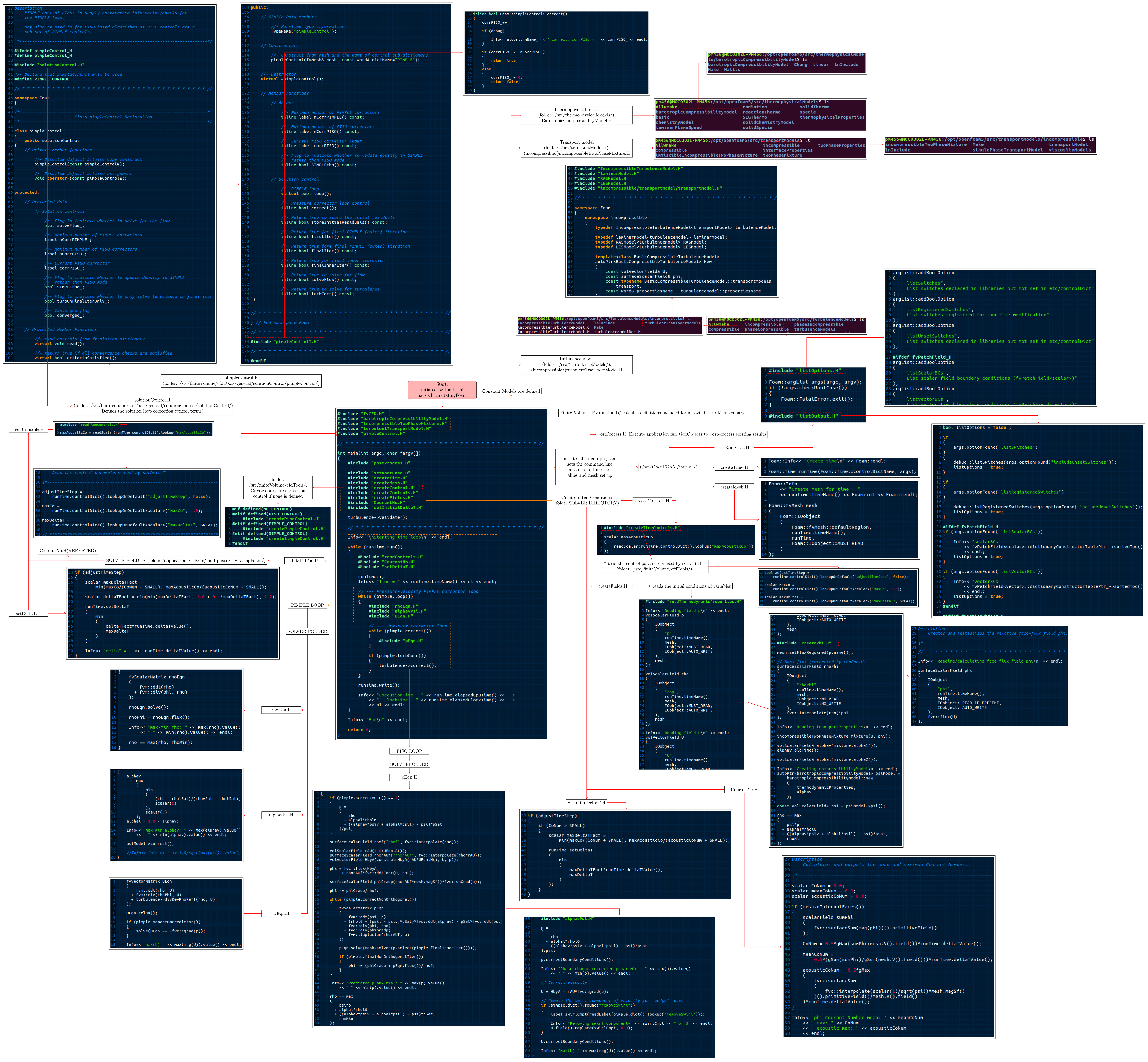
Start:
Initiated by the terminal call: cavitatingFoam
Constant Models are defined
Thermophysical model
(folder: /src/thermophysicalModels/):
BarotropicCompressibilityModel.H
Transport model
(folder: /src/transportModels/):
(incompressible/)incompressibleTwoPhaseMixture.H
Turbulence model
(folder: /src/TurbulenceModels/):
(incompressible/)turbulentTransportModel.H
pimpleControl.H
(folder: /src/finiteVolume/cfdTools/general/solutionControl/pimpleControl/)
solutionControl.H
(folder: /src/finiteVolume/cfdTools/general/solutionControl/solutionControl/)
Defines the solution loop correction control terms)
Finite Volume (FV) methods/ calculus definitions included for all avilable FVM machinary
postProcess.H: Execute application functionObjects to post-process existing results
Initiates the main program: sets the command line parameters, time variables and mesh set up
(/src/OpenFOAM/include/)
setRootCase.H
createTime.H
createMesh.H
folder:
/src/finiteVolume/cfdTools/.
Creates pressure correction control if none is defined
Create Initial Conditions
(folder:SOLVER DIRECTORY)
createControls.H
"Read the control parameters used by setDeltaT"
(folder: /src/finiteVolume/cfdTools/)
createFields.H
reads the initial conditions of variables
CourantNo.H
SetInitialDeltaT.H
readControls.H
CourantNo.H(REPEATED)
SOLVER FOLDER (folder:/applications/solvers/multiphase/cavitatingFoam/)
TIME LOOP
setDeltaT.H
PIMPLE LOOP
SOLVER FOLDER
rhoEqn.H
alphavPsi.H
UEqn.H
PISO LOOP
SOLVERFOLDER
pEqn.H