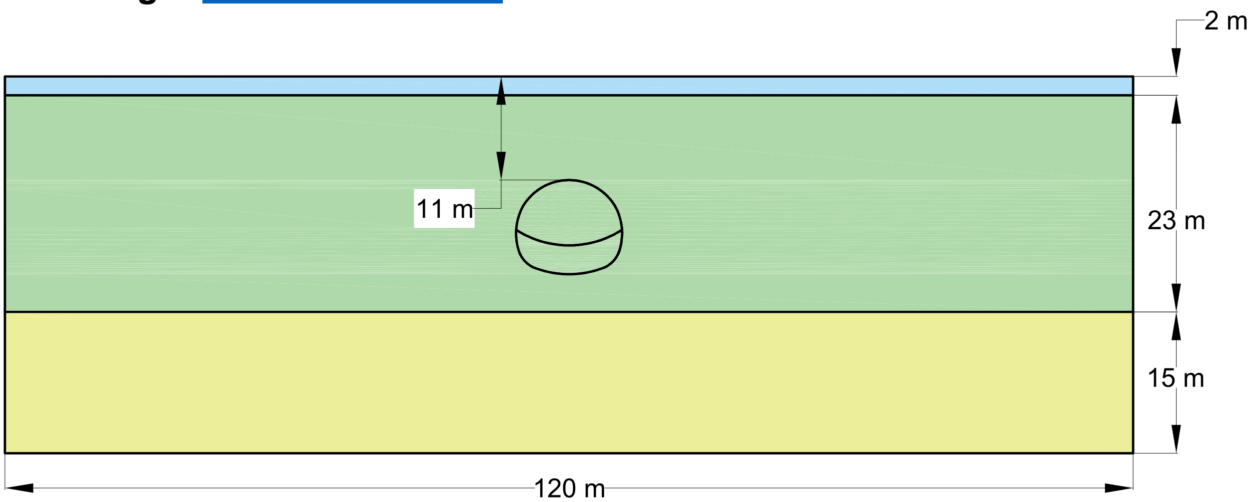
Sequential Tunnel Excavation

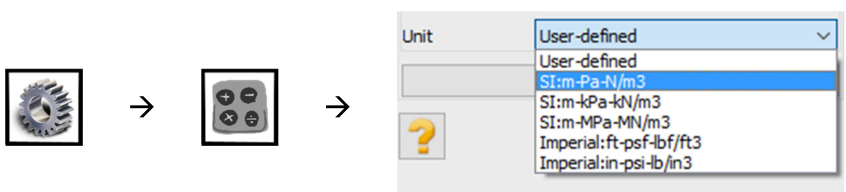
Created By: [Roozbeh Geraili Mikola, PhD, PE](#)
Email: adonis4geo@outlook.com
Web Page: www.geowizard.org



This tutorial illustrates the use of ***ADONIS*** for the analysis of the construction of a SEM/NATM tunnel. The SEM/NATM is a technique in which ground exposed by excavation is stabilized with shotcrete to form a temporary lining.

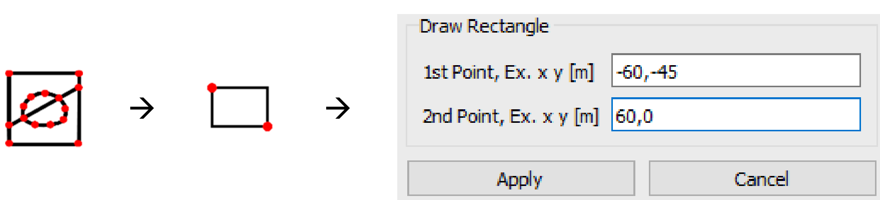
Select Unit

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

Select: 

Create Geometry

To create the model geometry select the ***Create Geometry*** button, then select ***Draw Rectangle*** and enter the first- and second-points' coordinates (-60,-45 and 60,0) and press ***Apply***.

