2015 4th OpenFOAM Korea Users' Community Conference (4th OKUCC)
Sept. 10 – 11, 2015, Hotel ICC (대전)

OpenFOAM 사용자 환경 개발 아이디어

9월 10일, 2015

김 군 홍
scurry1974m@gmail.com

경원이앤씨

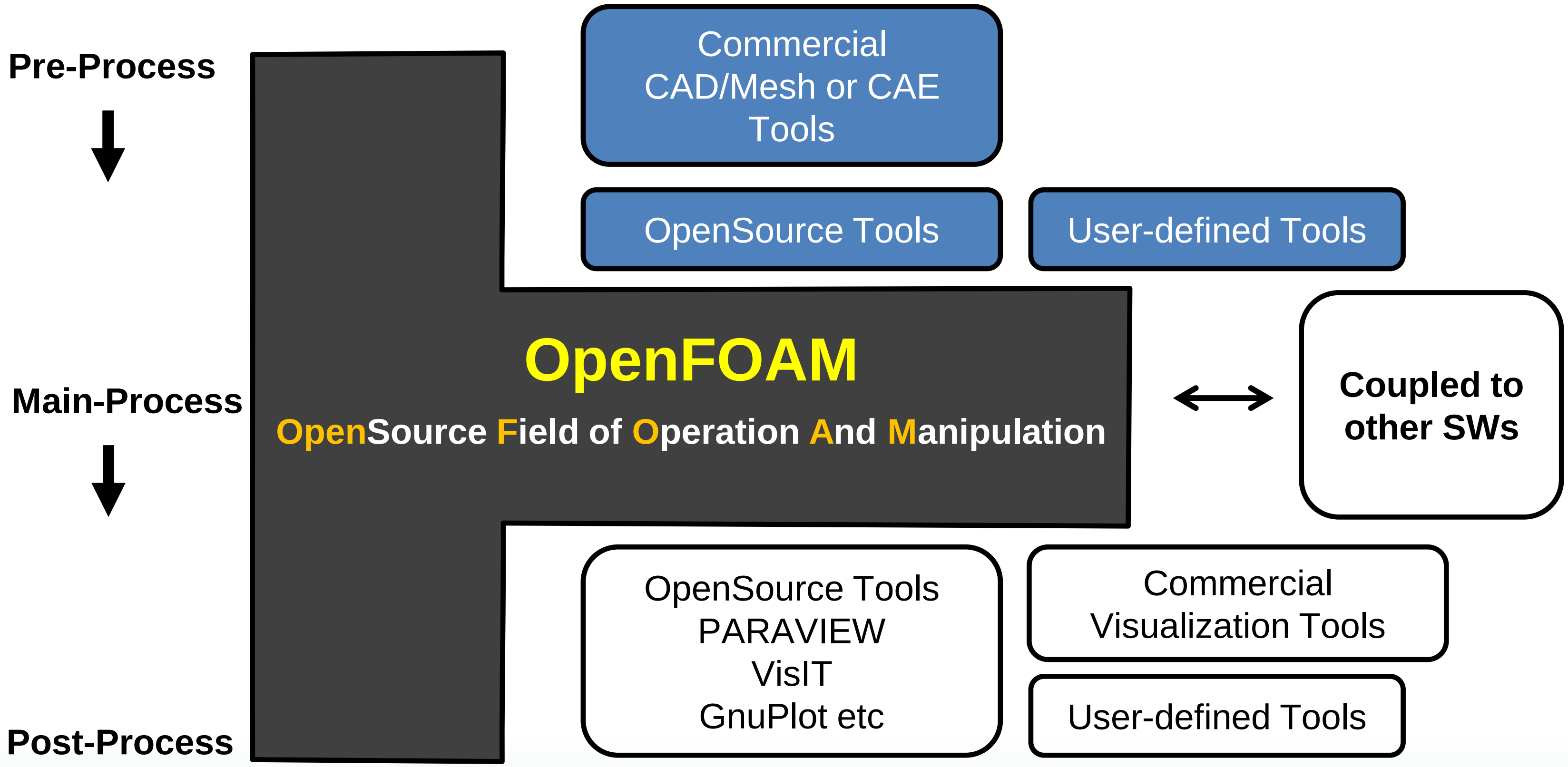
Summary

- **Developing idea on OpenFOAM-Styled GUI**
  - Interfacing btw binary and case
  - Familiar process
  - Common Terminology

- **Work-In-Progress(WIP)**
  - Visualizing geometry, mesh
  - opening or creating a simulation case
  - not yet, still progressing

