Tutorial: How to create GMSH meshes for application in DRUtES

J R Blöcher

April 17, 2018

Contents

1   Introduction

This tutorial aims to help users use GMSH software to create meshes to run simulations with DRUtES. It covers the creation of GMSH meshes for direct simulations and special cases for dual permeability. An example for to use Gmsh and DRUtEs for mesh optimization is also included.

We recommend to watch this youtube tutorial first. Here you learn how to create a 3D geometry and how to use square and round shapes as well as the use of transfinite lines for local mesh refinement.

We will be producing 2D meshes. This is why the z coordinate is always zero in both examples.

2   Example 1: Rectangular mesh with transfinite line

We want to create a 2D mesh for a rectangular domain. The domain is depicted in Fig. 1. Note, that here you already have to decide what length units to use. The same units have to be used when setting observation coordinates and units of material properties. We will use cm in this example.
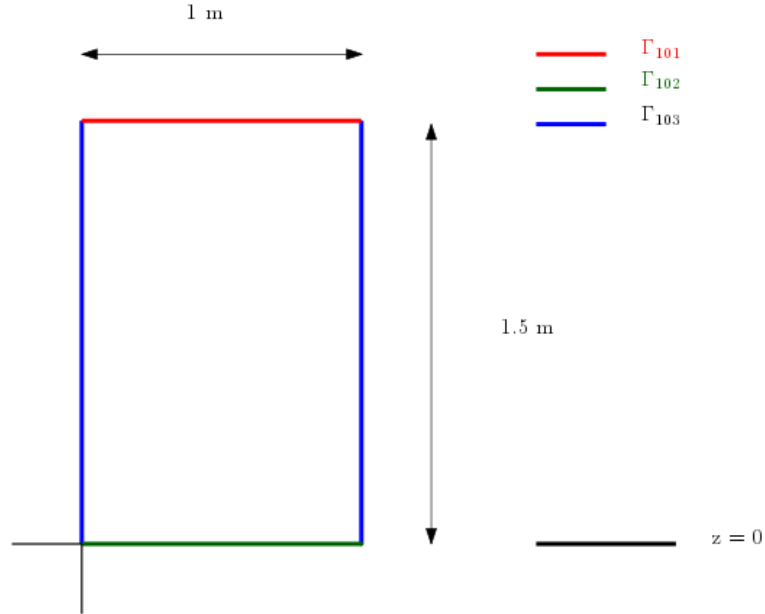

Figure 1:   Domain of the first example. The width is 1 m and the depth is 1.5 m. We have three boundaries: top (red), bottom (green) and the sides (blue. Our geodetic reference level is at the bottom of the profile with the origin at the left corner.
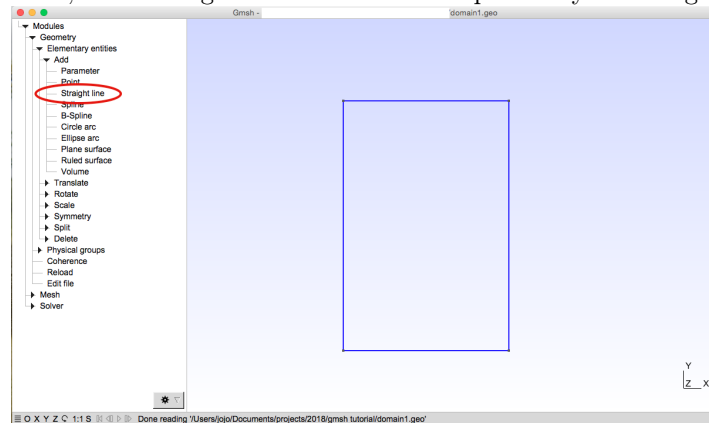
1. Open GMSH. Open Modules > Geometry > Elementary entities > Add > Point. The Z coordinate will always be 0. Set the point coordinates of the domain:
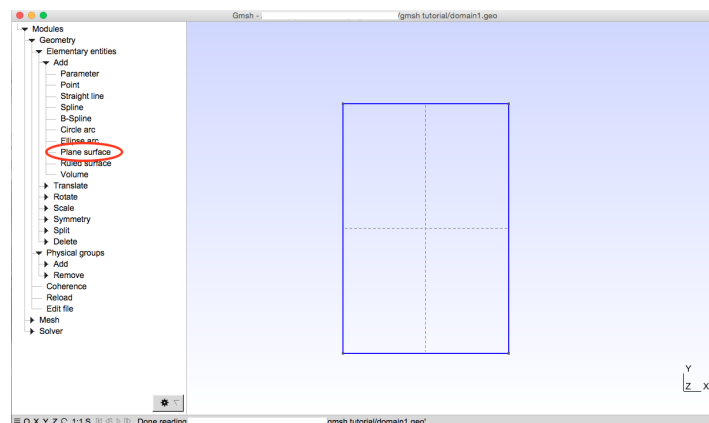
   - P1 (0,0)