Select: 

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

Type	Start Point	End Point
Line	-60,-2	60,-2
	-60,-25	60,-25

Select ***Draw Arc*** to draw material lines as listed below:

Type	Start Point	Middle Point	End Point	Num. of Segments
Arc	3.88,-20.23	4.69,-19.62	5.25,-18.77	6
	4.95,-13.87	0.00,-11.00	-4.95,-13.87	25
	-5.25,-18.77	-4.69,-19.62	-3.88,-20.23	6
	-3.88,-20.23	0.00,-21.00	3.88,-20.23	20
	5.25,-18.77	5.57,-17.55	5.63,-16.29	6
	5.63,-16.29	5.42,-15.05	4.95,-13.87	6
	-5.25,-18.77	-5.57,-17.55	-5.63,-16.29	6
	-5.63,-16.29	-5.42,-15.05	-4.95,-13.87	6
	-5.63,-16.29	0.00,-17.94	5.63,-16.29	25

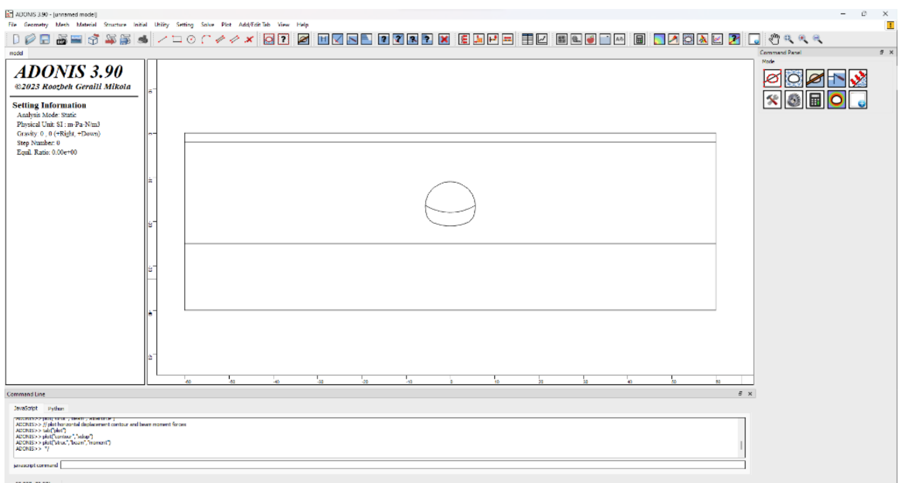
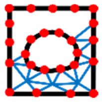
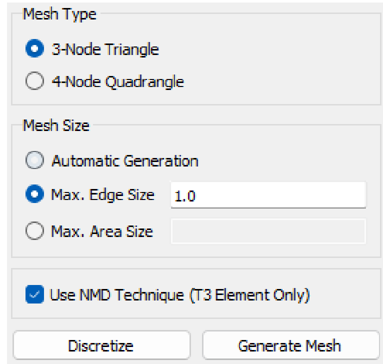


Figure 1- Model geometry.

Mesh

Now let's generate the finite element (FE) mesh. In ***ADONIS***, meshing is a simple two-step process. First you must discretize the boundaries, and then the Mesh can be generated. You can also configure various Mesh Setup parameters before generating the 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. Currently only two types of FE mesh is available 1) 3-Node triangular and 2) 4-Node quadrilateral elements. For this tutorial 3-Node triangular element is selected. Then 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 meh size. Now press ***Discretize*** then press ***Generate Mesh*** buttons to discretize the boundaries and generate FE mesh.

Select:  →  → 

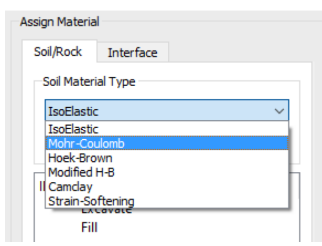
The finite element mesh is generated. ADONIS adopts "Nodal Mixed Discretization (NMD) methodology to overcome this problem. NMD is introduced by Detourney & Dzik (2006) as an improvement of Mixed Discretization (MD) technique.

Notes:

1. NMD is only available for T3 element (not Q4).
2. NMD is activated by default.

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.

Select:  → 

Now enter the following soil and rock properties for each layer as shown in Figures 2.

Table 1- Material properties.