Basic OpenFOAMing

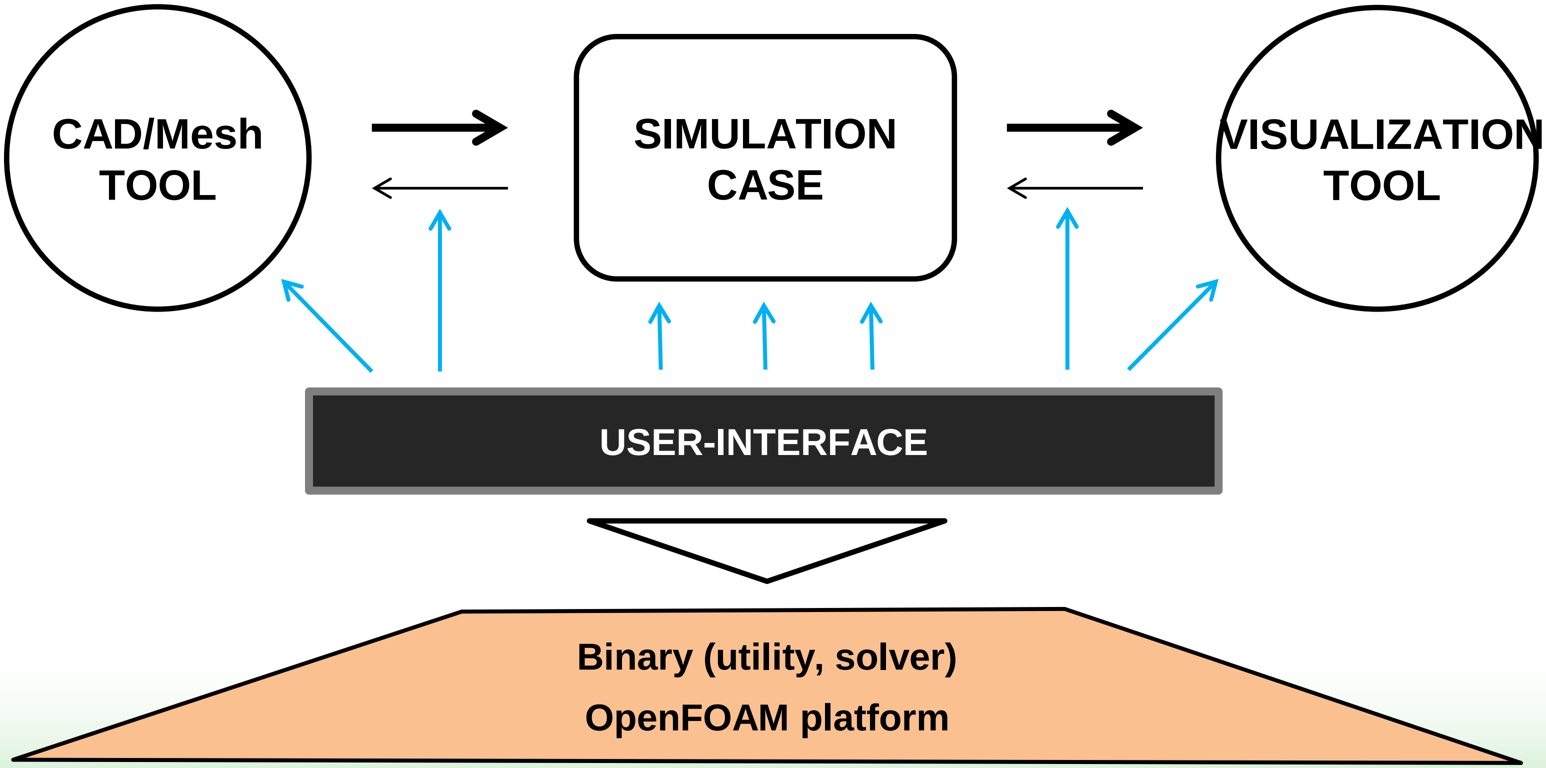
- **OpenFOAM's growing, despite inconveniences.**



