I started by creating the Cube by defining the points around and lines between them to then use the cube[] thing from gmsh:

```
cl1 = 10.0;
Point(1) = {-150,-150,-150,cl1};
Point(2) = {150,-150,-150,cl1};
Point(3) = {-150,150,-150,cl1};
Point(4) = {150,150,-150,cl1};

Line(1) = {1,2};
Line(2) = {3,4};
Line(3) = {1,3};
Line(4) = {2,4};

Line Loop(1)={1,4,-2,-3};
Plane Surface(1) = {1};



//extrude
cube[] = Extrude {0,0,300} {Surface{1};};
Delete{Volume{cube[1]};}
```

Before actually drawing the cube, I had to declare the outter coil points:

```
cl2 = 5.0;

Point(5) = {35,5,-50,cl2};
Point(6) = {45,5,-50,cl2};
Point(7) = {45,45,-50,cl2};
Point(8) = {-45,45,-50,cl2};
Point(9) = {-45,-45,-50,cl2};
Point(10) = {45,-45,-50,cl2};
Point(11) = {45,-5,-50,cl2};
Point(12) = {35,-5,-50,cl2};
Point(13) = {35,-35,-50,cl2};
Point(14) = {-35,-35,-50,cl2};
Point(15) = {-35,35,-50,cl2};
Point(16) = {35,35,-50,cl2};

Line(5) = {5,6};
Line(6) = {6,7};
Line(7) = {7,8};
Line(8) = {8,9};
Line(9) = {9,10};
Line(10) = {10,11};
Line(11) = {11,12};
Line(12) = {12,13};
Line(13) = {13,14};
Line(14) = {14,15};
Line(15) = {15,16};
Line(16) = {16,5};
```

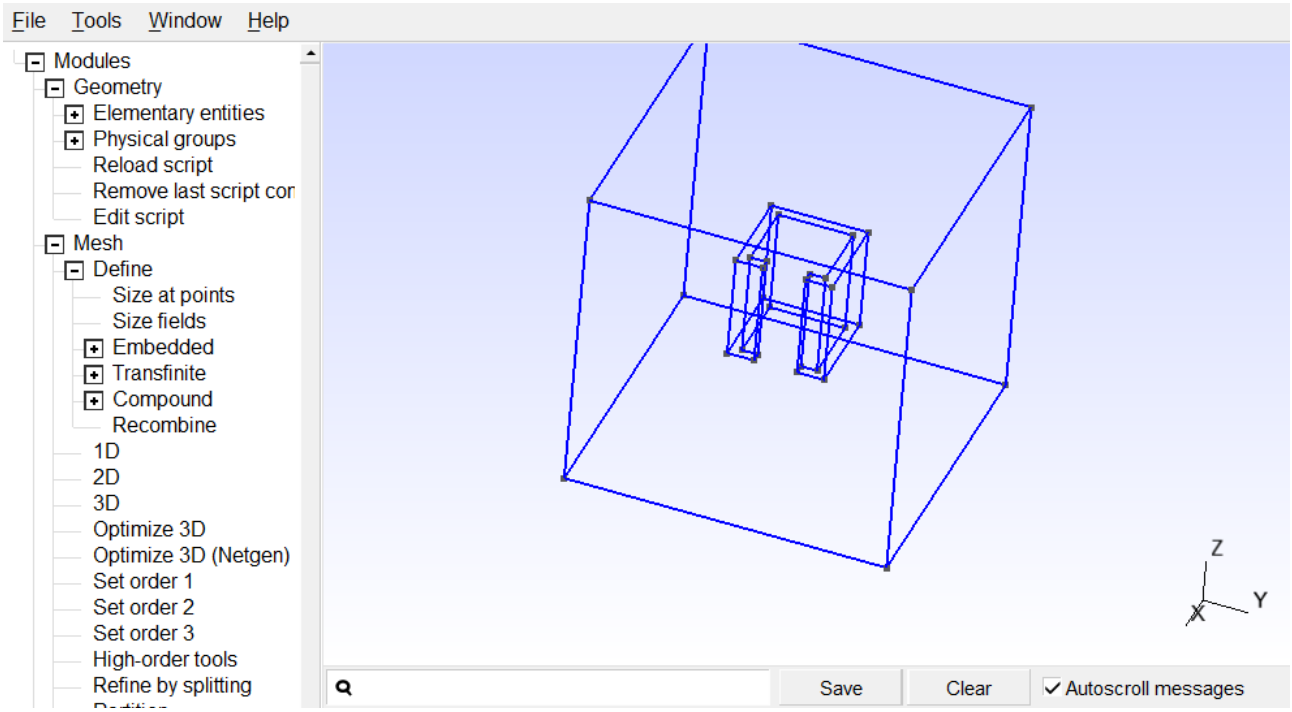
```
coil[] = Extrude {0,0,100} {Surface{2};};
Delete{Volume{coil[1]};}
```