Material ID	Material Name	Density (kg/m^3)	Young Modulus (Pa)	Poisson's Ratio	Cohesion (Pa)	Friction Angle (deg.)	Tensile Strength (Pa)	K0
1	Fill	1950	1.0x10^7	0.30	1.0x10^3	29	0.0	0.5
2	Soil	2200	1.0x10^8	0.25	3.9x10^4	35	0.0	0.6
3	Rock	2400	5.0x10^8	0.25	1.0x10^5	40	1.0x10^4	0.65

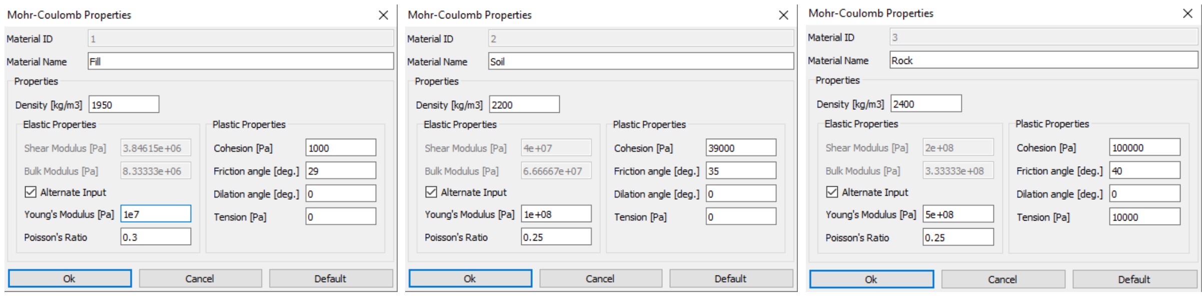


Figure 2- Create Mohr-Coulomb properties for each material.

To assign the created material to the proper region, select the Fill material (i.e. created material) from the material list and then use mouse and click on top layer in the main window and click on ***Apply*** Button, or press ***Enter*** (Figure 2). Repeat the sequence to create material for the rest of layers. Figure 3 shows the model configuration after assigning the materials.

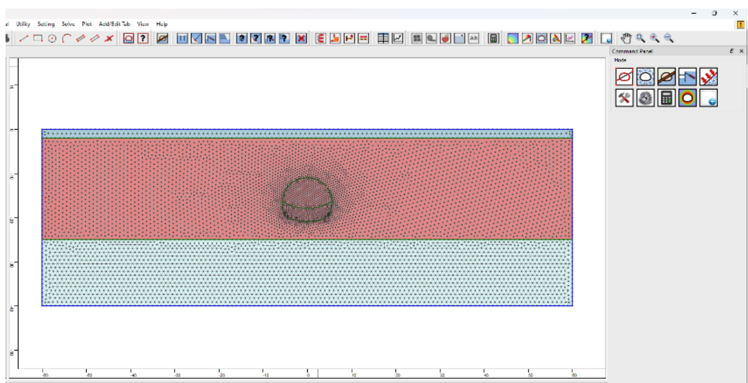
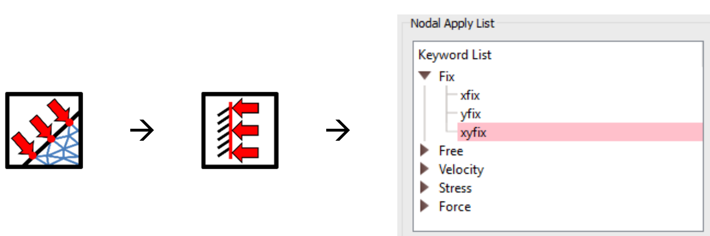


Figure 3- Model configuration after assigning the materials.

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 ***<u>xyfix</u>*** 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***.

Select:



Then select ***<u>xfix</u>*** 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 4.

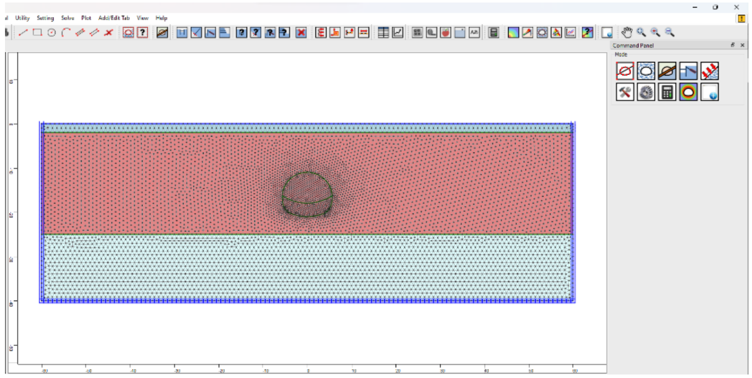
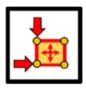


Figure 4- Model configuration after applying boundary condition.

