Quick Start Tutorial

Created By: **Roozbeh Geraili Mikola, PhD, PE**
Email: **adonis4geo@outlook.com**
Web Page: **www.geowizard.org**

30 , 30

Sxx = Syy = Szz = 30 MPa

R = 5

0 , 0

-30 , -30
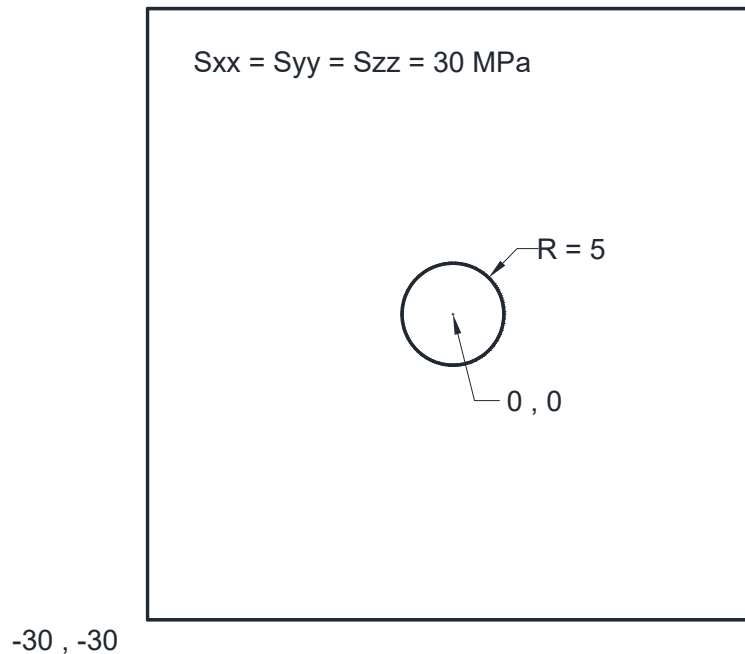
This tutorial will demonstrate some of the basic features of **ADONIS** using the simple model shown above.

Select Unit

First let's select the unit for this tutorial. This step is optional but it's recommended that user specify the unit before getting started. **ADONIS** accepts any consistent set of engineering units. Pleasure note that user should take great care when converting from one system of unit to another. **ADONIS** does not perform any conversion from unit to another. Examples of consistent sets of units for basic parameters are shown in Table 1.
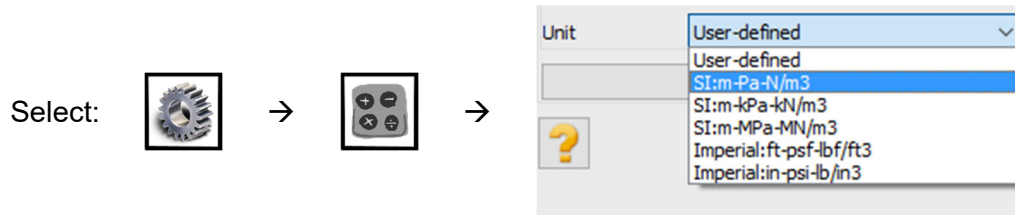
Table 1- System of units – mechanical parameters

	SI				Imperial	
Length	m	m	m	cm	ft	in
Density	kg/m^3	$10^3 \ kg/m^3$	$10^6 \ kg/m^3$	$10^6 \ g/m^3$	$slugs/ft^3$	$snails/in^3$
Force	N	kN	MN	Mdynes	lb_f	lb_f
Stress	Pa	kPa	MPa	bar	lb_f / ft^2	psi
Gravity	m / sec^2	m / sec^2	m / sec^2	cm / sec^2	ft / sec^2	in / sec^2
Stiffness*	Pa / m	kPa / m	MPa / m	bar / cm	lb_f / ft^3	lb_f / in^3

* Stiffness refers to normal and shear stiffnesses at interfaces.

