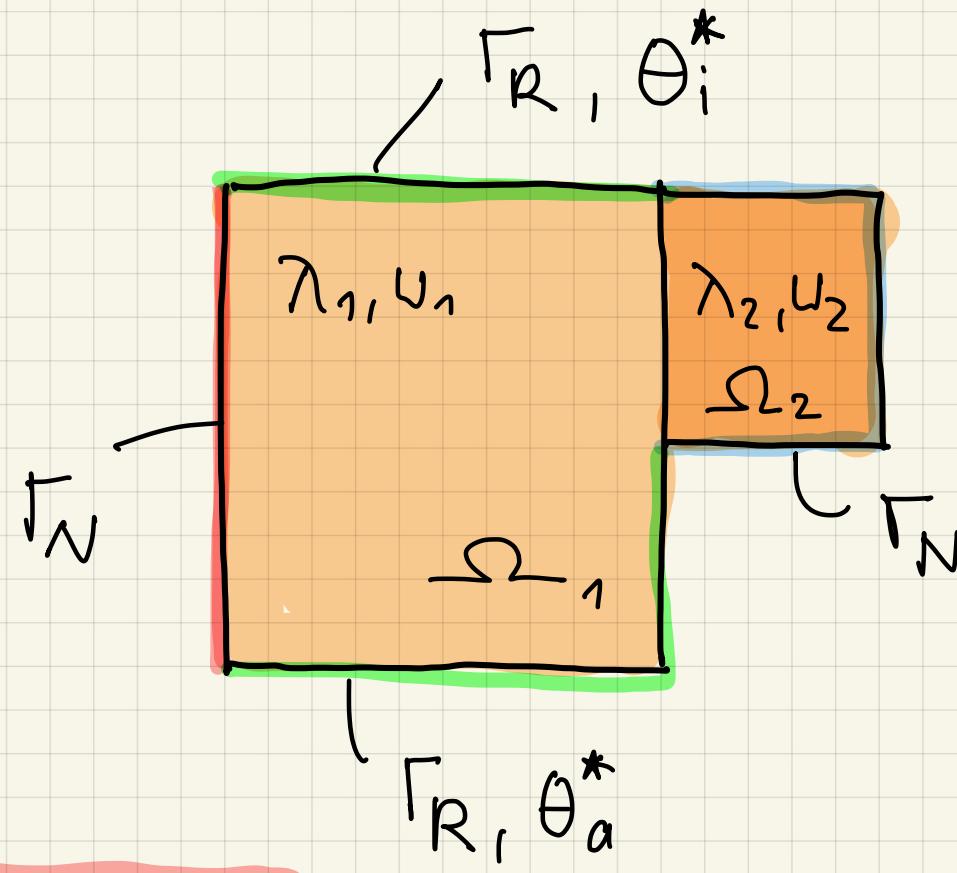


FEM for 2D problems

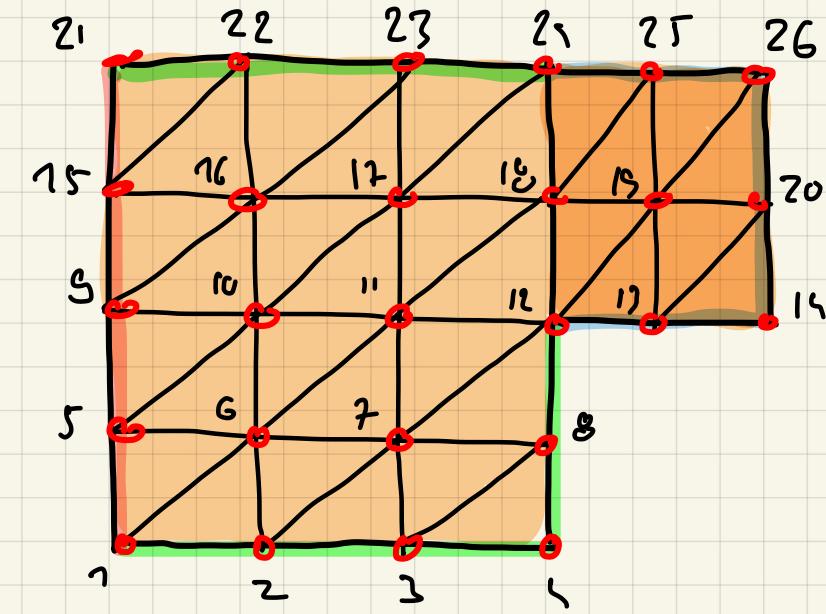
Mesh generation

# Objective



We have

- Domains } with different properties