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***.

Select:  → 

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:

$$value\_modified = value + vx \times x + vy \times y$$

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 Fill layer should be 0 and $\gamma_{Fill} \times h$ (e.g. 1950×9.81×2.0=38259 Pa). To specify such stress gradient, the ***initial stress*** and ***yvar*** values should be 0 and 17003 (i.e. $value\_modified = 0 + 0 \times x + \gamma_{Fill} \times y$) as shown in Figure 5. The same procedure will be applied for the initialization of the ***syy***, ***sxx*** and ***szz*** for both Clay and Sand layers. Table 2 summarizes the values required for each initialization command. The ***yvar*** value for ***sxx*** and ***szz*** should be applied based on each layer's K_0 value as shown in Table 2. Please make sure that proper range is used for each layer before click on ***Apply***.

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

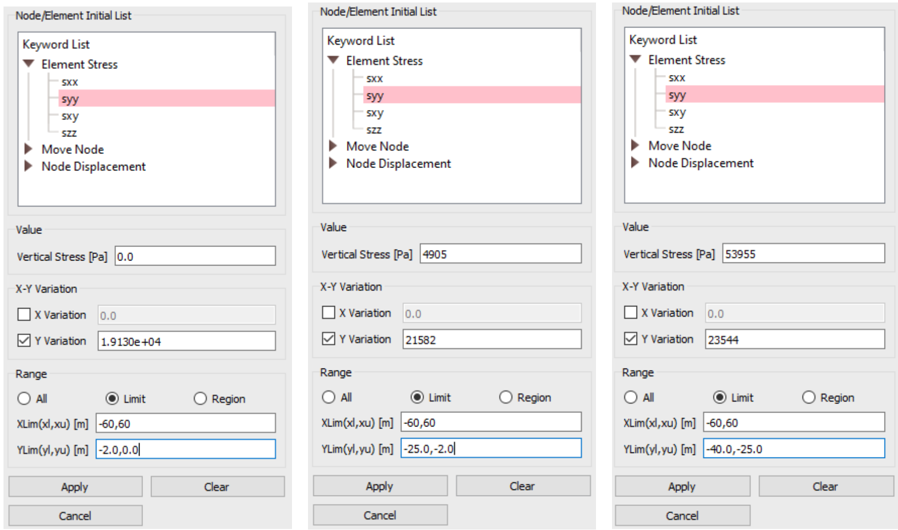


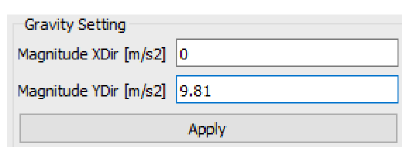
Figure 5- Stress initialization dialog box, initializing vertical stress for each layer.

Table 2- List of parameters for stress initialization.

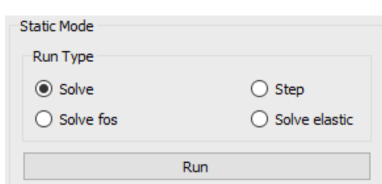
Stress Type	Initial Value (Pa)	Y-Variation Value (Pa)	X-Limit (m)	Y-Limit (m)
syy	0	19130	-60,60	-2,0
syy	4905	21582	-60,60	-25,-2
syy	53955	23544	-60,60	-40,-25
sxx	0	9564.8	-60,60	-2,0
sxx	2943	12949	-60,60	-25,-2
sxx	35071	15304	-60,60	-40,-25
szz	0	9564.8	-60,60	-2,0
szz	2943	12949	-60,60	-25,-2
szz	35071	15304	-60,60	-40,-25

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 9.81 m/s^2.

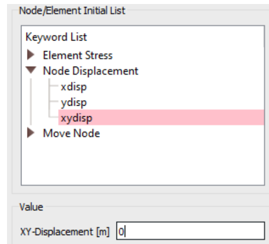
Note: Downward gravity is positive.

