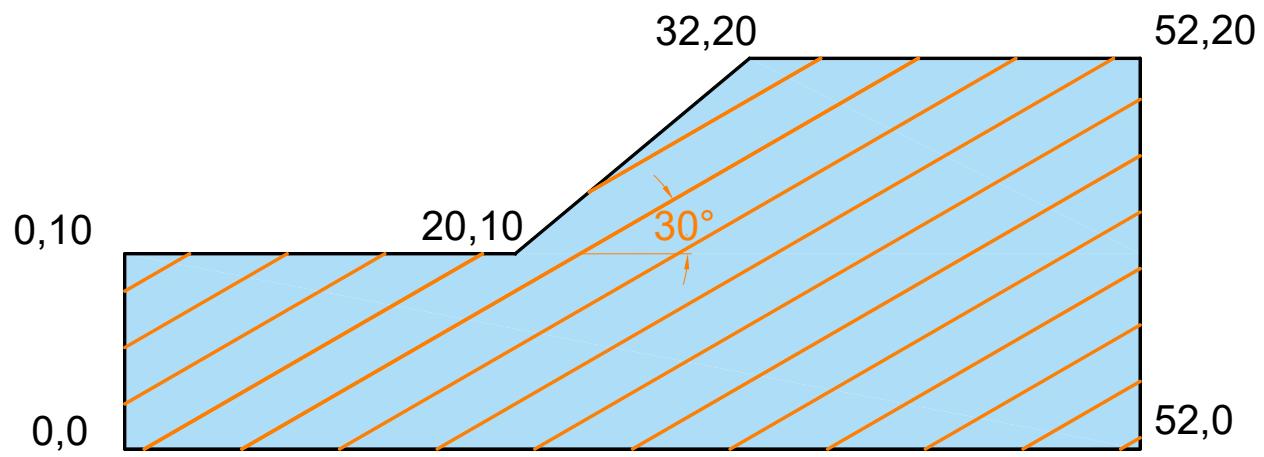# Stability Analysis of Jointed Rock Slope using Ubiquitous Joint Model

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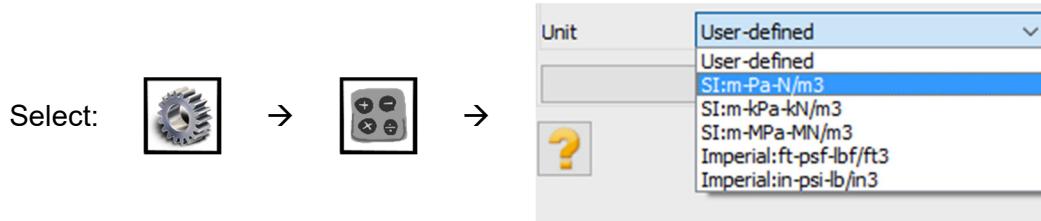***Email:*** adonis4geo@outlook.com
Web Page: www.geowizard.org

In this tutorial, **ADONIS** is used to determine the safety factor of a jointed rock slope by shear strength reduction method (SRM) considering Ubiquitous Joint model. This tutorial tries to cover the basics of setting up a model for an SRM analysis in **ADONIS**.

Select Unit

Let's select "SI: m-Pa-N/m³" as a system of unit from **Calculation Setting** under **Setting** button.

Create Geometry

Let's create the geometry as follows.

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

Select:

Draw Line

1st Point, Ex. x y [m]   0,0

2nd Point, Ex. x y [m]   52,0

Repeat the sequence for the rest of the lines until final geometry generated.

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 *0.5* 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:

Mesh Type

○ 3-Node Triangle

● 4-Node Quadrangle

Mesh Size

○ Automatic Generation

● Max. Edge Size   0.5

○ Max. Area Size

| Discretize | Generate Mesh |

The finite element mesh is generated.

