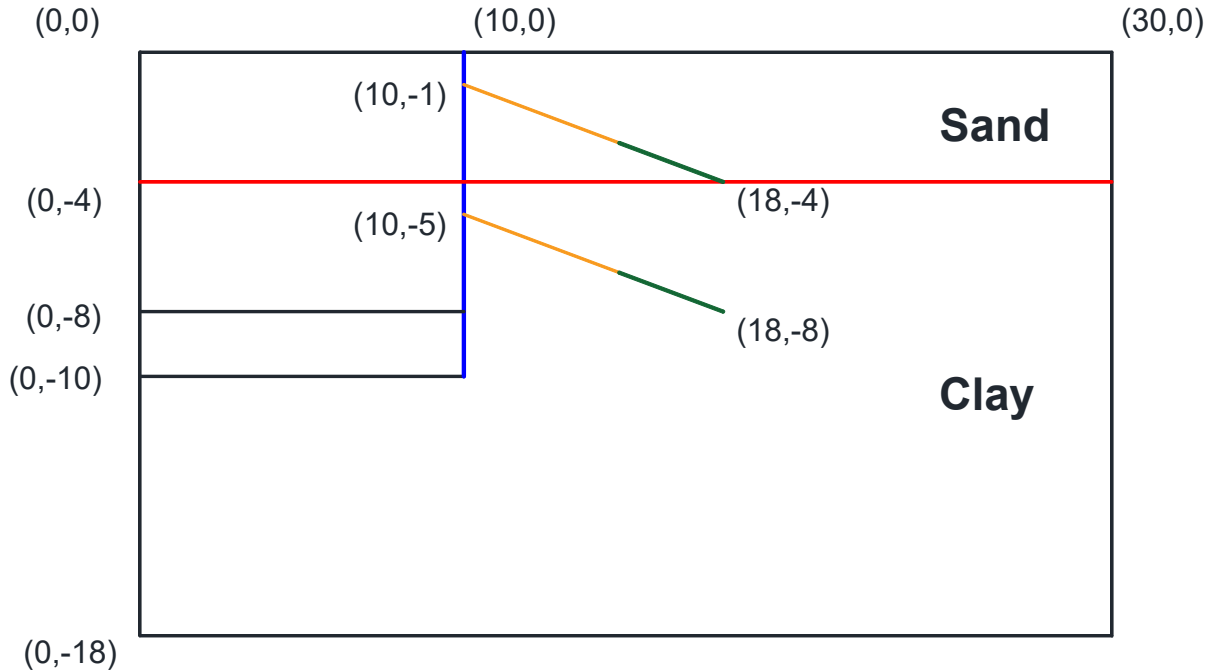


# Anchored Sheet Pile Wall

Created By: [Roozbeh Geraili Mikola, PhD, PE](#)

Email: [adonis4geo@outlook.com](mailto:adonis4geo@outlook.com)

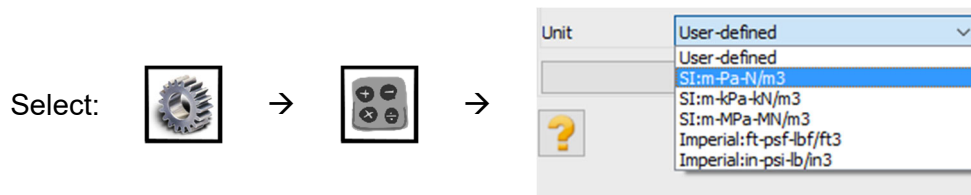
Web Page: [www.geowizard.org](http://www.geowizard.org)



This tutorial will demonstrate how to simulate the construction of an excavation support by a sheet pile wall anchored with tiebacks. This model is taken from **RS2** manual. The problem consists of two soil layers and two stages of excavation as shown in the figure above.

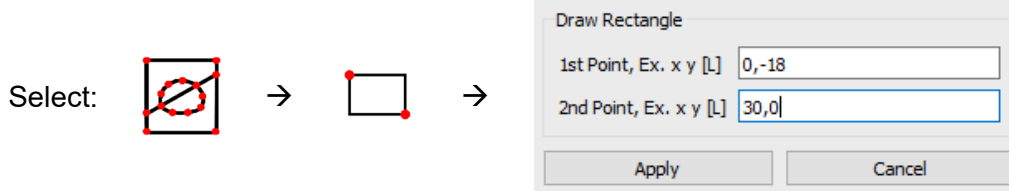
## Select Unit

Since we will be using Metric (Pa) units for this tutorial, let's select "SI: m-Pa-N/m<sup>3</sup>" from **Calculation Setting** under **Setting** button.



## Create Geometry

Select the **Create Geometry** button, then select **Draw Rectangle** and enter the first- and second-points' coordinates (0,-18 and 30,0) and press **Apply**.



Select **Draw Line** to draw material and boundary lines as listed below:

Type	Start Point	End Point
Line	0,-4	30,-4
	0,-10	10,-10
	0,-8	10,-8

In this tutorial, the sheet pile wall will be modeled by beam element sandwiched between two interface elements. It worth noting that, in **ADONIS** interface element, can only be applied on the boundary edges (i.e. edge that shared by only one active solid element). Therefore, to prepare the geometry, a new drawing capability named **Crack** introduced to **ADONIS** (i.e. after V2.75) which makes the elements along it to be separated (e.g. new sets of nodes for each side of the crack will be generated). Please note that each side of crack will be treated as open boundary and it is user's responsibility to add interface element between the edges otherwise model won't be converged during cycling. To add crack, simply click on **Crack** and specify start and end points and click on **Apply** Button, or press **Enter**.

Type	Start Point	End Point
Crack	10,-10	10,0

The geometry for this example has now been entered as shown in Figure 1.