Select:  →  → 

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.

Select:  → 

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.

Select:  →  → 

Perform the Excavation Sequence

A staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. The calculation phases are Plastic analyses, Staged construction. The three-dimensional arching effect is emulated by using the so-called β-method. The idea is that the initial stresses p_k acting around the location where the tunnel is to be constructed are divided into a part (1 - β) p_k that is applied to the unsupported tunnel and a part Deconfinement method that is applied to the supported tunnel. To define the calculation process, follow these steps:

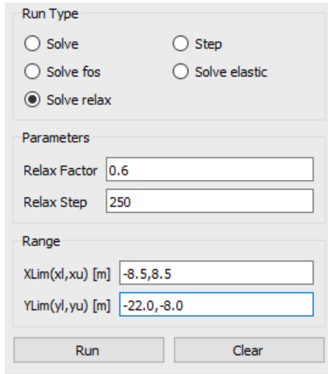
Stage 1: Excavate top heading

To excavate the top heading, 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 heading region and press ***Apply***.

Stage 2: Perform stress relaxation

To perform stress relaxation in ***ADONIS***, one can use the "***solve relax***" command, which is available under ***Run Type*** in the ***Calculation*** panel. In order to perform stress relaxation let's select the ***Solve Relax*** and modify the parameters. For this tutorial the assumption is that 60% of initial stress in the excavated soil should disappear before temporary support installation (so the remaining 40% is to be considered later). To achieve this the relax factor should be set to 0.6. The number of steps for stress relaxation to gradually be applied to the tunnel boundary is set to 250 by default. Then using mouse select the boundary that stress relaxation would be applied at (i.e. excavation boundary) or directly specify the range (i.e. xlim=[-8.5,8.5] and ylim=[-22.0,-8.0]) and click on ***Run*** button to start the cycle.

Notes: Relax step can be increased to have more smooth gradual stress relaxation but 250 step seem to be working for most of the cases.

Select:  → 

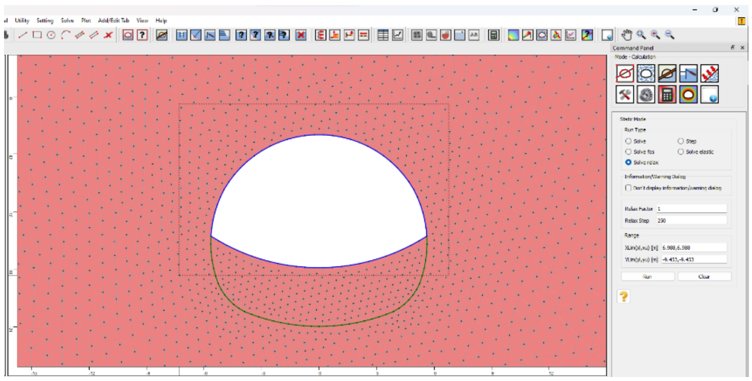


Figure 6- Use “solve relax” command to relax stress on the boundary of tunnel.

Stage 3: Install shotcrete support for top heading

In this stage 100 mm thick shotcrete (Table 3) will be installed at the crown of tunnel. To do that, select the ***Structures*** button from command panel, toolbar or menu, then select ***Beam*** element. Select ***Edge*** from ***Type*** group-box and after that ***Range*** from selection mode. Change the interface type to ***Both Sides*** from the dropdown. Now use mouse to select the range (or type xlim=[-8.0,8.0] and ylim=[-19.0,-10.0]) that shotcrete support should be installed and press ***Apply*** (Figure 7). Then click on ***Beam Element Properties*** and assign the shotcrete properties as shown in Figure 8. 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).

Note: In ***ADONIS***, the beam element formulation is a *plane-stress* formulation. If the beam is representing a structure that is continuous in the direction perpendicular to the analysis plane (e.g., a concrete tunnel lining), the value specified for E should be divided by $(1 - v^2)$ to account for plane-strain conditions.

Table 3- Shotcrete properties.

Thickness (m)	Young Modulus (Pa)	Poisson’s Ratio
0.1	15.0×10^9	0.2

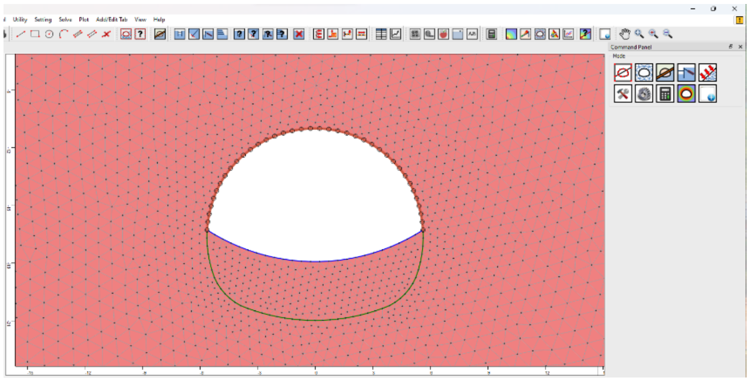


Figure 7- Install beam elements on tunnel's crown.

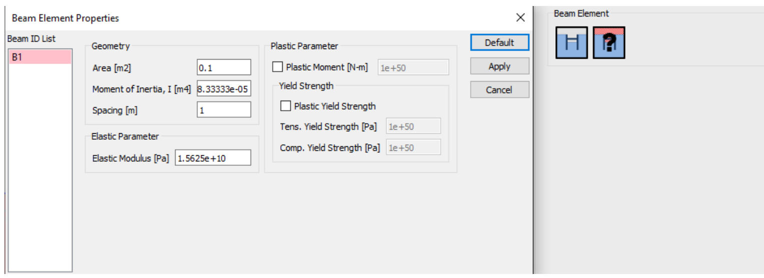


Figure 8- Assign shotcrete properties.

Stage 4: Excavate Bench

To excavate the bench, 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 bench region and press ***Apply***.

Stage 5: Install shotcrete support for bench

Repeat the sequence explained at stage 3 and install the beam element at the excavated part of the tunnel (i.e. bench region) as shown in Figure 9. There is no need for specifying the beam properties since similar shotcrete parameters will be used for the bench. 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).

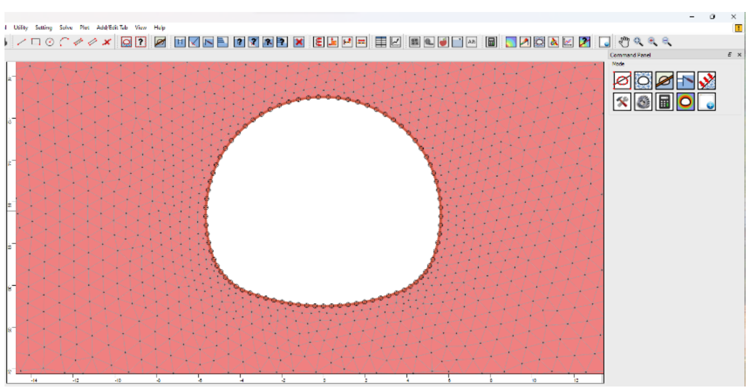
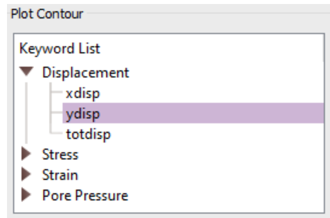


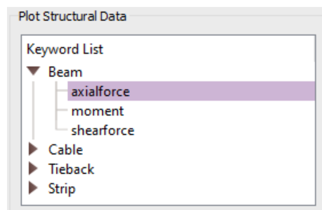
Figure 9- Install beam elements on tunnel's bench.

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 ***ydisp*** and press ***Apply***.

