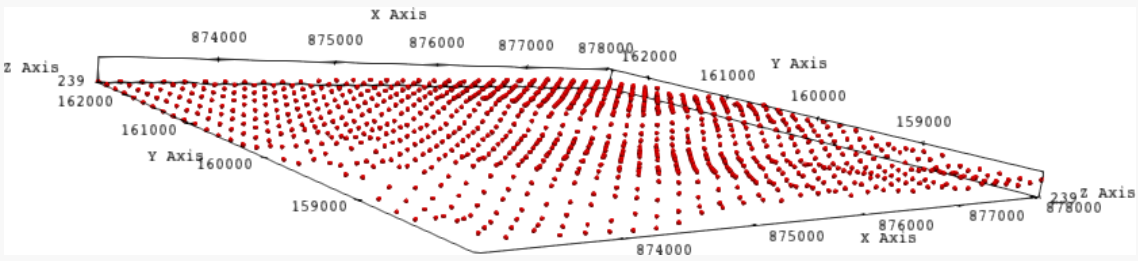
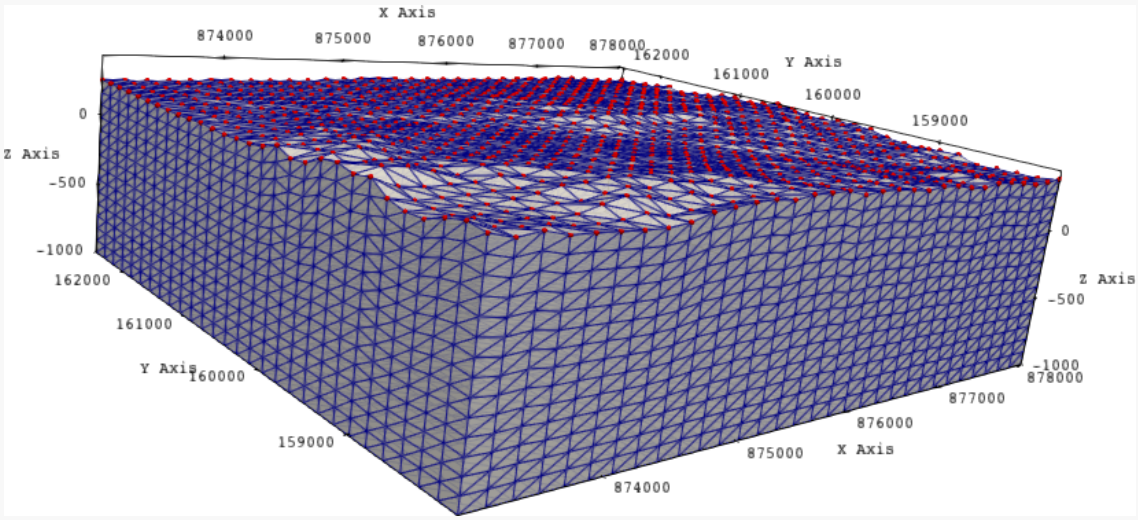
In this tutorial let us use `topIIvol_Mesher` to create a volume mesh from a point cloud cluster `./xyz/point-cloud-coarse.xyz` which contains $(x \times y) = 32 \times 29 = 928$ points. `topIIvol_Mesher` is a sequential computing tool, it will takes in a point-cloud as an input (`.xyz`) and generates volumetric meshes that can be extracted in Gmsh's `*.msh` format or medit's `*.mesh` format. Let us say we would like to create the volumeteric mesh `out-mesh.mesh`, with volume stretching upto a depth (z) of -1000 and z direction should be meshed with 15 layers.

To perform the meshing run the following in terminal:

```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 15 --depth -1000 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh mesh
```



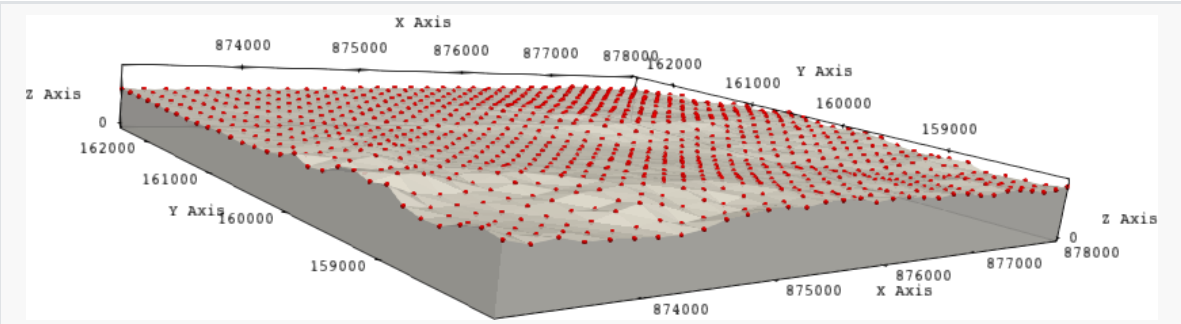
Initial point cloud with 928 data points