**Note:** In discretization and clean-up process **ADONIS** will remove the unconnected segments (dangling segments). So to generate boundary line that delineates the sheet pile wall a new dummy line (from 0,-10 to 10,-10) is added to the end of line to prevent it from being removed.

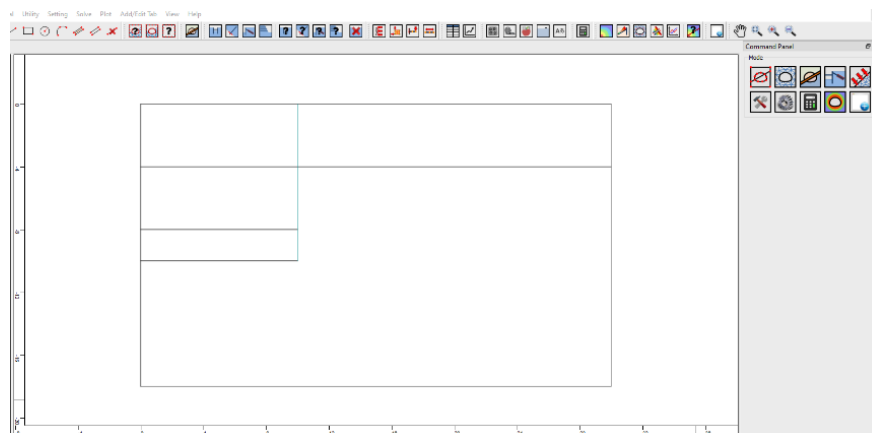
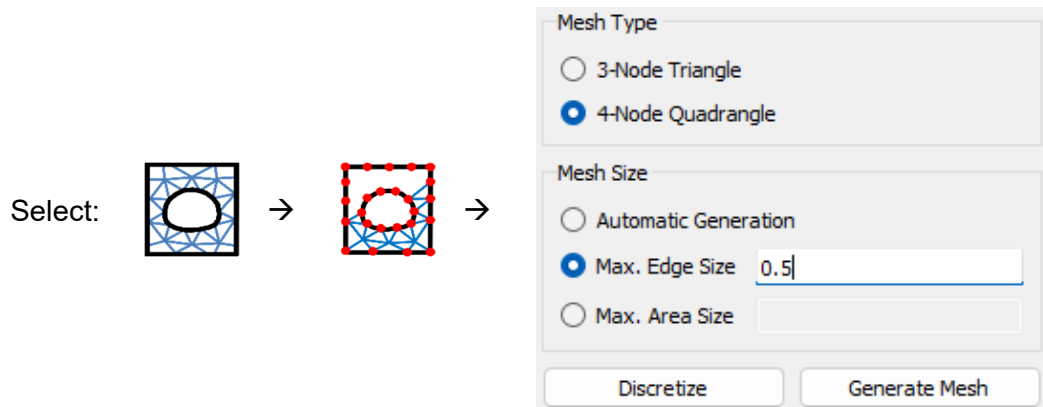


Figure 1- Model geometry.

## Mesh

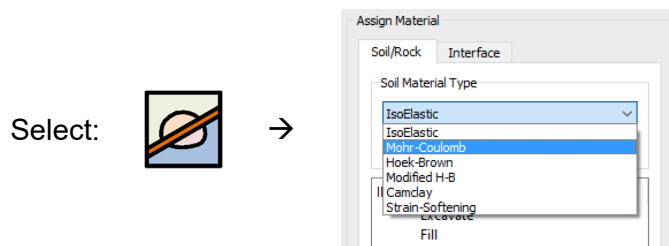
Select the **Discretize/Mesh** button in command panel or toolbar or the Mesh menu, then select **Mesh Property** button then select the element type. For this tutorial 4-Node quadrilateral element is selected. Select the **Discretize/Mesh** button in command panel or toolbar or the Mesh menu, then select **Discretize/Mesh** button. Enter **1.0** as **Max. Edge Size** in order to create finer mesh size. Now press **Discretize** then press **Generate Mesh** buttons to discretize the boundaries and generate FE mesh.



The finite element mesh is generated.

## Assign Soil Properties

To create a new material, select the **Assign Material/Excavate** button from command panel, toolbar or menu, then select **Soil/Rock** tab. From drop down select **Mohr-Coulomb** criteria then click on **Add Soil/Rock Material** button.



Now enter the following rock mass properties for both Sand and Clay soils as shown in Figures 2.

Material ID	Material Name	Density (kg/m <sup>3</sup> )	Young Modulus (Pa)	Poisson's Ratio	Cohesion (Pa)	Friction Angle (deg.)
1	Sand	1735	2.0x10 <sup>7</sup>	0.25	1.0x10 <sup>3</sup>	35
2	Clay	2040	1.5x10 <sup>7</sup>	0.25	1.0x10 <sup>4</sup>	25

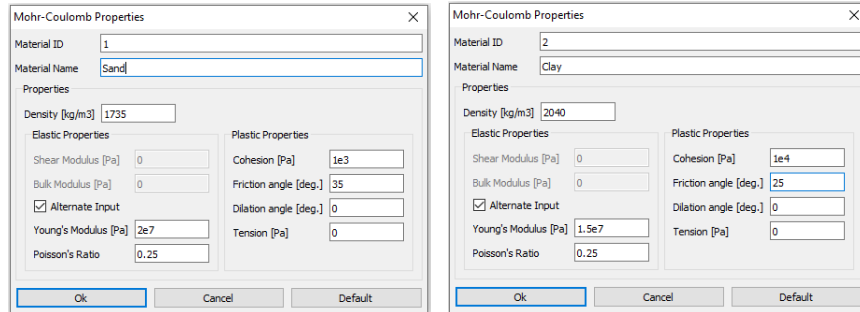


Figure 2- Create Mohr-Coulomb properties for each material.

