Add functionality to an existing solver

- We will now start with the `interFoam` solver and the `damBreak` tutorial, and add solid particles both to the solver and tutorial.
- The `solidParticleCloud` class is available in OpenFOAM-1.5.x, but it is not used anywhere in the distribution.
- You can find lots of user contributions and documentation at the OpenFOAM Wiki: `http://openfoamwiki.net`
- I have added a minimalistic tutorial of the `solidParticleCloud` class at: `http://openfoamwiki.net/index.php/Contrib_solidParticleFoam`
- Start by downloading and running the `solidParticleFoam` solver and the `Box` tutorial.
- Continue with the `solidParticleDamBreak` tutorial at the same page.
- A course project could be to describe the class in high detail, to add physics to the class (look at `dieselEngineFoam`) and add an injector. Descriptions and implementations should be added in the Wiki and in the svn (I will help you).