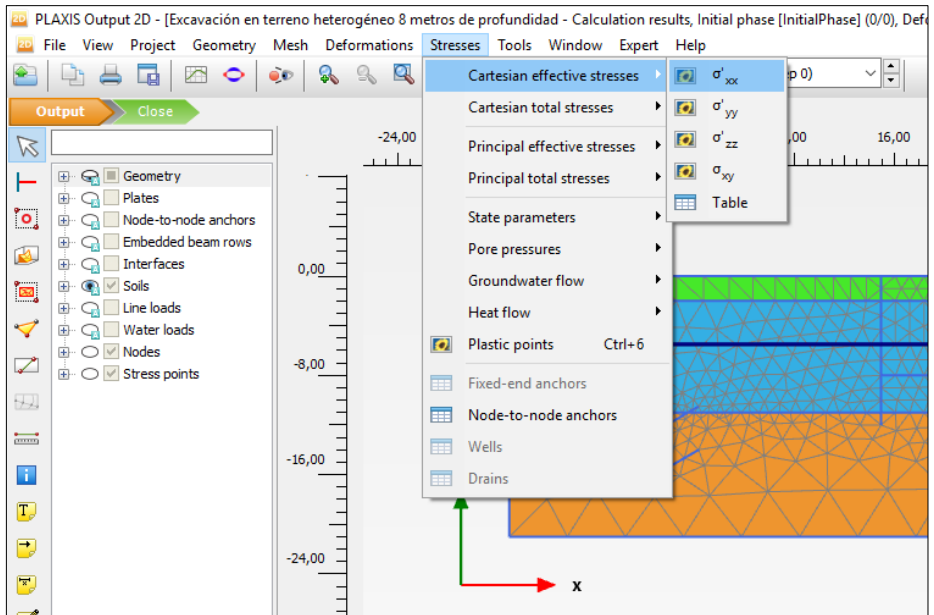


Figure 6. 75. Selection of horizontal effective stresses, σ'_{xx} , display

In Figures 6.76-6.79 the horizontal effective stress, σ'_{xx} , are shown for Initial Phase and Phases 1, 3 and 6.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

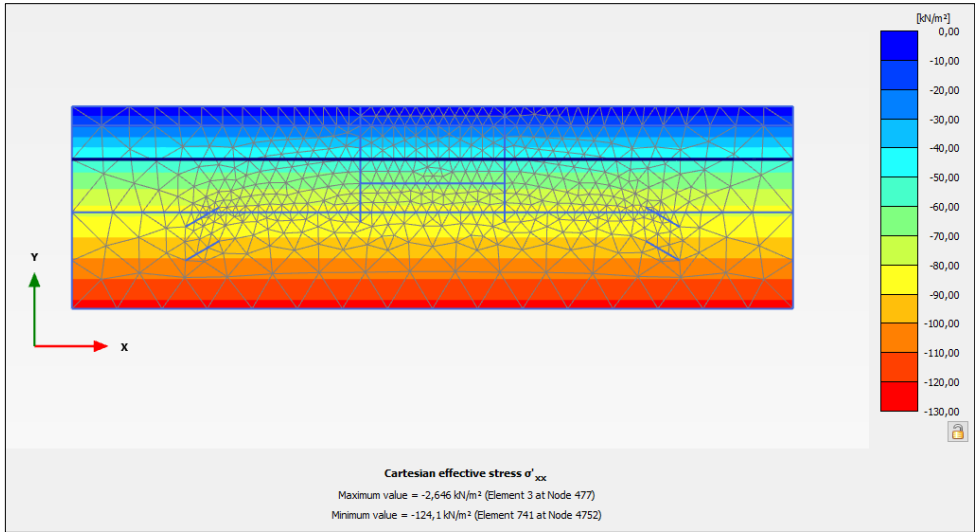


Figure 6. 76. Horizontal effective stresses, σ'_{xx} , in the Initial Phase

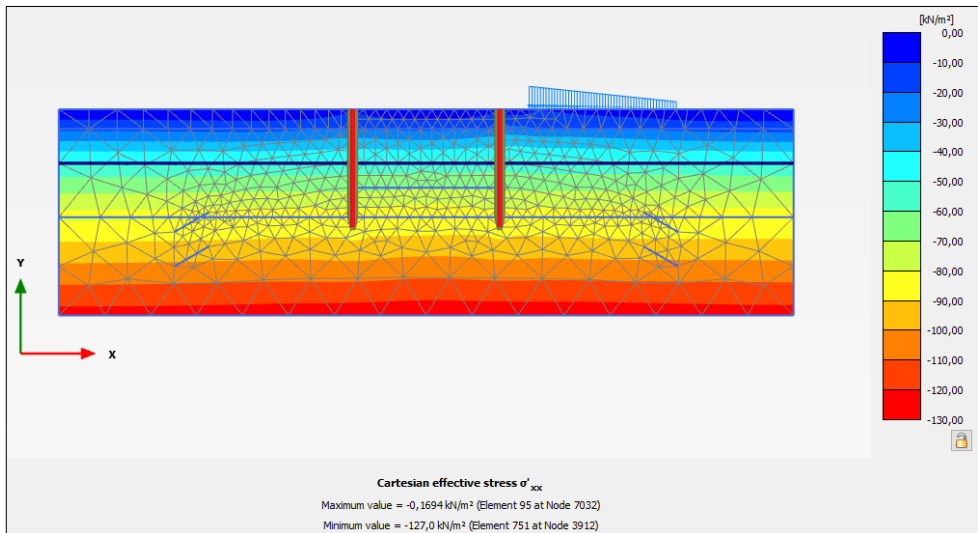


Figure 6. 77. Horizontal effective stresses, σ'_{xx} , in Phase 1

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

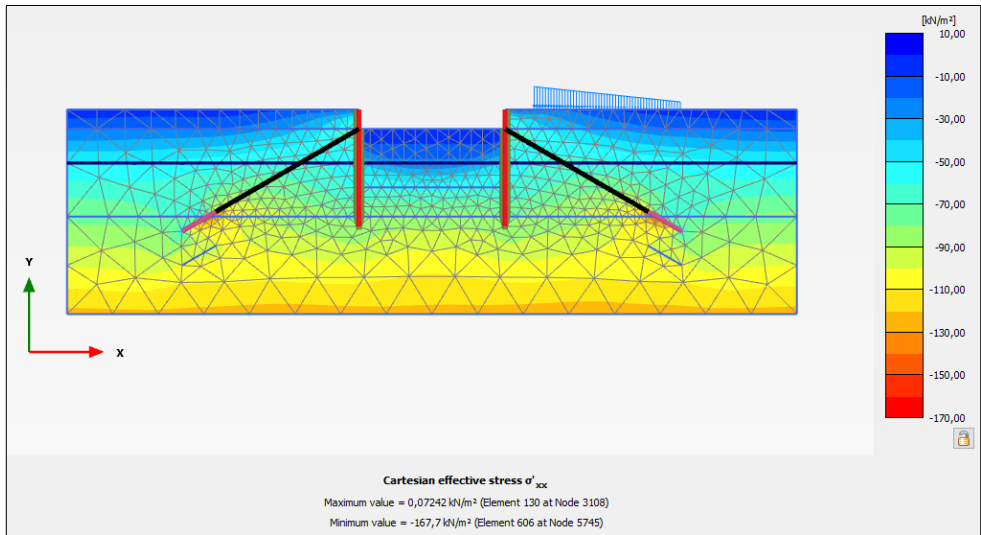


Figure 6. 78. Horizontal effective stresses, σ'_{xx} , in Phase 3

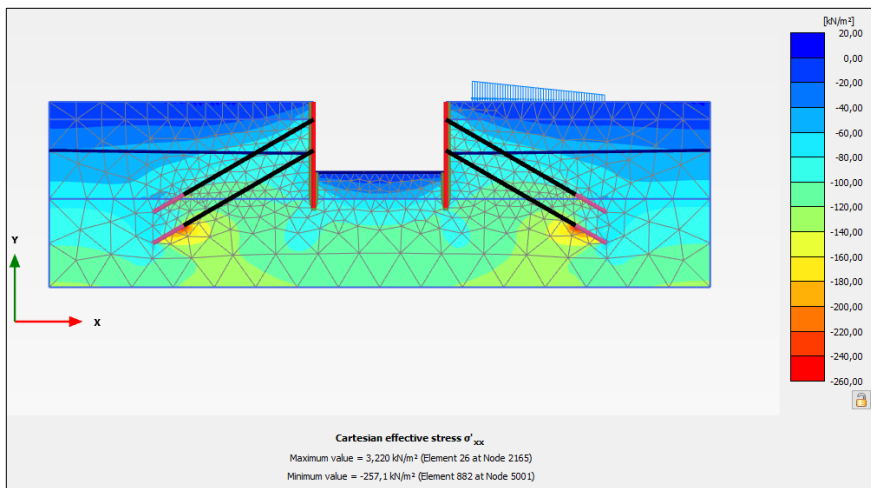


Figure 6. 79. Horizontal effective stresses, σ'_{xx} , in Phase 6

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

Finally, we are going to study the flow net for the last phase (Phase 6).