Now we need to assign the created material to the proper region. To do that, select the Sand material (i.e. created material) from the material list and then use mouse and click on the Sand regions in the main window and click on **Apply** Button, or press **Enter** (Figure 3). Repeat the sequence to create another material for Clay soil (Figure 4). Figure 5 shows the model configuration after assigning the materials (both Sand and Clay soils).

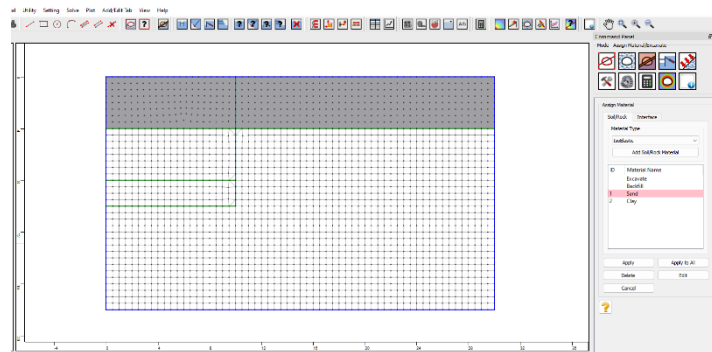


Figure 3- Assign sand soil material.

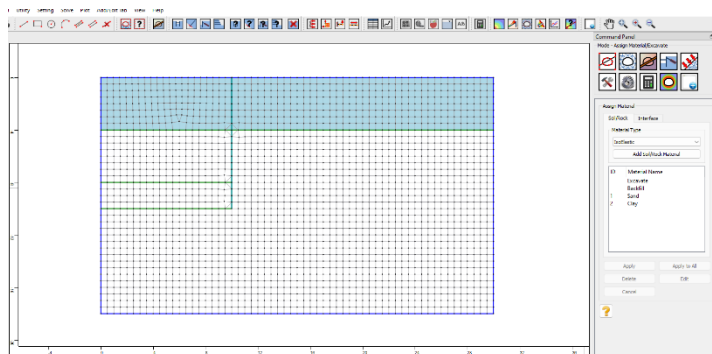


Figure 4- Assign clay soil material.

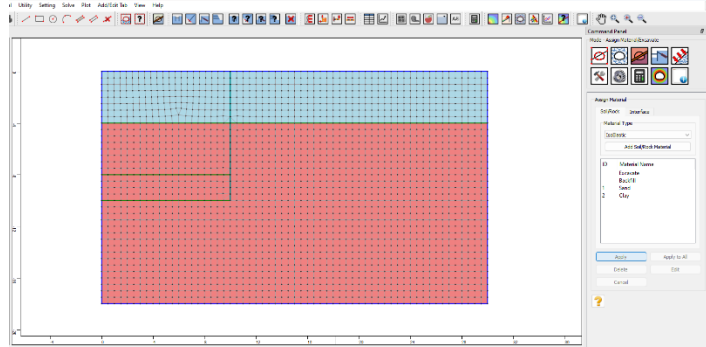
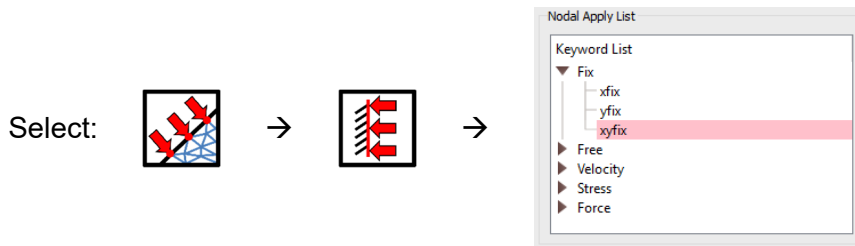


Figure 5- Model configuration after assigning the materials.

## Boundary Condition

In order to fix (i.e. zero displacement) the external boundary, select **Initial Condition** then select **Apply Boundary Condition** then from the "Keyword List" click on **xyfix** under **Fix** keyword. Then Use the mouse to select the nodes located on bottom of the model. When finished, click on **Apply** Button, or press **Enter**.



Then select **xyfix** and use the mouse to select the nodes located on left and right sides of the model. The boundary condition has been applied as shown in Figure 6.

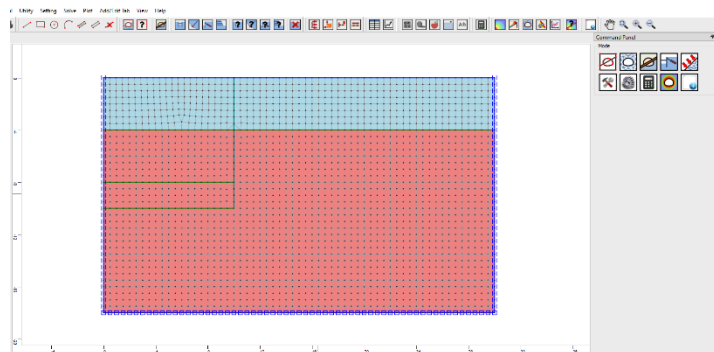


Figure 6- Model configuration after applying boundary condition.

## Install Sheet Pile Wall

As it's mentioned above, in this tutorial, sheet pile wall is modeled by beam structural elements. The beam element will be sandwiched between two interface elements. Note that beam element can be attached to the boundary edge only. So, to install the sheet pile wall at the right place (i.e. along 10,-10->10,0 crack line) you need to temporarily excavate (i.e. hide) the deep excavation part and install the sheet pile wall then backfill it later on. To do that, select the **Assign**

**Material/Excavate** button from command panel, toolbar or menu, then select **Excavate** from the material name list. Please make sure that **Reset Element after Excavation** checkbox is unchecked. This way you can make sure that initialized stress in the elements won't be zeroed out after backfill. Then use mouse and select three regions in the excavation zone and press **Apply**.