OPENFOAM is a registered trademark of OpenCFD Ltd. www.openfoam.org

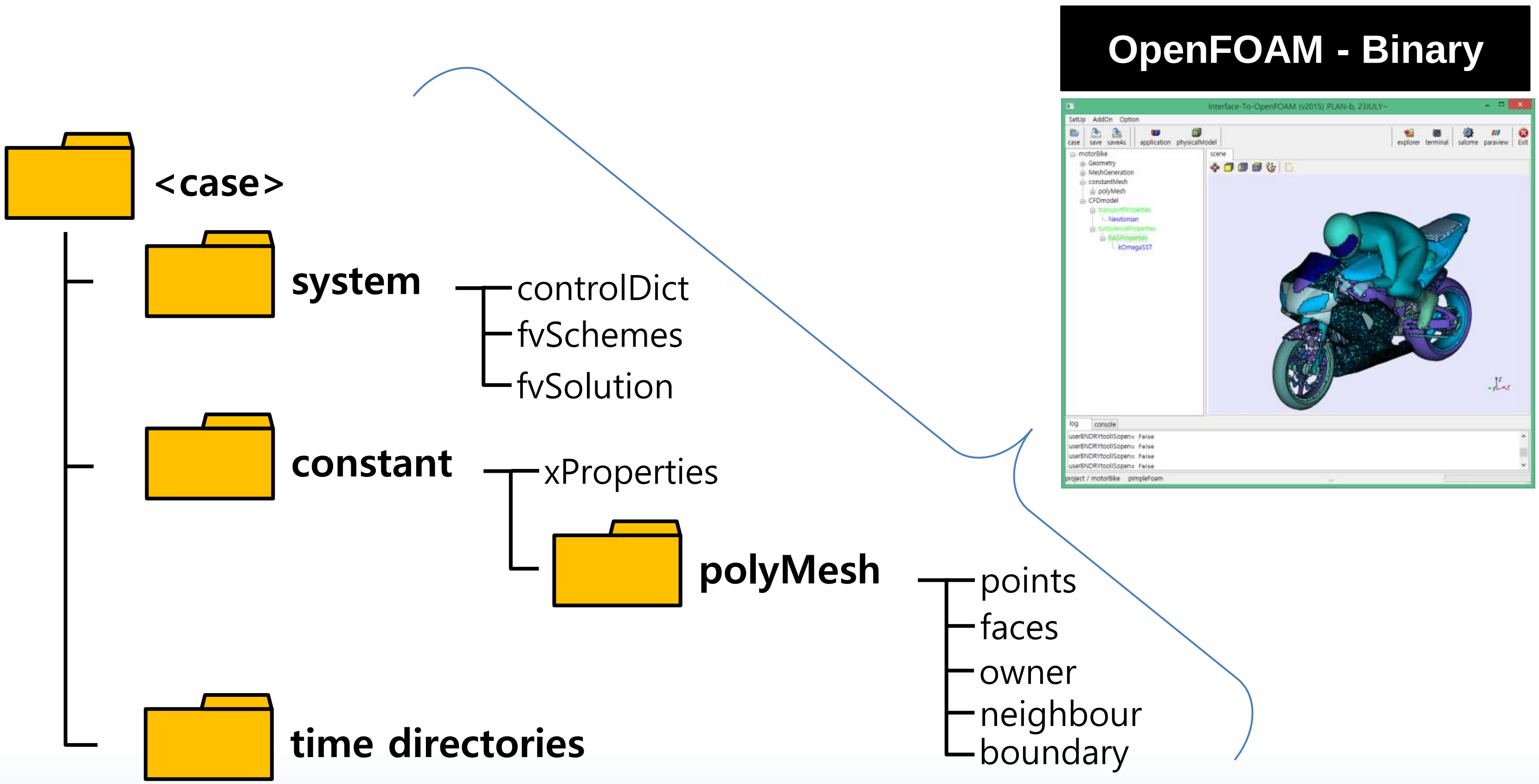
Big Picture

- **OpenFOAM-Interface to run binary, not library.**
  - User-processes consist of binaries..
  -Independent GUI could make-up a case, ready to run.

