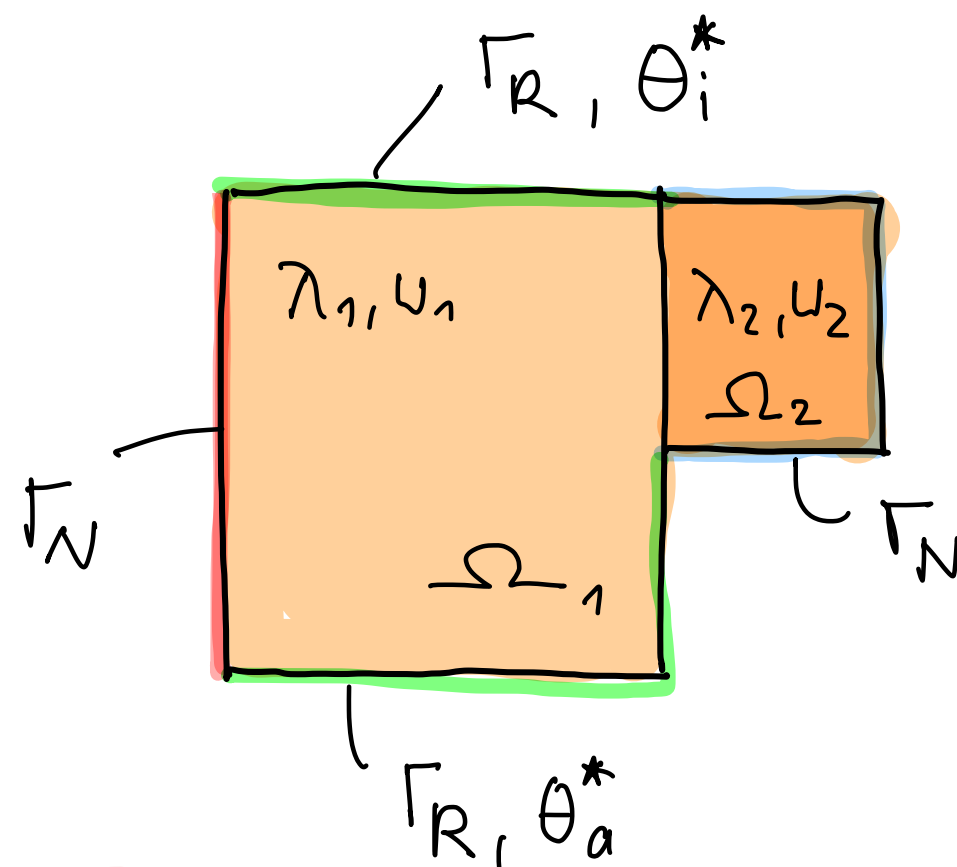


FEM for 2D problems

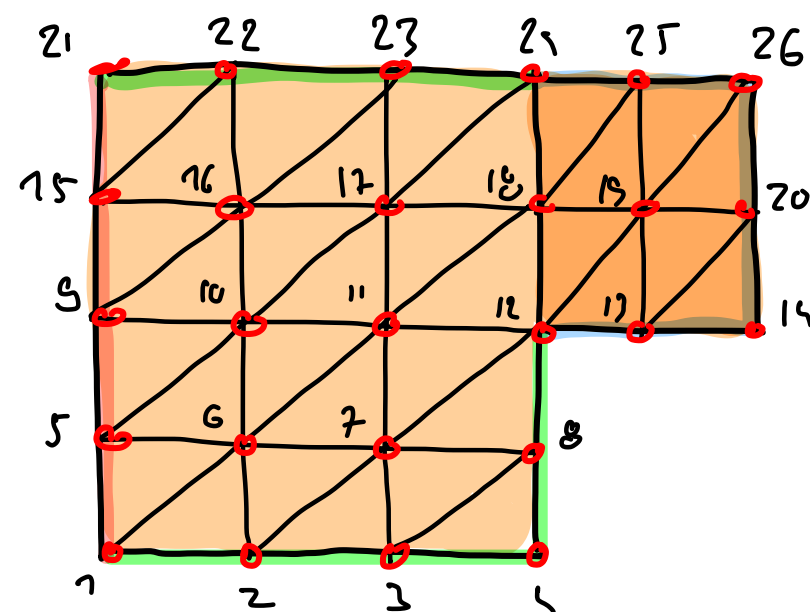
Mesh generation

