Shear Strength Reduction Tutorial

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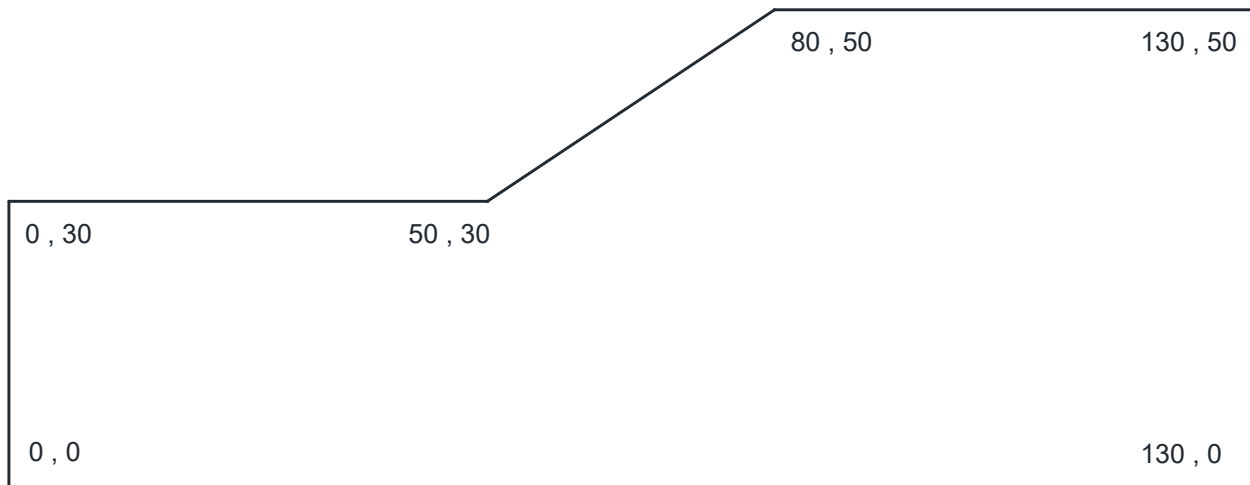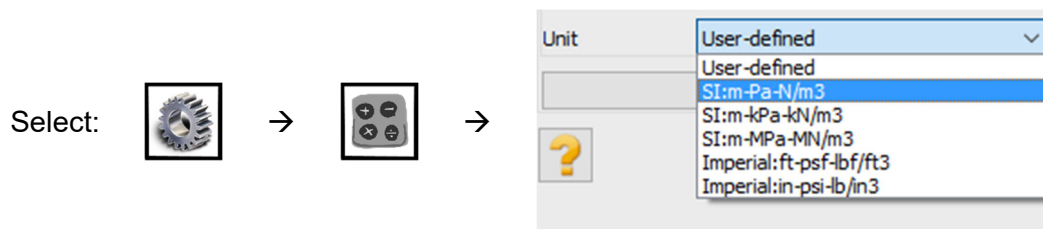

In this tutorial, ***ADONIS*** is used to determine the safety factor of a simple homogeneous slope using the shear strength reduction (SSR) method. This tutorial tries to cover the basics of setting up a model for an SSR analysis in ***ADONIS***.
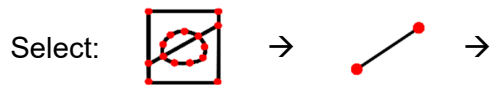
Select Unit

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***.
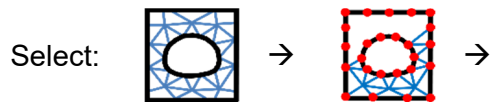
Select: → →

1st Point, Ex. x y [m]	0 0
2nd Point, Ex. x y [m]	130 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 *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.
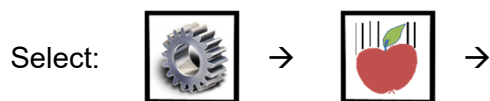
Select: → →

Mesh Type
○ 3-Node Triangle
● 4-Node Quadrangle

Mesh Size
○ Automatic Generation
● Max. Edge Size 1.0
○ 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.
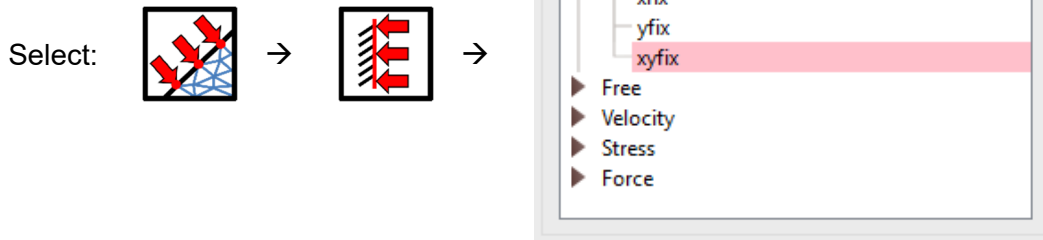
Select: → →

The boundary condition has been applied.

Properties

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: →
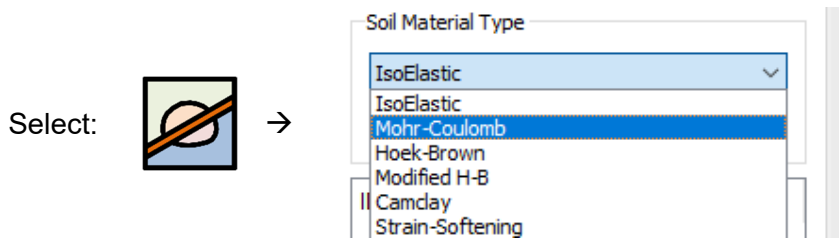
Now enter the following rock mass properties.

Material ID = 1
Material Name = Soil
Density = 1938.8 kg/m^3
Shear Modulus = 17857142.86 Pa
Bulk Modulus = 83333333.33 Pa
Cohesion = 5000 Pa
Friction = 30 deg.
Tensile Strength = 5000 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.
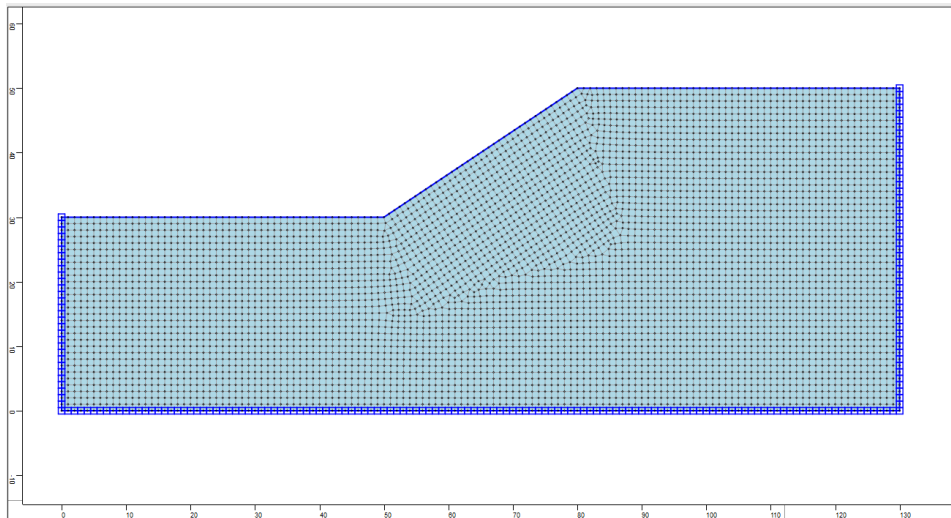

Figure 1- Initial mesh and boundary conditions.

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. Make sure to check **Tension** in order to include the tensile strength to 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: →
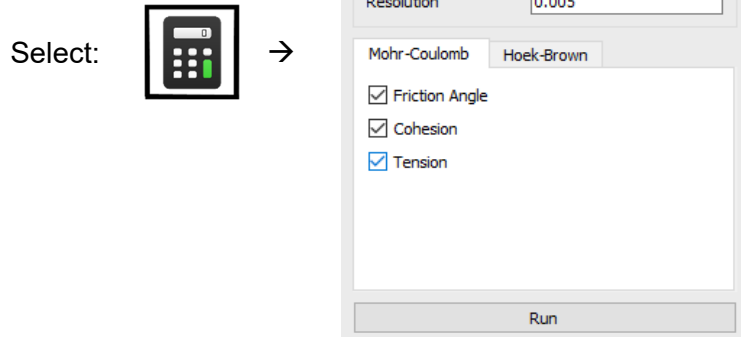
After you click on Run button, the program starts and perform an automatic FOS calculation. You should see the following screen with the critical factor of safety of 1.143 displayed at the top of the main window. Note that the same model is computed in **Slide** and **RS2**, two commercial programs developed by **Rocscience**, and the limit equilibrium safety factors are 1.14 and 1.15 respectively.
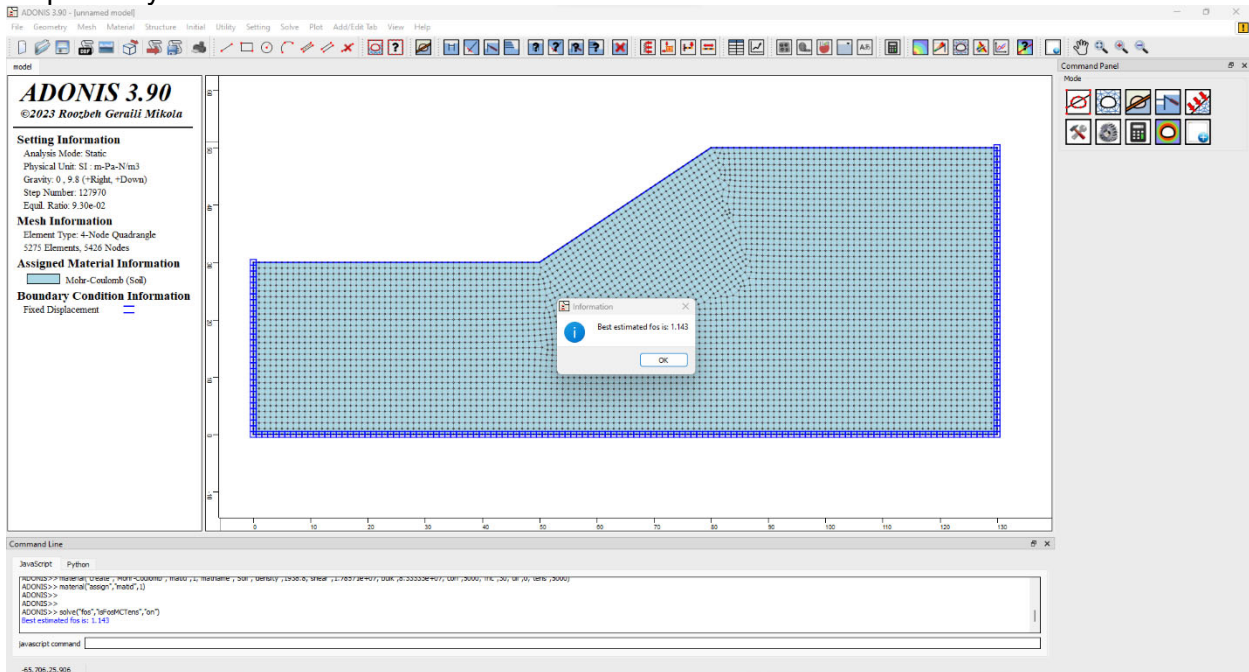

Figure 2- Displayed critical factor of safety at the end of run.