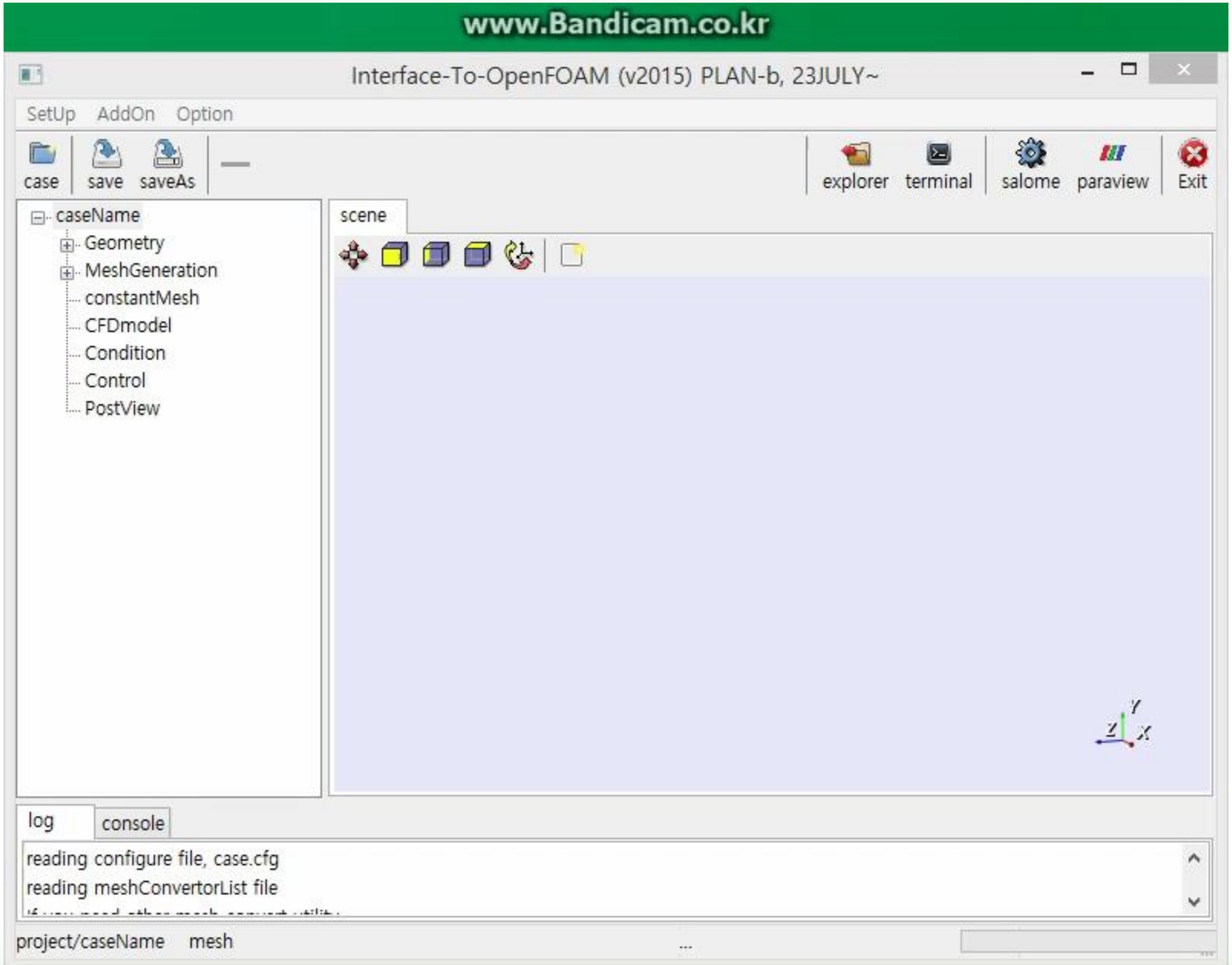


Coding Concept

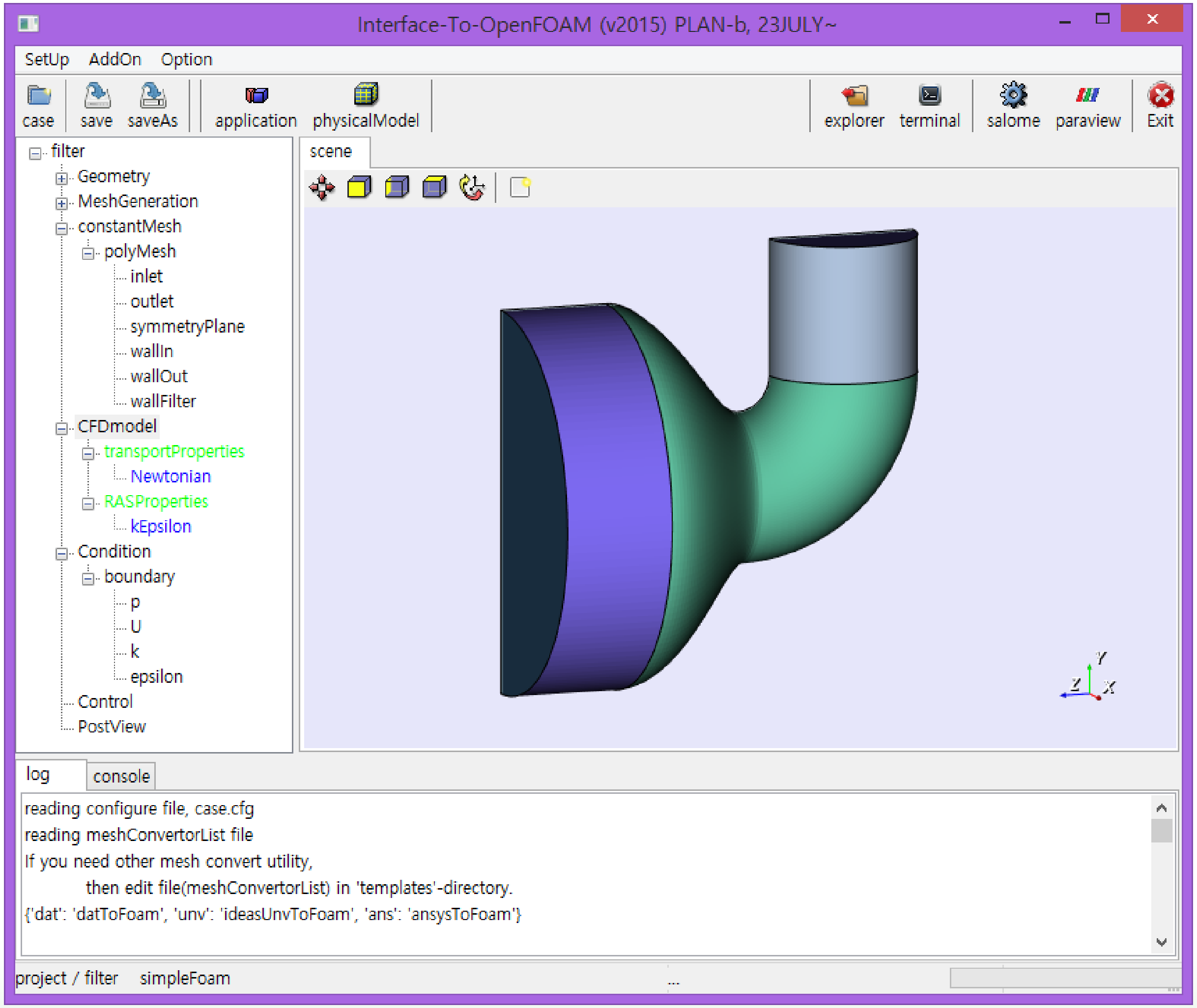
- **OpenFOAM-Styled Interface**
  - Friendly-Process, Common-Terminology



WIP(Work-In-Progress)