Volumeteric mesh generated from point cloud

Let us now create a mesh in which the volume stretches only upto -30 in z direction instead of -1000. This can be accomplished by using `--depth -30` instead of `--depth -1000`. Run the following in terminal:

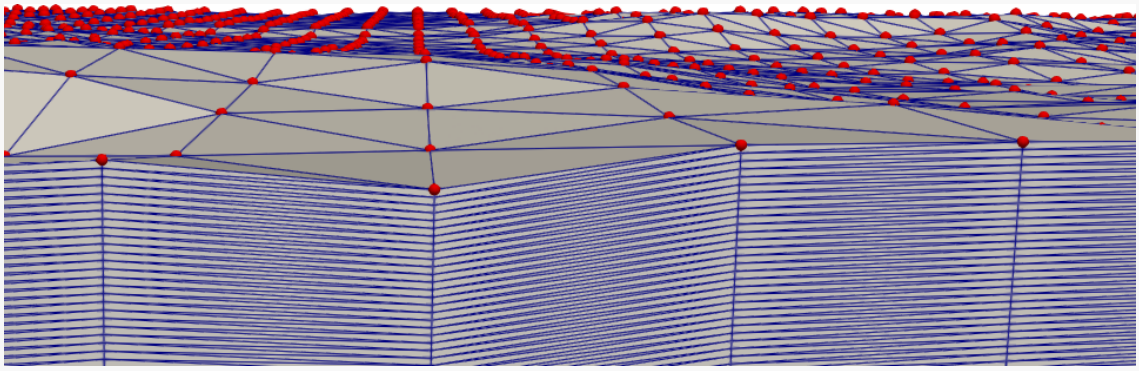
```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 15 --depth -30 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh mesh
```



Volumeteric mesh generated from point cloud with shallow depth

Additionally, let us create a volumetric mesh in which volume stretches upto -30 in z direction and contains 100 layers in z. Run the following in terminal:

```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 100 --depth -30 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh mesh
```



Volumeteric mesh generated from point cloud with fine layering

What else is there to try

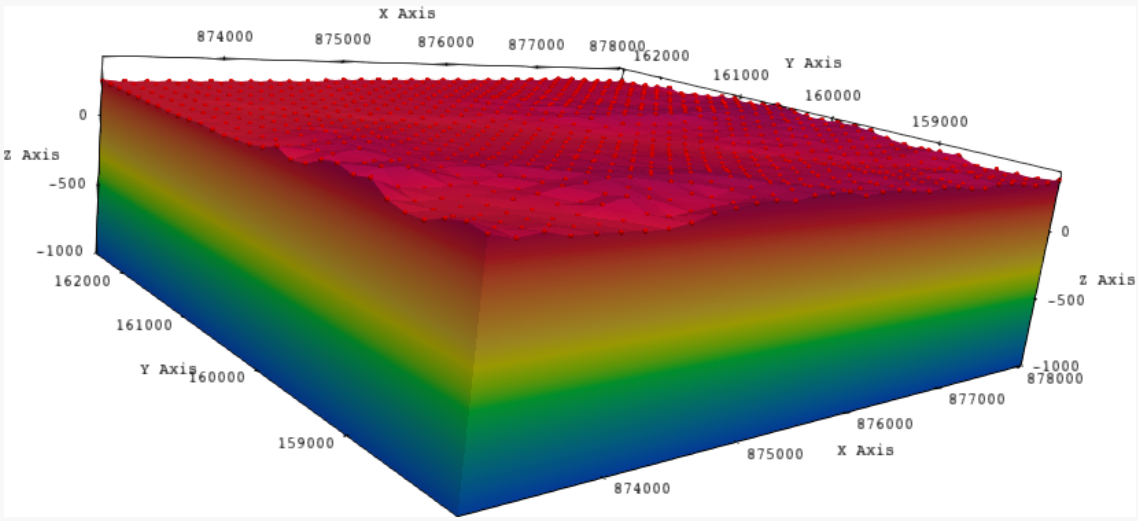
- In all the above three tutorials we saved the output mesh in `.mesh` format, to save the mesh in `.msh` format simply replace the flag `--mesh mesh` with `--mesh msh`. As such you should run

```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 15 --depth -1000 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh msh
```

```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 15 --depth -30 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh msh
```

```
topIIvol_Mesher --xpoints 32 --ypoints 29 --zpoints 100 --depth -30 \
--in ./xyz/point-cloud-coarse.xyz --out out-mesh.mesh --mesh msh
```

- You can try running these meshes with some finite element software e.g, try a simple Laplace equation with these meshes with.



Laplace solution on the mesh

To report bugs, issues, feature-requests contact:

- **mohd-afeef.badri@cea.fr**
- **mohd-afeef.badri@hotmail.com**