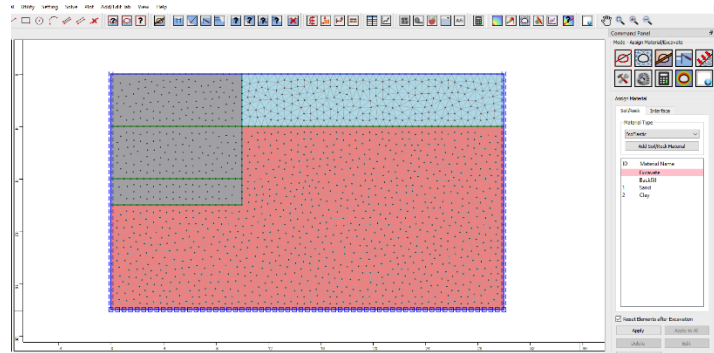
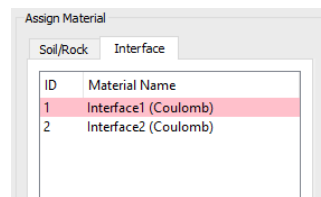


Figure 7- Selected zone to be hidden (temporarily excavated).

Then select the **Structures** button from command panel, toolbar or menu, then select **Beam** element. Select attach to **Edge with Interface** and after that **Range** from selection mode. Change the interface type to **Both Sides** from the dropdown. Now use mouse to select the range ( $xlim=[9.9,10.1]$  and  $ylim=[-10.1,0.1]$ ) that sheet pile wall should be installed and press **Apply** (Figure 8). Then click on **Beam Element Properties** and assign the sheet pile properties as shown in Figure 9. When user selects the “both sided interface”, ADONIS automatically sandwiches the beam elements between two interfaces. The First interface ID is always between beam element and visible solid element and second interface ID will be created between beam element and invisible solid element. Now you need to modify the interface properties. For this select the **Assign Material/Excavate** button from command panel, toolbar or menu, then select **Interface** tab. In the list you should see two interface elements. Select **Interface1 (Coulomb)** and press **Edit** button. Then adjust the following properties:

Material ID	Material Name	jkn (Pa/m)	jks (Pa/m)	Cohesion (Pa)	Friction Angle (deg.)
1	Interface1	$1.0 \times 10^8$	$1.0 \times 10^7$	0	20
2	Interface2	$1.0 \times 10^8$	$1.0 \times 10^7$	0	20

Select:



To backfill the temporarily excavated part, select the **Assign Material/Excavate** button from command panel, toolbar or menu, then select **Fill** from the material name list. Then use mouse and select three regions that were excavated previously and press **Apply** (Figures 10-11).

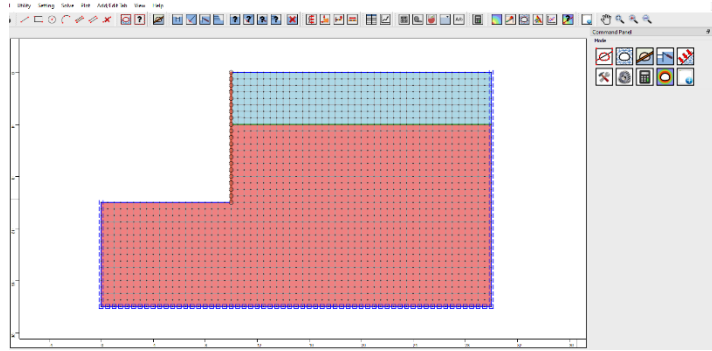


Figure 8- Install beam elements sandwiched between two interfaces.

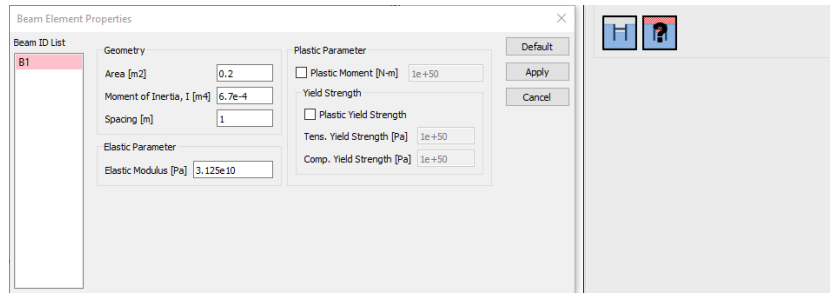


Figure 9- Assign sheet pile wall properties.

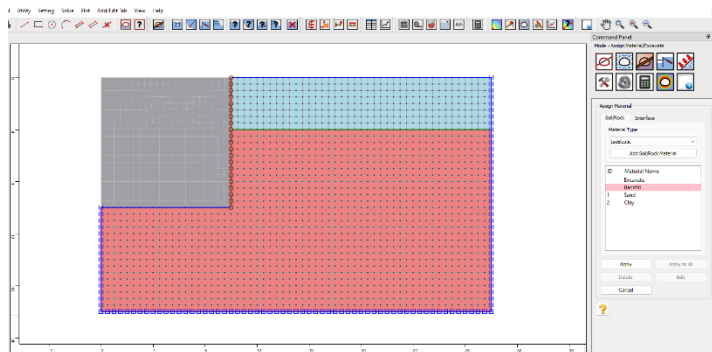


Figure 10- Backfill the excavated zone.