Where:

1 bar = 10^6 dynes / cm^2 = 10^5 N/m^2 = 10^5 Pa;
1 atm = 1.013 bars = 14.7 psi = 2116 lb_f / ft^2 = 1.01325 × 10^5 Pa;
1 slug = 1 lb_f - s^2 / ft = 14.59 kg;
1 snail = 1 lb_f -s^2 / in; and
1 gravity = 9.81 m/ s^2 = 981 cm / s^2 = 32.17 ft / s^2.
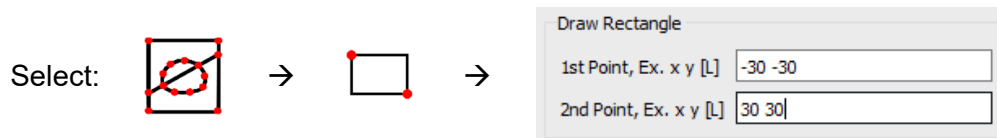
Since we will be using Metric (Pa) units for this tutorial, let's select "SI: m-Pa-N/m³" from *Calculation Setting* under *Setting* button.

Select: ![gear icon] → ![settings icon] →

Unit	User-defined ∨
	User-defined
?	SI:m-Pa-N/m3
	SI:m-kPa-kN/m3
	SI:m-MPa-MN/m3
	Imperial:ft-psf-lbf/ft3
	Imperial:in-psi-lb/in3

Create Geometry

Let's create the geometry as follows.

Select the *Create Geometry* button, then select *Draw Rectangle*. Then enter the first and second points' coordinates and press *Apply*.

Select: ![geometry icon] → ![rectangle icon] →

Draw Rectangle

1st Point, Ex. x y [L]	-30 -30
2nd Point, Ex. x y [L]	30 30

In order to draw the tunnel select *Draw Circle*. Then enter the center coordinate and radius of the tunnel and press *Apply*. Note: Number of segments for circle is 20 by default.

Select: ![geometry icon] → ![circle icon] →

Draw Circle

| Center, Ex. x y [m]	0 0
Radius [m]	5
Num. of Segment	20

The boundaries for this example have now been entered.

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 *1.5* 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: →
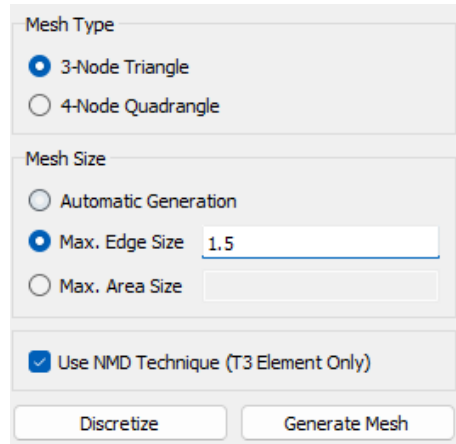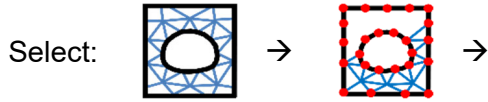
The finite element mesh is generated.

Low order elements tend to exhibit "locking" when undergoing plastic deformation, which leads to inaccurate estimates of loading capacity. 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. In order to activate NMD, check "Use NMD Technique", then click on *Apply* Button.

Notes:

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

Boundary Condition

In order to fix (i.e. zero displacement) the external boundary select *Assign 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 outside boundaries. When finished, click on *Apply* Button, or press *Enter*.
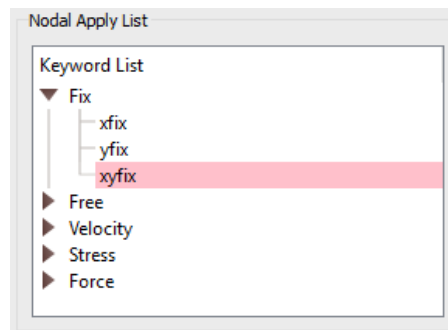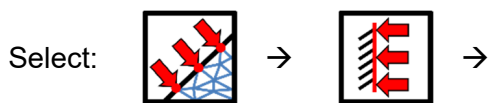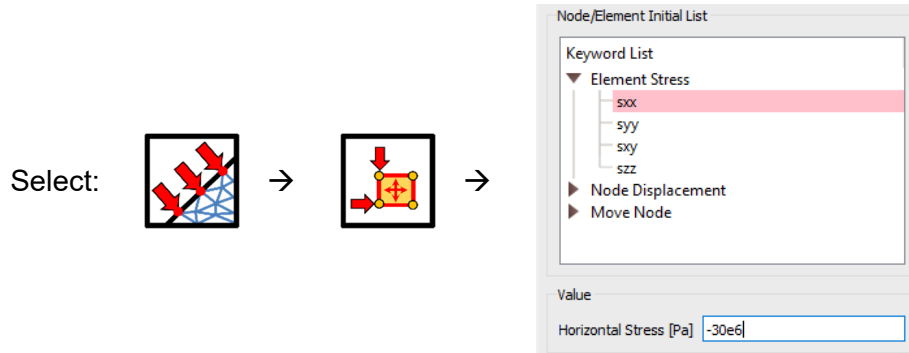
Select: →

The boundary condition has been applied.

In-Situ Stress

In order to assign constant in-situ stress to the model select *Assign Initial Condition* then select *Apply Initial Condition* then from the "Keyword List" click on *sxx* under *Element Stress*

keyword. In the stress value dialog enter Sxx = -30x10^6 Pa and click on **Apply** button. Next repeat the same sequence for **syy** and **szz** to assign vertical and out of plane stress to the model.

Select: → →

**Note:** compressive stresses are negative.

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 **IsoElastic** criteria then click on **Add Soil/Rock Material** button.
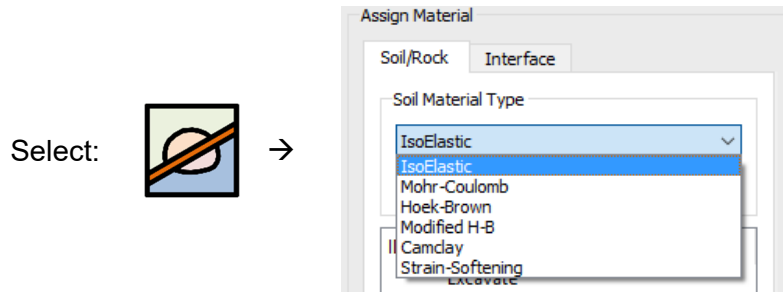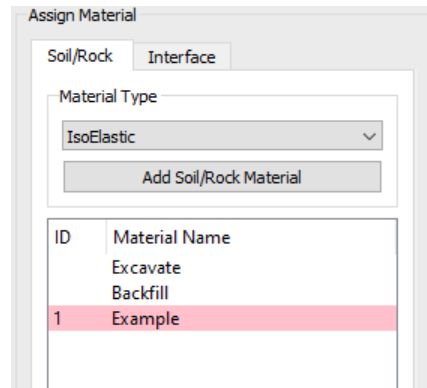
Select: →

Now enter the following rock mass properties.

Material ID = 1
Material Name = Example
Density = 2500 kg/m^3
Shear Modulus = 2.8x10^9 Pa
Bulk Modulus = 3.9x10^9 Pa
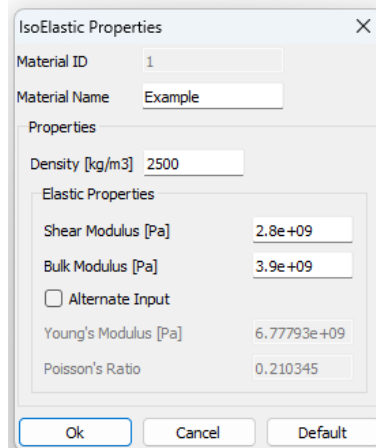
Now we need to assign the created material to the model. Select the create material (i.e. named Example) from the material list and click on **Apply**.