Set Gravity

In order to set the gravity, select *Setting* then select *Gravity Setting* and assign the gravity magnitude which is 0 and 9.8 m/s^2 for x and y directions respectively. Please note that positive value for y-direction indicates top to bottom.
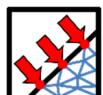
Select:

Gravity Setting

Magnitude XDir [m/s2]   0

Magnitude YDir [m/s2]   9.8

Apply

Boundary Condition

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 right boundary. When finished, click on *Apply* Button, or press *Enter*. Repeat the same sequence for left and bottom boundaries.
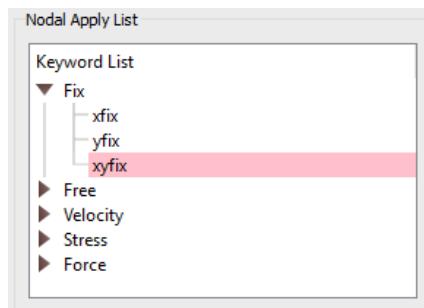
Select: → →

The boundary condition has been applied.

Properties

In this study, the strength reduction finite element analysis considering ubiquitous joint model is adopted to assess the stability of jointed rock slope. The ubiquitous-joint model represents the weak plane, such as weathering joints and bedding planes, orientation in an anisotropic plasticity model in a Mohr-Coulomb solid in which yielding is occurring in either along the weakness, the solid or both. The failure criteria depend on the orientation of weak plane, material properties of the solid or weak plane and the stress. The normal and shear stress are calculated at any orientation of the weak plane by giving the joint orientation for general failure then displacement, factor of safety and the stability of the excavation is evaluated by using Mohr-Coulomb failure criteria on the weak plane. Figure 1 illustrates the weak plane existing in a Mohr-Coulomb model and the global (x-y) and local (x'-y') coordinate frame.
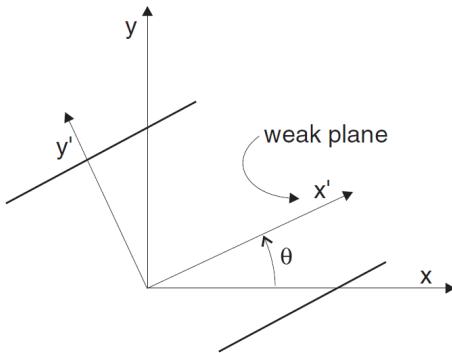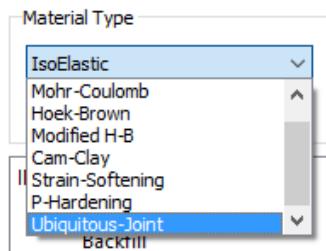

Figure 1- A weak plane oriented at an angle to the global frame.

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 **Ubiquitous-Joint** criteria then click on **Add Soil/Rock Material** button.
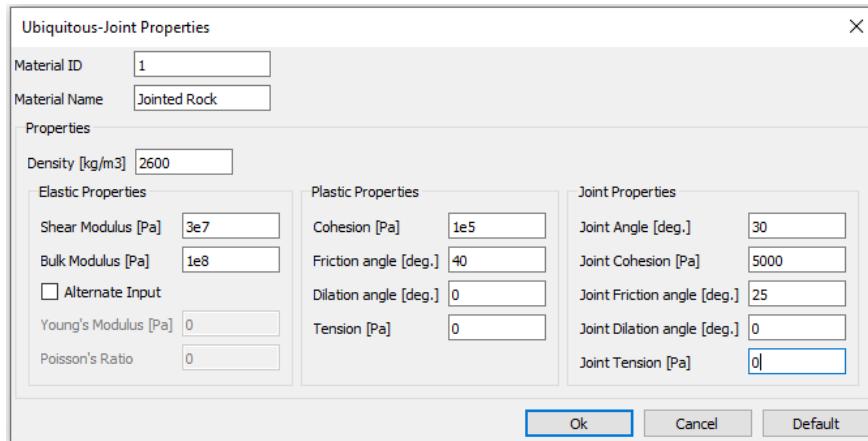
Select: →

Now enter the following jointed rock properties.