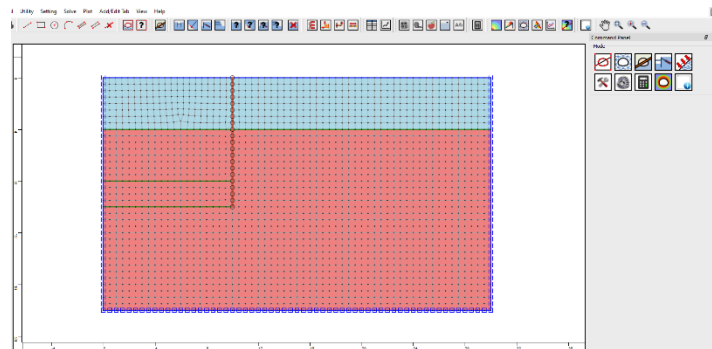


Figure 11- Model configuration after installing the sheet pile.

## Initialize the Stress Condition

In order to initialize the vertical and horizontal stress in the model select **Initial Condition** then select **Apply Initial Condition** then from the "Keyword List" click on **syy** under **Element Stress**.



In addition to constant stress, the **Initial** command will also take the keywords **xvar** and **yvar** which allow the initial stress to vary linearly over the specified range. The stresses vary linearly with distance from the global coordinate origin of x=0, y=0:

$$\text{value\_modified} = \text{value} + \text{vx} \times \text{x} + \text{vy} \times \text{y}$$

or

$$\text{sxx} = \text{sxx0} + \text{vx\_xx} \times \text{x} + \text{vy\_xx} \times \text{y}$$

$$\text{syy} = \text{syy0} + \text{vx\_yy} \times \text{x} + \text{vy\_yy} \times \text{y}$$

$$\text{szz} = \text{szz0} + \text{vx\_zz} \times \text{x} + \text{vy\_zz} \times \text{y}$$

where value is the stress component at origin and (x,y) is position of the gauss point in the global coordination. For example, the vertical stress at the top and bottom of the Sand layer should be 0 and  $\gamma_{\text{sand}} \times h$  (e.g.  $1735 \times 9.8 \times 4.0 = 68012$  Pa). To specify such stress gradient, the **initial stress** and **yvar** values should be 0 and 17003 (i.e.  $\text{value\_modified} = 0 + 0 \times \text{x} + \gamma_{\text{sand}} \times \text{y}$ ) as shown in Figure 12. The same procedure will be applied for the initialization of the **syy**, **sxx** and **szz** for both Clay and Sand layers. Table 1 summarizes the values required for each initialization command. K0=0.5 is assumed for this tutorial thus the **yvar** value for **sxx** and **szz** are equal to half the vertical stress gradients (i.e. **yvar**). Please make sure that proper range is used for each layer before click on **Apply** (i.e. ylim=[-4,0] for Sand layer and ylim=[-18,-4] for Clay layer).

**Note:** Compression stresses have a negative sign, in accordance with the general sign convention for internal stresses in **ADONIS**.

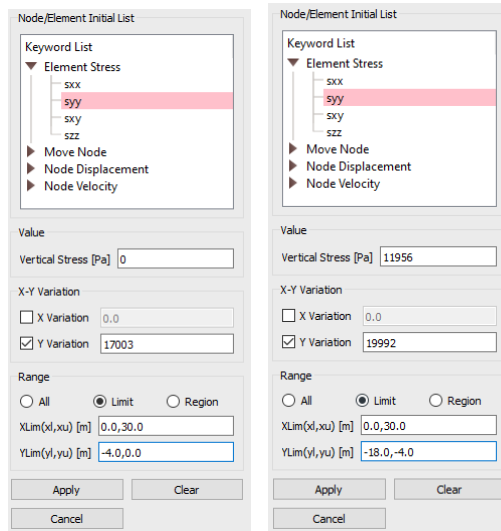


Figure 12- Stress initialization dialog box, initializing vertical stress for Sand and Clay layers.

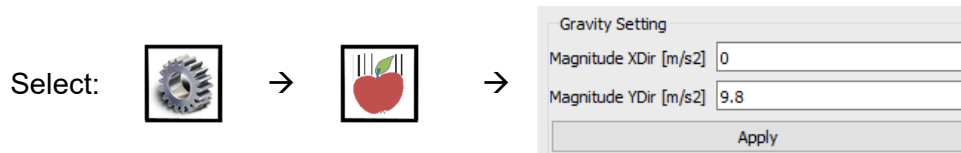
Table 1- List of parameters for stress initialization.

Stress Type	Initial Value (Pa)	Y-Variation Value (Pa)	X-Limit (m)	Y-Limit (m)
<b>syy</b>	0.0	17003.0	0,30	-4,0
<b>syy</b>	11956.0	19992.0	0,30	-18,-4

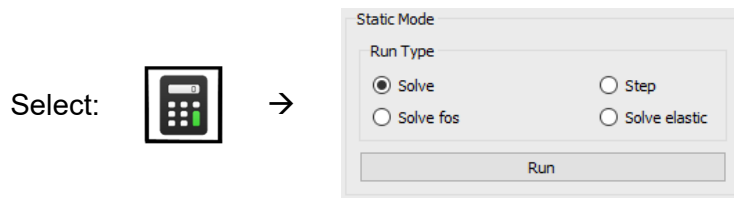


<b>sxx</b>	0.0	8501.5	0,30	-4,0
<b>sxx</b>	5978.0	9996.0	0,30	-18,-4
<b>szz</b>	0.0	8501.5	0,30	-4,0
<b>szz</b>	5978.0	9996.0	0,30	-18,-4

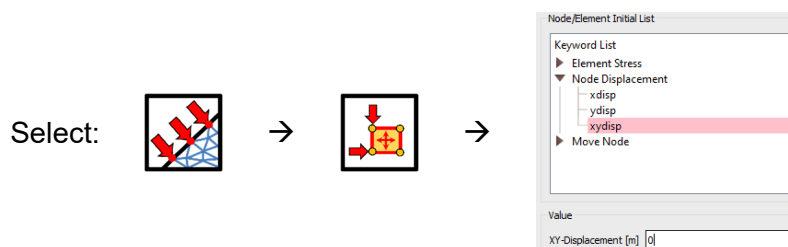
Before solving the mode to equilibrium let's specify the gravitational gradient value. For that, select **Setting** then **Gravity Setting** and set the magnitude of gravity in y-direction to 9.8 m/s<sup>2</sup>. Note: Downward gravity is positive.



