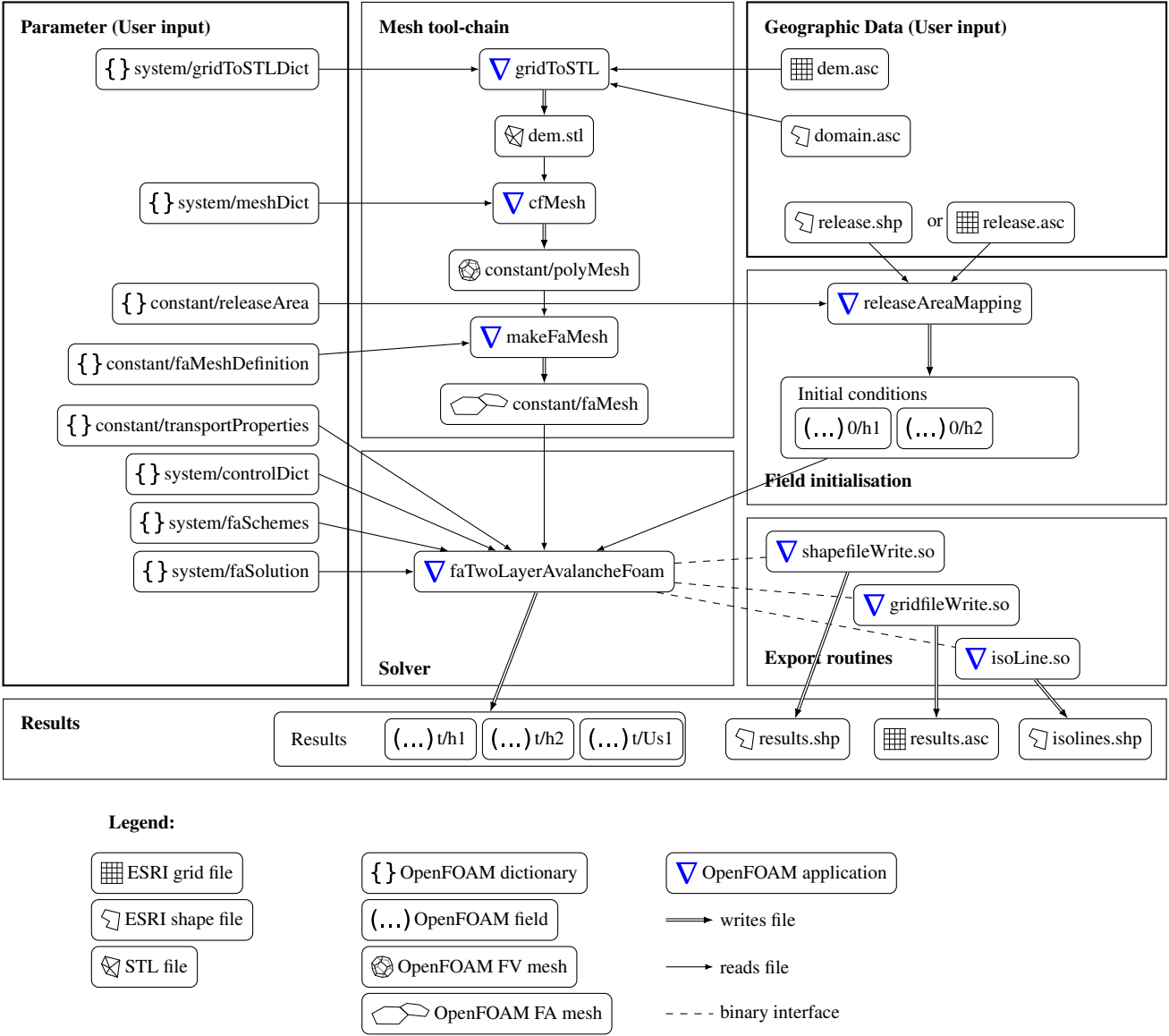
Parameter (User input)
system/gridToSTLDict
system/meshDict
constant/releaseArea
constant/faMeshDefinition
constant/transportProperties
system/controlDict
system/faSchemes
system/faSolution
Mesh tool-chain
gridToSTL
dem.stl
cfMesh
constant/polyMesh
makeFaMesh
constant/faMesh
Geographic Data (User input)
dem.asc
domain.asc
release.shp
or
release.asc
releaseAreaMapping
Initial conditions
(...) 0/h1
(...) 0/h2
Field initialisation
faTwoLayerAvalancheFoam
Solver
shapefileWrite.so
gridfileWrite.so
isoLine.so
Export routines
Results
Results
(...) t/h1
(...) t/h2
(...) t/Us1
results.shp
results.asc
isolines.shp
Legend:
ESRI grid file
ESRI shape file
STL file
OpenFOAM dictionary
OpenFOAM field
OpenFOAM FV mesh
OpenFOAM FA mesh
OpenFOAM application
writes file
reads file
binary interface