Material ID = 1
Material Name = Jointed Rock
Density = 2600 kg/m^3
Shear Modulus = 3.0E07 Pa
Bulk Modulus = 1.0E08 Pa
Cohesion = 1.0E05 Pa
Friction = 30 deg.
Tensile Strength = 0 Pa
Joint Angle = 30 deg.
Joint Cohesion = 5000 Pa
Joint Friction = 25 deg.
Joint Tensile Strength = 0 Pa

Now select the create material (i.e. named Example) from the material list and click on **Apply** to assign the selected material to the solid elements.
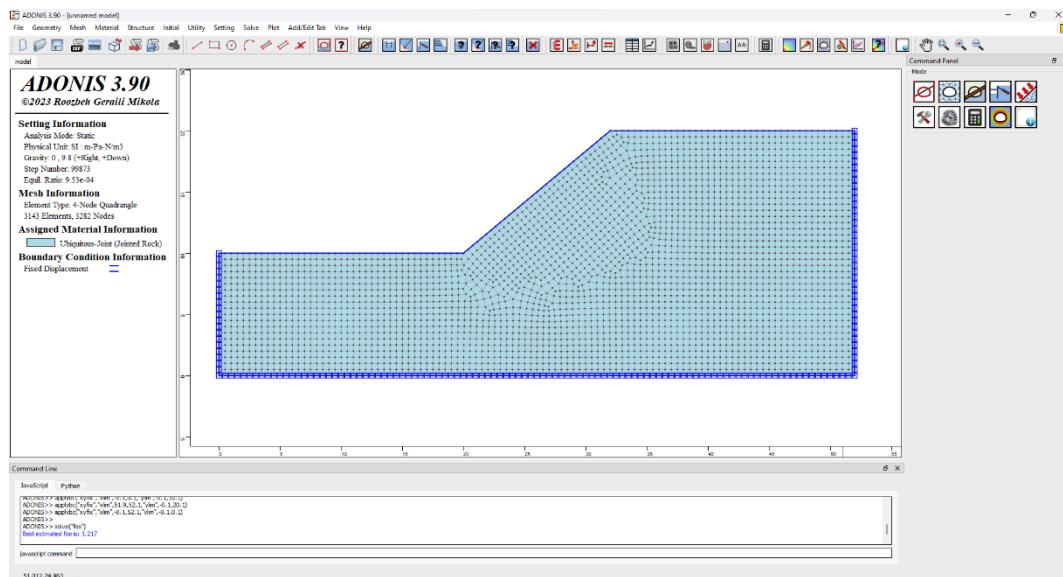

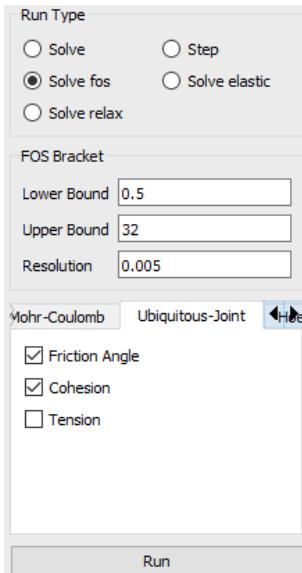Figure 2- Initial mesh and boundary conditions.

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 fos** radio button. In current analysis **Tension** mode is not included to ignore the tensile strength in the factor of safety (FOS) calculation. The lower, upper as well as resolution value can be modified. For this tutorial default values are considered.