Select:  →  → 

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

Select:  →  → 

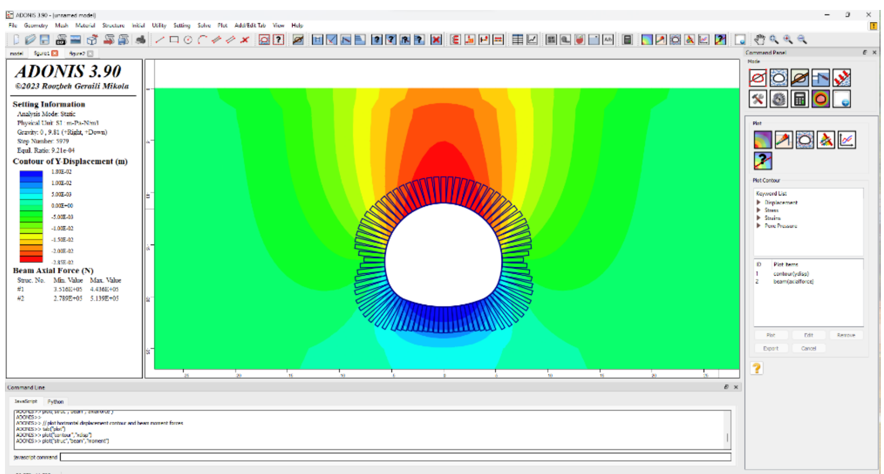


Figure 10- Vertical displacement contour and beam axial force plots.

The same process can be followed to plot the vertical displacement contour and beam elements moment force as shown in Figures 10 and 11.

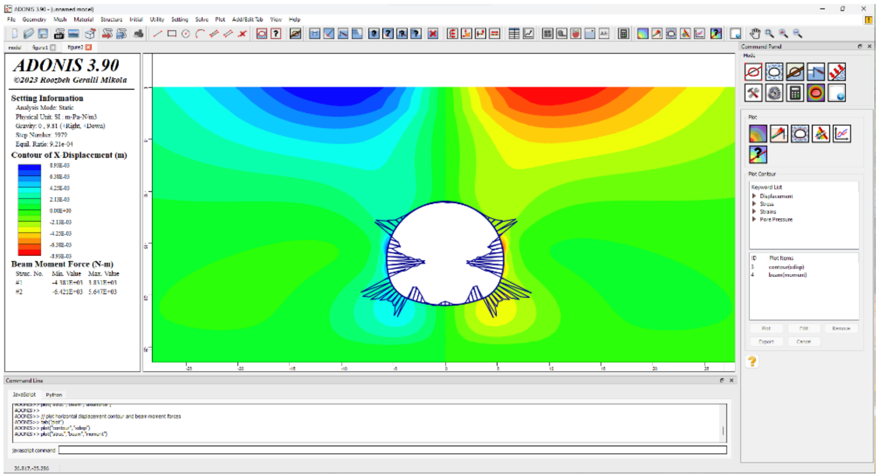


Figure 11- Horizontal displacement contour and beam moment force plots.

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",-60,-40,"endPoint",60,0)
line("startPoint",-60,-2,"endPoint",60,-2)
line("startPoint",-60,-25,"endPoint",60,-25)

arc("startPoint",3.88,-20.23,"midPoint",4.69,-19.62,"endPoint",5.25,-18.77,"numSeg",6)
arc("startPoint",4.95,-13.87,"midPoint",0.00,-11.00,"endPoint",-4.95,-13.87,"numSeg",25)
arc("startPoint",-5.25,-18.77,"midPoint",-4.69,-19.62,"endPoint",-3.88,-20.23,"numSeg",6)
arc("startPoint",-3.88,-20.23,"midPoint",0.00,-21.00,"endPoint",3.88,-20.23,"numSeg",20)
arc("startPoint",5.25,-18.77,"midPoint",5.57,-17.55,"endPoint",5.63,-16.29,"numSeg",6)
arc("startPoint",5.63,-16.29,"midPoint",5.42,-15.05,"endPoint",4.95,-13.87,"numSeg",6)
arc("startPoint",-5.25,-18.77,"midPoint",-5.57,-17.55,"endPoint",-5.63,-16.29,"numSeg",6)
arc("startPoint",-5.63,-16.29,"midPoint",-5.42,-15.05,"endPoint",-4.95,-13.87,"numSeg",6)
arc("startPoint",-5.63,-16.29,"midPoint",0.00,-17.94,"endPoint",5.63,-16.29,"numSeg",25)

// discretize/mesh
discretize("maxedge",1.0)
gmsh("maxedge",1.0,"elemtype","T3","useNMD","on")

