```
        "<root0>"
        "<root1>"
        ...
    );
```

where <`nRoots`> is the number of roots.

Each of the *processorN* directories should be placed in the case directory at each of the root paths specified in the *decomposeParDict* dictionary. The *system* directory and *files* within the *constant* directory must also be present in each case directory. Note: the files in the *constant* directory are needed, but the *polyMesh* directory is not.

3.4.5 Post-processing parallel processed cases

When post-processing cases that have been run in parallel the user has two options:

- reconstruction of the mesh and field data to recreate the complete domain and fields, which can be post-processed as normal;
- post-processing each segment of decomposed domain individually.

3.4.5.1 Reconstructing mesh and data

After a case has been run in parallel, it can be reconstructed for post-processing. The case is reconstructed by merging the sets of time directories from each *processorN* directory into a single set of time directories. The reconstructPar utility performs such a reconstruction by executing the command:

```
reconstructPar
```

When the data is distributed across several disks, it must be first copied to the local case directory for reconstruction.

3.4.5.2 Post-processing decomposed cases

The user may post-process decomposed cases using the paraFoam post-processor, described in section 6.1. The whole simulation can be post-processed by reconstructing the case or alternatively it is possible to post-process a segment of the decomposed domain individually by simply treating the individual processor directory as a case in its own right.

3.5 Standard solvers

The solvers with the OpenFOAM distribution are in the *$FOAM_SOLVERS* directory, reached quickly by typing `sol` at the command line. This directory is further subdivided into several directories by category of continuum mechanics, *e.g.* incompressible flow, combustion and solid body stress analysis. Each solver is given a name that is reasonably descriptive, *e.g.*icoFoam solves incompressible, laminar flow. The current list of solvers distributed with OpenFOAM is given in the following Sections.

3.5.1 ‘Basic’ CFD codes

laplacianFoam Solves a simple Laplace equation, e.g. for thermal diffusion in a solid.

potentialFoam Potential flow solver which solves for the velocity potential, to calculate the flux-field, from which the velocity field is obtained by reconstructing the flux.

scalarTransportFoam Solves the steady or transient transport equation for a passive scalar.

3.5.2 Incompressible flow

adjointShapeOptimizationFoam Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids with optimisation of duct shape by applying "blockage" in regions causing pressure loss as estimated using an adjoint formulation.

boundaryFoam Steady-state solver for incompressible, 1D turbulent flow, typically to generate boundary layer conditions at an inlet, for use in a simulation.

icoFoam Transient solver for incompressible, laminar flow of Newtonian fluids.

nonNewtonianIcoFoam Transient solver for incompressible, laminar flow of non-Newtonian fluids.

pimpleFoam Transient solver for incompressible, turbulent flow of Newtonian fluids, with optional mesh motion and mesh topology changes.

SRFPimpleFoam Large time-step transient solver for incompressible, turbulent flow in a single rotating frame.

pisoFoam Transient solver for incompressible, turbulent flow, using the PISO algorithm.

shallowWaterFoam Transient solver for inviscid shallow-water equations with rotation.

simpleFoam Steady-state solver for incompressible, turbulent flow, using the SIMPLE algorithm.

porousSimpleFoam Steady-state solver for incompressible, turbulent flow with implicit or explicit porosity treatment and support for multiple reference frames (MRF).

SRFSimpleFoam Steady-state solver for incompressible, turbulent flow of non-Newtonian fluids in a single rotating frame.

3.5.3 Compressible flow

rhoCentralFoam Density-based compressible flow solver based on central-upwind schemes of Kurganov and Tadmor with support for mesh-motion and topology changes.

rhoPimpleFoam Transient solver for turbulent flow of compressible fluids for HVAC and similar applications, with optional mesh motion and mesh topology changes.

rhoSimpleFoam Steady-state solver for turbulent flow of compressible fluids.

rhoPorousSimpleFoam Steady-state solver for turbulent flow of compressible fluids, with implicit or explicit porosity treatment and optional sources.

3.5.4 Multiphase flow

cavitatingFoam Transient cavitation code based on the homogeneous equilibrium model from which the compressibility of the liquid/vapour "mixture" is obtained, with optional mesh motion and mesh topology changes.

compressibleInterFoam Solver for 2 compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

compressibleInterFilmFoam Solver for 2 compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach and surface film modelling.

compressibleMultiphaseInterFoam Solver for n compressible, non-isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach.

driftFluxFoam Solver for 2 incompressible fluids using the mixture approach with the drift-flux approximation for relative motion of the phases.

interFoam Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

interMixingFoam Solver for 3 incompressible fluids, two of which are miscible, using a VOF method to capture the interface, with optional mesh motion and mesh topology changes including adaptive re-meshing.

