Study questions

1. How do you use the `interIsoFoam` solver in OpenFOAM?

    - To use the solver properly, we need to set the volume fractions using the `setAlphaField` utility. Then we need to add an `alpha.*` dictionary in the `fvSolution` file. This dictionary will specify the settings of the `interIsoFoam` solver.

2. How do we solve the governing equations of interfacial flows in `interIsoFoam`?

    - In the `interIsoFoam` solver we employ the isoAdvector algorithm for the advection of the interface and use the PIMPLE algorithm for the the pressure-velocity coupling.

3. How do we ensure a balance between the pressure and the gravitational force in the hydrostatic case?

    - We need to use a proper discretization of the gravitational force using the knowledge of the interface position.

4. How do we use the `interFlow` solver?

    - The solver is similar to the `interIsoFoam` solver. It needs an additional dictionary in the `transportpropoerties` file called `surfaceForces`. This dictionary will then determine which model we use in the solver.

5. What is the difference between the `interIsoFoam` and the `interFlow` solver?

    - The `interFlow` solver can be used so that it resembles the `interIsoFoam` solver. The `interFlow` solver offers an easier and faster way of testing new models for handling interfacial forces.

6. How do we add an extension to the acceleration model of the TwoPhaseFlow library?

    - We should place the extension in the `src/surfaceForces/accelerationForce/` folder. Similar thing can be added for surface force models in the corresponding folder.