In order to display the equipotential lines, we select ‘*Stresses*’, ‘*Pore pressures*’ and, finally, we choose ‘*Groundwater head*’ option. Figure 6.80 shows the equipotential lines for Phase 6.

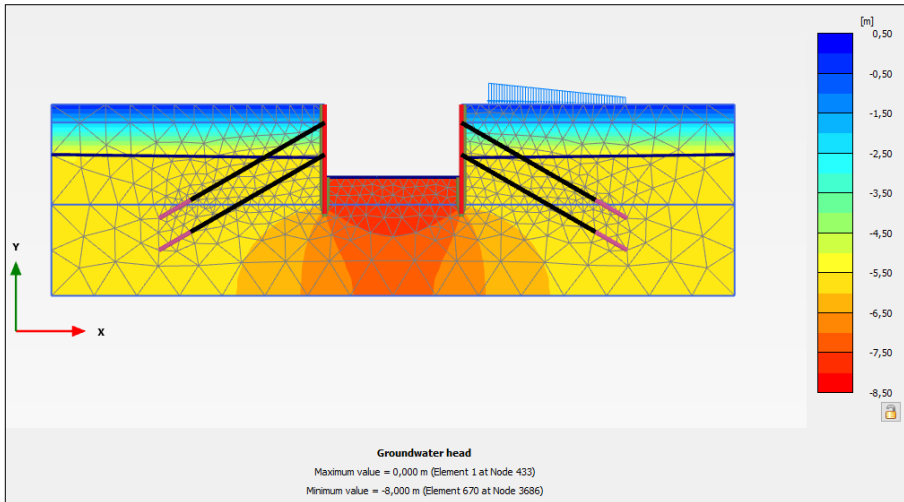


Figure 6. 80. Equipotential lines display in Phase 6

As can be observed, PLAXIS Output proposes a default legend that, occasionally, should be modified (click with the right mouse button on the legend, ‘*Legend settings*’ options, Figure 6.81), since those points above the phreatic level are automatically given the potential value of its position. Therefore, to show a strict flow net, we must only present those equipotential values between -5.5 (initial phreatic level) and -8 (excavation bottom value), Figure 6.82.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

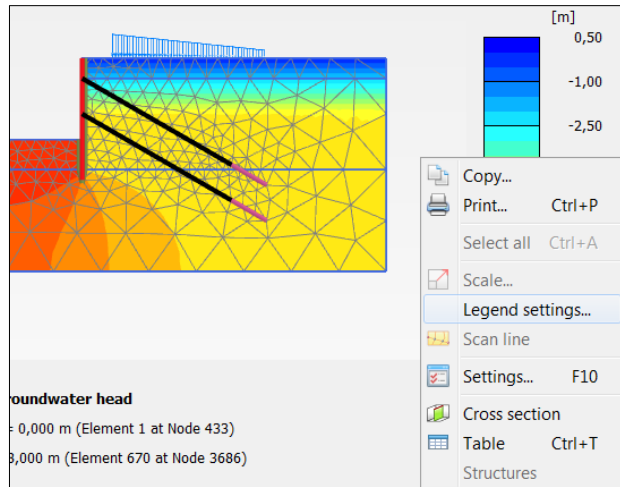


Figure 6. 81. Legend adjustment selection

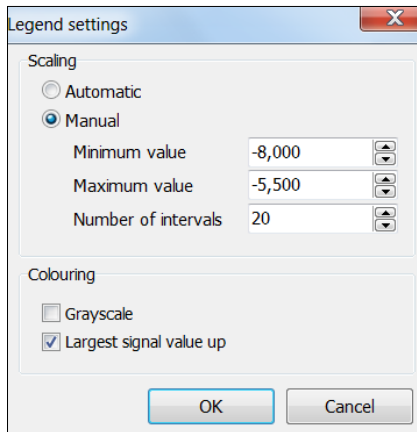


Figure 6. 82. Manual assignment of the parameters of the legend

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

The flow net is displayed in Figure 6.83.

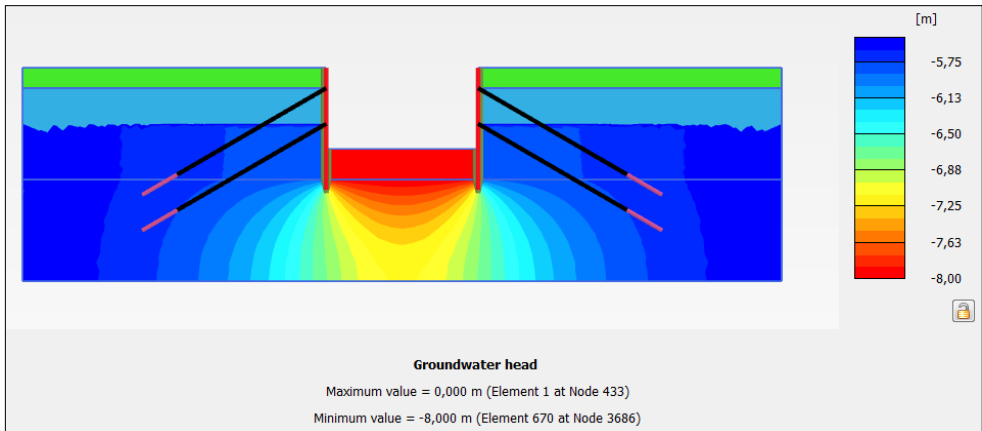


Figure 6. 83. Equipotential line representation in Phase 6

If we want to know the pore pressure distribution, we select ‘Stresses’, ‘Pore pressures’, P_{water} option (Figure 6.84).

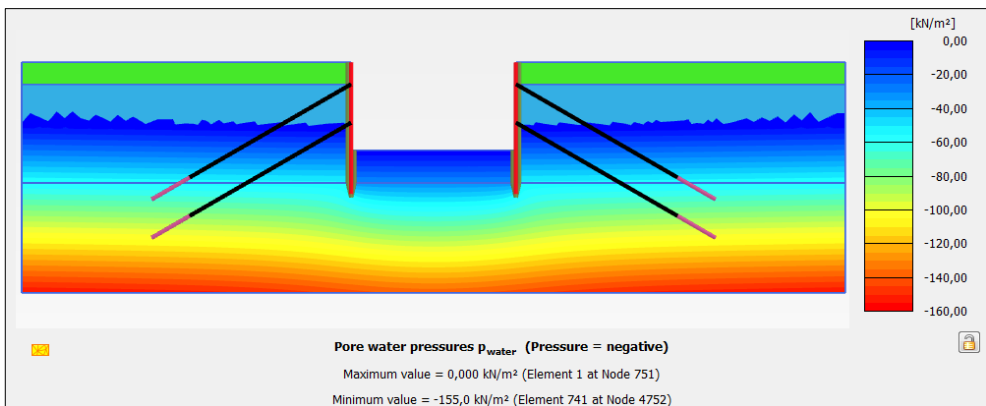


Figure 6. 84. Pore pressure representation in Phase 6

To display the velocity vectors in the net flow, we select ‘Stress’, ‘Groundwater flow’, $|q|$ (select ‘Arrows’ options), Figures 6.85-6.87.