There is one last thing to do to complete our simple model. We need to excavate the tunnel and for that, select **Excavate** in **Soil/Rock** tab under the Assign Material then click inside the tunnel area and click on **Apply**.
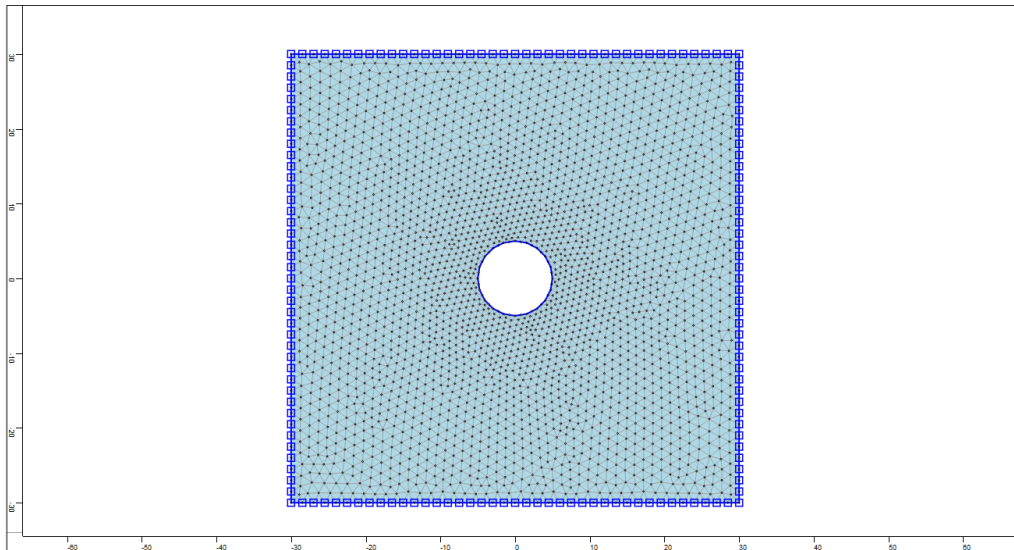

Figure 1- Model's mesh in ADONIS after excavation.

Solve Model

The model is now ready to run. In order to start the solving process select **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. When completed, you will be ready to view the results in Interpret.