- Boundaries }

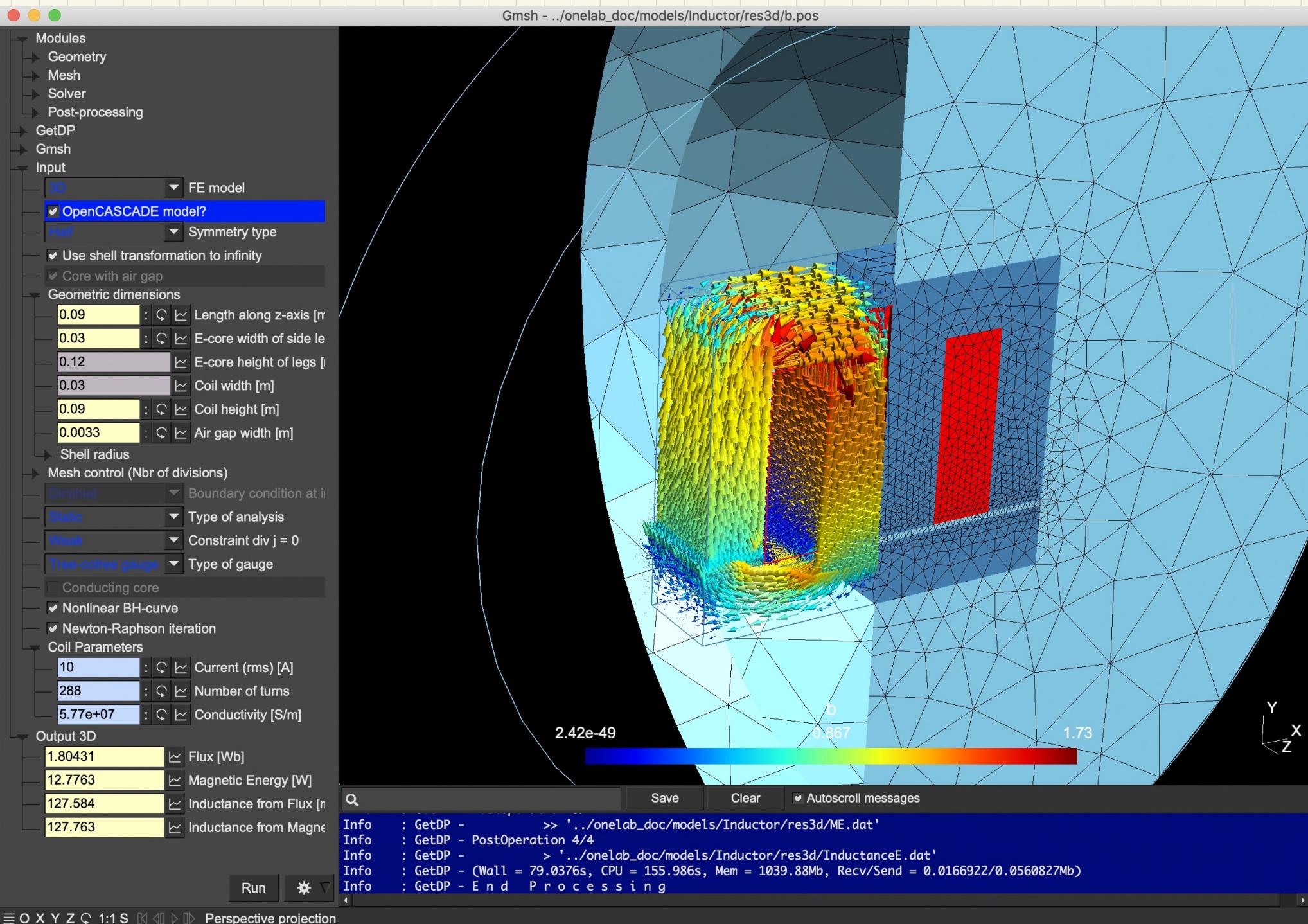


We need

- Nodes
  - Connectivity
    - Triangles
    - Lines on boundary
- (Robin-BCs)

