

builtin:
7_19.OpenFOAM

Information Properties

Properties

Apply Reset Delete ?

Search ... (use Esc to clear text)

- Cache Mesh
- Include Sets Groups Only
- Include Zones Patch Names
- Interpolate volFields Extrapolate Patch

Update GUI

Use VTKPolyhedron

- Mesh Parts
 - AMI1 - patch
 - AMI2 - patch
 - atmosphere - patch
 - bottom - patch
 - center - patch
 - inlet - patch
 - outlet - patch
 - outside - patch
 - amicylinder - cellSet
 - amicylinder - cellZone

- Volume Fields
 - k
 - nut
 - omega
 - p
 - U

