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)

NOTE: this list does not show all the environment variables set by Openfoam®

Echoing Important Environment Variables in OpenFOAM®	
echo *environment variable*	will show you the path of the *environment variable* Example: echo **$WM_PROJECT_DIR** will show you $HOME/OpenFOAM/OpenFOAM-2.1.x (if the installed version is 2.1.x and was installed in the default installation directory $HOME/OpenFOAM)

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`

OpenFOAM® QUICK REFERENCE GUIDE

Important Shell-Aliases in OpenFOAM®	
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/1/ **case/2/** . . . **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***

OpenFOAM® QUICK REFERENCE GUIDE

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®

OpenFOAM® QUICK REFERENCE GUIDE

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

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

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