OpenFOAM Quick-Reference

.mavt.ethz.ch

Important Environment Variables

$WM_PROJECT_DIR – path to the OpenFOAM installation
$WM_PROJECT_USER_DIR – OpenFOAM user directory
$FOAM_TUTORIALS – OpenFOAM tutorials
$FOAM_SRC – source-tree of OpenFOAM libraries
$FOAM_APP – source-tree of OpenFOAM applications
$FOAM_APPBIN – directory with the applications
$FOAM_USER_APPBIN – directory with the applications created by the user
$FOAM_LIBBIN – directory with the libraries provided by OpenFOAM
$FOAM_USER_LIBBIN – directory with the libraries created by the user
$FOAM_RUN – directory where the user can put his/her cases

Important Shell-Aliases

run – *cd* to `$FOAM_RUN`
src – *cd* to `$FOAM_SRC`
app – *cd* to `$FOAM_APP`
util – *cd* to `$FOAM_APP/utilities`
sol – *cd* to `$FOAM_APP/solvers`

Definitions Used Here

case – relative or absolute path to the case

Basic Case Structure

***case*/** – the case directory
+ 0/ – contains initial and boundary conditions
+ constant/ – constant data
  + polyMesh/ – contains the grid data
  + transportProperties – viscosity
+ system/ – run-time control / numerics
  + controlDict – run-time control
  + fvSchemes – numerical schemes
  + fvSolution – linear solvers

***case*/0/** – contains for each variable a file defining the initial and boundary conditions. May also contain initial and boundary conditions for a moving grid.
***case*/constant/polyMesh/** – contains the grid data for a non-moving grid. The files are: `boundary`, `faces`, `neighbour`, `owner`, `points`.
***case*/constant/transportProperties** – defines the viscosity (also for non-Newtonian fluids)
***case*/system/controlDict** – sets start-/endtime, time-step size, output control etc. Also allows to load general “plugins” and apply “function-objects” to compute forces acting on a surface.
***case*/system/fvSchemes** – defines the numerical schemes to be used for each differential operator
***case*/system/fvSolution** – selects the solvers to be used for the linear equation systems for each variable which is solved for using an implicit scheme.

Uniform Invocation Syntax

app* -case *case – start *app* (the case is in *case*). Sometimes more options/arguments are accepted.
app – start *app* directly if you are already in *case*
***app* -help** – display short help message for *app*
***app* -doc** – open documentation for *app*
***app* -srcDoc** – open source-documentation for *app*

Important (Basic) Solvers

laplacianFoam – solve Laplace equation, suitable for e.g. thermal diffusion in solids
potentialFoam – potential flow solver, suitable for generating good initial conditions
scalarTransportFoam – solve scalar transport equation, suitable for e.g. postprocessing
icoFoam – solves the incompressible Navier-Stokes equations for Newtonian fluids (for laminar flows)
turbFoam – solves the incompressible RANS equations using turbulence modelling for Newtonian fluids (turbulent flows)
icoDyMFoam/turbDyMFoam – for dynamic mesh cases
simpleFoam – steady-state solver for the incompressible Navier-Stokes equations for non-Newtonian fluids
interFoam – solver for 2 incompressible, immiscible fluids

Important Utilities

blockMesh – creates the mesh defined by *case/constant/polyMesh/blockMeshDict*
decomposePar – splits the case for parallel run, controlled by *case/system/decomposeParDict*
reconstructPar – reassembles the decomposed solution of a parallel run
paraFoam – starts ParaView to visualize the results
touch *case*/*caseName*.foam && paraview --data=*case*/*caseName*.foam – starts ParaView with an alternative (better) plugin

Structure of a Solver

***appName*/** – the directory with the source code
+ *appName*.C – the main program
+ createFields.H – declarations and initializations of all fields
+ Make/ – compilation instructions
  + files – list of source/output files
  + options – compilation options

***appName*/*appName*.C** – the actual solver code
***appName*/createFields.H** – declares all the field variables and initializes the by (usually) reading the initial conditions from a file.
***appName*/Make/files** – names all the source (.C) files, one file per line. The last line should read `EXE=$(FOAM_USER_APPBIN)/`*appName* to specify the name and location of the output file.
***appName*/Make/options** – specifies directories to search for include files and libraries to link the solver against. The former are specified in the variable EXE_INCS, the latter in EXE_LIBS. Lines have to be continued using the \ character