To run the model 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: →

After you click on Run button, the program starts and perform an automatic FOS calculation. You should see the following screen with the critical factor of safety of 1.217 displayed at the top of the main window. Note that the same model is computed with **FLAC**, a commercial program developed by **Itasca**, and the factor of safety is calculated to be 1.217.
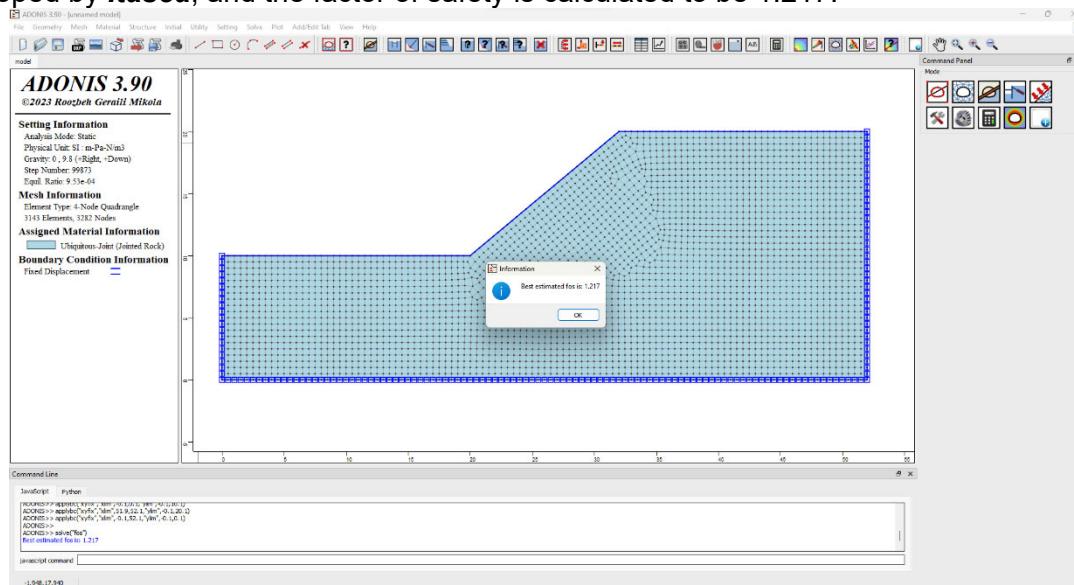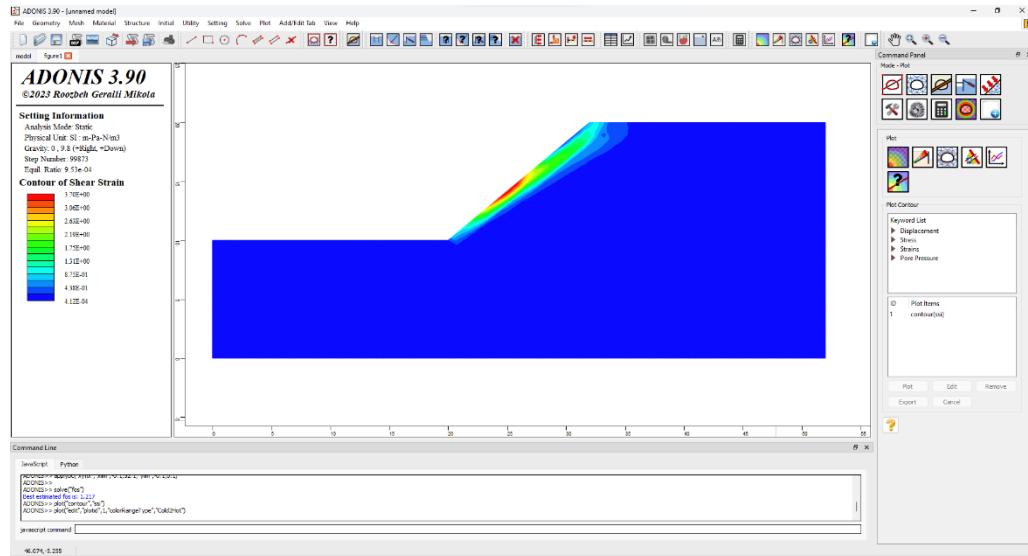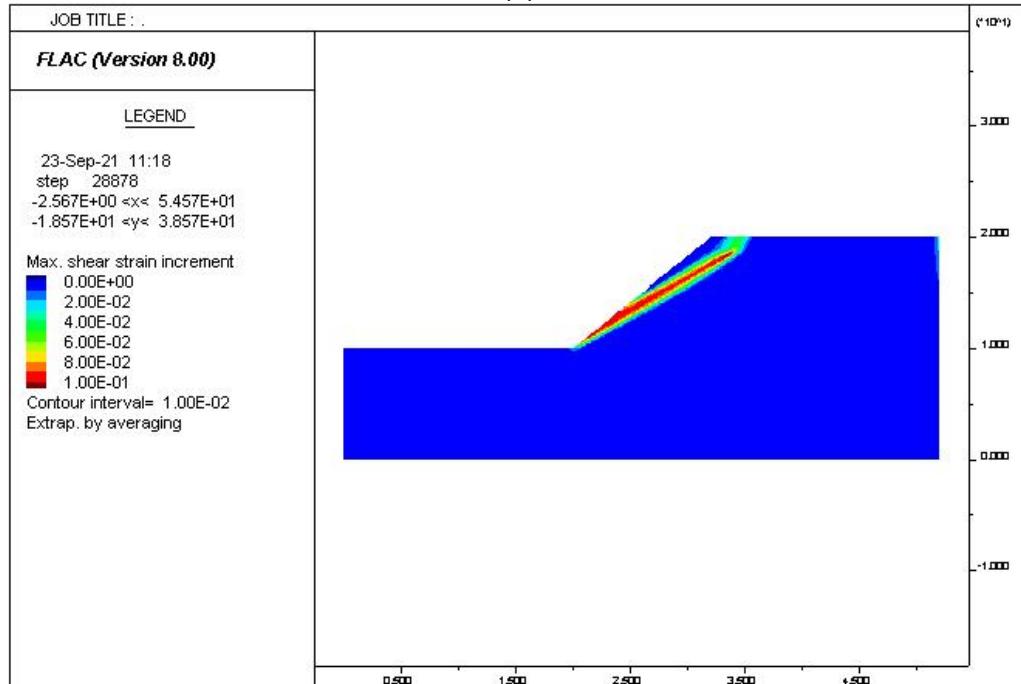

Figure 3- Displayed critical factor of safety at the end of run.

Post Processing

Now you plot the displacement and strain contours to check the critical failure surface. For instance, the following figure shows the comparison of maximum shear strain contour between *ADONIS* and *FLAC*.

(a)

(b)

Figure 4- Contour plot of maximum shear strain using a) *ADONIS* b) *FLAC*.

Script

The commands for this tutorial are listed below.

```
newmodel()

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

line("startPoint",0,0,"endPoint",52,0)
line("startPoint",52,0,"endPoint",52,20)
line("startPoint",52,20,"endPoint",32,20)
line("startPoint",32,20,"endPoint",20,10)
line("startPoint",20,10,"endPoint",0,10)
line("startPoint",0,10,"endPoint",0,0)

discretize("maxedge",0.5)
gmsh("maxedge",0.5,"elemtype","Q4")

set("gravity",0,9.8)

material("create","Ubiquitous-Joint","matid",1,"matname","Jointed
Rock","density",2600,"shear",3e+07,"bulk",1e+08,"coh",100000,"fric",40,"jangle",30,"jcoh",5000,"j
fric",25)
material("assign","matid",1)

applybc("xyfix","xlim",-0.1,0.1,"ylim",-0.1,10.1)
applybc("xyfix","xlim",51.9,52.1,"ylim",-0.1,20.1)
applybc("xyfix","xlim",-0.1,52.1,"ylim",-0.1,0.1)

solve("fos")
```