- P2 (0, 150)

- P3 (100, 0)

- P4 (100,150)
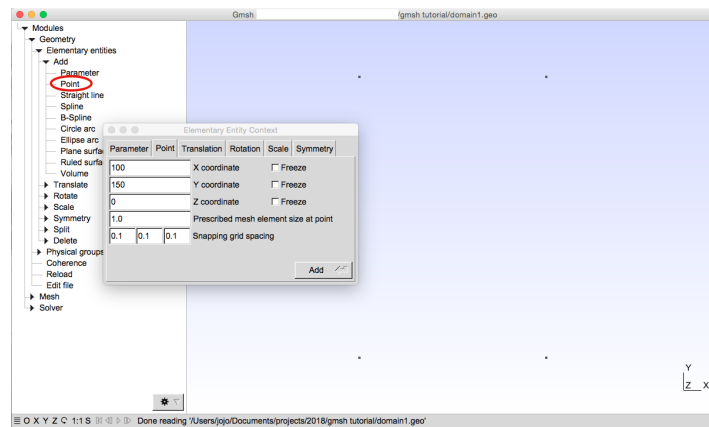
2. Next, add Straight lines between the points by selecting straight lines below Point


3. Next, add a Plane surface. Select the rectangle. Two dashed lines should appear in the center of the rectangle.


4. Next, we want to define Physical groups. This includes boundaries used for boundary conditions and materials. Note, the order of physical groups is very important. You need to define your material layers first, then boundaries. Select Physical group: Add Surface and name it soil, untick automatic numbering and give it number 1.

untick box
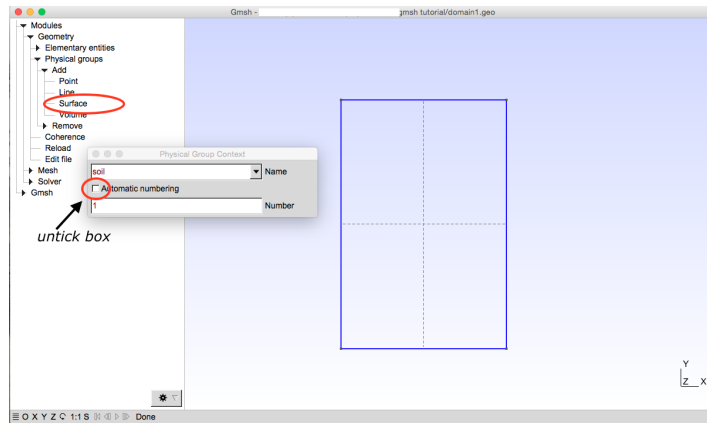
5. Next, select Physical group line and select first the top line and name it 'top' and give it number '101'. This has to be the same ID as the later configuration files. Hit e to finish your selection. Select the bottom line and name it 'bottom' with Number '102'. Hit e again. Now select both sides and name it 'sides' with Number '103'. After you finish you should see the physical group of each geometry when hoovering over them.
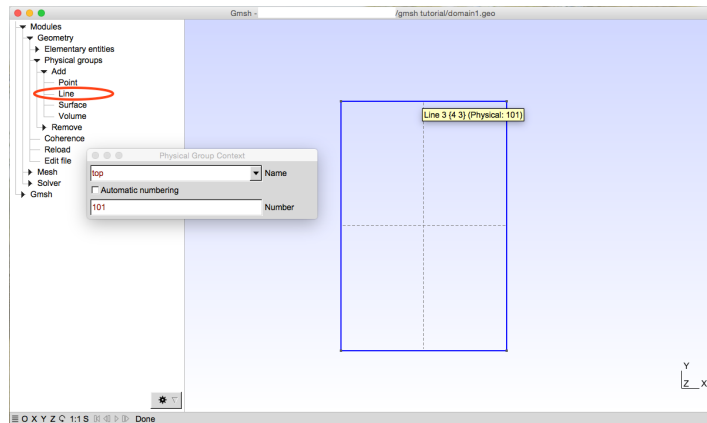
6. Our domain is ready to be meshed. To generate a mesh select Mesh and click on 2D. A mesh will appear. Probably not in yellow (these are my personal setting), rather black. We can see that the mesh is automatically a triangulation and quite fine. This is due to our settings in the geometry. We can see details of our mesh in Tools > Statistics and we can see that our mesh consist of more than 39000 triangles.

3
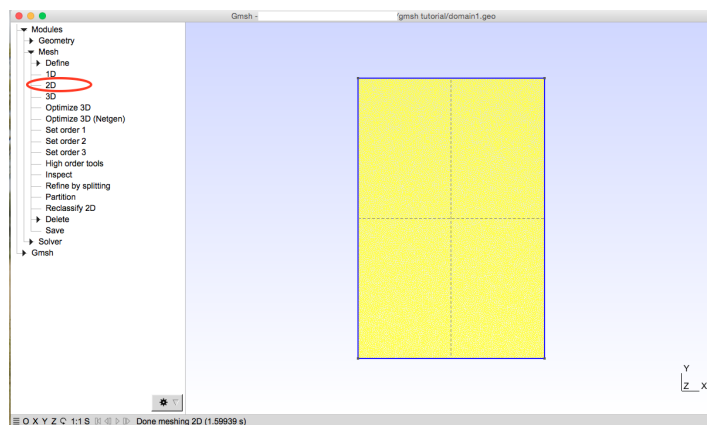
7. The exact number may differ between gmsh versions. In Geometry there is the option to 'Edit file'. This will open the .geo file in a text editor. There you can see the definition of our mesh. The fourth number in our points is 1. This defines the density of our mesh. The picture below shows what i see when I open the file in TextWrangler. The black background and green writing are again my personal settings.
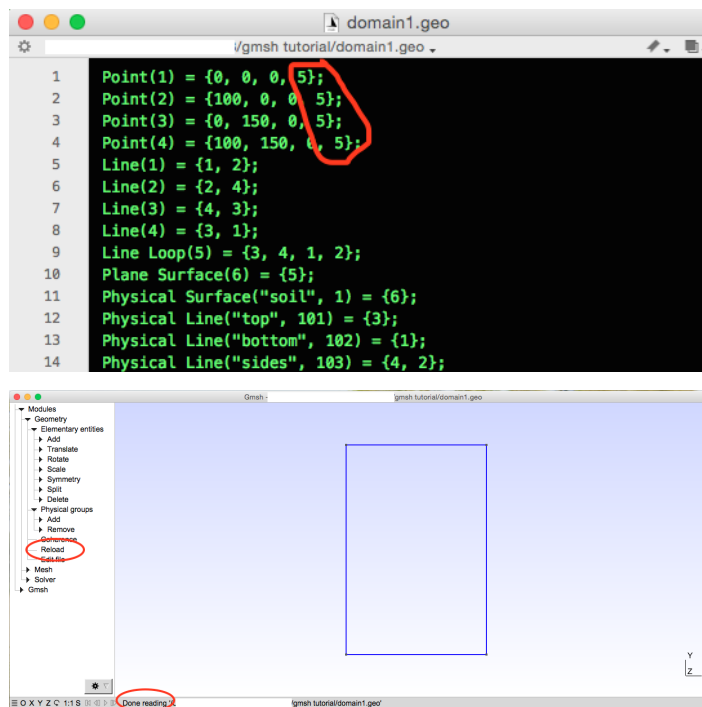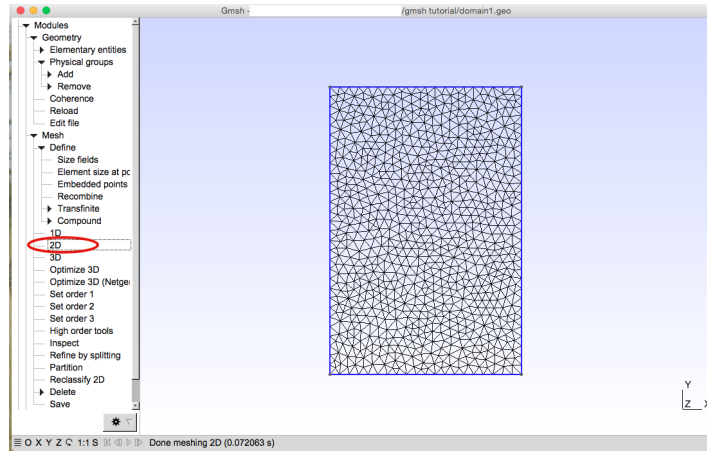

8. There are different ways to change your mesh density. First, change the fourth number in the text editor, to let's say 5. This will generate a much coarser mesh. Before you generate the mesh in GMSH, make sure you click on **'Reload'** in Geometry. Otherwise you generate errors. The mesh is now black for better visibility.