Objective



We have

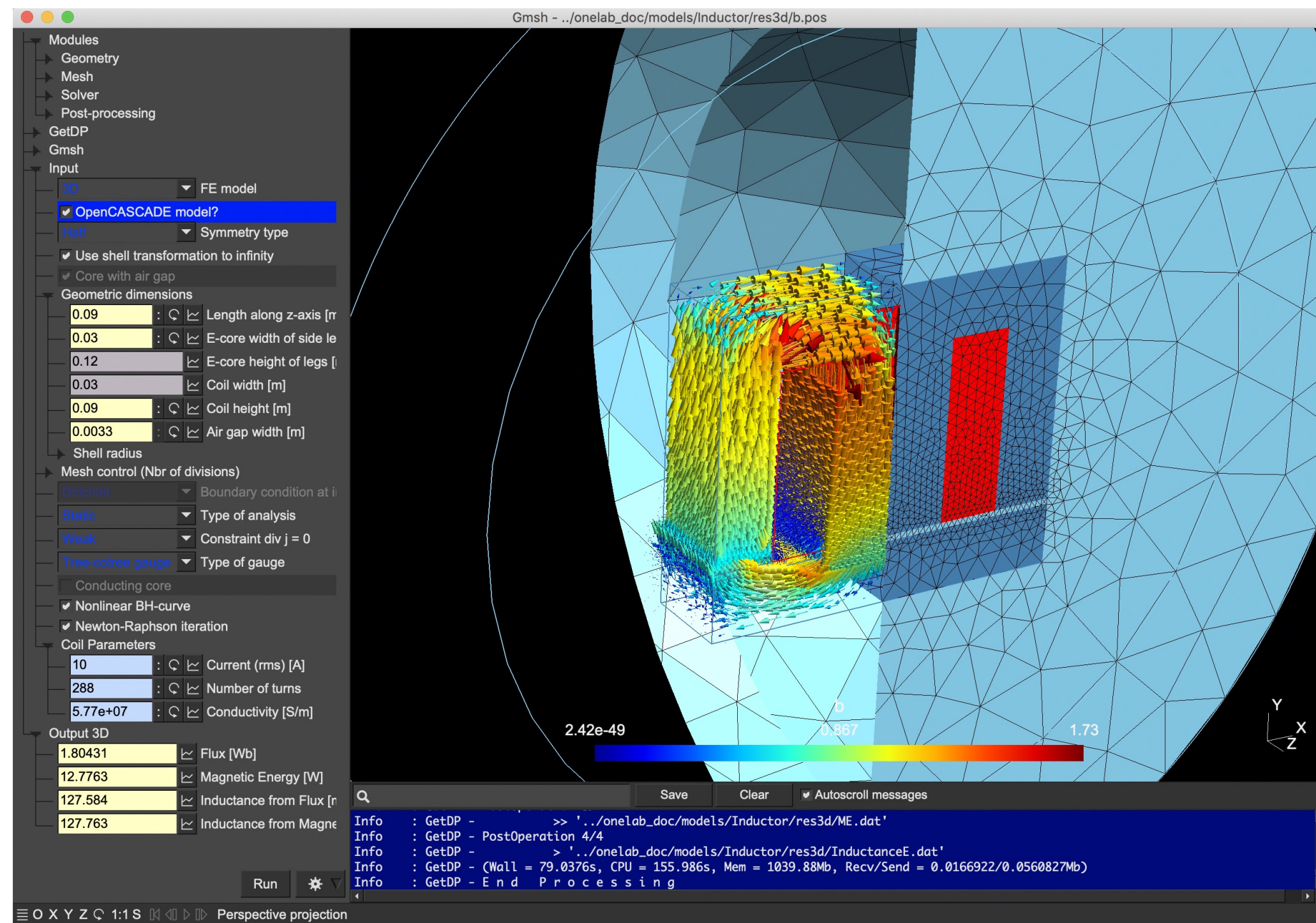
- Domain
 - Boundaries
- } With different properties