interPhaseChangeFoam Solver for 2 incompressible, isothermal immiscible fluids with phase-change (e.g. cavitation). Uses a VOF (volume of fluid) phase-fraction based interface capturing approach, with optional mesh motion and mesh topology changes including adaptive re-meshing.

multiphaseEulerFoam Solver for a system of any number of compressible fluid phases with a common pressure, but otherwise separate properties. The type of phase model is run time selectable and can optionally represent multiple species and in-phase reactions. The phase system is also run time selectable and can optionally represent different types of momentun, heat and mass transfer.

multiphaseInterFoam Solver for n incompressible fluids which captures the interfaces and includes surface-tension and contact-angle effects for each phase, with optional mesh motion and mesh topology changes.

potentialFreeSurfaceFoam Incompressible Navier-Stokes solver with inclusion of a wave height field to enable single-phase free-surface approximations, with optional mesh motion and mesh topology changes.

twoLiquidMixingFoam Solver for mixing 2 incompressible fluids.

twoPhaseEulerFoam Solver for a system of 2 compressible fluid phases with one phase dispersed, e.g. gas bubbles in a liquid including heat-transfer.

3.5.5 Direct numerical simulation (DNS)

dnsFoam Direct numerical simulation solver for boxes of isotropic turbulence.

3.5.6 Combustion

chemFoam Solver for chemistry problems, designed for use on single cell cases to provide comparison against other chemistry solvers, that uses a single cell mesh, and fields created from the initial conditions.

coldEngineFoam Solver for cold-flow in internal combustion engines.

engineFoam Transient solver for compressible, turbulent engine flow with a spray particle cloud.

fireFoam Transient solver for fires and turbulent diffusion flames with reacting particle clouds, surface film and pyrolysis modelling.

PDRFoam Solver for compressible premixed/partially-premixed combustion with turbulence modelling.

reactingFoam Solver for combustion with chemical reactions.

rhoReactingBuoyantFoam Solver for combustion with chemical reactions using a density based thermodynamics package with enhanced buoyancy treatment.

rhoReactingFoam Solver for combustion with chemical reactions using density based thermodynamics package.

XiEngineFoam Solver for internal combustion engines.

XiFoam Solver for compressible premixed/partially-premixed combustion with turbulence modelling.

3.5.7 Heat transfer and buoyancy-driven flows

buoyantPimpleFoam Transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer.

buoyantSimpleFoam Steady-state solver for buoyant, turbulent flow of compressible fluids, including radiation, for ventilation and heat-transfer.

chtMultiRegionFoam Solver for steady or transient fluid flow and solid heat conduction, with conjugate heat transfer between regions, buoyancy effects, turbulence, reactions and radiation modelling.

thermoFoam Solver for energy transport and thermodynamics on a frozen flow field.

3.5.8 Particle-tracking flows

coalChemistryFoam Transient solver for compressible, turbulent flow, with coal and limestone particle clouds, an energy source, and combustion.

DPMFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase, with optional mesh motion and mesh topology changes.

MPPICFoam Transient solver for the coupled transport of a single kinematic particle cloud including the effect of the volume fraction of particles on the continuous phase. Multi-Phase Particle In Cell (MPPIC) modeling is used to represent collisions without resolving particle-particle interactions, with optional mesh motion and mesh topology changes.

particleFoam Transient solver for the passive transport of a single kinematic particle cloud, with optional mesh motion and mesh topology changes.

reactingParcelFoam Transient solver for compressible, turbulent flow with a reacting, multiphase particle cloud, and surface film modelling.

rhoParticleFoam Transient solver for the passive transport of a particle cloud.

simpleReactingParcelFoam Steady state solver for compressible, turbulent flow with reacting, multiphase particle clouds and optional sources/constraints.

sprayFoam Transient solver for compressible, turbulent flow with a spray particle cloud, with optional mesh motion and mesh topology changes.

3.5.9 Discrete methods

dsmcFoam Direct simulation Monte Carlo (DSMC) solver for, transient, multi-species flows.

mdEquilibrationFoam Solver to equilibrate and/or precondition molecular dynamics systems.

mdFoam Molecular dynamics solver for fluid dynamics.

3.5.10 Electromagnetics

electrostaticFoam Solver for electrostatics.

magneticFoam Solver for the magnetic field generated by permanent magnets.

mhdFoam Solver for magnetohydrodynamics (MHD): incompressible, laminar flow of a conducting fluid under the influence of a magnetic field.

3.5.11 Stress analysis of solids

solidDisplacementFoam Transient segregated finite-volume solver of linear-elastic, small-strain deformation of a solid body, with optional thermal diffusion and thermal stresses.

solidEquilibriumDisplacementFoam Steady-state segregated finite-volume solver of linear-elastic, small-strain deformation of a solid body, with optional thermal diffusion and thermal stresses.