Finally, we can create surface loops in order to get the volumes:

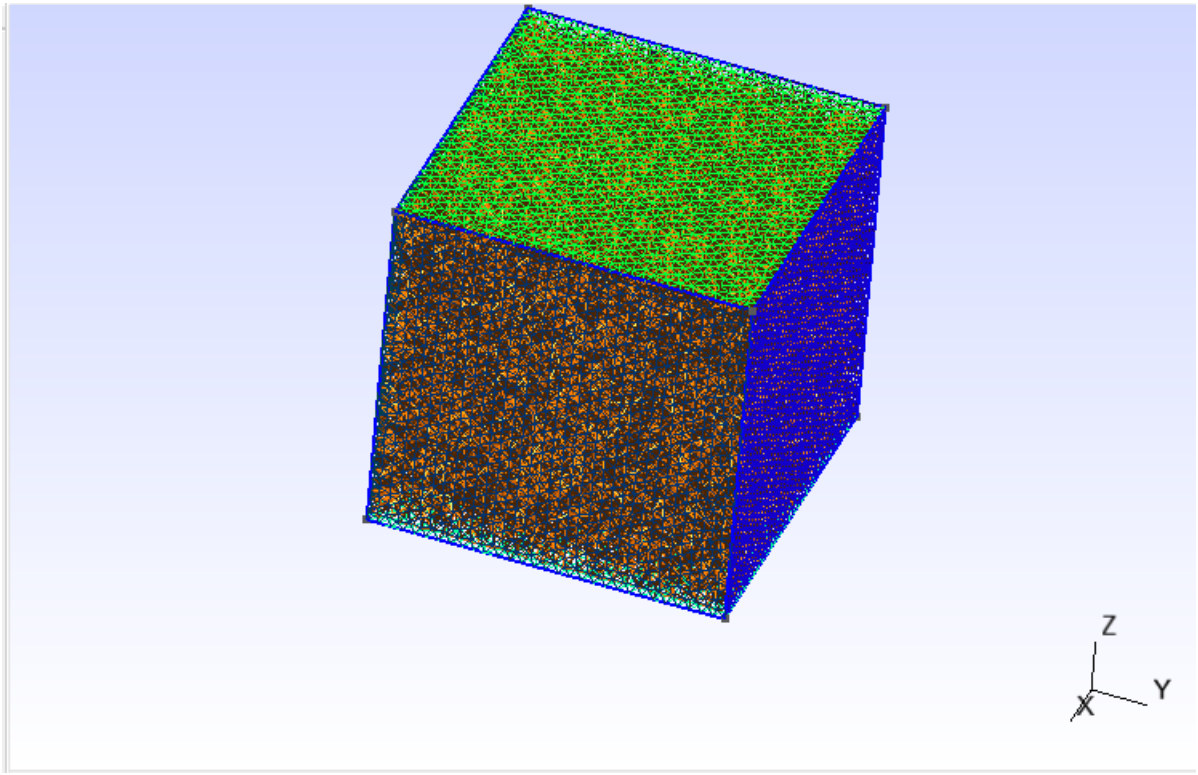
```
Surface Loop(1) = {1, cube[0], cube[2], cube[3], cube[4], cube[5]};
Surface Loop(2) = {2, coil[0], coil[2], coil[3], coil[4], coil[5], coil[6], coil[7], coil[8], coil[9], coil[10], coil[11], coil[12], coil[13]};

// DEFINE PHYSICAL VOLUMES
Volume(100) = {2};
Volume(200) = {1,2};
```

Opening this with gmsh filename.geo:



Click on 3D Mesh:



After executing dolfin-convert and the python script, I can open the files in paraview: