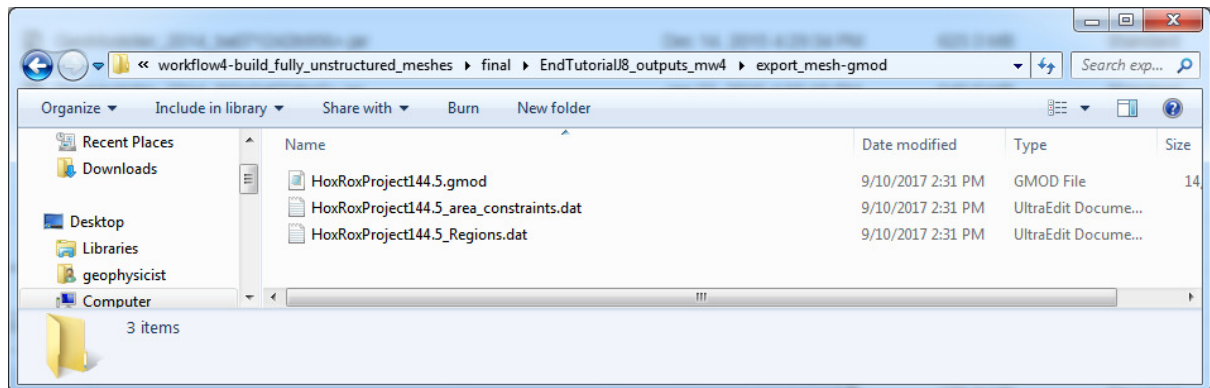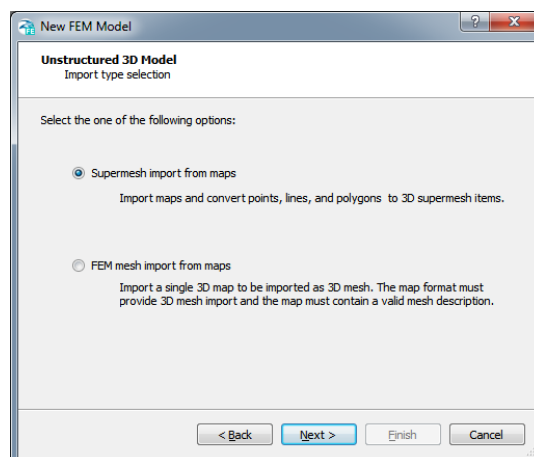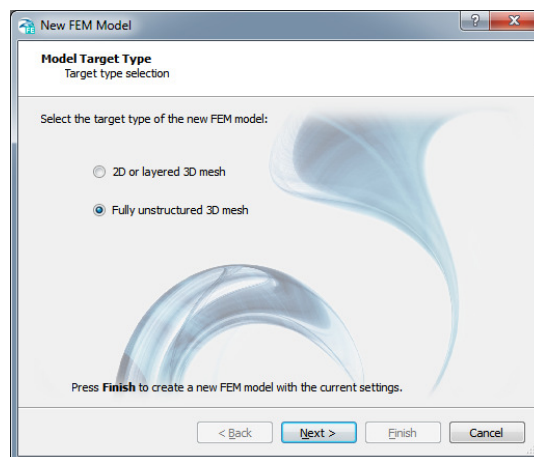# Create a FEFLOW Supermesh from a .gmod file (ASCII)

As exported from GeoModeller, let's now use the .gmod and two .dat files in FEFLOW

Quick Steps: In FEFLOW

1. File > New
2. > Fully unstructured 3D mesh (tick), next
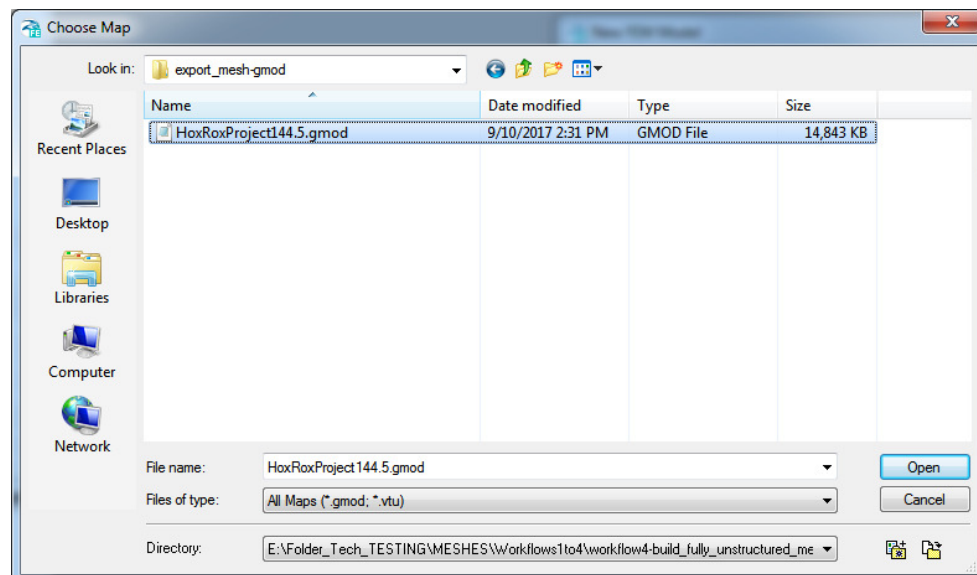
3. > Supermesh import from maps (tick), next

4. Click Add New Map (from top right)

5. Files of type: ".gmod" and then, Browse to the .gmod file

6. Open

7. Click Add New Map (from top right)

8. Files of type: ".dat" and then, Browse to the regions.dat file

9. Open

You have now Loaded items 1, and 2 (in the list below) as a + Add New Map.

10. Next, load item 2a (*area_constraints.dat) as a table linked (Define SQL) to item 1 (.gmod) *this is where you need to drop and drag the line to join them (joining "SURFACE" to TYPE")

1) • Alps_geological_model.gmod contains the native geological model in terms of structural surfaces (e.g. 3D boundaries of model regions including interfaces between geological formations and outer boundaries). All these surfaces are characterized by triangulated surfaces.

   └─ Alps_area_constraints.dat is an optional ASCII-based input file. This file contains the names of the 3D boundaries and values to constrain the element areas of the boundary triangles.

2) • Alps_Regions.dat is a *Region Marker* to mark geological units by an unique identifier. Regions are enclosed by the triangulated surfaces of the geological input model. The file contains an additional column with values to constraint the volume of tetrahedron within each geological unit.
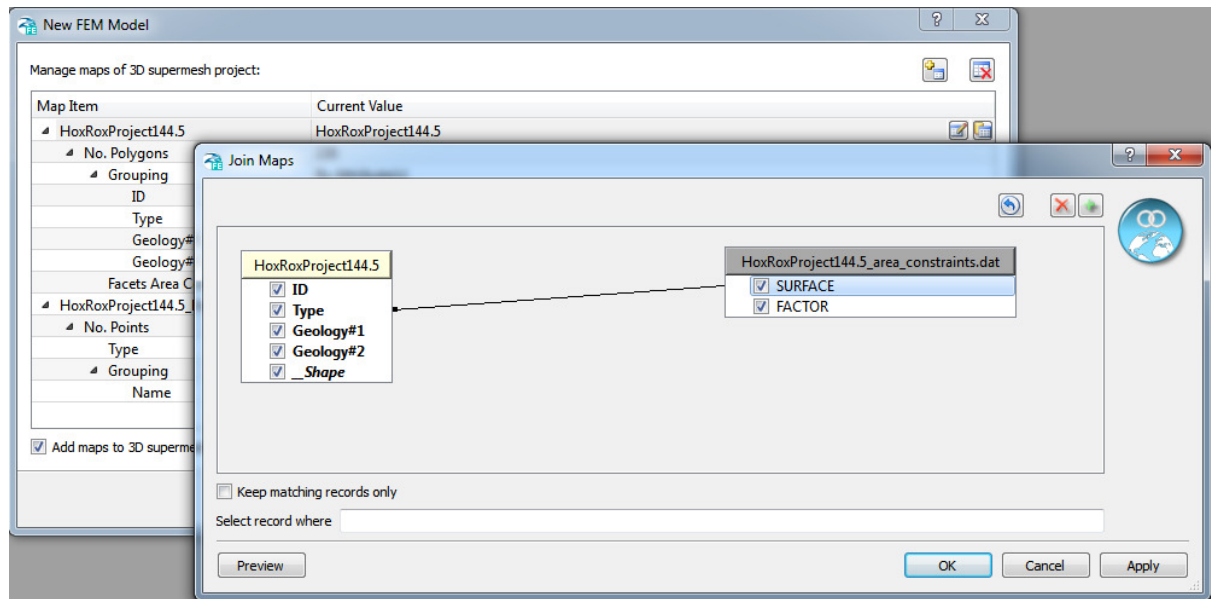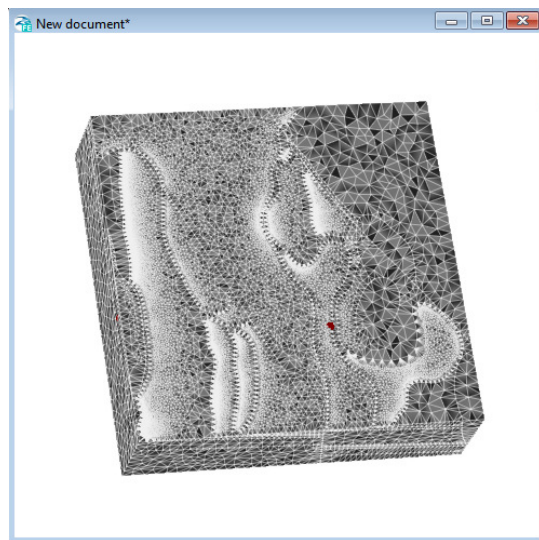
11. Once the 2 maps, and 1 table are loaded, then you need to work carefully through the table (opening tabs, etc), and changing terms and values So that the summary of parameters should eventually look like this:
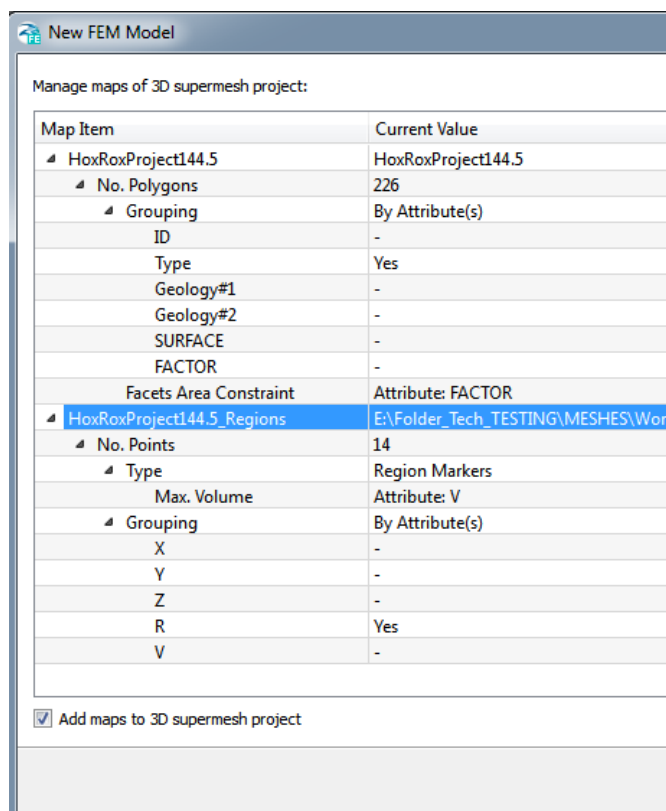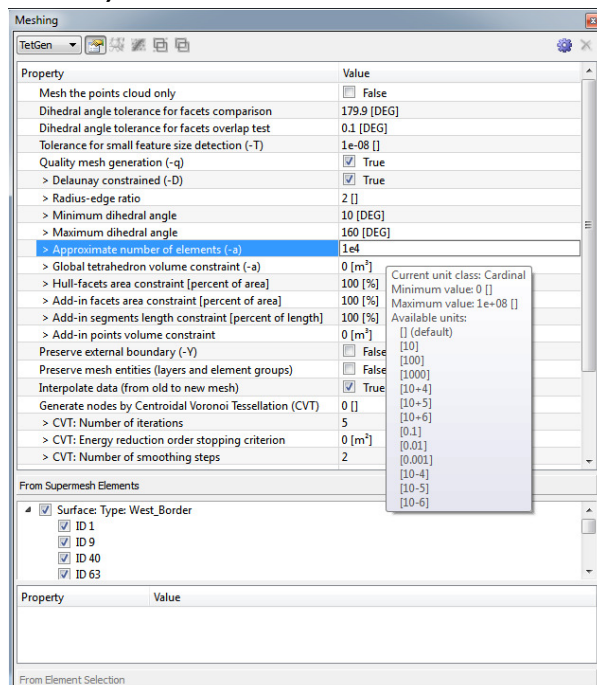

>Finish

Supermesh is done:

Last steps:
[Is to make the .fem file using the TetGen plug-in]

12. from the Main Menu: View > Panels > Meshing
-the properties table should drop down on the right-side

13. In here, choose "Approximate # Elements" and type <u>1e4,</u> and return (becomes 100000)

14. Last: Generate * (click the cog) and wait (approx 2-3 minutes).

FINISHED *Your finite element mesh file (.fem) is created, in FEFLOW, With Rocktype attributes ready