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

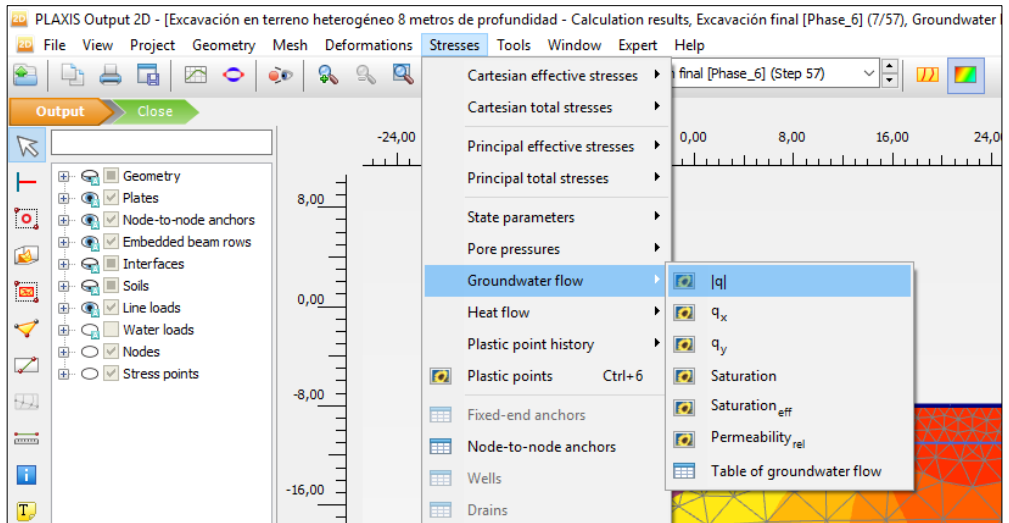


Figure 6. 85. Velocity vectors representation selection

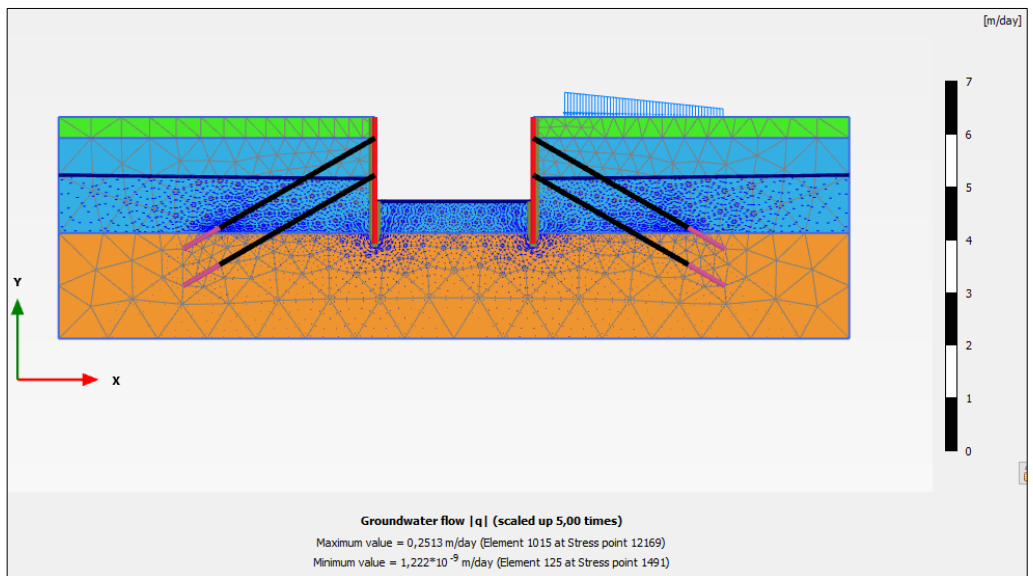


Figure 6. 86. Velocity vectors of the flow net in Phase 6

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

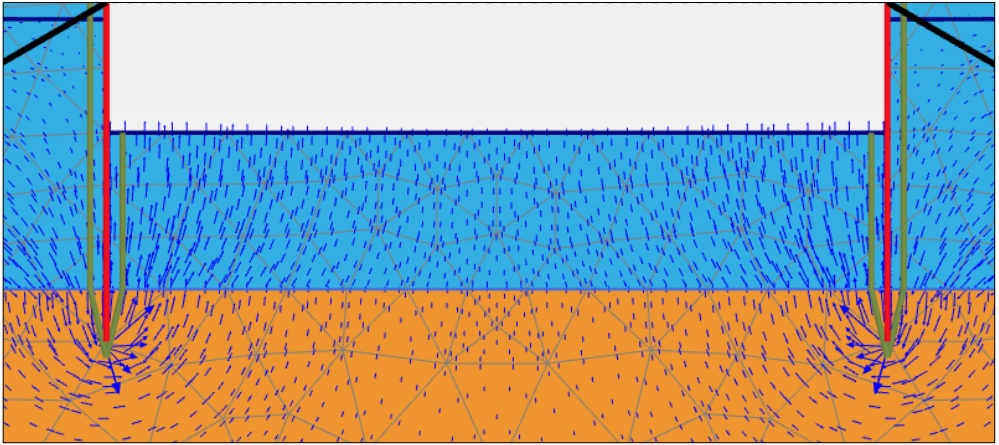


Figure 6. 87. Velocity vectors of the flow net in Phase 6 (zoom below the excavation)

In order to know the groundwater flow in the excavation, we use ‘*Cross section*’ option (Figure 6.88).

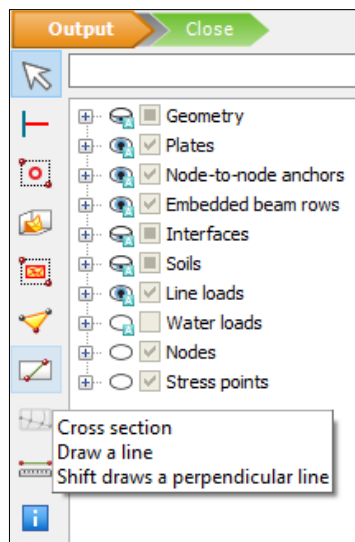


Figure 6. 88. ‘*Cross section*’ option selection

Exercise 06. Excavation in heterogeneous soil between two diaphragm walls

We choose coordinates (15, -8) and (30, -8) to define the section (button of the excavation). Although it is possible to select these coordinates in the workspace, it is better to introduce them in the emerging window (Figure 6.89).

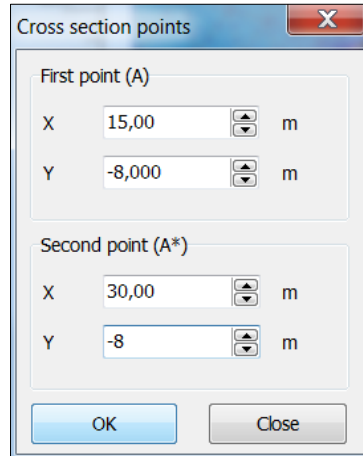


Figure 6. 89. Coordinate selection for the cross section where we want to obtain the groundwater flow

The discharge flow at the excavation button appears in Figure 6.90, and it is $0.63 \text{ m}^3\text{7day/m}$.

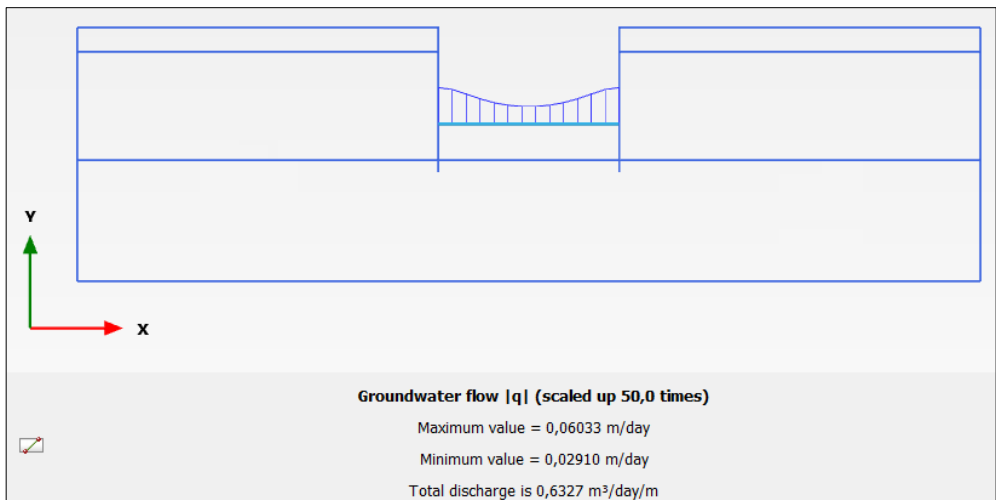


Figure 6. 90. Groundwater flow at the excavation button due to the flow net

Exercise 07

Pile foundation under bridge deck support

7.1 INTRODUCTION

On a terrain made of three soil layers, an embankment is built (with three layers of two meters thickness) to place a bridge deck support and a pile foundation. The different soil layers consist on two materials (sand and clay).

