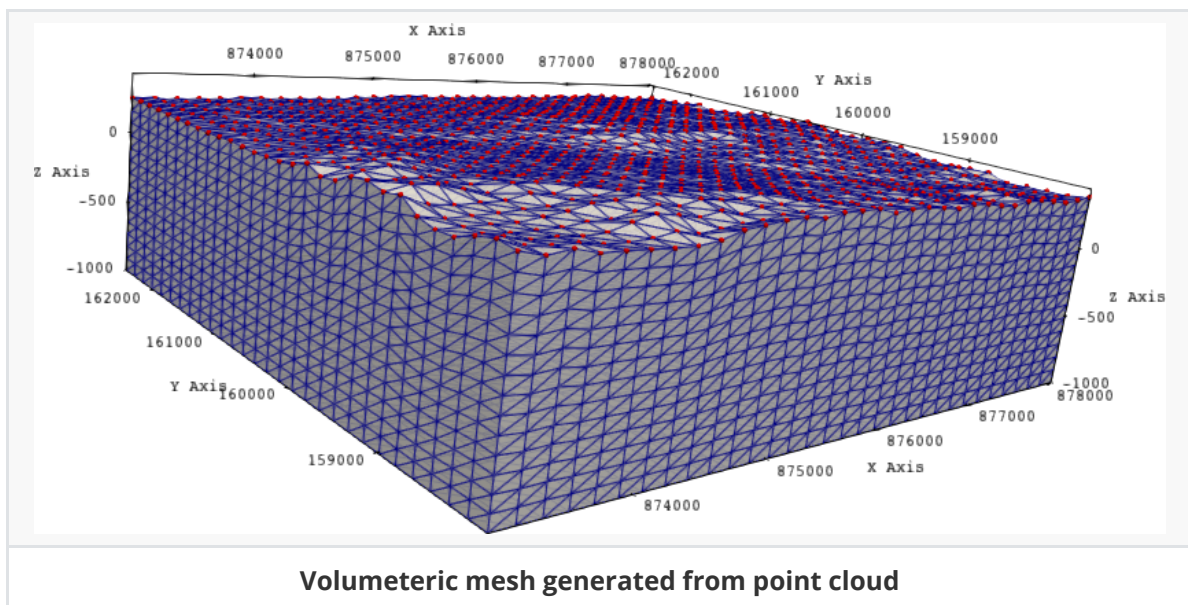
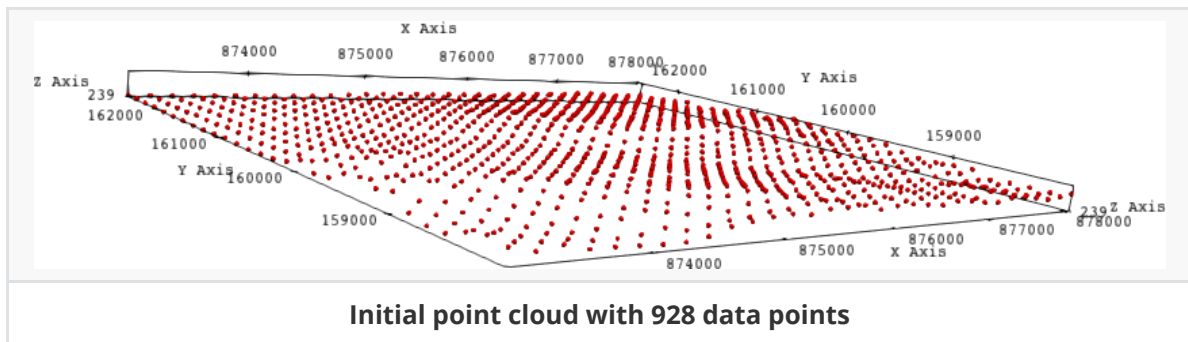


In this tutorial let us use `topIIvol_Mesher` to create a volume mesh from a point cloud cluster `./xyz/point-cloud-coarse.xyz` which contains $(x \times y) = 32 \times 29 = 928$ points.

`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 and `*.mesh` format. Let us say we would like to create the volumetric mesh `out-mesh.mesh` , with volume stretching upto a depth (z) of -1000 and z direction should be meshed with 15 layers.

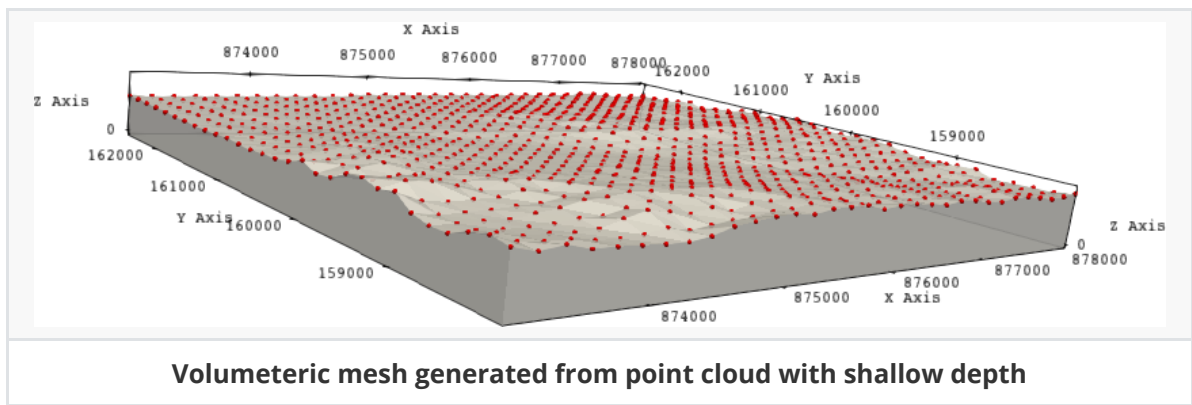
To perform the meshing run the following in terminal:

