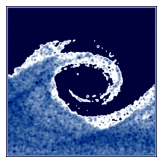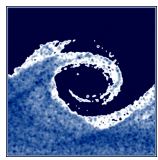# Laboratory tasks

1. Perform simulation of an airfoil.

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/airFoil2D $FOAM_RUN
cd $FOAM_RUN/airFoil2D
# Study files with ls, less, find, etc.
checkMesh > log.checkMesh.1
renumberMesh -overwrite >log.renumberMesh
checkMesh > log.checkMesh.2
# Study the log files!
meld log.checkMesh*
# Study the difference!
simpleFoam > log.simpleFoam.1
# Study new files with ls, less, find, etc.
# Study the solver logfile!
tail log.simpleFoam.1
# Note that the solution is converged.
```

Laboratory tasks

2. Create a changeDictionary file to change flow velocity.

```
head -n 17 system/controlDict > system/changeDictionaryDict
sed -i 's/controlDict/changeDictionaryDict/' system/changeDictionaryDict
echo "dictionaryReplacement
{
    U
    {
        boundaryField
        {
            inlet
            {
                freestreamValue uniform (15 -3 0);
            }
            outlet
            {
                freestreamValue uniform (15 -3 0);
            }
        }
    }
}" >> system/changeDictionaryDict
cat system/changeDictionaryDict
```

Laboratory tasks

3. Change boundary condition and continue simulation.

```
cp 350/U 350/U.org
changeDictionary
meld 350/U*
# Find all differences!
simpleFoam > log.simpleFoam.2
# Study the solver logfile!
tail log.simpleFoam.2
# Note that the solution is NOT converged.
cp system/controlDict system/controlDict.org
sed -i 's/500/1000/' system/controlDict
meld system/controlDict*
simpleFoam > log.simpleFoam.3
# Study the solver logfile!
tail log.simpleFoam.3
# Note that the solution is now converged.
paraFoam
```
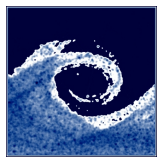
Laboratory tasks

4. Visualize streamlines for the two converged solutions!

- Stream tracer, Line source, Y axis
- Time = 350 s and 783 s

5. Plot residuals and number of iterations using gnuplot!

```
cat log.simpleFoam.* > log.simpleFoam
foamLog log.simpleFoam # Study files in logs directory!
gnuplot
```

```
plot 'logs/pIters_0' w l, 'logs/UxIters_0' w l, 'logs/UyIters_0' w l
set grid
plot 'logs/p_0' w l lw 3 t 'pressure', 'logs/Ux_0' w l lw 3 t 'x-velocity'
set term png size 800,450 font Arial 16
set out 'residuals.png'
set logscale y
set title 'Convergence of solution'
set xlabel 'Time [s]'
set ylabel 'Initial residuals'
replot
exit
```

```
display residuals.png
```
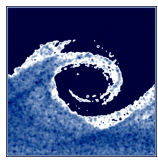
Laboratory tasks

6. Perform a simplified motorbike simulation and visualize results.

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/motorBike $FOAM_RUN
cd $FOAM_RUN/motorBike
sed -i 's/20 8 8/5 2 2/' constant/polyMesh/blockMeshDict
cp $FOAM_TUTORIALS/resources/geometry/motorBike.obj.gz constant/triSurface/
surfaceFeatureExtract > log.surfaceFeatureExtract
blockMesh > log.blockMesh
snappyHexMesh -overwrite > log.snappyHexMesh
cp -r 0.org 0
potentialFoam -noFunctionObjects -writep > log.potentialFoam
simpleFoam >log.simpleFoam
vorticity
foamCalc mag U
foamCalc components U
paraFoam
# Stream tracer, Line source, e.g. z axis
# Slice, e.g. y plane
# Study and learn features of ParaView, e.g. Contour, Clip, Opacity!
# Enable all Volume Fields, e.g. vorticity!
```
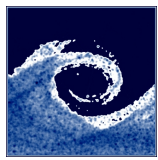
Assignments

1. Some utilities need dictionary files, e.g. *autoRefineMesh*. Where can you find an example dictionary file in OpenFOAM 2.2.2 installation?

2. What are the dimensions of field p in the *airFoil2D* case?

3. What is the average of velocity magnitude over the inlet patch in *airFoil2D*?

4. What is the integrated volume flow over the inlet patch in *airFoil2D*?

5. What are the dimensions of field B in the mhdFoam tutorial called *hartmann*?

6. Is the lift coefficient positive or negative for (15 -3 0) freestream velocity in *airFoil2D* case?
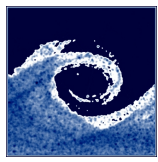
Homework

1. Read OpenFOAM UserGuide chapter 3.5-3.7.

2. Perform *motorBike* simulation again at home. Try to create better visualization in ParaView, save colourful images and send images to teachers by e-mail.

3. Use sample utility to post-process results of *motorBike* simulation. You can read OpenFOAM UserGuide chapter 6.5 if you need help. Use gnuplot to create png images of profiles.