Documentation: BHE meshing tool

About:

The BHE meshing tool is capable of creating a 3D prism mesh of the subsurface with an arbitrary number of horizontal and parallel layers, element thickness and material groups as well as an arbitrary number of BHEs, respectively line elements, with different borehole diameters. Here, the BHE center nodes are always surrounded by 6 nodes.

The tool works as follows:

1. Reading the input file
2. Creating a GMSH geometry file according to specifications made in the input file
3. Calling GMSH for creating the 2D surface mesh
4. Importing the mesh created in step 3
5. Extruding the surface mesh according to layer specifications made in the input file
6. Generating BHE elements according to specifications made in the input file
7. Writing the corresponding mesh file in OGS5 format
8. Writing the corresponding geometry file in OGS5 format

Prerequisites:

In order to use the BHE meshing tool, you'll need the binary *bhe_meshing_tool.exe* and a copy of *gmsh.exe* in the folder you are working in.

Usage:

The meshing tool is called with **bhe_meshing_tool.exe** *inputfile* for creating a complete 3D mesh. The meshing tool is called with **bhe_meshing_tool.exe** *inputfile -2D* for creating the surface 2D mesh. This is useful for the inspection of the surface mesh before it will be extruded.

For the example at hand this would read **bhe_meshing_tool.exe** *example.inp* or **bhe_meshing_tool.exe** *example.inp -2D*.

Input file: example.inp

```
WIDTH 100
LENGTH 200
DEPTH 90
BOX 50 100 50
// BOX -1 -1 -1
ELEM_SIZE 5 20
LAYER 0 10 1
LAYER 1 20 2
LAYER 2 10 4
BHE 0 -10 100 0 -50 0.063
BHE 1  10 100 0 -30 0.063
```

- WIDTH: Domain width in x-direction. Note that the model is centered in the x-direction. In the example above, the domain would extend from x=-50..50
- LENGTH: Domain length in y-direction. Note that the origin is at the front boundary. In the example above, the domain would extend from y=0..200

- DEPTH: Domain thickness in negative z-direction. Note that the surface is always at z=0, so that in the example above the domain would extend from z=0..-90. This parameter only affects the creation of the geometry file. The mesh will be extruded according to the definition of layers (c. LAYER keyword below)
- BOX: Here, a refinement box can be defined. Parameters are
  y-coordinate of the box front
  length of the box
  width of the box
  In the example above, the box would extend from x=-25..25 and y=50..150. If the model shall not contain a refinement box, the values have to be replaced with *-1*
- ELEM_SIZE: Element size at the boundary of the refinement box and the outer boundaries. If no refinement box is employed, the first value will be overrun.
- LAYER: Specification of each layer with following parameters
  material group
  number of elements in this layer
  thickness of elements in this layer
  The thickness of layer is then number of elements times element thickness. Make sure that the definition of domain depth, layers and vertical extent of BHEs match!
- BHE: Here, the BHEs are defined with following parameters
  Number of BHE (this is only for counting the BHEs, NOT the material group; the material group will be generated automatically)
  x-coordinate of BHE center
  y-coordinate of BHE center
  z-coordinate of BHE top end
  z-coordinate of BHE bottom end
  You have to make sure, that there are layers respectively nodes on the z-coordinates of BHE top and bottom ends. Otherwise, the algorithm which is generating the BHE elements will make use of the nearest node in z-direction.

<u>Output:</u>

After execution of the meshing tool, you'll find the following files in your working folder:

- *example.bhe.msh*: This is the OGS mesh file. In order to use it for running a simulation, you'll have to rename it respectively delete the *.bhe* from the filename

```
#FEM_MSH
$PCS_TYPE
NO_PCS
$NODES
50512
0 -50 0 0
1 50 0 0
2 50 200 0
3 -50 200 0
[…]
50510 -4.9577 100.738 -90
50511 4.96093 100.753 -90
```

```
$ELEMENTS
97330
0 0 pris 1092 122 503 2324 1354 1735
1 0 pris 1091 504 121 2323 1736 1353
2 0 pris 1178 387 233 2410 1619 1465
[…]
24318 0 pris 11502 11834 11973 12734 13066 13205
24319 0 pris 11503 11971 11832 12735 13203 13064
24320 1 pris 13412 12442 12823 14644 13674 14055
24321 1 pris 13411 12824 12441 14643 14056 13673
[…]
72958 1 pris 36142 36474 36613 37374 37706 37845
72959 1 pris 36143 36611 36472 37375 37843 37704
72960 2 pris 38052 37082 37463 39284 38314 38695
72961 2 pris 38051 37464 37081 39283 38696 38313
72962 2 pris 38138 37347 37193 39370 38579 38425
[…]
97278 2 pris 48462 48794 48933 49694 50026 50165
97279 2 pris 48463 48931 48792 49695 50163 50024
97280 3 line 8 1240
97281 3 line 1240 2472
[…]
97308 3 line 34504 35736
97309 3 line 35736 36968
97310 4 line 15 1247
97311 4 line 1247 2479
[…]
97329 4 line 23423 24655
#STOP
```

