Folder structure and *pre-defined File Names* in OpenFOAM

Folder	Sub-Folders	Files / Folder	Files / Folder	Files / Folder	Files / Folder	Explanation
BaseDir/	constant/					
	Allclean					File to remove unwanted files: polyMesh, time folders, files created by checkMesh / topoSet
	Allrun					Issue all commands from a single file: generate mesh, set I.C. / B.C., solve
		polyMesh/	-	-		-
			points			Points generated after meshing. Note: most properties are defined at the cell centroids
			faces			Faces of the mesh
			owner			Internal faces Those faces that connect two cells (and it can never be more than two). Cell-1: **owner**, Cell-2: **neighbour**
			neighbour			
			faceZones			Not created by blockMesh, ansysToFoam creates this file
			cellZones			For multi-zone [e.g. air + porous] cases
			boundary			B.C. data: e.g. boundary faces - attached to only 1 cell, hence have only an owner
			set/			topoSet operation files
				highAspectRatioCells		The utility foamToVTK converts the failed sets to VTK format.
				nonOrthoFaces		foamToVTK -faceSet nonOrthoFaces
				wrongOrientedFaces		
				skewFaces		
				unusedPoints		foamToVTK -pointSet unusedPoints
		triSurface/				
			*.obj or *.obg.gz or *.stl			Primarily used by snappyHexMesh (sHM)
			*.eMesh			Edge Mesh generated by utility surfaceFeatureExtract [no surface data]
		boundaryData/				
			patchName/	0/		
					U	Specifying interpolation data on a patch such as non-uniform inlet B.C. Depending upon the field variable to be interpolated on boundary named "patchName".
					k	
					epsilon	
		transportProperties (replaced by thermophysicalProperties)				Defines fluid properties for incompressible solvers
		thermophysicalProperties				Ideal gas or non-ideal behaviour: properties of a fluid when using the energy equation (with compressible solvers).
		turbulenceProperties				Selection of turbulence modelling method
		thermalProperties				Thermal cond, specific heat capacity
		chemistryProperties				Combustion modeling chemistry
		dynamicMeshDict				Settings specific to pimpleDyMFoam
		MRFZones				For moving reference frame calculations
		environmentalProperties				Value of gravity in case of buoyant flows
		combustionProperties				Fuel properties, combustion / chemical reaction related data
		mechanicalProperties				For structural mechanics

Folder	Sub-Folders	Files / Folder	Files / Folder	Files / Folder	Files / Folder	Explanation
BaseDir/	constant/					
		fluid_region_name/				
		dynamicMeshDict				Compressible flows, mesh motion - pimpleDyMFoam
		regionProperties				Defines the solid and fluid regions as defined in topoSetDict
			polyMesh/			Mesh data for this (fluid) zone: created by splitMesh utility
			g			Gravity
			radiationProperties			Radiation settings
			thermophysicalProperties			Transport properties
			turbulenceProperties			Turbulence setting such as coefficient of k-ε model
		solid_region_name/				
			polyMesh/			Mesh data for this (solid) zone: created by splitMesh utility
			radiationProperties			Radiation settings
			thermophysicalProperties			Transport properties such as thermal condutivity and density
		g				Direction of gravity
		phaseProperties				Two-phase flows [.air and .water are the phases]
		thermophysicalProperties.air				
		thermophysicalProperties.water				
		turbulenceProperties.air				
		turbulenceProperties.water				
BaseDir/	VTK/					faulty sets written by foamToVTK
BaseDir/	*.OpenFOAM	* = Name of the folder containing OpenFOAM data				To open a case in ParaView
BaseDir/	*.vtk	[Default location, though folder path can be specified to be different]				Convert *.eMesh to vtk format to view edge data in Paraview [which cannot read *.eMEsh]
BaseDir/	0/	-				-
		cellToRegion				created by splitMeshRegions: This file defines all the patches of a particular region.
		p				Initialization of pressure field
		U				Initialization of velocity field
		T				Initialization of temperature field
		D				Displacement field for structural calculations
		mut				Turbulence viscosity & wall functions
		nut				turbulent viscosities: Selection of wall function model is specified through this file
		nuTilda				Turb. visc.- Spallarat Allmaras Model
		k				Turbulent Kinetic Energy: TKE
		epsilon				Turbulent Eddy Dissipation rate
		omega				For k-ω model: eddy dissipation frequency

