1 OpenFOAM Quick Reference Guide

Important Environment Variables in OpenFOAM

$WM_PROJECT_DIR	Path to the OpenFOAM installation
$WM_PROJECT_USER_DIR	Path to OpenFOAM user directory
$FOAM_TUTORIALS	Path to OpenFOAM tutorials
$FOAM_SRC	Path to source code directory of OpenFOAM libraries
$FOAM_APP	Path to source code directory of OpenFOAM applications
$FOAM_APPBIN	Path to directory with the compiled OpenFOAM applications
$FOAM_USER_APPBIN	Path to directory with the OpenFOAM applications created by the user
$FOAM_LIBBIN	Path to directory with the compiled OpenFOAM libraries
$FOAM_USER_LIBBIN	Path to directory with the OpenFOAM libraries created b the user
$FOAM_RUN	Path to directory where the user can put his/her cases (recommended)

Important Shell-Aliases in OpenFOAM

foam	cd $WM_PROJECT_DIR
foamApps or app	cd $FOAM_APP
foamSol or sol	cd $FOAM_SOLVERS
foamTuts or tut	cd $FOAM_TUTORIALS
foamUtils or util	cd $FOAM_UTILITIES
foamsrc	cd $FOAM_SRC/$WM_PROJECT
foam3rdParty	cd $WM_THIRD_PARTY_DIR
foamfv	cd $FOAM_SRC/finiteVolume
lib	cd $FOAM_LIBBIN
run	cd $FOAM_RUN
src	cd $FOAM_SRC
wmSET	. $WM_PROJECT_DIR/etc/bashrc
wmUNSET	. $WM_PROJECT_DIR/etc/config/unset.sh

NOTE: this list does not show all the aliases set by Openfoam

Basic Case Structure in OpenFOAM

case/
    + 0/
    + constant/
      + polyMesh/
    + system/
    + timedirectory/
	case directory general hierarchical structure

case/0/	contains for each variable a file defining the initial and boundary conditions
case/constant/	contains files specifying physical properties for the application concerned
case/constant/polyMesh	contains the polyhedral mesh information
case/system/	for setting parameters associated with the numerics and run-time control. It contains at least the following 3 files: controlDict where run control parameters are set including start/end time, time step and parameters for data output; fvSchemes where discretization schemes used in the solution may be selected; and, fvSolution where the equation solvers, tolerances and other algorithm controls are set
case/timedirectory	contains the results for each saved time-step. It stores the solution of the corresponding time-step to the corresponding time directory timedirectory

Important Solvers in OpenFOAM

potentialFoam	potential flow solver which can be used to generate starting fields for full Navier-Stokes solvers
icoFoam	transient solver for incompressible laminar Newtonian flows
simpleFoam	steady-state solver for incompressible turbulent/laminar flows
pisoFoam	transient solver for incompressible turbulent/laminar flows
rhoCentralFoam	transient density-based compressible flow solver based on central upwind schemes of Kurganov and Tadmor
sonicFoam	transient solver for trans-sonic/supersonic, laminar or turbulent flow of a compressible gas
interFoam	transient solver for 2 incompressible, isothermal immiscible fluids using VOF (volume of fluid) phase fraction based interface capturing approach
XiFoam	solver for compressible premixed/partially-premixed combustion with turbulence modelling
buoyantPimpleFoam	transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer
icoUncoupledKinematicParcelFoam	transient solver for the passive transport of a single kinematic particle cloud
solidDisplacementFoam	transient segregated finite-volume solver of linear-elastic, small strain deformation of a solid body, with optional thermal diffusion and thermal stresses

NOTE: this list does not show all the solvers available in Openfoam

Invoking a Solver in OpenFOAM

solver_name -case *casedir*	starts the solver, casedir is the path of the directory where the case files are located. Sometimes more options/arguments are needed/accepted
solver_name	starts the solver directly if you are already in the case directory. Sometimes more options/arguments are needed/accepted
solver_name -help	shows a short help message for the solver

Important Utilities in OpenFOAM

blockMesh	a multi-block mesh generator
snappyhexmesh	automatic split hex mesher
checkMesh	check validity of a mesh
decomposePar	automatically decomposes a mesh and fields of a case for parallel execution of Openfoam

Important Utilities in OpenFOAM

reconstrucPar	reconstructs a mesh and fields of a case that is decomposed for parallel execution of Openfoam
setFields	set values on a selected set of cells/patch faces through a dictionary
mapFields	maps volume fields from one mesh to another
paraFoam	starts paraview to visualize the results

Invoking Utilities in OpenFOAM

utility_name	starts the utility directly if you are already in the case directory. Sometimes more options/arguments are needed/accepted
utility_name -help	shows a short help message for the utility

Structure of a Solver in OpenFOAM

appName/ 　- appName.C 　- createFields.H 　+ Make/ 　　- files 　　- options	appName directory general hierarchical structure
appName/appName.C	the actual solver code
appName/createFields.H	declares all the field variables
appName/Make/files	names all the source files (.C), one 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_INC, the later in EXE_LIBS

Compiling a new solver in OpenFOAM

wmake	to compile solver. Use this command in appName directory
wclean	remove dependency list and compiled solver. Use this command in appName directory