The first layer is built in a 3-day period and, once it is finished, we wait 25 days to build the second layer. This second layer is also built in a 3-day period, and the third layer is started once 50 days have passed. Finally, the last layer is built also in a 3-day period and it is kept for 75 days. The three layers consist on sand material with the same properties as the sand in the foundation soil.

Once the embankment is built, a cutting is excavated with a total depth of 2 meters, a width in the bottom of the cutting of 6 meters and 2H:1V slopes in both sides. Afterwards, the foundation and the support are built.

Finally, part of consolidated embankment is removed to create a slope (2H:1V) that allows the construction of the linear work that will be placed under the bridge, and the back of the support vertical plate is filled.

7.2 GEOTECHNICAL PARAMETERS, AND PLATE AND PILES PROPERTIES

Tables 7.1 and 7.2 show the geotechnical parameters of the sand and the clay that appear in the three layers of the initial terrain, as well as in the three embankment layers (sand). The position and thickness of the layers are defined in the following points.

Exercise 07. Pile foundation under bridge deck support

Geotechnical parameters of sand

Material	Sand
Constitutive model	Mohr-Coulomb
Material behaviour	Drained
Soil unit weight above phreatic level (γ_{unsat})	17.5 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	20 kN/m ³
Young's modulus (E')	31500 kPa
Poisson's ratio (ν')	0.28
Effective soil cohesion (c')	0 kPa
Effective friction angle (φ')	32°
Effective soil cohesion (c')	0°
Horizontal hydraulic conductivity K_x	0.4 m/day
Vertical hydraulic conductivity K_v	0.1 m/day
R_{inter} (strength reduction factor inter.)	0.7

Table 7. 1. Geotechnical parameters of sand

Geotechnical parameters of clay

Material	Clay
Constitutive model	Mohr-Coulomb
Material behaviour	No drenado tipo A
Soil unit weight above phreatic level (γ_{unsat})	19 kN/m ³
Soil unit weight below phreatic level (γ_{sat})	19 kN/m ³
Young's modulus (E')	13500 kPa
Poisson's ratio (ν')	0.33
Effective soil cohesion (c')	5 kPa
Effective friction angle (φ')	22.5°
Effective soil cohesion (c')	0°
Horizontal hydraulic conductivity K_x	$5.5 \cdot 10^{-4}$ m/day
Vertical hydraulic conductivity K_v	$2.1 \cdot 10^{-4}$ m/day
R_{inter} (strength reduction factor inter.)	0.5

Table 7. 2. Geotechnical parameters of clay

Exercise 07. Pile foundation under bridge deck support

Tables 7.3 and 7.4 present the properties of support plates and piles.

Properties of support plates

Material type	Elastic
Isotropic	Yes
Normal stiffness (EA)	$3 \cdot 10^7$ kN/m
Flexural rigidity (EI)	$2.5 \cdot 10^6$ kN/m ² /m
Unit weight (w)	24 kN/m/m
Poisson's ratio (ν)	0

Table 7. 3. Material properties of support plates

Properties of the piles

Young's modulus (E)	$3 \cdot 10^7$ kN/m ²
Unit weight (γ)	14 kN/m ³
Pile type	Pre-defined
Type of pre-defined pile	Solid circular
Diameter	0.6 m
Spacing out-of-plane ($L_{spacing}$)	2 m
Axial skin resistance	Linear
Initial axial skin resistance ($T_{skin,start,max}$)	10 kN/m
Final axial skin resistance ($T_{skin,end,max}$)	100 kN/m
Base resistance (F_{max})	1000 kN

Table 7. 4. Material properties of piles

7.3. RESOLUTION OF THE PROBLEM WITH PLAXIS 2D

7.3.1 Introduction of the project parameters

The name of the problem can be 'Pile foundations under bridge deck support'. In 'Model' tab we check if 'Plane strain' option is selected in model and '15-Noded' in elements. We define the workspace contour with the values in Table 7.5.

x_{min}	0	m
x_{max}	70	m
y_{min}	-25	m
y_{max}	9	m

Table 7. 5. Project contours

In ‘Constants’ tab we keep the default values.

7.3.2 Definition of the initial stratigraphy and the three embankment layers

We only create a borehole in point (0, 0). We add 6 layers in ‘Modify soil layers’ window. The thickness of each layer is presented in Figure 7.1 (natural terrain consists on layer 4, 5 and 6, while the embankment consists on layers 3, 2 and 1).

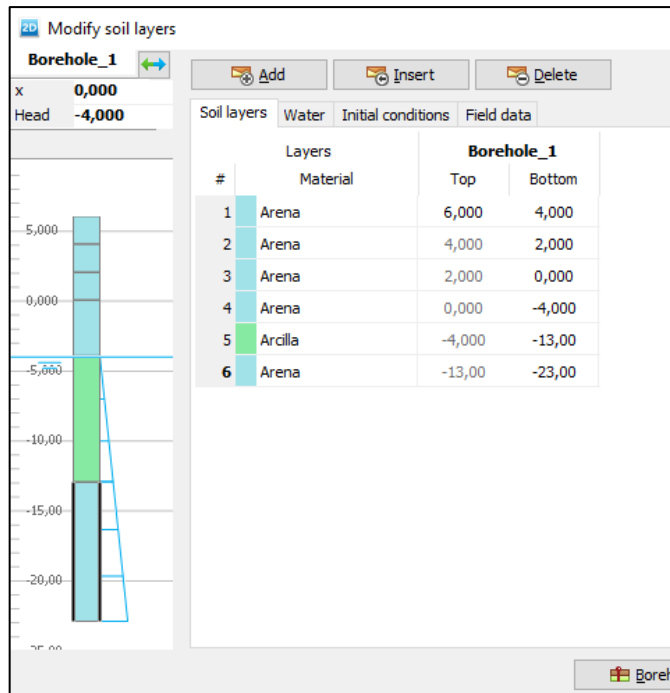


Figure 7. 1. First and only borehole to define initial terrain and embankment

The phreatic level is located 4 meters below the initial ground surface. Once the different layers have been created, we must create the two materials to model soil and, afterwards, those for plates and piles. In order to define both sand and clay, we click on ‘Show materials’ button. In ‘Set type’, we select ‘Soil and interface’, ‘New’ and introduce the geotechnical parameters of sand first, (Table 7.1) and then, those of clay (Table 7.2).

Now, we have to define the material to model support plates. In ‘Set type’, we select ‘Plates’, ‘New’ and introduce the properties of plates (Table 7.3). Finally, we define

Exercise 07. Pile foundation under bridge deck support

the material for piles. Again in ‘*Set type*’, we select ‘*Embedded beam row*’, ‘*New*’ and introduce the material in Table 7.4.

The three embankment layers (2-meter thickness each) are sand. The first layer of initial soil (4 meters) and the deeper one (10 meters) are also sand material, while the middle one (9-meter thickness) is the only one that presents clay (Figure 7.2).

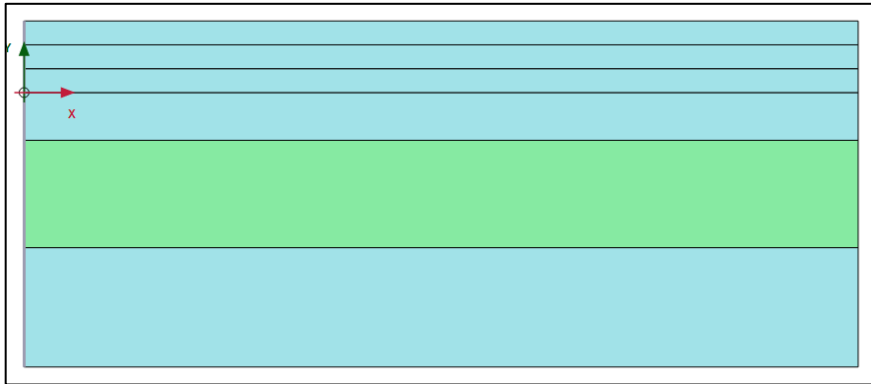


Figure 7. 2. Stratigraphy of initial terrain with the three embankment layers

7.3.3. Definition of support, piles, interfaces and lines to excavate and carry out the cutting

We go to ‘*Structure*’ geometry mode. We draw two lines to create the support plates. These lines have the coordinates presented in Table 7.6.