// create new materials
material("create","Mohr-Coulomb","matid",1,"matname","Fill","density",1950,"shear",3.84615e+06,"bulk",8.33333e+06,"coh",1e3,"fric",29,"dil",0,"tens",0)
material("create","Mohr-Coulomb","matid",2,"matname","Soil","density",2200,"shear",4e+07,"bulk",6.66667e+07,"coh",3.9e4,"fric",35,"dil",0,"tens",0)
material("create","Mohr-Coulomb","matid",3,"matname","Rock","density",2400,"shear",2e+08,"bulk",3.33333e+08,"coh",1e5,"fric",40,"dil",0,"tens",1.0e4)

// assign materials
material("assign","matid",1,"region",0.0,-1.0)
material("assign","matid",2,"region",0.0,-9.0)
material("assign","matid",2,"region",0.0,-13.0)
material("assign","matid",2,"region",0.0,-18.5)
material("assign","matid",3,"region",0.0,-32.0)

// assign boundary condition
applybc("xfix","xlim",-60.1,-59.9,"ylim",-40.1,0.1)
applybc("xfix","xlim",59.9,60.1,"ylim",-40.1,0.1)
applybc("xyfix","xlim",-60.1,60.1,"ylim",-40.1,-39.9)

// initialize stress condition
```

```
var grav = 9.81
var dsyy1 = 1950*grav
var isyy1 = 0.0
var dsyy2 = 2200*grav
var isyy2 = dsyy2*2.0 - dsyy1*2.0
var dsyy3 = 2400*grav
var isyy3 = dsyy3*(2.0+23.0) - dsyy1*2.0 - dsyy2*23.0
var k01 = 0.5
var k02 = 0.6
var k03 = 0.65
initial("syy",isyy1,"yvar",dsyy1,"xlim",-60,60,"ylim",-2,0)
initial("syy",isyy2,"yvar",dsyy2,"xlim",-60,60,"ylim",-25,-2)
initial("syy",isyy3,"yvar",dsyy3,"xlim",-60,60,"ylim",-40,-25)
initial("sxx",k01*isyy1,"yvar",k01*dsyy1,"xlim",-60,60,"ylim",-2,0)
initial("sxx",k02*isyy2,"yvar",k02*dsyy2,"xlim",-60,60,"ylim",-25,-2)
initial("sxx",k03*isyy3,"yvar",k03*dsyy3,"xlim",-60,60,"ylim",-40,-25)
initial("szz",k01*isyy1,"yvar",k01*dsyy1,"xlim",-60,60,"ylim",-2,0)
initial("szz",k02*isyy2,"yvar",k02*dsyy2,"xlim",-60,60,"ylim",-25,-2)
initial("szz",k03*isyy3,"yvar",k03*dsyy3,"xlim",-60,60,"ylim",-40,-25)

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

// solve to initialize
solve()

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

// excavate the top region (sand)
excavate("region",0.0,-13.0)

// solve with 60% stress relaxation (40% of load remains)
solve("relax","relaxFactor",0.6,"relaxStep",250,"xlim",-8.5,8.5,"ylim",-22.0,-8.0)

// install shotcrete
var beam_nu = 0.2; // shotcrete poisson's ratio
var beam_th = 0.1; // shotcrete thickness
var beam_b = 1.0; // out of plane length
var beam_area = beam_th * beam_b; // shotcrete area
var beam_I = (beam_b*Math.pow(beam_th,3))/12.0; // shotcrete moment of inertia
var beam_ymod = 15e9 / (1-Math.pow(beam_nu,2)); // E should be divided by (1 – nu^2) to account
for plane-strain conditions.
structure("drawbeam","beamid",1,"xlim",-6.5,6.5,"ylim",-16.3,-10.5)
structure("material","beamid",1,"area",beam_area,"I",beam_I,"ymod",beam_ymod)

// solve to equilibrium
solve()

// excavate bench
excavate("region",0.0,-19.5)

// install shotcrete
structure("drawbeam","beamid",1,"xlim",-6.5,6.5,"ylim",-21.0,-16.3)

// solve to equilibrium
solve()

// plot vertical displacement contour and beam axial forces
plot("contour","ydisp")
plot("struc","beam","axialforce")

// plot horizontal displacement contour and beam moment forces
tab("plot")
plot("contour","xdisp")
plot("struc","beam","moment")
```