9. The second option to change the meshing involves the use of transfinite lines. We will use a transfinite line at the top boundary to make the mesh finer again. We go to Mesh > Define > Line. We enter Number of points: 50 and we select the top boundary and hit e. Then we hit mesh 2D. and we can see that the mesh is a lot finer.
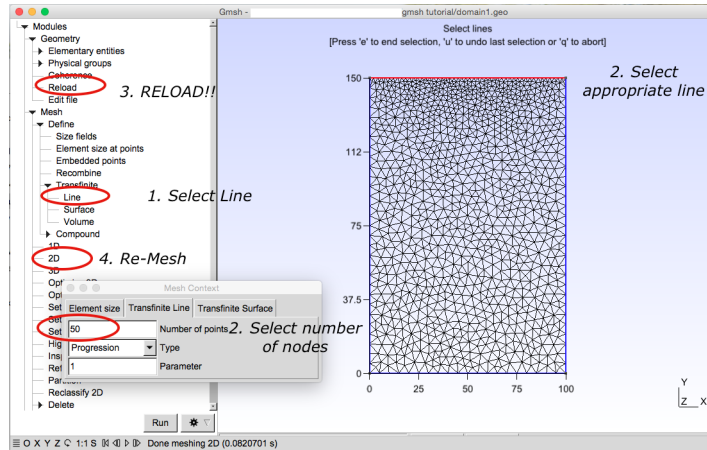
To use the mesh in DRUtES it has to be renamed to **mesh.msh**. For water flow K_{11} and K_{22} need to be defined. For a homogeneous medium they can have th same value. The number of boundaries are now 3. You need to define the boundaries 101 (top), 102 (bottom) and 103 (sides).

3 Example 2: Layered mesh with cylindric lense

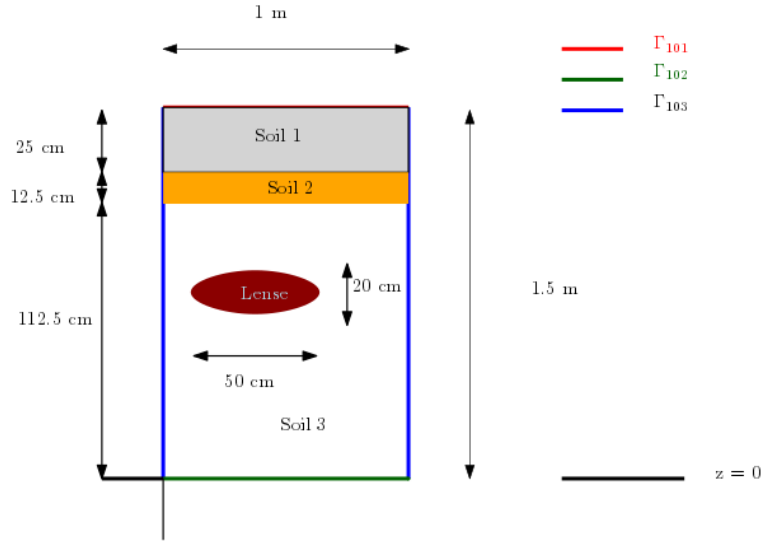The next domain contains 4 different soils, one of which is a lense (Fig. 2).

Figure 2: Domain of the soil wit three layers and a lense

1. Open GMSH. Open Modules > Geometry > Elementary entities > Add > Point. The Z coordinate will always be 0. Set the point coordinates of the domain:
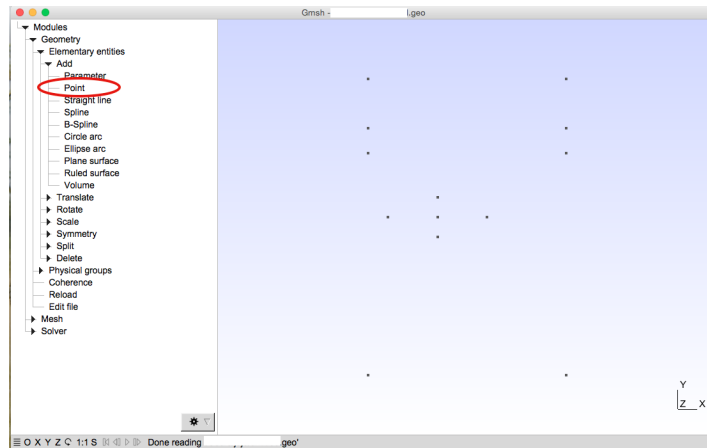
   - P1 (0,0)