Folder	Sub-Folders	Files / Folder	Files / Folder	Files / Folder	Files / Folder	Explanation
BaseDir/	0/	-				-
		G				Incident radiation field G for P1 model
		I				Radiation intensity in case of fvDOM model
		Qr				Radiative heat flux for P1 model
		alphat				alphat = nu/Pr + nut/Prt: turbulent thermal diffusivity
		h				Free surface flows
		hTotal				
		hU				
		alpha.vapour				Multi-phase flows
		rho				
		p_rgh				Buoyancy driven flows
		betavSolid				Porosity in the solid region between 0 and 1. If betavSolid file does not exist, set to 1.
		fluid_region_name/		Nnamed as per zones defined in **topoSetDict,** created by **splitMesh:** all field files from the 0 folder are copied into the region folders		
			cellToRegion			Created by topoSet utility
			p			Initialization of pressure and value at boundaries and interfaces
			U			Initialization of velocity and value at boundaries and interfaces
			AoV			Area of Volume ratio: for porous domains only
			htcConst			Contant HTC inside porous domains - in combination with AoV
			fvOptions			fvOptions specific to a domain such as porous domain
			...			
		solid_region_name/				
			cellToRegion			Created by topoSet utility
			p			Initialization of pressure and value at boundaries and interfaces
			U			Initialization of velocity and value at boundaries and interfaces
			...			
BaseDir/	system/					
		blockMeshDict				Geometry and block mesh setting
		fvSolution/	PISO			Setting of PISO solver
		fvSchemes				Discretization schemes
		fvOptions				Special options to solver such as heat source, porosity of a zone...
		fluid_region_name/				Folders for each region has to be created by the user, should contain default files.
			fvSolution/	PISO		Settings for PISO solver
			changeDictionaryDict			Boundary and initial condition change
			fvSchemes			Discretization schemes

Folder	Sub-Folders	Files / Folder	Files / Folder	Files / Folder	Files / Folder	Explanation

BaseDir/	system/				
			fvOptions		Specifying porosity, momentum source, mass source...
		solid_region_name/			Folders for each region has to be created by the user, should contain default files.
			fvSolution/	PISO	Settings for PISO solver
			fvSchemes		Discretization schemes
			fvOptions		Specifying heat source or sink...
			changeDictionaryDict		Change boundary condition definitions
		controlDict			Time steps for reading & writing data
		createPatchDict			Create new patches from boundary faces
		cuttingPlane			Planes to extrapolate data in vtk format
		decomposeParDict			Domain decomposition - parallel computing
		extrude2DMeshDict			Setting for extrdue2DMesh utility
		foamyQuadMeshDict			Setting for foamyQuadMesh utility
		forceCoeffs			
		mapFieldsDict			Setting for interpolating / extrapolating data
		meshQualityDict			Setting for quality checking parameters for meshQuality utility
		readFields			Read results from other simulations
		refineMeshDict			Refine Mesh (works with setTopo utility)
		sampleDict			Required for 'sample' utility
		setFieldsDict			To specify a non-uniform initial condition
		snappyHexMeshDict			Settings for mesh generation using snappyHexMesh utility
		splitMeshRegionDict			splitMeshRegions - different regions are created
		streamLines			
		surfaceFeatureExtractDict			Extract geometry from a STL geometry data
		surfaceFeatureExtractDictDefaults			
		topoSetDict			topoSet - different cellsets per region are created, dictionaries created elsewhere have
		wallBoundedStreamLines			
BaseDir/	logs/				Files created by foamLog utility
		Ux_0, Ux_1, Ux_2 ...UxFinalRes 0			Convergence history for Velocity Field
		p 0, pFinalRes 0, pIters 0 ... p 1, pFinalRes 1, pIters 1			Convergence history for Pressure Field
		CourantMax_n, CourantMean_n: n = 0, 1, 2, ...			Log file for Courant Number
BaseDir/	output/	t1/			Time step -1
		phi			Contains face fluxes that are needed to yield a perfect restart.
		uniform			Used for uniform information in a parallel simulation

Folder	Sub-Folders	Files / Folder	Files / Folder	Files / Folder	Files / Folder	Explanation
BaseDir/	output/					
		t2/				Time step -2
		...				
		tn/				Time step -n
BaseDir/	postProcessing/	File created by solver if "cuttingPlane" dictionary found in system folder				

The solvers with the OpenFOAM distribution are in the $FOAM SOLVERS directory, reached quickly by typing app at the command line.