Now click on **Solve the Model** button from toolbar, menu or command panel then select **Solve** radio button from the **Run Type** and click on **Run** button. The **ADONIS** Compute engine will proceed in running the analysis. Model should reach equilibrium after several steps.



After reaching to equilibrium let's zero out the displacement in the model to get ready for the next stages. For this, select **Initial Condition** then select **Apply Initial Condition** then from the "Keyword List" click on **xydisp** (means both x- and y- displacements) under **Nodal Displacement** keyword and enter 0.0 in XY-Displacement value box then press **Apply**. This way you can reset both x- and y- displacements to zero.



## Perform the Excavation Sequence

Now let's excavate the top part of excavation (i.e. Sand) and install the tieback element. Select the **Assign Material/Excavate** button from command panel, toolbar or menu, then select **Excavate** from the material name list. Then use mouse and select the top region (i.e. Sand) in the excavation zone and press **Apply**. Then select the **Structures** button from command panel, toolbar or menu, then select **Tieback** element. Make sure to select **From Struc. Node at Point** from dropdown for the start point (to make sure the start point is attached to existing sheet pile wall structural node) and then enter 10,-1 and 18,-4 for first and second end points (Figure 13). Make sure the pretension force and grouted length ratio are set to 2.0x10<sup>4</sup> N and 0.4 (i.e. 40% grouted) respectively then press Apply. Next click on "**Set Tieback Property**" button in Command Panel and assign the tieback properties as shown in Figure 14.

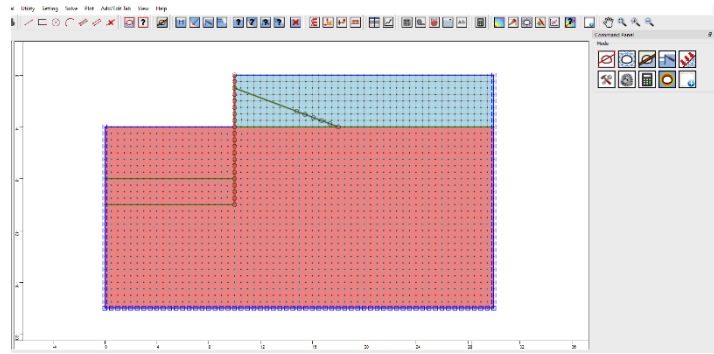


Figure 13- Excavate the top region and install the first tieback.

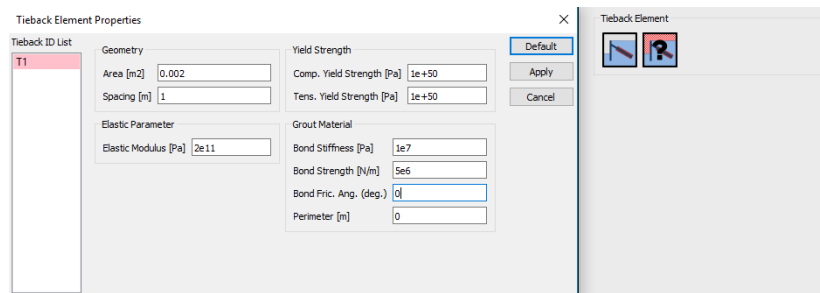


Figure 14- Assign the first tieback properties.

Then select **Solve the Model** button from toolbar, menu or command panel and solve the model until reaching to equilibrium (i.e. select **Solve** and press **Run** button). Now repeat the same sequence and excavate the second part of excavation zone (i.e. Clay) and install the second tieback and solve to equilibrium as shown in the Figures 15 and 16. The properties for the second tieback is similar to the first one except the end points that starts from 10,-5 to 18,-8. Also make sure you choose different ID for the second tieback (e.g. ID = 2).

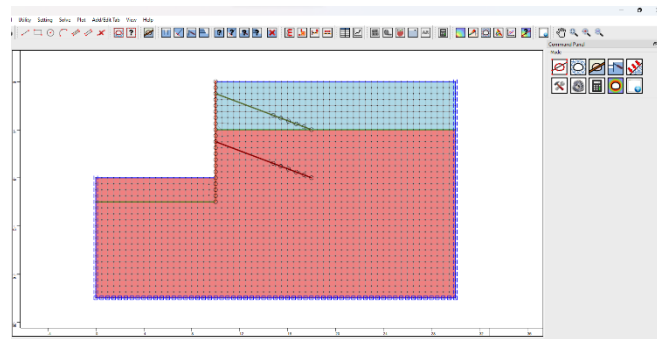


Figure 15- Excavate the bottom region and install the second tieback.