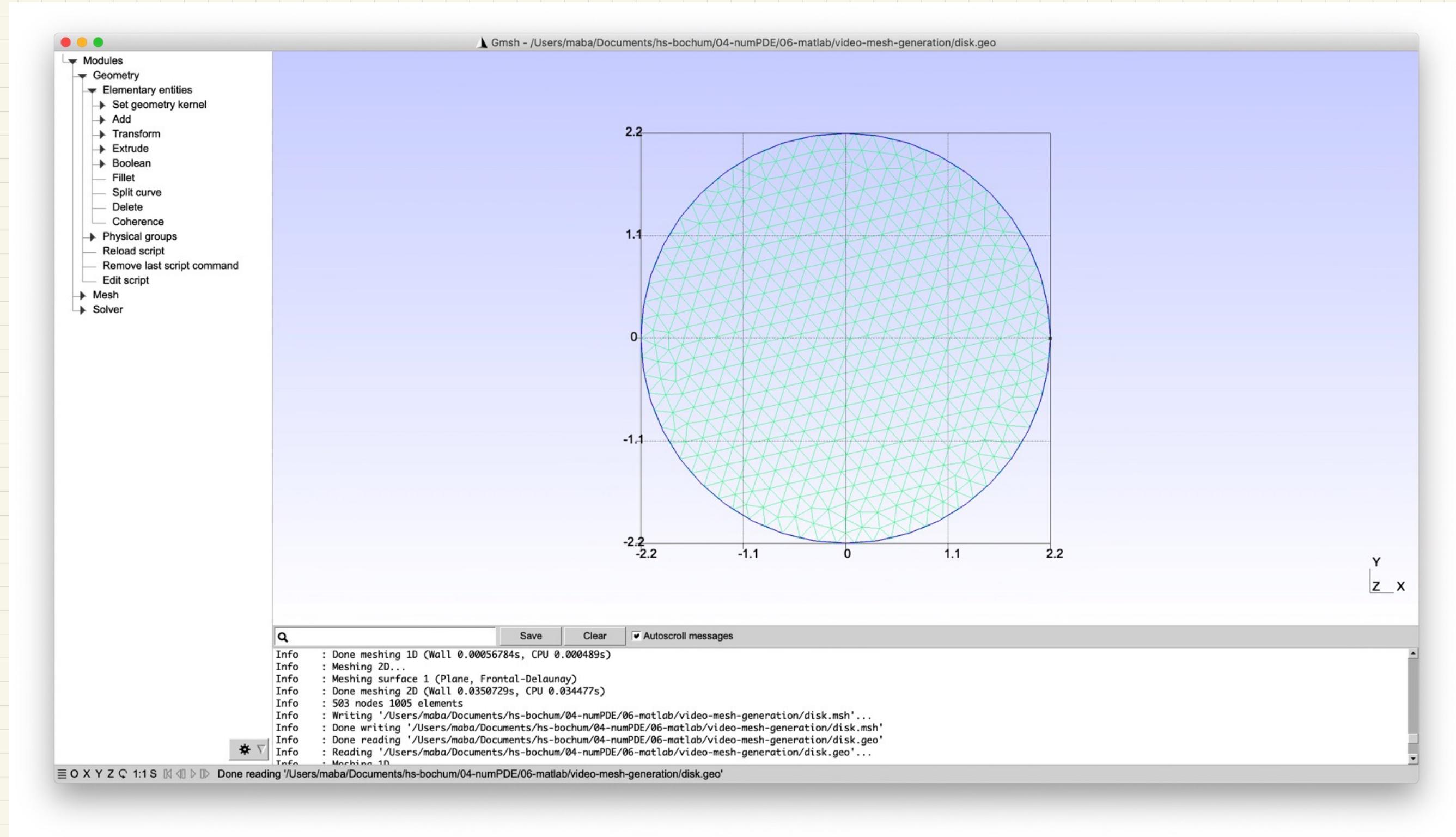
} What belongs where?