Line	X _{initial}	Y _{initial}	X _{final}	Y _{final}
1	37 m	6 m	37 m	4 m
2	33 m	4.3 m	37 m	4.3 m

Table 7. 6. Coordinates for the two lines that define the plates

We click on line 1 with the right mouse button, select ‘*Create*’ and then ‘*Plate*’. We select line 1 again with the right mouse button. In ‘*Plate*’ we click on ‘*Set material*’ and finally we choose the material we have previously created for this. We do the same for line 2. We also must create positive and negative interfaces for both plates.

Exercise 07. Pile foundation under bridge deck support

In order to create the piles, we draw two vertical lines with the coordinates presented in Table 7.7.

Line	X_{initial}	Y_{initial}	X_{final}	Y_{final}
3	34 m	4.3 m	34 m	-18 m
4	36 m	4.3 m	36 m	-18 m

Table 7. 7. Coordinates for the two lines that define the piles

We can see that both piles are separated by 2 meters. We click on line 3 with the right mouse button, select ‘*Create*’ and then ‘*Embedded beam row*’. Then, we select again line 3 with the right mouse button, in ‘*Embedded beam row*’ we click on ‘*Set material*’ and choose the material we have created before for this. We do the same for line 4.

Finally, we define the lines that will define the cutting and the excavation. The line coordinates are presented in Table 7.8.

Excavation for plates and piles

Line	X_{initial}	Y_{initial}	X_{final}	Y_{final}
5	27 m	6 m	31 m	4 m
6	37 m	4 m	41 m	6 m

Excavation for plates and piles

Line	X_{initial}	Y_{initial}	X_{final}	Y_{final}
7	31 m	4 m	23 m	0 m

Table 7. 8. Coordinates of lines for cutting and excavation

Exercise 07. Pile foundation under bridge deck support

Figure 7.3 show plates, piles, interfaces and excavation and cutting lines.

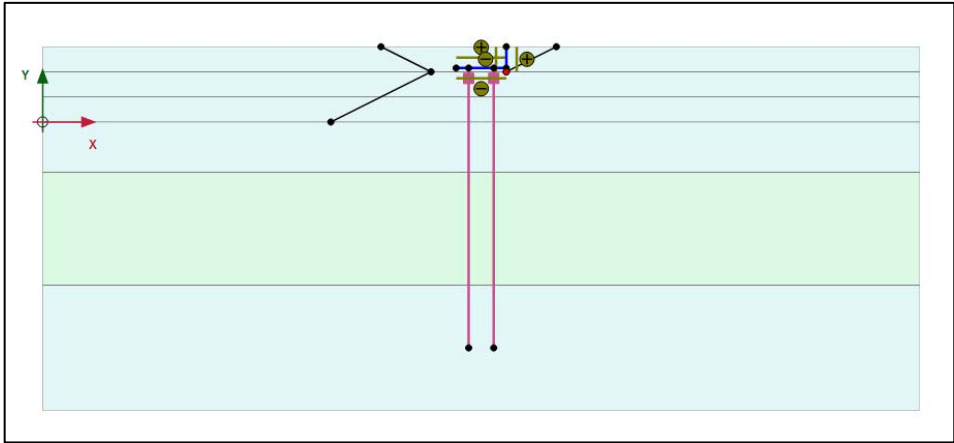


Figure 7. 3. Definition of plates, piles, interfaces and cutting and excavation lines

7.3.4. Problem meshing

We go to 'Mesh' geometry mode, click on 'Generate mesh' and choose a medium element definition. Figure 7.4 shows the created mesh.

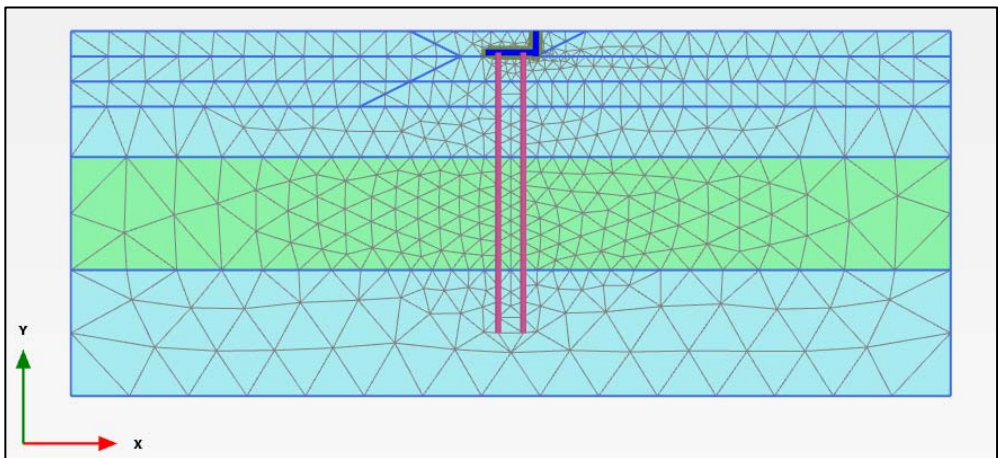


Figure 7. 4. Problem meshing

7.3.5. Project phases

The project presents 11 phases:

- **Initial Phase.** The only activated elements are the three terrain layers, so we must deactivate the three embankment layers and check that plates, piles and interfaces are also deactivated. In '*Calculation type*' we choose '*K0 procedure*'. In '*Loading type*' we select '*Staged construction*', and finally, in '*Pore pressure*' we select '*Phreatic*' option.
- **Phase 1.** The first embankment layer is built. We activate then the first embankment layer and check that the rest of elements are still deactivated. In '*Calculation type*' we choose '*Consolidation*', in '*Loading type*' we select '*Staged construction*', in '*Pore pressure*' we choose '*Phreatic*' option and in '*Time interval*' we write 3 days.
- **Phase2.** Consolidation of the first embankment layer. This phase is the same as Phase 1, but with a different time interval (25 days).
- **Phase 3.** Construction of the second embankment layer. For this we activate this second layer. The other options are the same as in Phase 1.
- **Phase 4.** Consolidation of the second embankment layer. This phase is the same as Phase 3, but with a different time interval (50 days).
- **Phase 5.** Construction of the third embankment layer. For this we activate this third layer. The rest of the options are the same as in Phase 3.
- **Phase 6.** Consolidation of the third embankment layer. This phase is the same as Phase 5, but with a different time interval (75 days).
- **Phase 7.** Excavation for the installation of plates and piles. We must deactivate the soil between lines 5 and 6 (Table 7.8, Figure 7.5). The rest of the options are the same as in Phases 1, 3 and 5.

Exercise 07. Pile foundation under bridge deck support

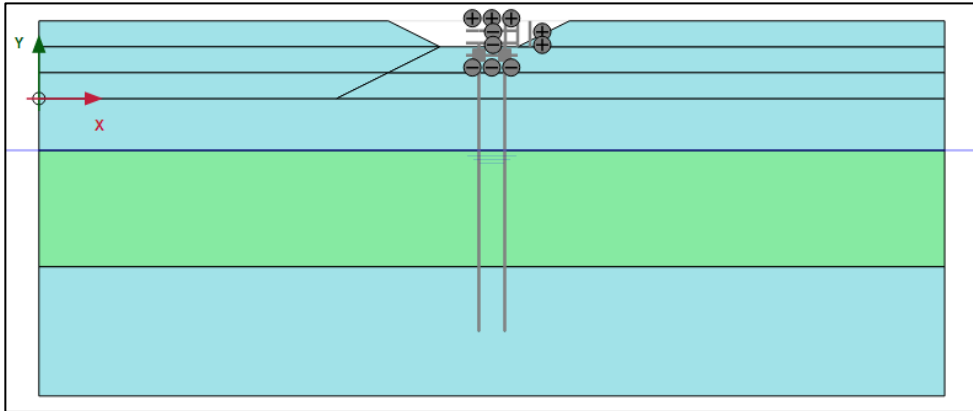


Figure 7. 5. Phase 7. Excavation for the installation of plates and piles

- **Phase 8.** Construction of plates and piles. We must activate plates, interfaces and piles. In 'Calculation type', we choose 'Consolidation', in 'Loading type', we select 'Staged construction', in 'Pore pressure' we choose 'Phreatic' option and in 'Time interval' we write 10 days (Figure 7.6).

