Laboratory Session 4

Open-Source CFD Course 2021

Miklós BALOGH and Josh DAVIDSON

2021

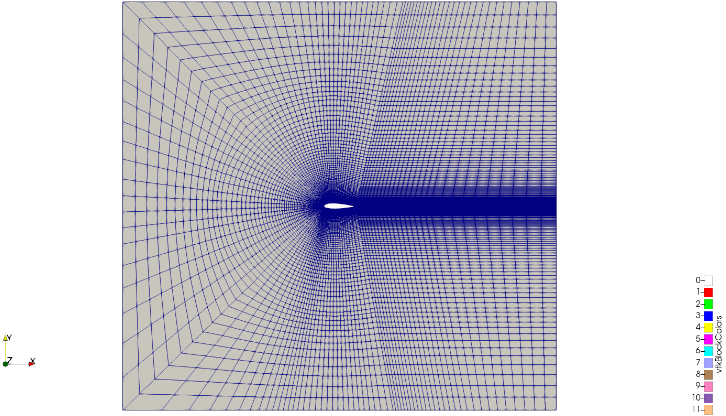
airFoil2D case

Laboratory Session 4

Miklós BALOGH and Josh DAVIDSON

Laboratory

Assignments



Laboratory tasks

1. Perform a simulation of flow around an airfoil:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/airFoil2D $FOAM_RUN
cd $FOAM_RUN/airFoil2D

### Study files with ls, less, find, etc. ###
#-------------------------------------------#
# Study the log files!
checkMesh > log.checkMesh.1
renumberMesh -overwrite >log.renumberMesh
checkMesh > log.checkMesh.2

# Study the difference!
meld log.checkMesh*

# Study the solver logfile!
simpleFoam > log.simpleFoam
tail log.simpleFoam

# Note that the solution is converged.
```

2. Perform 2nd simulation with a different flow velocity.

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/airFoil2D $FOAM_RUN/airFoilv2
cd $FOAM_RUN/airFoilv2
```

3. Although for this simple case we could change the boundary conditions manually, we will instead create a *changeDictionary* file to automate the change :

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

Laboratory tasks

4. Change boundary condition and run simulation:

```
cp 0/U 0/U.org
changeDictionary
meld 0/U*

renumberMesh -overwrite >log.renumberMesh

simpleFoam > log.simpleFoam

# Study the solver logfile!


paraFoam
```

Laboratory tasks

5. Visualize streamlines for the two converged solutions
6. Plot residuals and number of iterations using gnuplot:

```
foamLog log.simpleFoam
# Study files in logs directory! Than execute gnuplot:
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
```

7. Follow the steps below to 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/' system/blockMeshDict
cp $FOAM_TUTORIALS/resources/geometry/motorBike.obj.gz constant/triSurface/
surfaceFeatureExtract > log.surfaceFeatureExtract
blockMesh > log.blockMesh
snappyHexMesh -overwrite > log.snappyHexMesh
mkdir 0
cd 0
cp -r ../0.orig/* .
cd ..
potentialFoam > log.potentialFoam
simpleFoam >log.simpleFoam
vorticity
Q
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, Q!
```

Assignments

1. Provide a copy of your streamlines diagram(s) from Task 5.
2. Provide a copy of your "residuals.png" file from Task 6.
3. What are the dimensions of field p in the airFoil2D case?
4. What is the average of velocity magnitude over the inlet patch in airFoil2D?
5. What is the integrated volume flow over the inlet patch in airFoil2D?
6. Is the lift coefficient positive or negative for (15 -3 0) freestream velocity in airFoil2D case? What is its value?
7. Some utilities need dictionary files. If for example you wanted to use the *extrudeMesh* utility, where can you find an example dictionary file, *extrudeMeshDict* in your OpenFOAM installation?
8. What are the dimensions of field B in the mhdFoam tutorial called hartmann?