* Install openFoam and PyFoam
* Download these files
* Open constant folder and unzip cmp you will get a directory called polymesh after extraction delete that tar file
* Run ***paraFoam*** in each and every directory and then check for any discrepancies
* Run ***checkMesh*** in each directory and check for orthogonality
* Now run ***pyFoamPlotRunner.py pimpleFoam*** in each and every directory i.e case