Laboratory tasks

1. Study directories and files in your OpenFOAM installation. Use command-line to list files and file contents.
2. Perform a lid-driven cavity simulation.

```
mkdir -p $FOAM_RUN
cp -r $FOAM_TUTORIALS $FOAM_RUN
cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity
# study files with ls, find, less and other commands
blockMesh
icoFoam
# study output of icoFoam solver
# study new files with ls, find, less and other commands
paraFoam
```

3. Visualize pressure distribution using paraFoam.
4. Visualize velocity vectors.
    1. Cell Centers from the Filter->Alphabetical menu (Vertex cells)
    2. Glyph from the Filter->Alphabetical
    3. Adjust Scale Mode
5. Visualize streamlines (Stream Tracer from the Filter menu).

Assignments

1. How many folders are inside OpenFOAM 2.2.2 $FOAM_APP folder?
2. How many PDF files are present in OpenFOAM 2.2.2 installation?
3. Where can you find icoFoam.C?
4. What is the symbolic form of UEqn in icoFoam.C?
5. What is the temporal derivative scheme in cavity tutorial?
6. What other temporal derivative schemes are possible?

Homework

1. Install OpenFOAM to your computer (e.g. using VirtualBox).
2. Read OpenFOAM user guide from chapter 2.1 until chapter 2.1.5.

Note

Online user guide: http://www.openfoam.org/docs/user/tutorials.php

UserGuide.pdf: `$WM_PROJECT_DIR/doc/Guides-a4/`