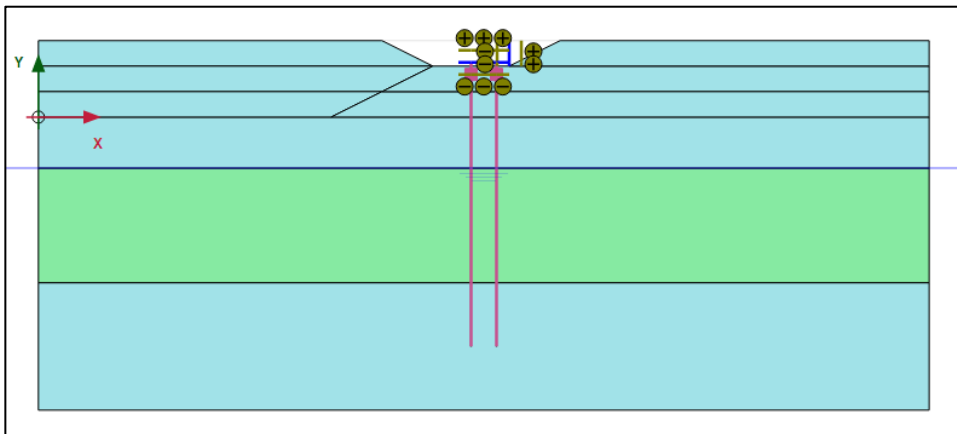


Figure 7. 6. Phase 8. Activation of plates, interfaces and piles

- **Phase 9.** Cutting construction. We must deactivate the polygons on the left of lines 5 and 7 (Table 7.8). We activate the portion of soil in the right of

Exercise 07. Pile foundation under bridge deck support

the vertical plate. The rest of the options are the same as in Phase 8 (Figure 7.7).

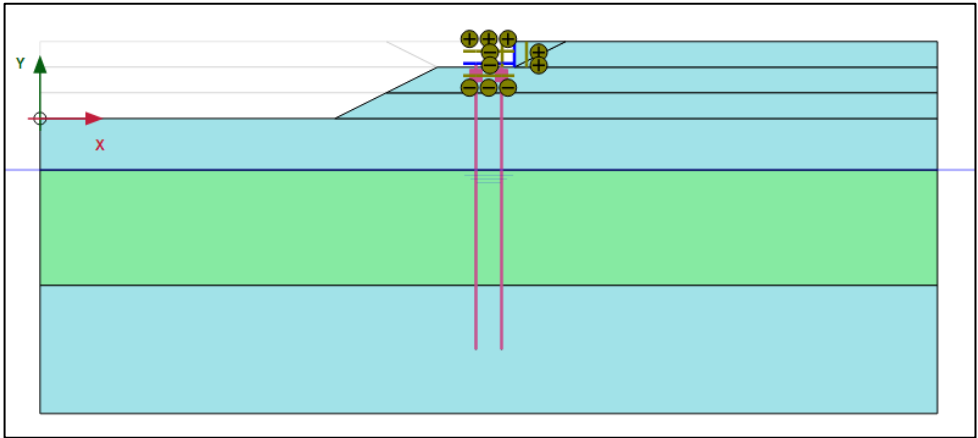


Figure 7. 7. Phase 9. Deactivation of cutting and activation of soil portion on the right of the vertical plate

- **Phase 10.** Final consolidation time. In ‘*Calculation type*’ we choose ‘*Consolidation*’. In ‘*Loading type*’ we select ‘*Minimum pore pressure*’. In ‘*Pore pressure*’ we select ‘*Phreatic*’ option. In ‘*P-stop*’ we write 10 kPa.

Once all phases have been defined, we click on ‘*Calculate*’ button, and we display results in PLAXIS Output by clicking on ‘*View calculation results*’.

7.3.6. Results of the problem for the Final Consolidation Phase

All the representations in this point are referred to the last project phase (Final Consolidation Phase). Figure 7.8 present the deformed mesh of the problem.

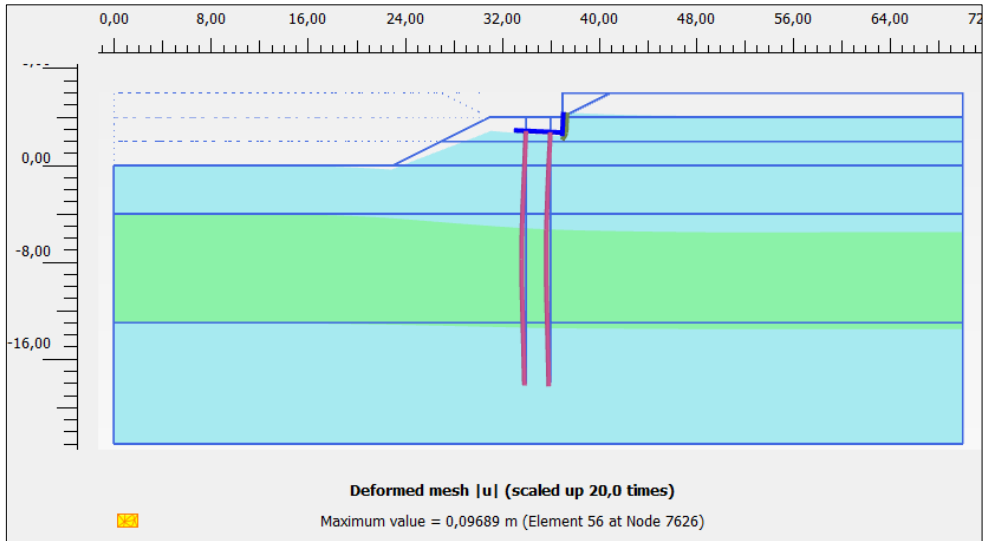


Figure 7. 8. Deformed mesh in the Final Consolidation Phase

Total displacements, $|u|$, horizontal displacements, u_x , and vertical displacements, u_y , are presented in Figure 7.9, 7.10 and 7.11 respectively.