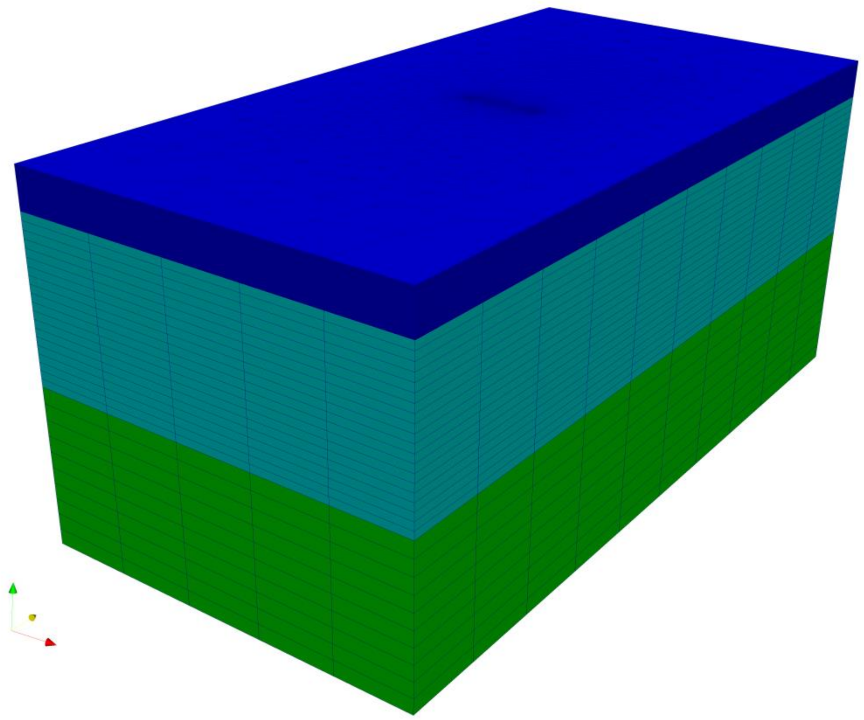

Fig. 1: 3D mesh

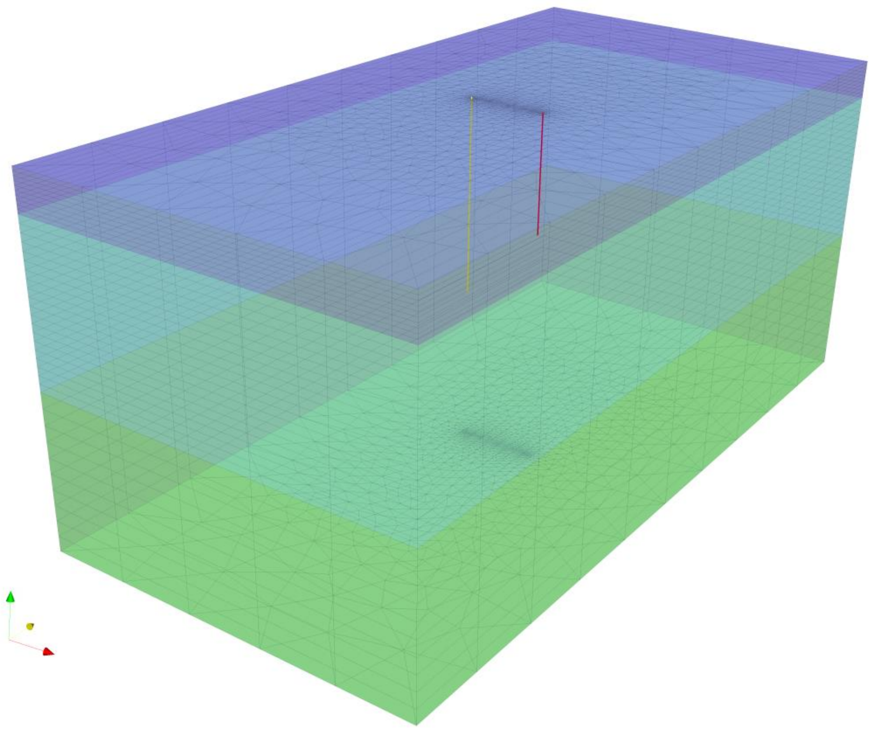

Fig. 2: 3D mesh, line elements inside

- *example.geo*: This is the geometry input file for GMSH
- *example.gli*: This is the OGS geometry file. The six surfaces of the cube are named *top, bottom, left, right, inflow, outflow*. Important definitions for the BHEs are the top and bottom points as well as the polylines, which are named in a systematic manner:

```
#POINTS
0 -50 0 0
1 50 0 0
[…]
8 -10 100 0 $NAME BHE0_top
9 -10 100 -50 $NAME BHE0_bottom
10 10 100 0 $NAME BHE1_top
11 10 100 -30 $NAME BHE1_bottom
[…]
#POLYLINE
$NAME
ply_BHE0
$POINTS
8
9
#POLYLINE
$NAME
ply_BHE1
$POINTS
10
11
[…]
#STOP
```

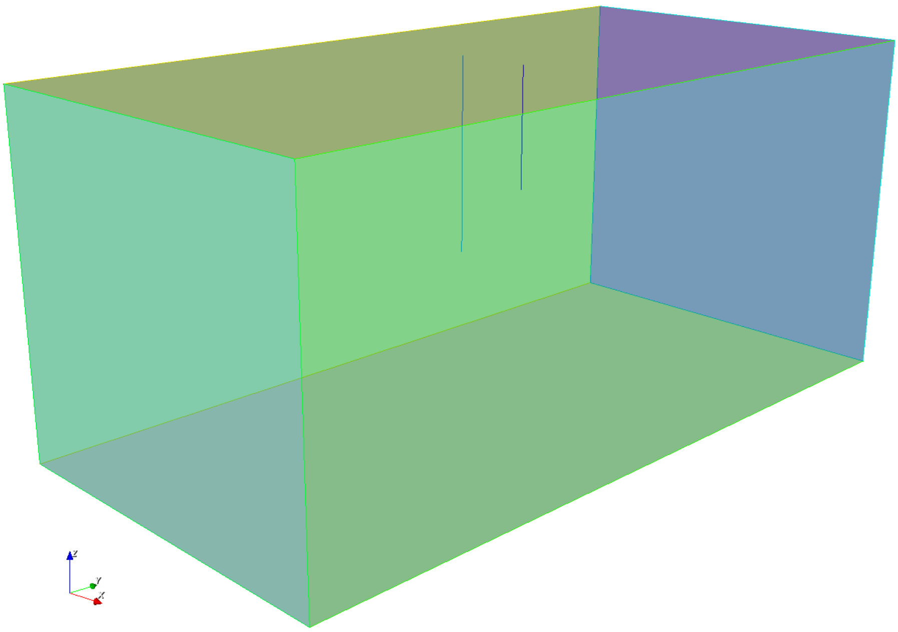

Fig. 3: Geometry

- *example.msh*: This is the 2D surface mesh generated by GMSH. Note that this is NOT the OGS mesh file!

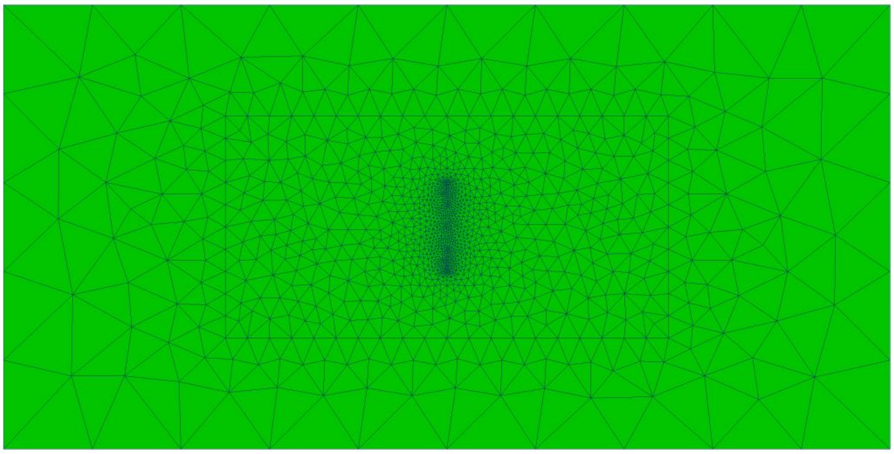

Fig. 4: Surface mesh