Select: →
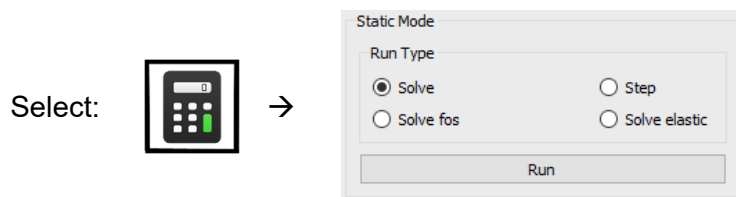
Post Processing

After solving, the results are shown under the *Plot Geometry/Results* in the command panel. 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. In order to plot horizontal displacement contour click on *Plot Geometry/Results* button then click on *Plot Contour*, under *Displacement* select *xdisp* and press *Apply*.
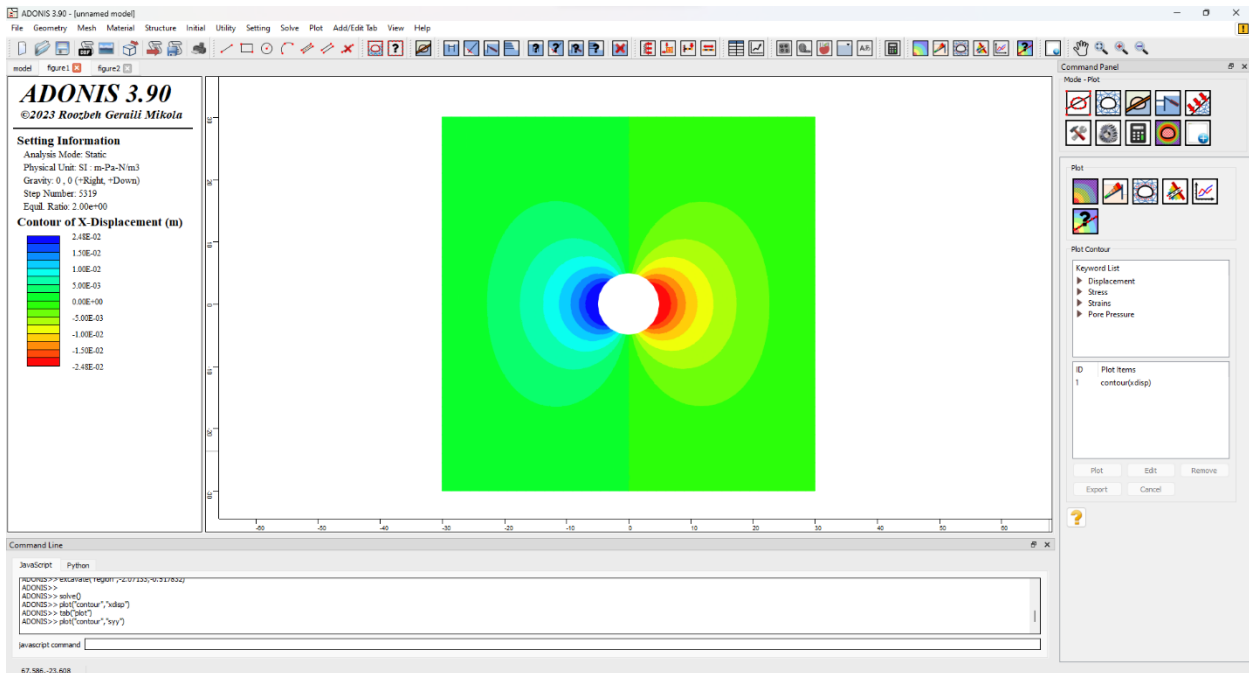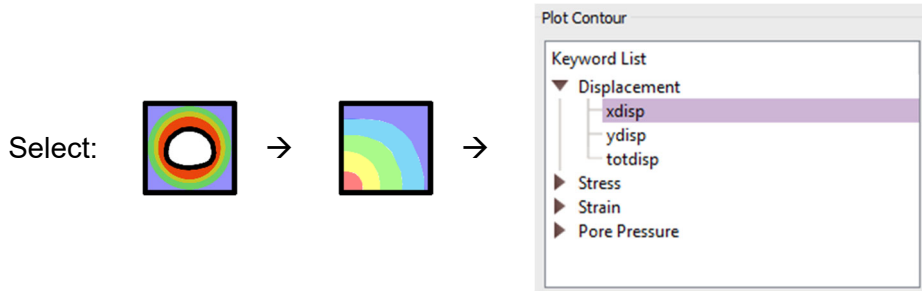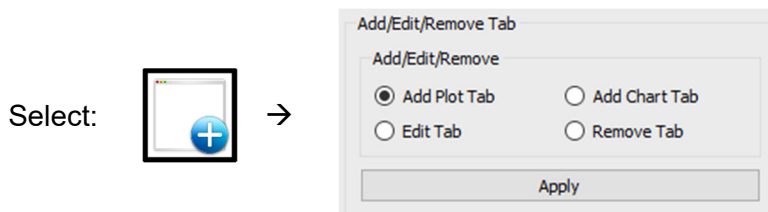

Figure 2- Horizontal displacement plot (unit: meter)