Exercise 07. Pile foundation under bridge deck support

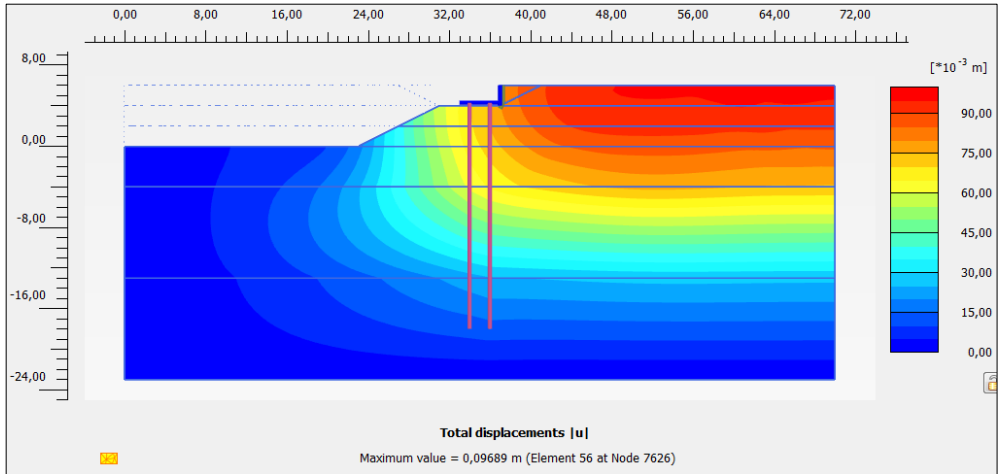


Figure 7. 9. Total displacements, $|u|$, in the Final Consolidation Phase

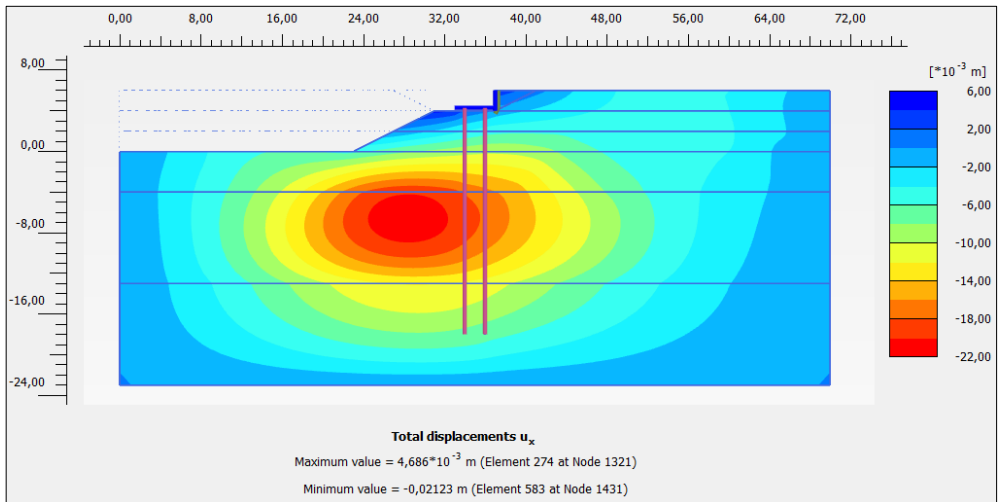


Figure 7. 10. Horizontal displacements, u_x , in the Final Consolidation Phase

Exercise 07. Pile foundation under bridge deck support

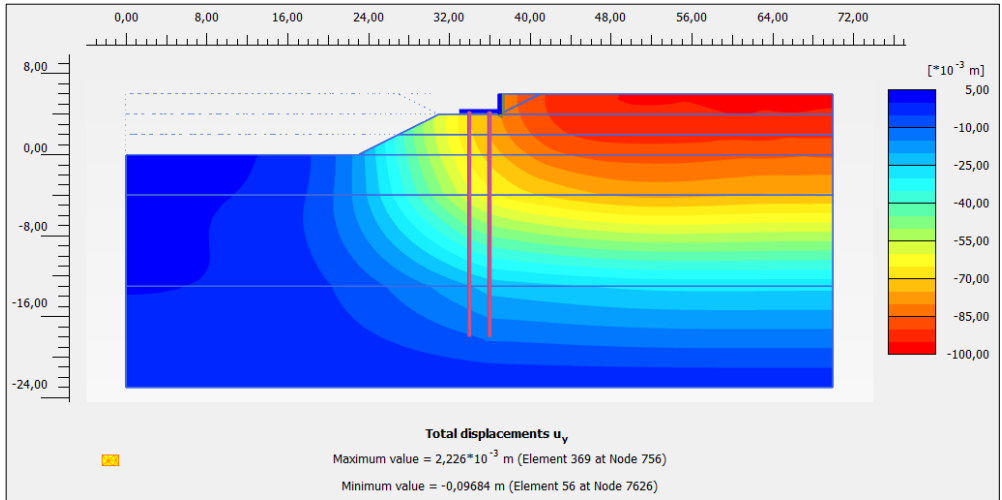


Figure 7. 11. Vertical displacements, u_y , in the Final Consolidation Phase

'Plastic points' (those which have reached a plastic state from the efforts point of view) are shown in Figure 7.12.

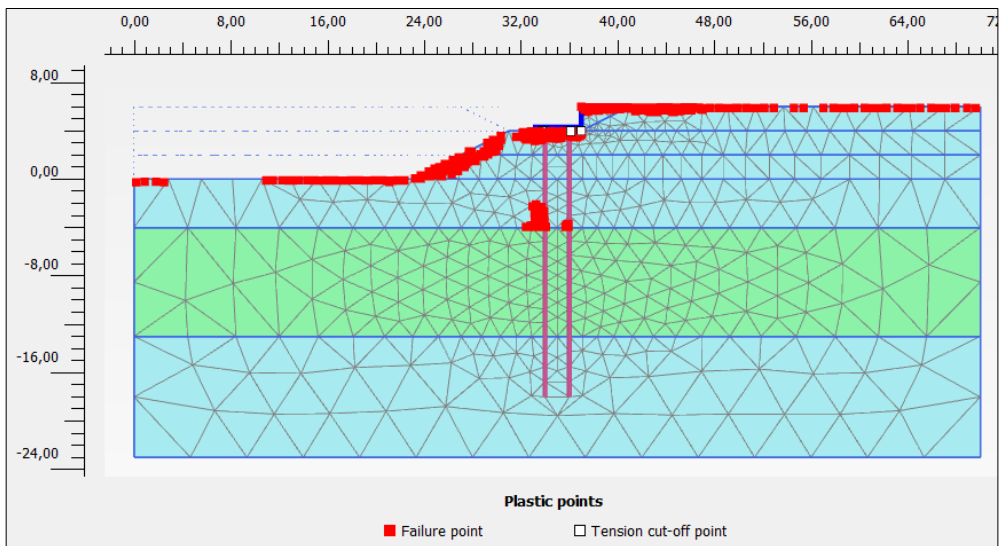


Figure 7. 12. 'Failure points' (red) and 'Tension cut-off points' (white) in the Final Consolidation Phase

Exercise 07. Pile foundation under bridge deck support

Figure 7.13 presents the excess of pore pressure.

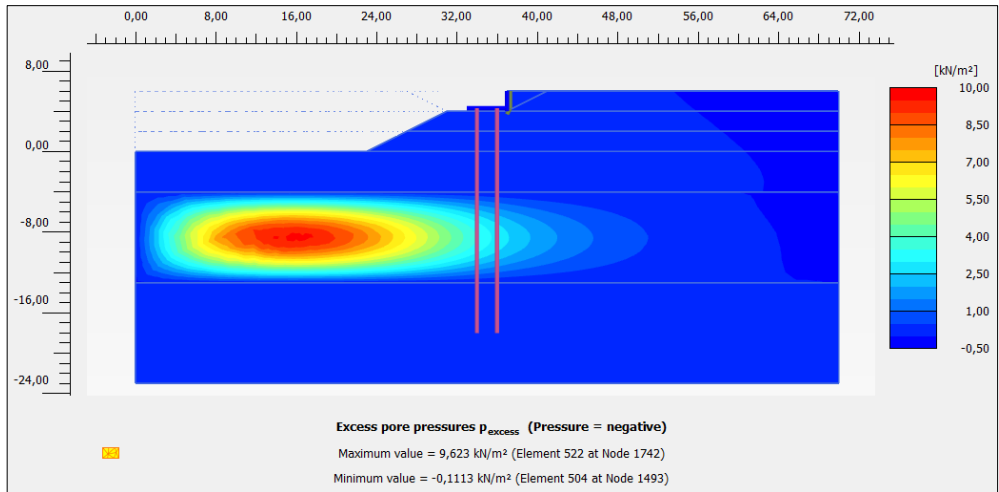


Figure 7. 13. Excess of pore pressure in the Final Consolidation Phase

Bending moments, axial forces and shear forces on the left pile are shown in Figures 7.14, 7.15 and 7.16, respectively.