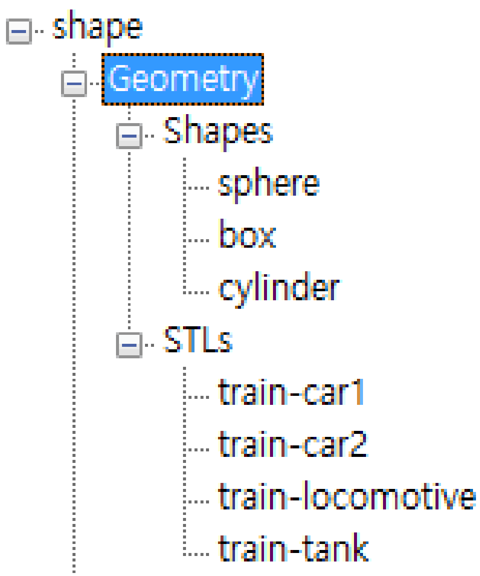


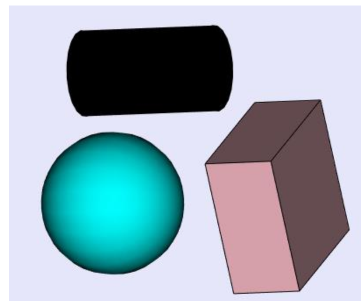
WIP(Work-In-Progress)