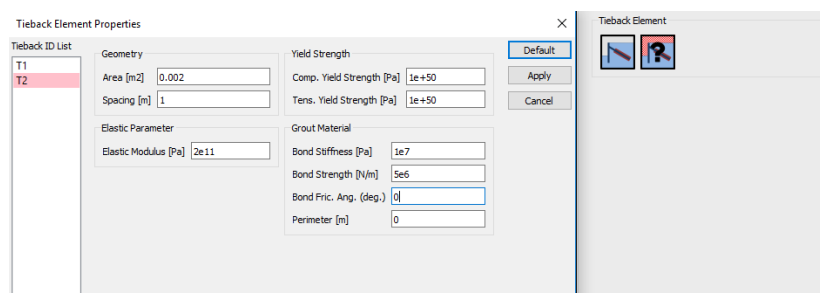
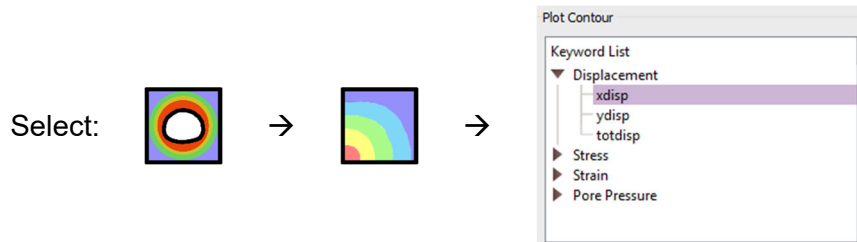


Figure 16- Assign the first tieback properties.

## Post Processing

After reaching to equilibrium in the final stage, the tools under the **Plot Geometry/Results** in the command panel can be used to plot the results. An extensive set of options is available for plotting the results of the finite element analysis generated by **ADONIS**. These include contouring of the nodal and gauss point variables, strain and stress values, forces in the structural elements and highlighting of plastic zones. Lines or color fills can be selected for all contour plots. To plot horizontal displacement contour, click on **Plot Geometry/Results** button then click on **Plot Contour**, under **Displacement** select **xdisp** and press **Apply**.



Now in order to plot the axial force in the tiebacks click on **Plot Structural Data** and select **axialforce** under **Tieback** and press **Apply**.

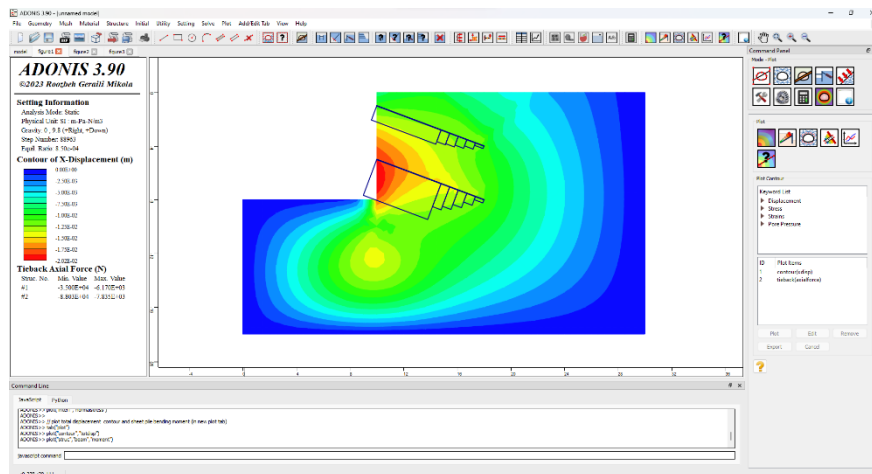
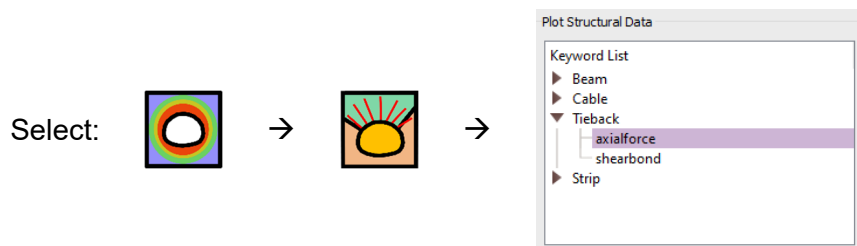


Figure 17- Horizontal displacement contour and tieback axial force plots.

The same process can be followed to plot the vertical displacement contour, interface normal stress as well as sheet pile moment force as shown in Figures 17-19.

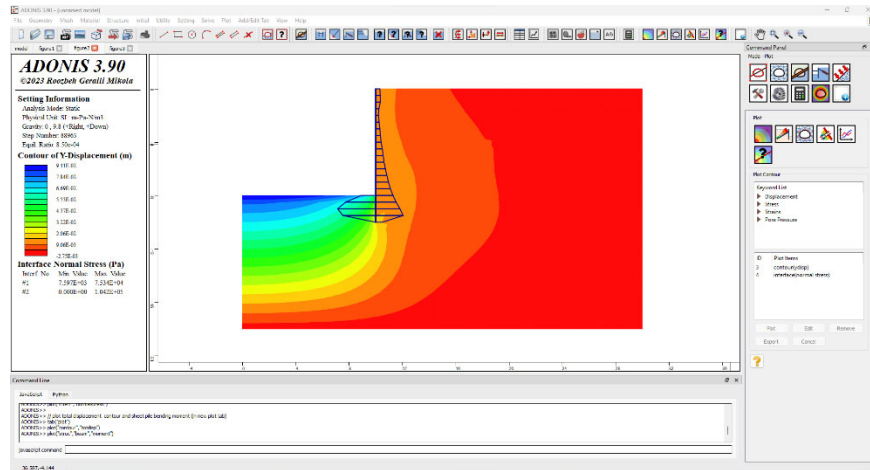


Figure 18- Vertical displacement contour and interfaces normal stress plots.

