

/home/julian/OpenFOAM
/leia2/src/vof/foamGeometry.H



```
graph TD; A["/home/julian/OpenFOAM  
/leia2/src/vof/foamGeometry.H"] --> B["fvMesh.H"]; A --> C["surfaceFields.H"]; A --> D["volFields.H"]; A --> E["vector"]; A --> F["set"]; A --> G["algorithm"]; A --> H["sigFpe.H"];
```

fvMesh.H

surfaceFields.H

volFields.H

vector

set

algorithm

sigFpe.H