# Gmsh



- Powerful 2D/3D mesh generator + postprocessor
- Since 1997 — mature but a bit tricky to use

# Demo : Disk



# Import to Matlab

```
M = readmsh('file');
```

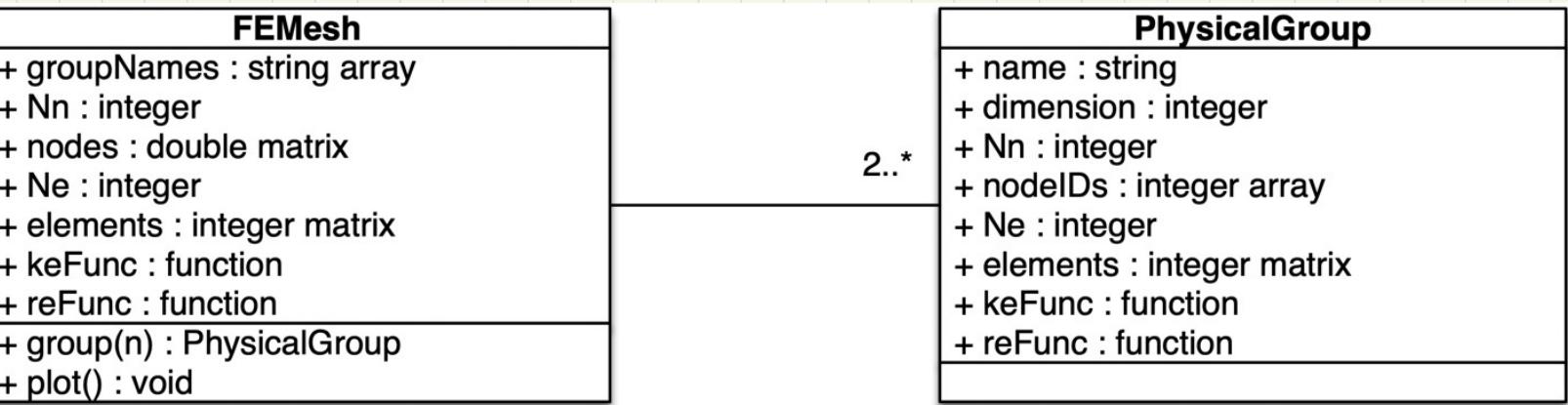
M is an FEMesh object with

M.nodes  
M.elements  
M.keFunc  
M.reFunc

} As before

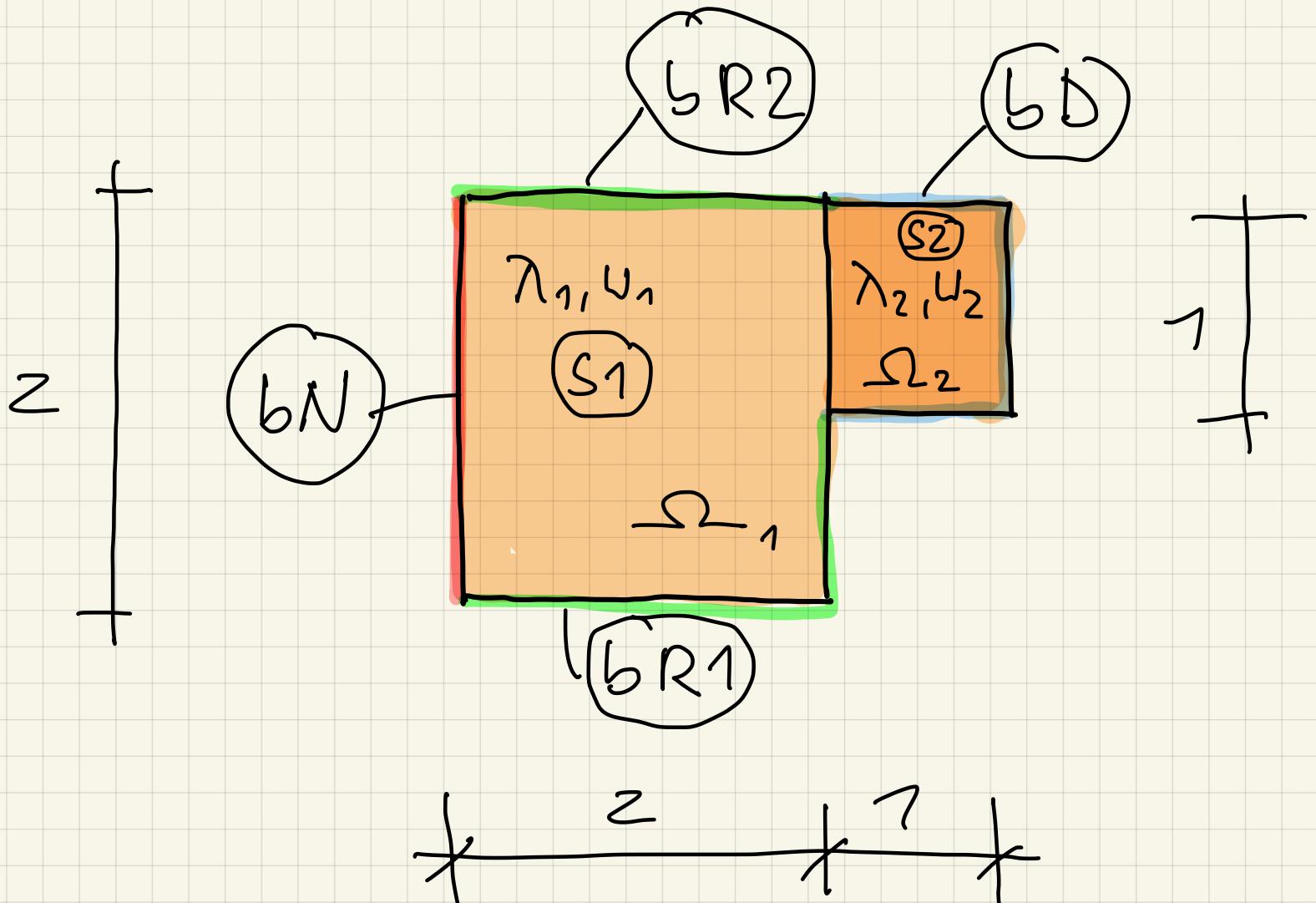
and Physical groups (surfaces, edges, user defined)