   - P2 (0, 150)

   - P3 (100, 0)

   - P4 (100,150)

   and the dividing the soil layers:

   - P5 (0,112.5)

   - P6 (100, 112.5)

   - P7 (0,125)

   - P8 (100, 125)

   and the outside of the lense and the center point:

   - P9 (10,80)

   - P10 (60, 80)

   - P11 (35,70)

   - P12 (35, 90)

   - P13 (35, 80)

2. Define the lines for outside boundary. We will do the ellipse in the next step. For this, add straight lines from Open GMSH. Open Modules > Geometry > Elementary entities > Add > Straight Line. Be careful to only connect two adjacent points. If you skip points, you will not be able to define the surfaces correctly.
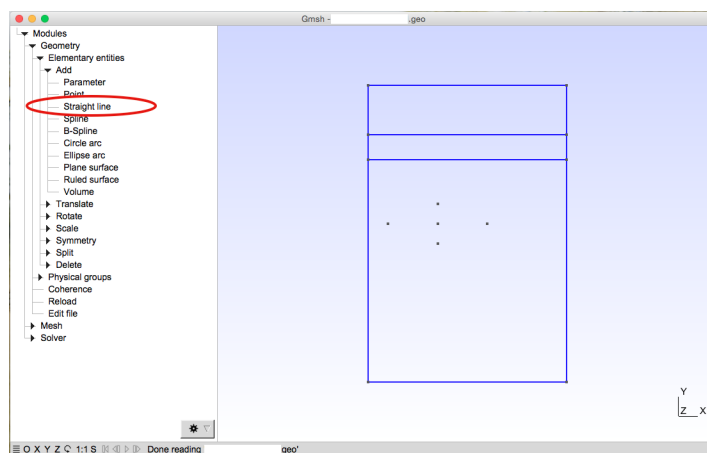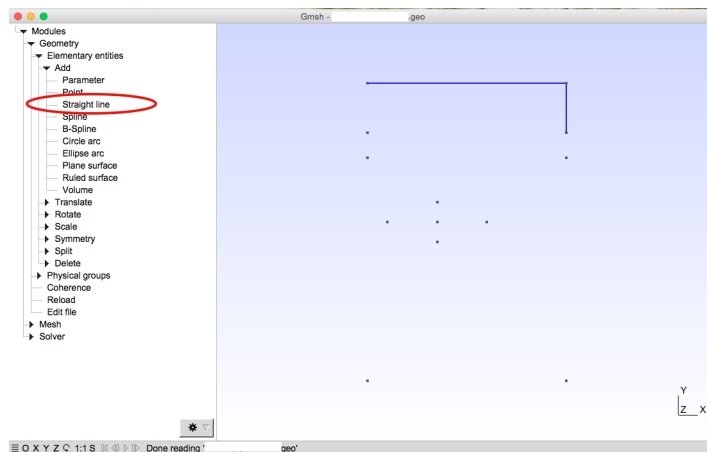

3. Define the outside of the lense. You require four connecting elliptic arcs. You will define four separate objects by selecting the elliptic point in a different order. Select Ellipse arc.