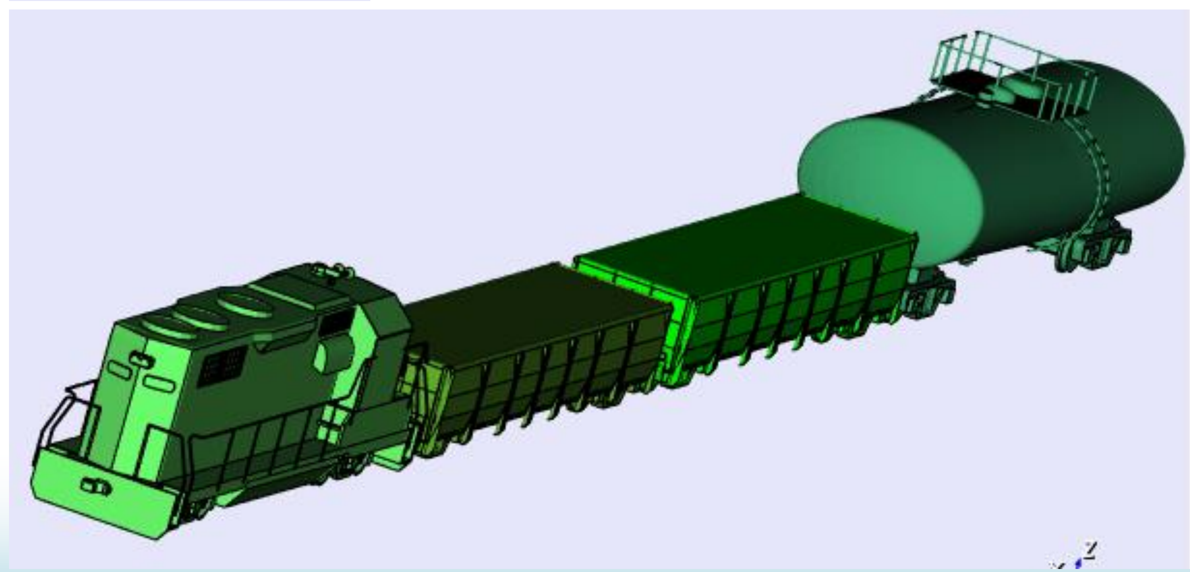


Geometry

- **Simple shape: Box, Cylinder, Sphere**
- **STL: import / export**

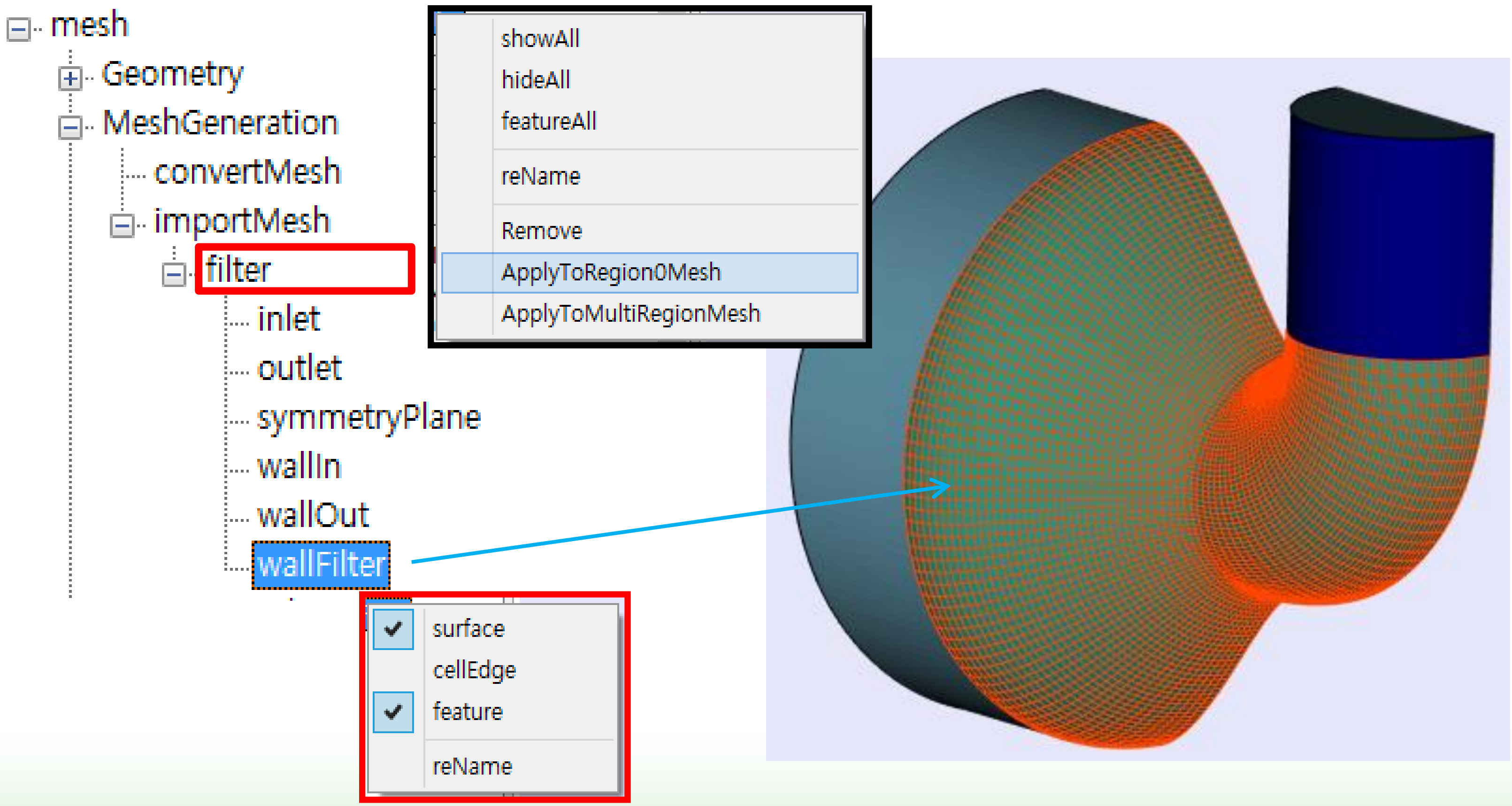






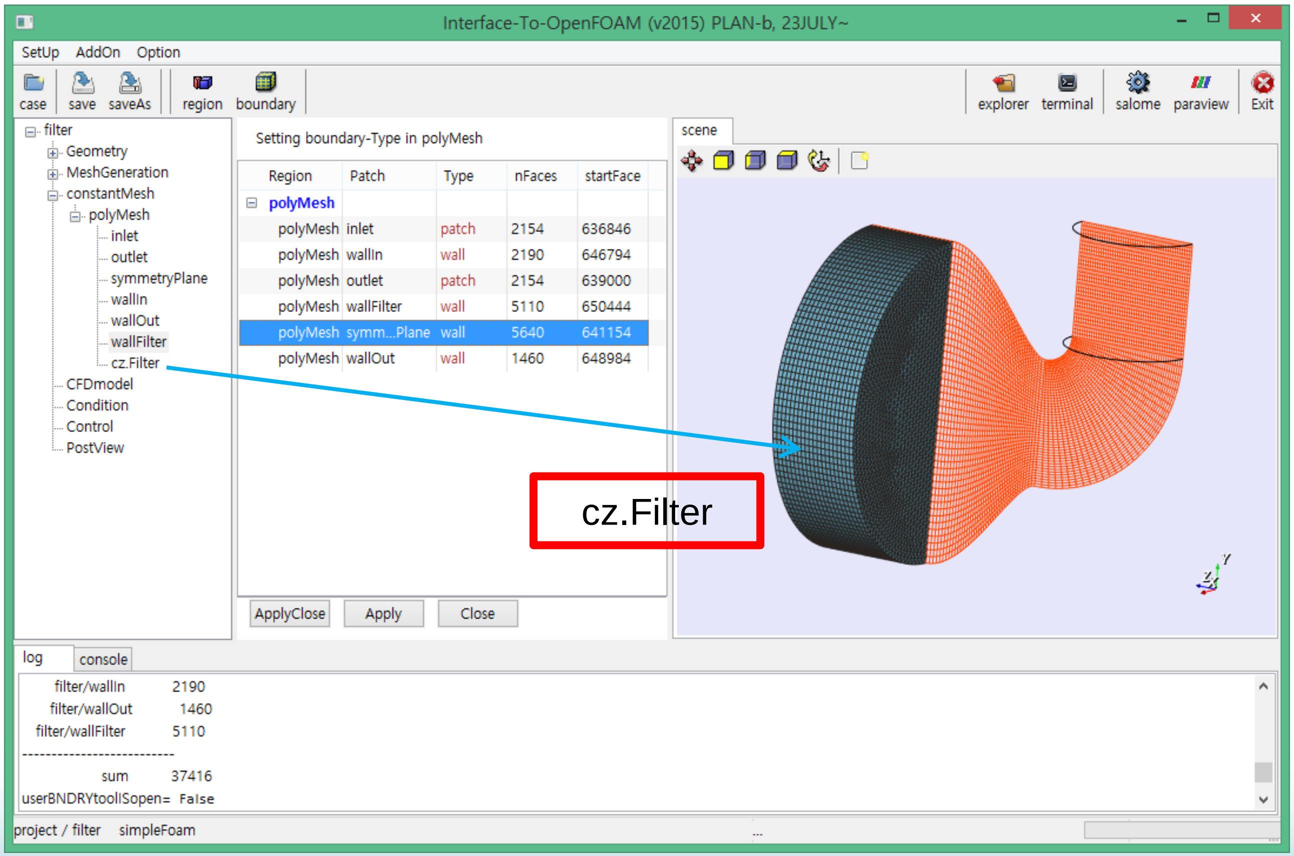
MeshGeneration

- Import polyMesh, Convert Mesh
- Not generating mesh



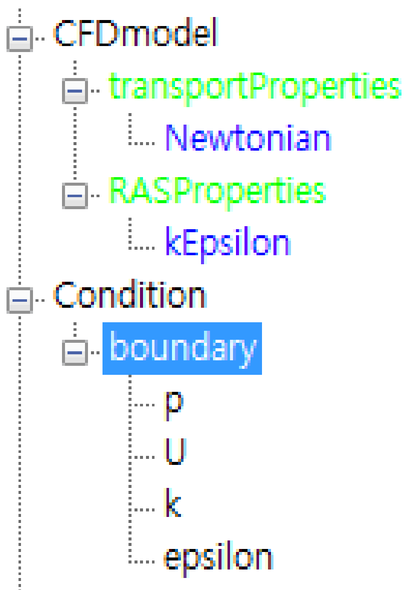
constant/polyMesh

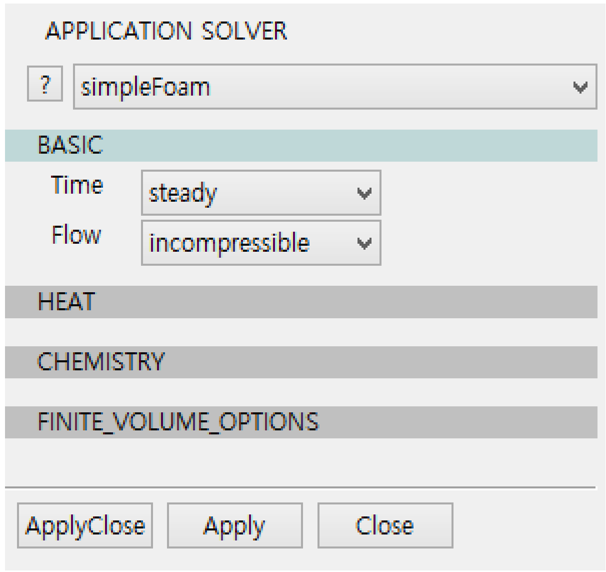
- **Edit boundary-types**
- **Display Zones (cell/face/point)**

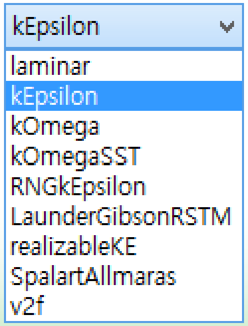


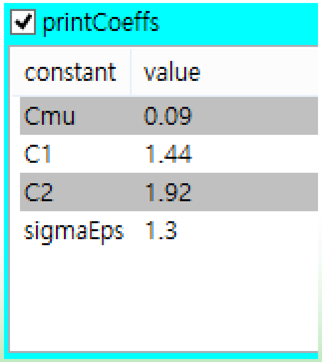
CFDmodel

- **Selecting Solver (, application) is very important !**
- **OpenFOAM-ish Terminology**









Further work is going.

- CFDmodel is still floating.
- What shall I do?

Diversity of user-meshing

- **More than 21 Grid-SWs currently being used.**

SIG: Grid Generation Software
35 people (I guess, most use OF.)

21 different grid generators in 35.

1 person uses at least 2.

Impressions of the 10th OpenFOAM Workshop blog-pointwise.com, posted by Claudio M. Pita

Diversity of Standard Solvers

- **77 solvers and forever growing.**

Basic' CFD codes
laplacianFoam
potentialFoam
scalarTransportFoam

Multiphase flow
bubbleFoam
cavitatingFoam
compressibleInterFoam
interFoam
interDyMFoam
interMixingFoam
interPhaseChangeFoam
LTSInterFoam
MRFInterFoam
MRFMultiphaseInterFoam
multiphaseInterFoam
porousInterFoam
settlingFoam
twoLiquidMixingFoam
twoPhaseEulerFoam

'Incompressible flow
adjointShapeOptimizationFoam
boundaryFoam
channelFoam
icoFoam
MRFSimpleFoam
nonNewtonianIcoFoam
pimpleDyMFoam
pimpleFoam
pisoFoam
porousSimpleFoam
shallowWaterFoam
simpleFoam
SRFSimpleFoam
windSimpleFoam

Direct numerical simulation (DNS) and large eddy simulation (LES)
dnsFoam

Compressible flow
rhoCentralFoam
rhoCentralDyMFoam
rhoPimpleFoam
rhoPorousMRFLTSPimpleFoam
rhoPorousMRFSimpleFoam
rhoPorousMRFPimpleFoam
rhoSimplecFoam
sonicDyMFoam
sonicFoam
sonicLiquidFoam

Diversity of Standard Solvers

- **Fundamentally a tool for PDEs**

Combustion
chemFoam
coldEngineFoam
dieselEngineFoam
dieselFoam
engineFoam
fireFoam
PDRFoam
reactingFoam
rhoReactingFoam
XiFoam

Particle-tracking flows
coalChemistryFoam
icoUncoupledKinematicParcelDyMFoam
icoUncoupledKinematicParcelFoam
LTSReactingParcelFoam
porousExplicitSourceReactingParcelFoam
reactingParcelFilmFoam
reactingParcelFoam
uncoupledKinematicParcelFoam

Heat transfer and buoyancy-driven flows
buoyantBaffleSimpleFoam
buoyantBoussinesqPimpleFoam
buoyantBoussinesqSimpleFoam
buoyantPimpleFoam
buoyantSimpleFoam
buoyantSimpleRadiationFoam
chtMultiRegionFoam

Molecular dynamics methods
mdEquilibrationFoam
mdFoam

Stress analysis of solids
solidDisplacementFoam
solidEquilibriumDisplacementFoam

Direct simulation Monte Carlo methods
dsmcFoam

Finance
financialFoam

Electromagnetics
electrostaticFoam
magneticFoam
mhdFoam

Diversity of Standard Utilities

- **Among 174 utils, Pre-utils (9%), Mesh-utils (41%)**

Pre-processing

applyBoundaryLayer
applyWallFunctionBoundaryConditions
boxTurb
changeDictionary
createExternalCoupledPatchGeometry
dsmcInitialise
engineSwirl
faceAgglomerate
foamUpgradeCyclics
foamUpgradeFvSolution
mapFields
mdInitialise
setFields
viewFactorsGen
wallFunctionTable

Mesh generation

blockMesh
extrudeMesh
extrude2DMesh
extrudeToRegionMesh
foamyHexMesh
foamyHexMeshBackgroundMesh
foamyHexMeshSurfaceSimplify
foamyQuadMesh
snappyHexMesh

Mesh manipulation

attachMesh
autoPatch
checkMesh
createBaffles
createPatch
deformedGeom
flattenMesh
insideCells
mergeMeshes
mergeOrSplitBaffles
mirrorMesh
moveDynamicMesh
moveEngineMesh
moveMesh
objToVTK
orientFaceZone
polyDualMesh
refineMesh
renumberMesh
rotateMesh
setSet
setsToZones
singleCellMesh
splitMesh
splitMeshRegions
stitchMesh
subsetMesh
topoSet
transformPoints
zipUpMesh

Diversity of Standard Utilities

- **Post-utils (25%)**

Other mesh tools
autoRefineMesh
collapseEdges
combinePatchFaces
modifyMesh
PDRMesh
refineHexMesh
refinementLevel
refineWallLayer
removeFaces
selectCells
splitCells

Post-processing turbulence
createTurbulenceFields
R

Post-processing patch data
patchAverage
patchIntegrate

Post-processing graphics
ensightFoamReader

Post-processing velocity fields
Co
enstrophy
flowType
Lambda2
Mach
Pe
Q
streamFunction
uprime
vorticity

Post-processing scalar fields
pPrime2

Sampling post-processing
probeLocations
sample

Post-processing Lagrangian simulation
particleTracks
steadyParticleTracks

Post-processing stress fields
stressComponents

Post-processing data converters
foamDataToFluent
foamToEnsight
foamToEnsightParts
foamToGMV
foamToTecplot360
foamToVTK
smapToFoam

Post-processing at walls
wallGradU
wallHeatFlux
wallShearStress
yPlusLES
yPlusRAS

Diversity of Standard Utilities

- **STL-utils (17%), Parallel-utils(2%), Thermo-utils(3%), Mis.(3%)**

Surface mesh (e.g. STL) tools

surfaceAdd
surfaceAutoPatch
surfaceBooleanFeatures
surfaceCheck
surfaceClean
surfaceCoarsen
surfaceConvert
surfaceFeatureConvert
surfaceFeatureExtract
surfaceFind
surfaceHookUp
surfaceInertia
surfaceLambdaMuSmooth
surfaceMeshConvert
surfaceMeshConvertTesting
surfaceMeshExport
surfaceMeshImport
surfaceMeshInfo
surfaceMeshTriangulate
surfaceOrient
surfacePointMerge
surfaceRedistributePar
surfaceRefineRedGreen
surfaceSplitByPatch
surfaceSplitByTopology
surfaceSplitNonManifolds
surfaceSubset
surfaceToPatch
surfaceTransformPoints

Parallel processing

decomposePar
redistributePar
reconstructParMesh

Thermophysical-related utilities

adiabaticFlameT
chemkinToFoam
equilibriumCO
equilibriumFlameT
mixtureAdiabaticFlameT

Miscellaneous utilities

expandDictionary
foamDebugSwitches
foamFormatConvert
foamHelp
foamInfoExec
patchSummary

Concluding Remark

- The variety of using OpenFOAM makes a GUI-developer crazy.
- A lot of knowledge(, heavy-thinking) are essential to minimize trial and error.
- Until now, the goal is to code a simple GUI being familiar to OF-users.