Case is set to run with *simpleFoam* and *kEpsilon* turbulance model. It can be changed in *FoamSetup* directory. To run tutorial do next steps:

1. Use attached python script *ahmed.py* or create your own python geometry scrip for *Salome* and apply changes explained on thread: *http://openfoamwiki.net/index.php/Thread:Talk:Sig_Numerical_Optimization/Salome_and_cfMesh_parametrization* (thread is printed in PDF inside tutorial folder: *Salome and cfMesh parametrization - OpenFOAMWiki.pdf*)
2. Edit *salomePythonScriptReader:*
   1. To locate *Salome* installation folder if needed
   2. To choose between *parametric* (multiple script run with change to python geometry file) way and *non-parametric* (script gets initiated only ones) way. Parametric is set by default.

   Or if you created your own geometry file edit all that is needed (files names and locations). Script is thoroughly explained on: *http://openfoamwiki.net/index.php/Thread:Talk:Sig_Numerical_Optimization/Salome_and_cfMesh_parametrization* (thread is printed in PDF inside tutorial folder: *Salome and cfMesh parametrization - OpenFOAMWiki.pdf*)
3. Run *salomePythonScriptReader* with command:
   ./[PATH]/salomePythonScriptReader ./PATH]/PYTHON_SCRIPT_NAME.py
   for *ahmed.py* runned from directory *Parametic:*
   ./salomePythonScriptReader ./ahmed.py
4. Run *stlCombine* with command:
   ./stlCombine [PATH TO DIRECTORY WITH .stl FILES] [PATH TO OUTPUT DIRECTORY]
   for this tutorial particularly, runned from directory *Parametic*:
   ./stlCombine ./ahmed_25/constant/triSruface/ ./ahmed_25/ or
   ./stlCombine ./ahmed_25/constant/triSruface/ ./ahmed_35/
   Created file is called *merged.stl* and **it needs to be located in base case directory.** In these tutorial it is:
   ./[PATH]/Parametric/ahmed_25/merged.stl
   ./[PATH]/Parametric/ahmed_35/merged.stl
5. Enter case base directory (*ahmed_25* or *ahmed_35*) and run *cartesianMesh* (cfMesh's cartesian mesher - *http://www.c-fields.com/Media/Default/PDF/User%20Guide%20-%20cfMesh%20v1.0.pdf*)
   Changing mesh parameters can be done in *meshDict* file located in *system* directory. All parameters of that file are very good explained in its documentation (*http://www.c-fields.com/technical-area/downloads/documentation-cfmesh#C1*)
6. You are ready to run OpenFOAM simulation! (Just check case settings: *controlDict, fvSchemes, ...*)