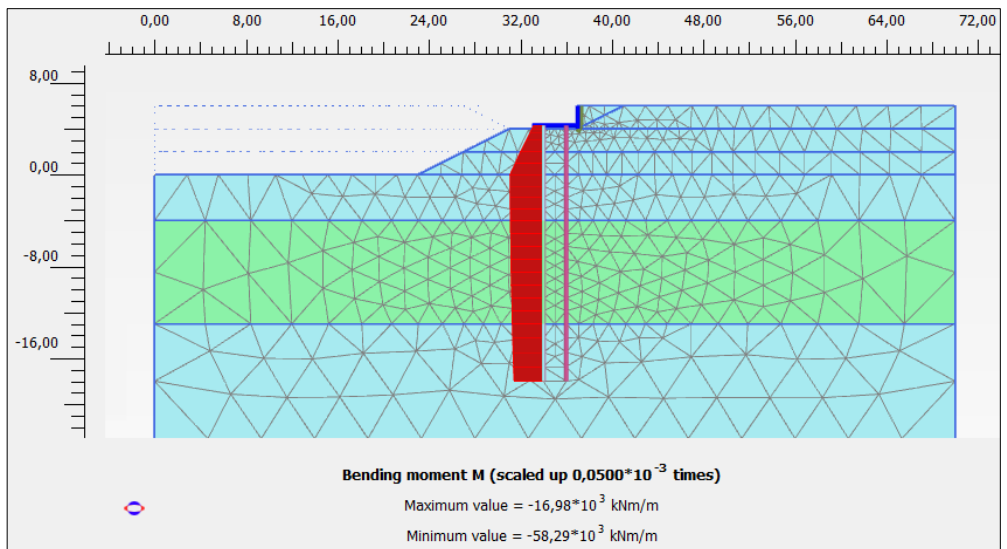


Figure 7. 14. Bending moments on the left pile in the Final Consolidation Phase

Exercise 07. Pile foundation under bridge deck support

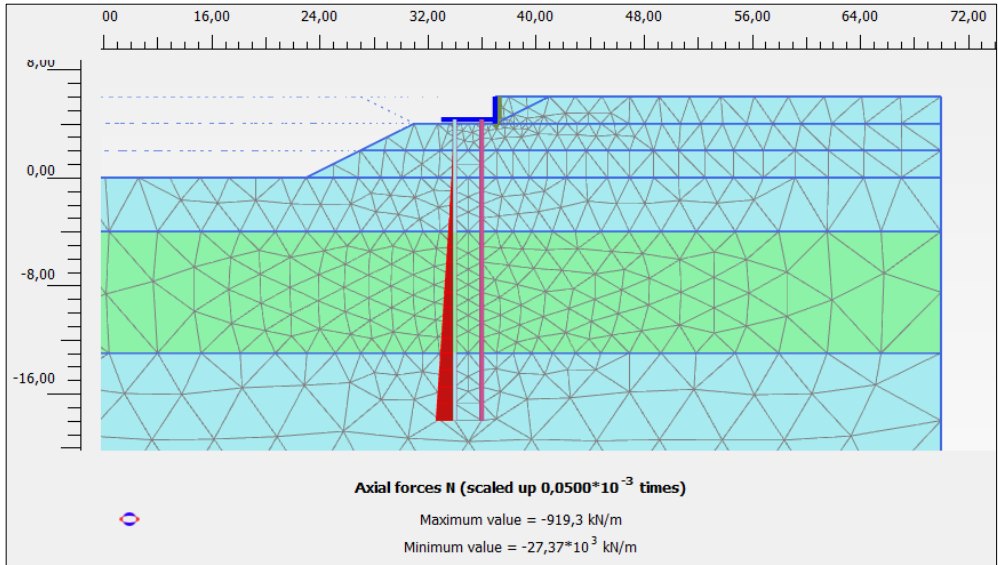


Figure 7. 15. Axial forces on the left pile in the Final Consolidation Phase

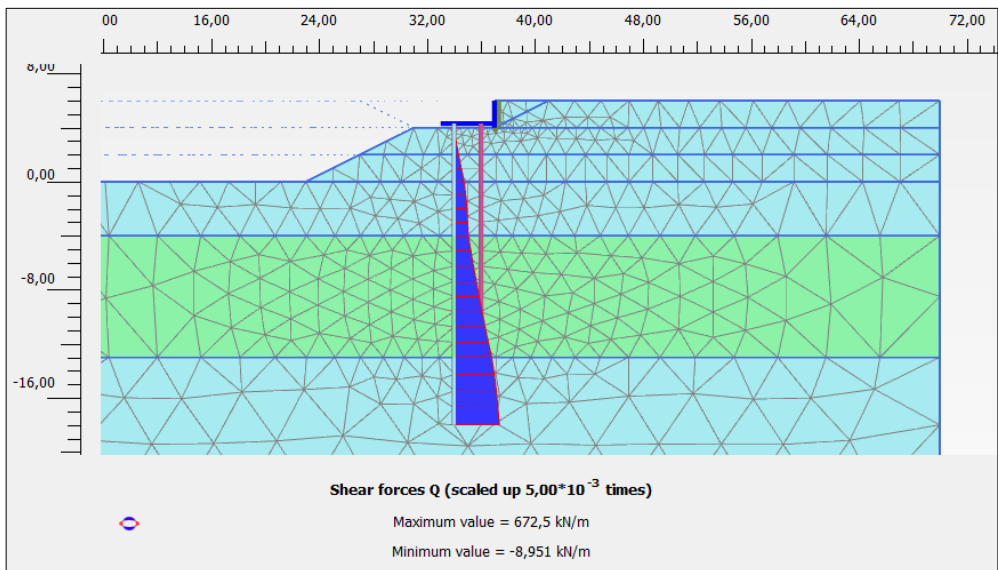


Figure 7. 16. Shear forces on the left pile in the Final Consolidation Phase

Exercise 07. Pile foundation under bridge deck support

Bending moments, axial forces and shear forces on the right pile are shown in Figures 7.17, 7.18 and 7.19.

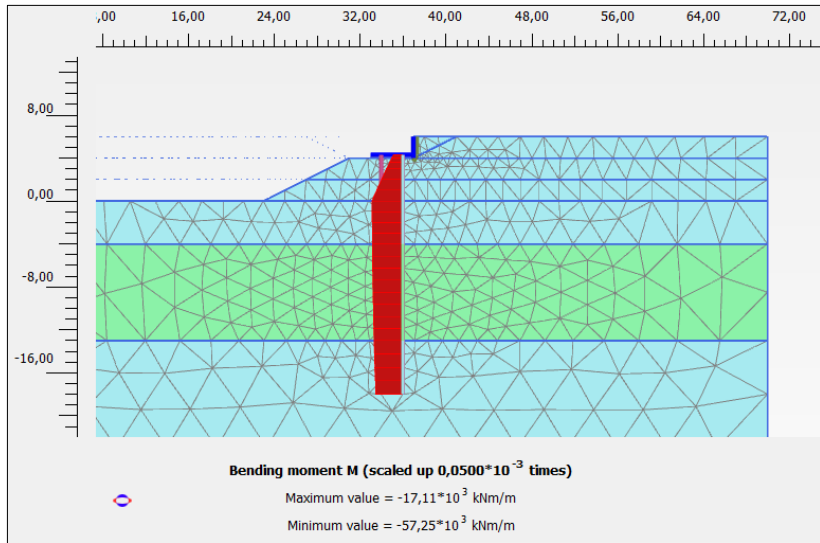


Figure 7. 17. Bending moments on the right pile in the Final Consolidation Phase

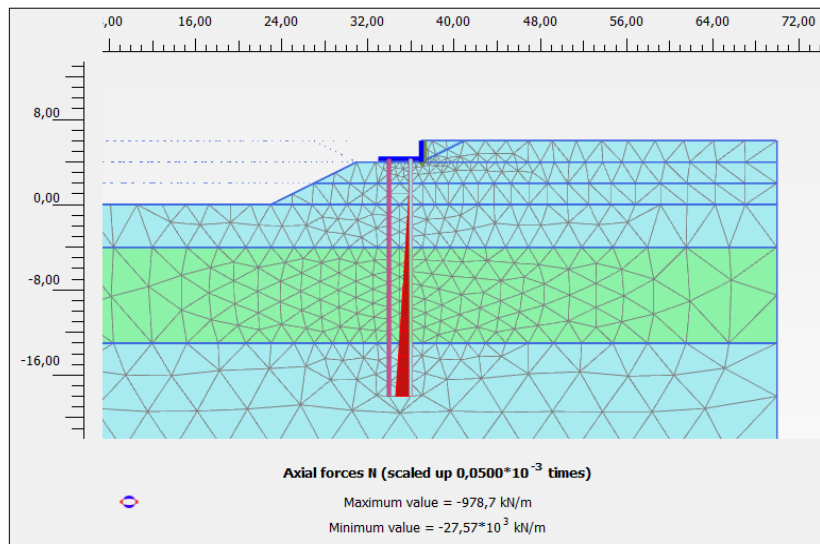


Figure 7. 18. Axial forces on the right pile in the Final Consolidation Phase

Exercise 07. Pile foundation under bridge deck support

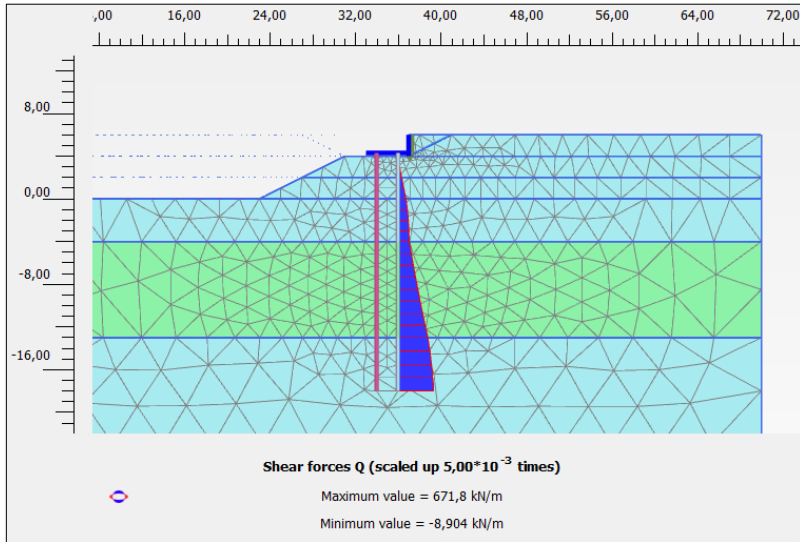


Figure 7. 19. Shear forces on the right pile in the Final Consolidation Phase



Universidad
Politécnica
de Cartagena

edicionesUPCT