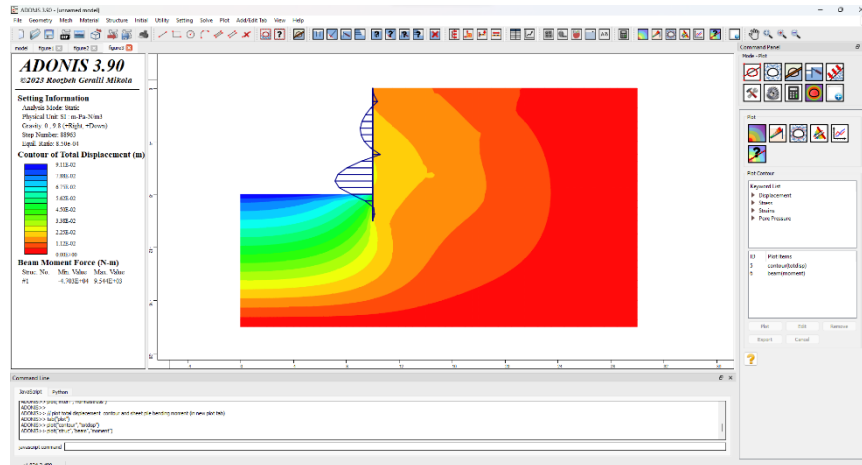


Figure 19- Total displacement contour and beam element bending moment plots.

## Model's Script

The commands for this tutorial are listed below:

```
// create new/fresh model
newmodel()

// set unit
set("unit", "stress-pa")

// create geometry
rect("startPoint", 0, -18, "endPoint", 30, 0)
line("startPoint", 0, -4, "endPoint", 30, -4)
line("startPoint", 0, -10, "endPoint", 10, -10)
line("startPoint", 0, -8, "endPoint", 10, -8)
crack("startPoint", 10, -10, "endPoint", 10, 0)

// discretize/mesh
discretize("maxedge", 0.5)
gmsh("maxedge", 0.5, "elementype", "Q4")

// create new materials
material("create", "Mohr-Coulomb", "matid", 1, "matname", "Sand", "density", 1735, "shear", 8e+06, "bulk", 1.33333e+07, "coh", 1000, "fric", 35, "dil", 0, "tens", 0)
material("create", "Mohr-Coulomb", "matid", 2, "matname", "Clay", "density", 2041, "shear", 6e+06, "bulk", 1e+07, "coh", 10000, "fric", 25, "dil", 0, "tens", 0)
```

```

// assign materials
material("assign","matid",1,"region",28,-2)
material("assign","matid",1,"region",9,-1.5)
material("assign","matid",2,"region",7,-7)
material("assign","matid",2,"region",12,-7.5)
material("assign","matid",2,"region",9,-9)

// assign boundary condition
applybc("xfix","xlim",-0.1,0.1,"ylim",-18.1,0.1)
applybc("xfix","xlim",29.9,30.1,"ylim",-18.1,0.1)
applybc("xyfix","xlim",-0.1,30.1,"ylim",-18.1,-17.9)

// hide the excavation zone to be able to install the sheet pile
excavate("region",2.99756,-1.85243,"reset","off")
excavate("region",4.74894,-6.33193,"reset","off")
excavate("region",5.89408,-9.39685,"reset","off")

// install the sheet pile wall
structure("drawliner","beamid",1,"iftype","bothSides","ifid1",1,"ifid2",2,"xlim",9.9,10.1,"ylim",-10.1,0.1)

// assign/modify the beam and interface elements properties
// use near zero modulus for structural element and glue the interfaces (to create initial condition)
structure("material","beamid",1,"area",0.2,"I",6.7e-4,"ymod",3.125e10)
imaterial("assign","Coulomb","ifid",1,"matname","Interface1","jkn",1e+08,"jks",1e+07,"friction",20)
imaterial("assign","Coulomb","ifid",2,"matname","Interface2","jkn",1e+08,"jks",1e+07,"friction",20)

// backfill the temporarily excavated zones
backfill("region",2.99756,-1.85243,"reset","off")
backfill("region",4.74894,-6.33193,"reset","off")
backfill("region",5.89408,-9.39685,"reset","off")

// initialize stress condition (k0=0.5)
initial("syy",0,"yvar",17003,"xlim",0,30,"ylim",-4,0)
initial("syy",11956,"yvar",19992,"xlim",0,30,"ylim",-18,-4)
initial("sxx",0,"yvar",8501.5,"xlim",0,30,"ylim",-4,0)
initial("sxx",5978,"yvar",9996,"xlim",0,30,"ylim",-18,-4)
initial("szz",0,"yvar",8501.5,"xlim",0,30,"ylim",-4,0)
initial("szz",5978,"yvar",9996,"xlim",0,30,"ylim",-18,-4)

// apply gravity force
set("gravity",0,9.8)

// solve to initialize
solve()

// reset the displacement to zero
initial("xydisp",0)

// excavate the top region (sand)
excavate("region",6.5,-2.5)

// install the first tieback (start node will be attached to existing sheet pile node)
structure("drawtieback","tieid",1,"fromstrucnodeatpoint",10.000,-1.000,"topoint",18,-4,"pretens",2e4,"grouted",0.4,"segnum",5)
structure("material","tieid",1,"area",0.002,"ymod",2e+11,"kbond",1e+07,"sbond",5e+06)

// solve to equilibrium
solve()

// excavate the bottom region (clay)
excavate("region",6.5,-6.0)

// install the second tieback (start node will be attached to existing sheet pile node)
structure("drawtieback","tieid",2,"fromstrucnodeatpoint",10.000,-5.000,"topoint",18,-8,"pretens",2e4,"grouted",0.4,"segnum",5)
structure("material","tieid",2,"area",0.002,"ymod",2e+11,"kbond",1e+07,"sbond",5e+06)

// solve to equilibrium
solve()

// plot horizontal displacement contour and tieback axial forces
plot("contour","xdisp")
plot("struc","tieback","axialforce")

```

```
// plot vertical displacement contour and interface normal stresses (in new plot tab)
tab("plot")
plot("contour","ydisp")
plot("interf","normalstress")

// plot total displacement contour and sheet pile bending moment (in new plot tab)
tab("plot")
plot("contour","totdisp")
plot("struc","beam","moment")
```