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 maximum shear strain contour click on *Plot Geometry/Results* button then click on *Plot Contour*, under *Strains* select *ssi* and press *Apply*.
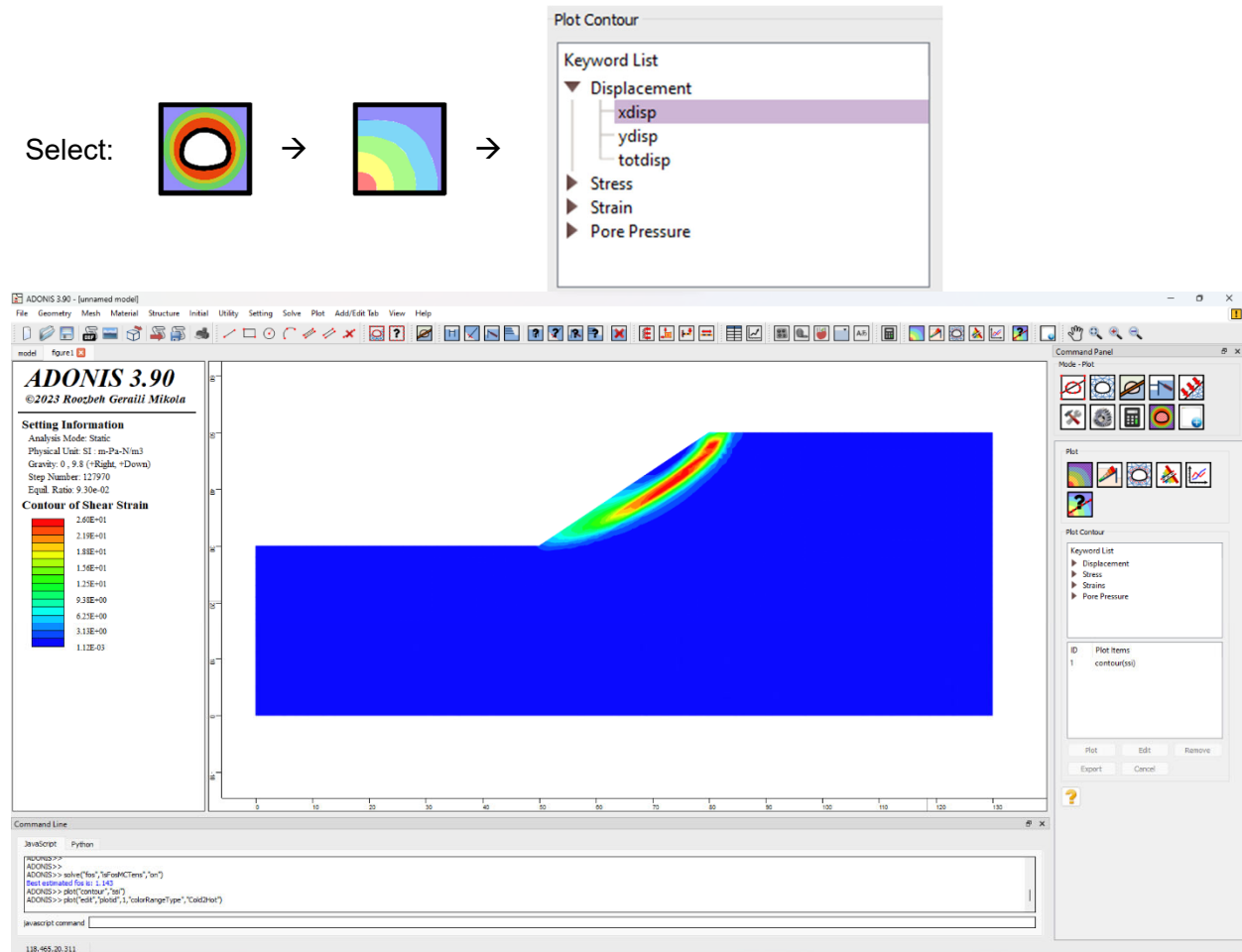

Figure 3- Contour plot of maximum shear strain.

Script

The commands for this tutorial are listed below.

```
newmodel()

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

line("startPoint",0,0,"endPoint",130,0)
line("startPoint",130,0,"endPoint",130,50)
line("startPoint",130,50,"endPoint",80,50)
line("startPoint",80,50,"endPoint",50,30)
line("startPoint",50,30,"endPoint",0,30)
line("startPoint",0,30,"endPoint",0,0)

discretize("maxedge",1.0)
gmsh("maxedge",1.0,"elemtype","Q4")

set("gravity",0,9.8)

applybc("xyfix","xlim",129.383,132.649,"ylim",-1.464,51.348)
applybc("xyfix","xlim",-6.418,132.874,"ylim",-6.418,0.788)
applybc("xyfix","xlim",-1.013,0.563,"ylim",-5.743,30.629)
```

```
material("create","Mohr-
Coulomb","matid",1,"matname","Soil","density",1938.8,"shear",1.78571e+07,"bulk",8.33333e+07,"coh"
,5000,"fric",30,"dil",0,"tens",5000)
material("assign","matid",1)


solve("fos","isFosMCTens","on")
```