We need

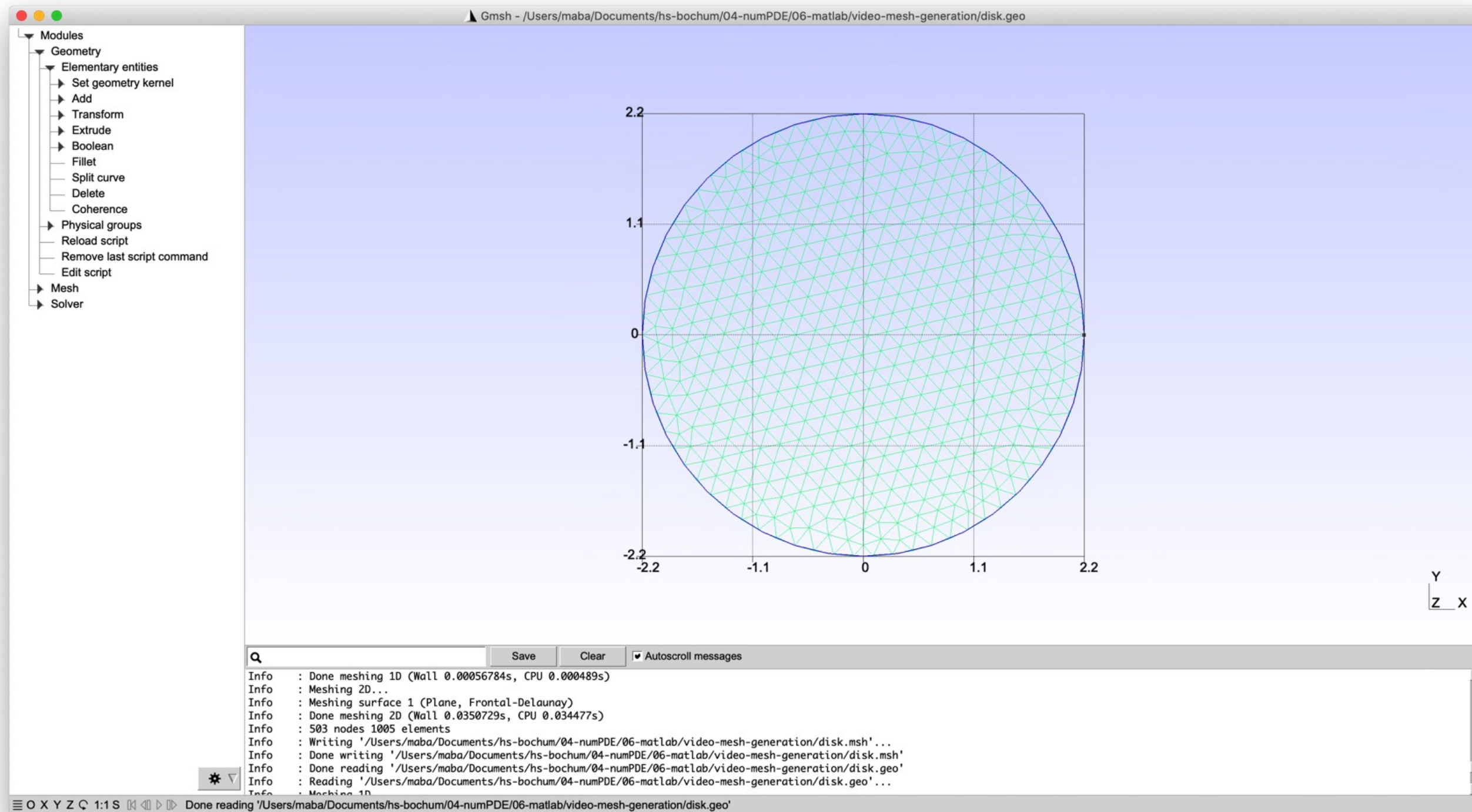
- Nodes
 - Connectivity
 - Triangles
 - Lines on boundary (Robin-BCs)
- } What belongs where?

