by flux directly (see subsection 2.4), in order to specify pressure boundary conditions through velocity boundary conditions, e.g. OpenFOAM’s fixedFluxPressure boundary condition, the OpenFOAM’s function of constrainPressure has to be called before solving pressure equation (see line 3 in Listing 2). For non-orthogonal mesh, a non-orthogonal correction algorithm (line 4 in Listing 2) is commonly adopted to improve accuracy for gradient computation. The number of non-orthogonal correction is specified by nNonOrthogonalCorrectors key in PIMPLE sub-dictionary in the system/fvSolution file.

**Listing 2** Implementation of pressure equation 5 with OpenFOAM (in pEqn.H).

```cpp
surfaceScalarField rhorAUf("rhorAUf", fvc::interpolate(rho*permeability/mu));
surfaceScalarField phig("phig", (fvc::interpolate(rho)*rhorAUf * g) & mesh.Sf());
constrainPressure(p, rho, U, phig, rhorAUf);
while (pimple.correctNonOrthogonal()){
    fvScalarMatrix pEqn(
        porosity*rho*betaT*fvm::ddt(p) - fvm::laplacian(rhorAUf,p)
        -porosity*rho*alphaP*fvc::ddt(T) + fvc::div(phig)
    );
    pEqn.solve();
}
```

4. The velocity field is calculated explicitly using latest pressure field based on Darcy’s law (Equation 1). Instead of calculating the velocity directly, we implement an indirect approach based on OpenFOAM’s function fvc::reconstruct to reconstruct the velocity field from the computed mass flux (see Listing 3), which performs higher numerical stability and benefits from the flux conservation characteristics of the finite volume method. In addition, boundary conditions of velocity field have to be updated (line 3 in Listing 3) if OpenFOAM’s fixedFluxPressure boundary condition is applied for pressure field.


```cpp
phi = phig + pEqn.flux();
U = permeability/mu*fvc::reconstruct(phi/rhorAUf);
U.correctBoundaryConditions();
```

5. Thermodynamic properties of fluid are updated by the thermophysical model after solving temperature and pressure field. The implementation code snippet is shown in Listing 4, in which thermo.correct() is used to update temperature and pressure value for all the calculating nodes. Then the thermodynamic properties of the fluid, for example density ($\rho$), at each nodes are calculated based on IAPWS-IF97 (see line 2-6 in Listing 4).