   - For the first ellipse arc select: P11, then center point P13, then right point P10 and left point P9. It will create an arc between P11 and P9.
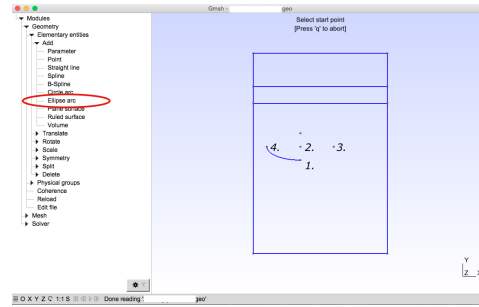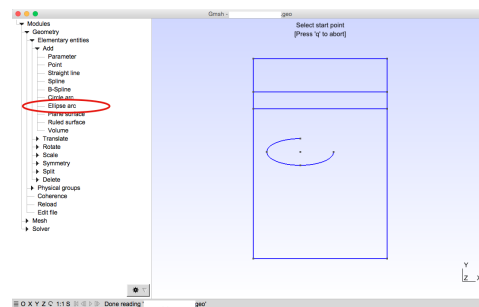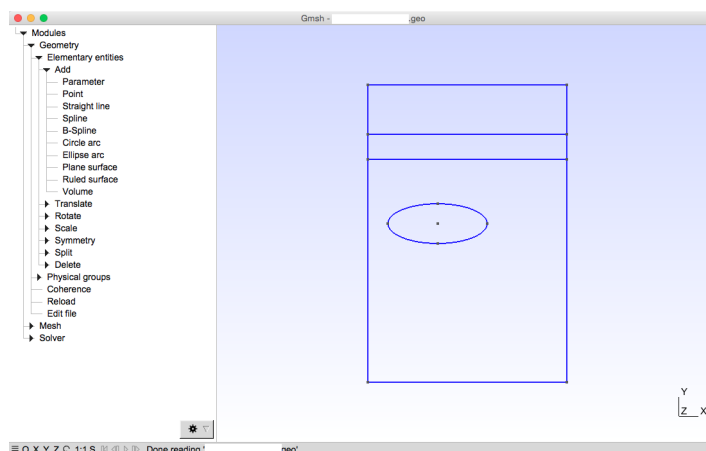
- For the second ellipse arc select: P11, then center point P13, P9 and lastly P10. This will create an arc between P11 and P10.
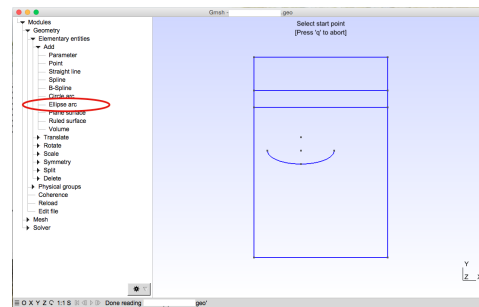
- For the third arc select: P12, then center point P13, P10 and P9. This creates an arc between P12 and P9.


- For the last arc select: P12, then center point P13, P9 and P10. This creates the last arc.


4. Next, we want to define the Plane surfaces. We will define 4 surfaces for each soil. For each surface select the bounding lines. Be careful to exclude the lense for the large soil

3. In the end you should have defined **four plane surfaces**. Each can be defined by cross appearing in the center of each geometry.
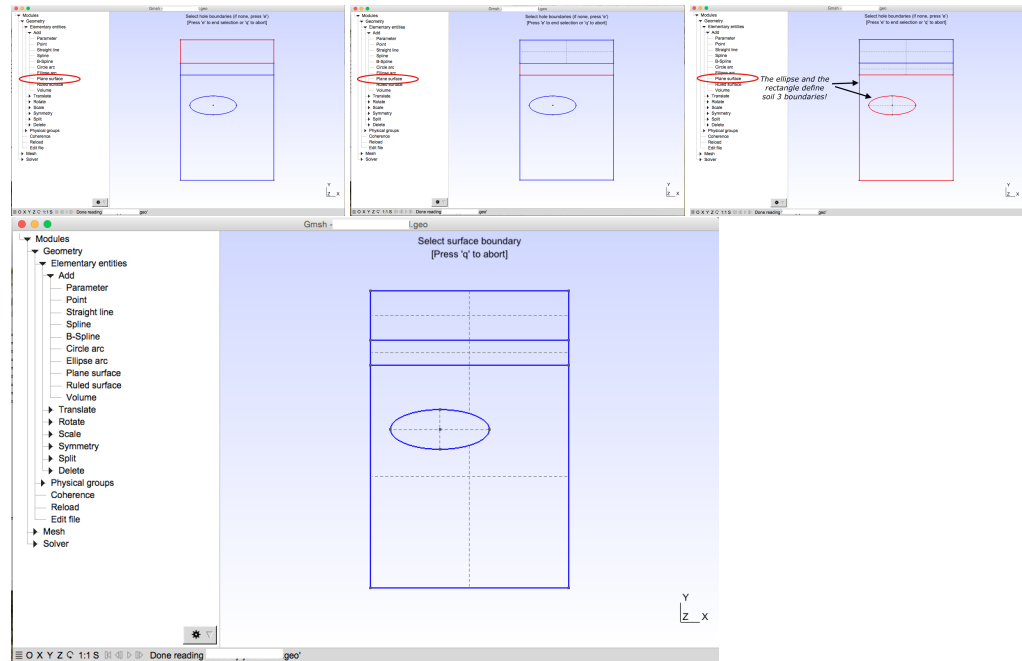