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	}	As before
m.elements		
m.keFunc		
m.reFunc		

and Physical groups (surfaces, edges, user defined)

p.nodeIDs
p.elements
p.keFunc
p.reFunc

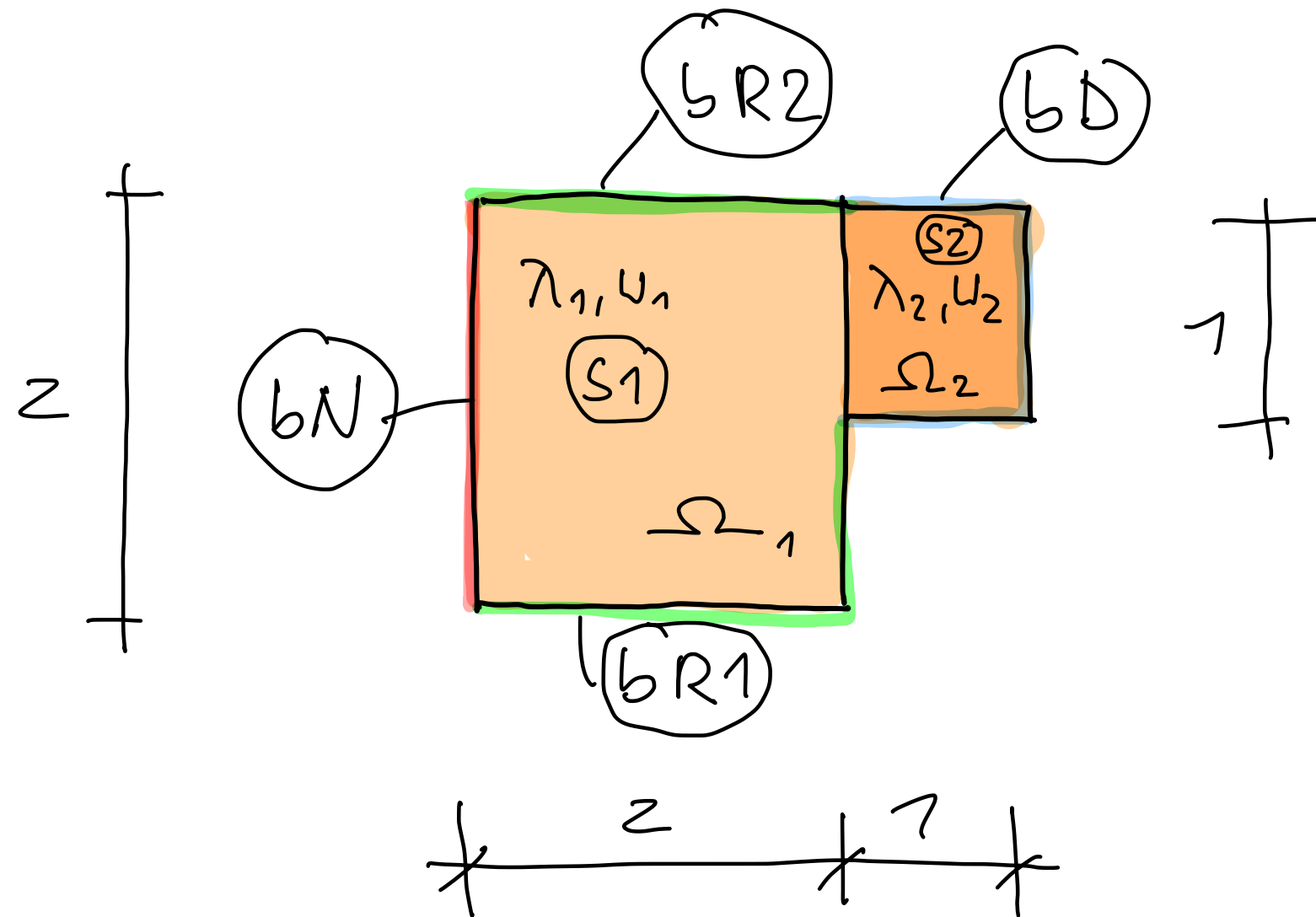
FEMesh
+ groupNames : string array
+ Nn : integer
+ nodes : double matrix
+ Ne : integer
+ elements : integer matrix
+ keFunc : function
+ reFunc : function
+ group(n) : PhysicalGroup
+ plot() : void

2..*

PhysicalGroup
+ name : string
+ dimension : integer
+ Nn : integer
+ nodeIDs : integer array
+ Ne : integer
+ elements : integer matrix
+ keFunc : function
+ reFunc : function

Composed geometry

○ - Physical group



		Γ_N			
1	Γ_D	$\lambda=1$	$\lambda=4$	$\lambda=2$	Γ_D
1		$\lambda=3$	$\lambda=0.5$	$\lambda=2$	
1		$\lambda=1$	$\lambda=5$	$\lambda=1$	
		Γ_N			
		1	1	1	

$\omega = 1$ überall