In order to plot vertical stress in the separate plot tab, click on *Add/Edit Tab* select *Add Plot Tab* radio button and click *Apply*. Now you can go ahead and select *syy* under *Stress* and press *Apply*.
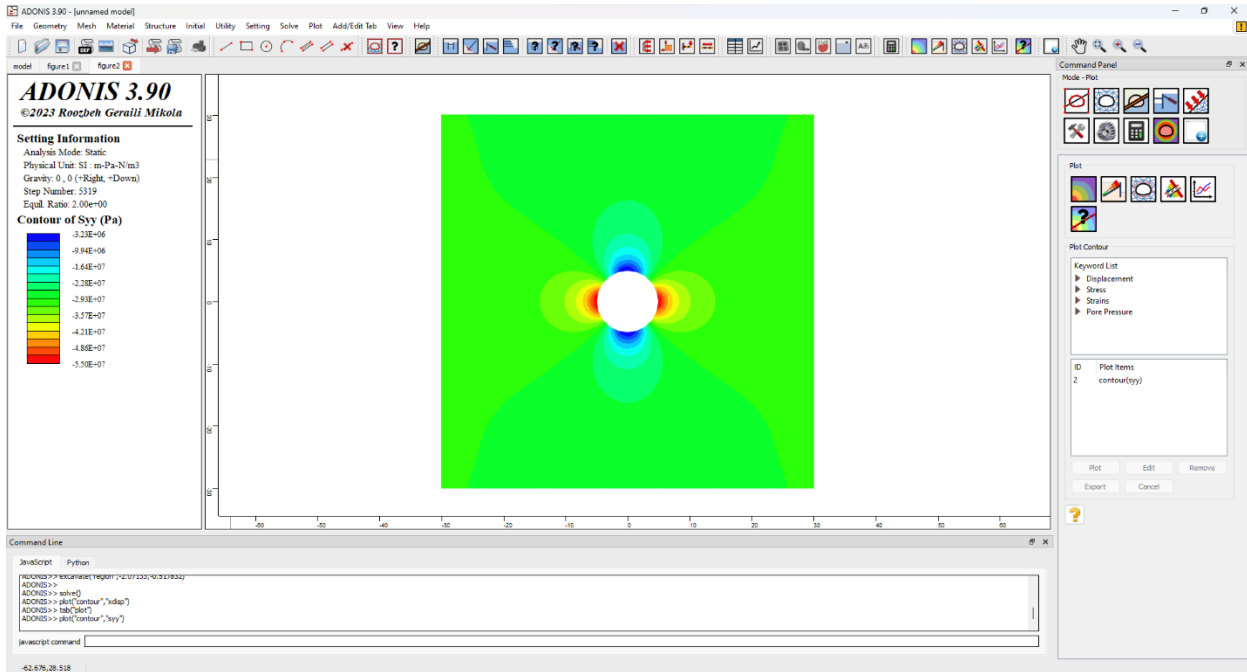
Figure 3- Horizontal stress plot (unit: Pa)

Script

After finishing the model you will be able to save the generated script by **ADONIS** in the text file.

Select:

The commands for this tutorial are listed below.

```javascript
newmodel()

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

rect("startPoint",-30,-30,"endPoint",30,30)
circle("centerPoint",0,0,"radius",5,"numSeg",20)

discretize("maxedge",1.5)
gmsh("maxedge",1.5,"elemtype","T3","useNMD","on")

material("create","IsoElastic","matid",1,"matname","Example","density",2500,"shear",2.8e+9,"bulk",3.9e+9)
material("assign","matid",1)

applybc("xyfix","xlim",-34.384,42.255,"ylim",-32.520,32.520)

initial("sxx",-30e6)
initial("syy",-30e6)
initial("szz",-30e6)

excavate("region",-2.07133,-0.517832)

solve()
plot("contour","xdisp")
tab("plot")
plot("contour","syy")
```