5. Next, define physical surfaces in physical groups. You need to define the surfaces before you can define the lines. This is due to how DRUtES reads the mesh. It is important to remember the numerical ID of each surface for simulations in DRUtES.
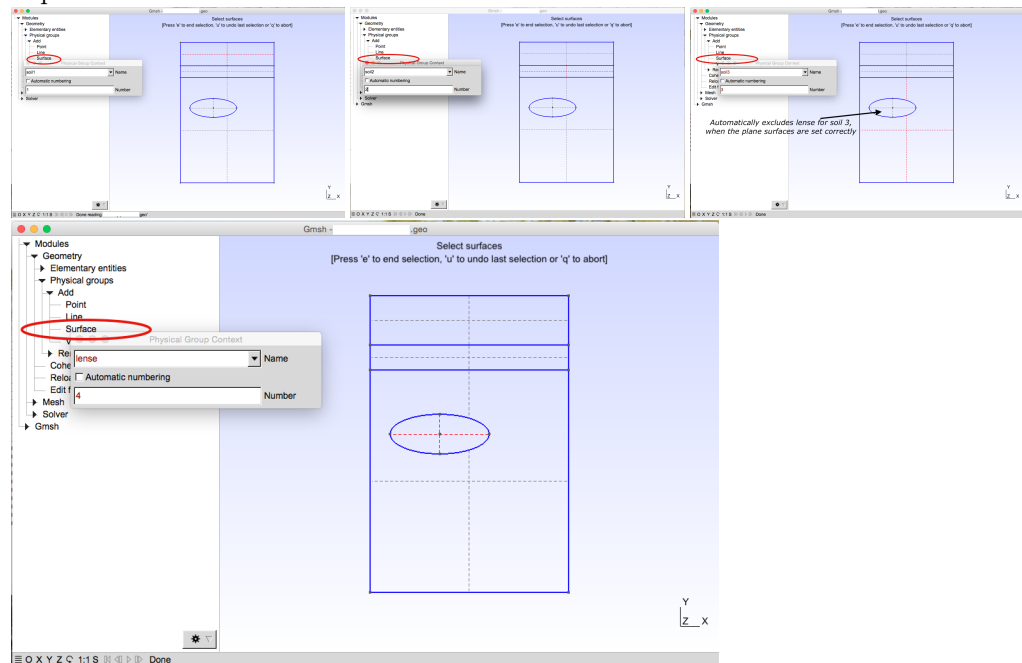
6. Define, physical lines. These define all of your boundary conditions. If the lense was impermeable material, the ellipse could also be defined with a physical line and be used as a boundary condition. We will define only three boundary conidtions: top (ID 101), bottom (ID 102) and sides (ID 103). For the sides each of the six side lines need to be selected.
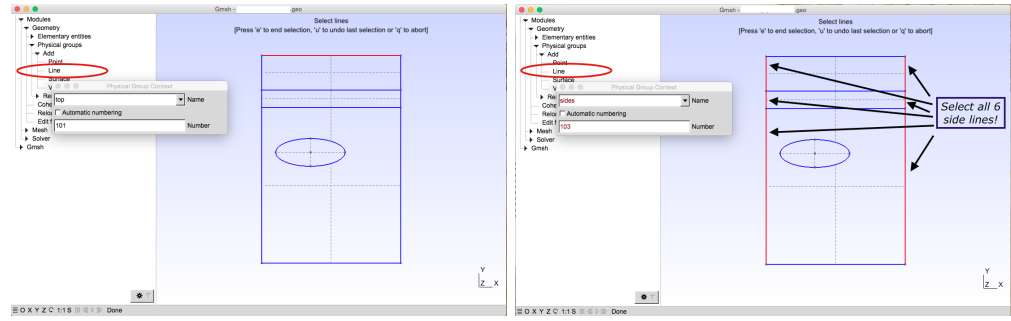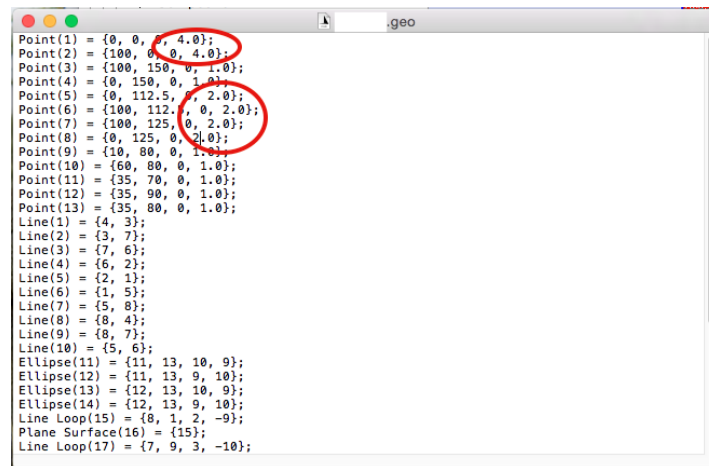
7. Now, the domain is ready to be meshed. For this, select Mesh and click on 2D.
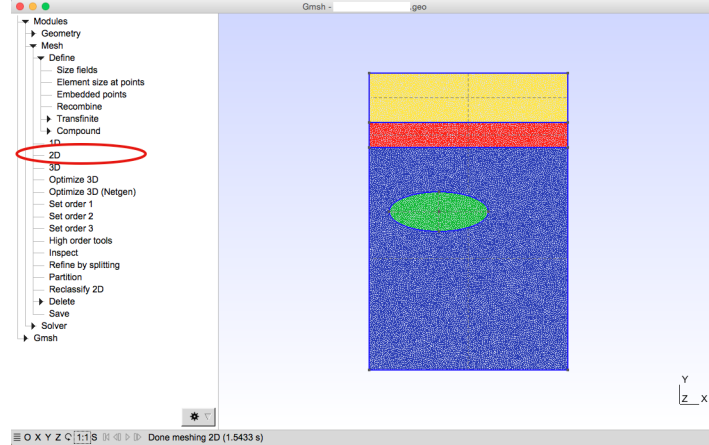

8. The mesh is again, quite fine and will be likely to take too much time. We will do two things. First, we open the gmsh file in a text editor and change the fourth number (Default is 1.0)

   (a) for Point 1 and 2 to 4.0.

   (b) for Point 5,6,7 and 8 to 2.0.

Reload and click on mesh **2D** again. This reduces the mesh to around 12500 triangles. We can see that the mesh is finest at the top at the soil-atmosphere interface and between soil 3 and the lense as we can expect a sharp interface as well. The mesh density can easily be adapted by changing the mesh density in a text editor or to define additional transfinite lines (see Example 1).

10
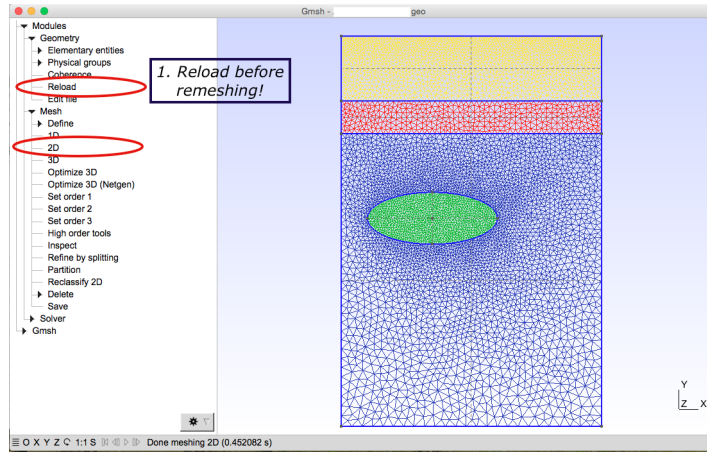
9. To use this mesh, rename it mesh.msh and place it into the mesh folder. Remember, you defined four materials. You need to define four materials in all drutes configuration file. The first line will correspond to the soil material defined as 1 etc.

4 Postprocessing with gmsh

GMSH can also be used for postprocessing. I recommend selecting gmsh as output for observation times. This is still under development and only works for pressure head output:

Make sure you are in directory drutes-dev and merge the mesh input with the gmsh output:

```
cat drutes.conf/mesh/mesh.msh out/RE_matrix_press\_head-[1-9]* > hout.msh
```

In GMSH, better output can be achieved with following settings. Select Options.

- in Post-pro consider increasing frame duration

- Untick Draw value scales horizontally

- Select the data in View in General:

  - Make the View name shorter

  - Intervals type: Select Filled iso-values

  - One can define the limits of the output in Range mode. Set to custom and change as desired

  - Make Use Adapt visualization grid is unticked

- in Axes: Change None to Simple axes