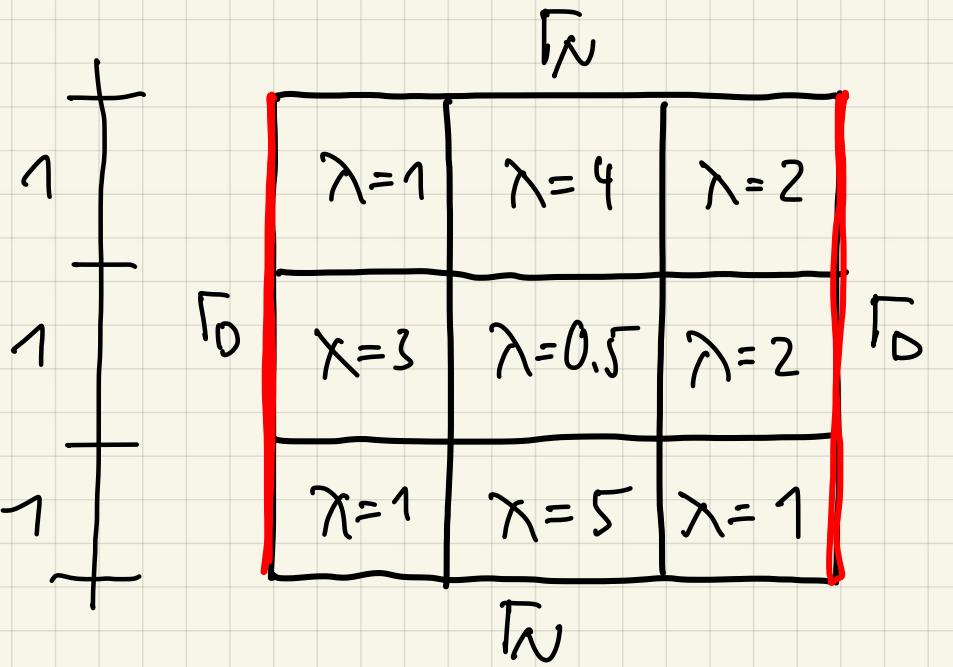
P.nodeIDs  
P.elements  
P.keFunc  
P.reFunc



# Composed geometry

O - Physical group





$$+ \frac{1}{1} \mid \frac{1}{1} \mid \frac{1}{1} +$$

$\omega = 1$  überall