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



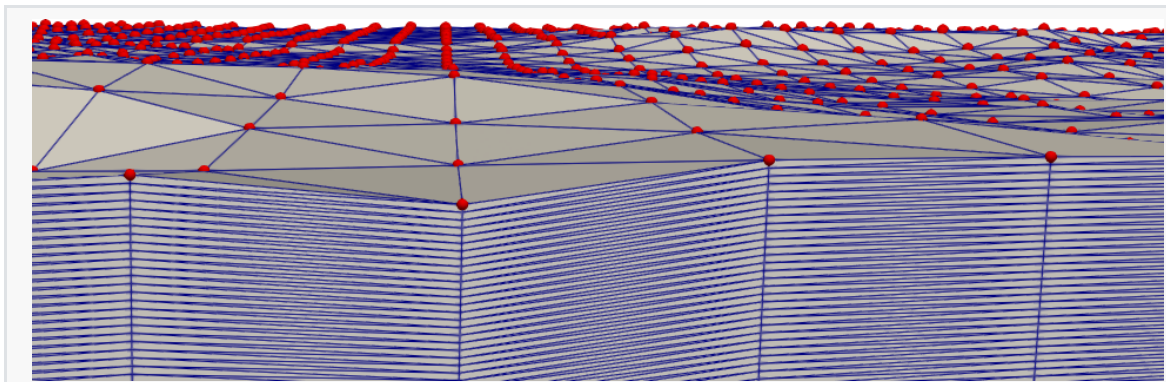
Let us now create a mesh in which the volume stretches only upto -30 in z direction instead of -1000 . This can be accomplished by using `--depth -30` instead of `--depth -1000`. Run the following in terminal:

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



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
```



What else is there to try

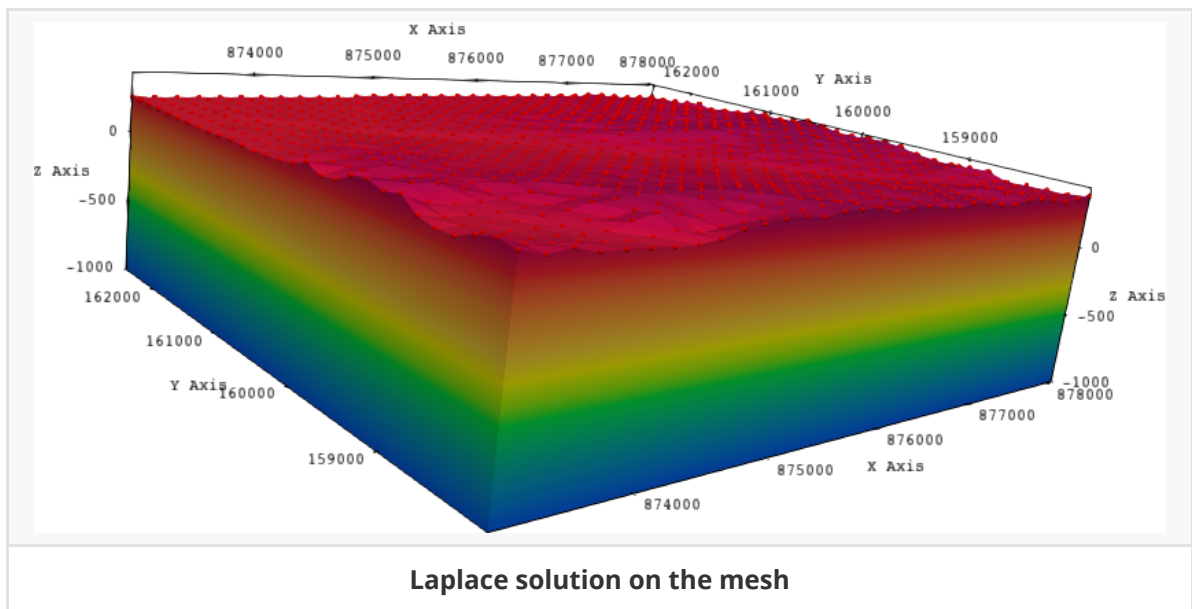
- In all the above three tutorials we saved the output mesh in `.mesh` format, to save the mesh in `.msh` format simply replace the flag `--mesh mesh` with `--mesh msh`. As such you should run

```
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.



To report bugs, issues, feature-requests contact:

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