Other tutorials

- We will now learn how to use a number of useful utilities and libraries. Some of them are described in the UserGuide, and some of them have been discussed in the Forum. If you find a utility, solver or library that is not described anywhere, you can include a description of it in your tutorial. I would prefer if you then use my LaTeX slide template, so that I can easily use it in coming courses. This will of course contribute to your grade.
- At the end of this part we will learn how to search for additional tutorials and other information on the use of different applications and libraries.

The mapFields utility

- The mapFields utility maps the results of one case to another case. You will use this utility when you do the tutorials in the UserGuide.
- Usage:

```
mapFields <source dir> [-consistent] [-parallelSource]
    [-sourceTime scalar] [-case dir] [-parallelTarget]
    [-help] [-doc] [-srcDoc]
```

  (You get this information by simply typing `mapFields -help`)
- The time used for the mapping is specified by `startFrom/startTime` in the *target* case.
- The flag `-consistent` is used if the geometry and boundary conditions are identical in both cases. This is useful when modifying the mesh density of a case. For non-consistent cases a `mapFieldsDict` dictionary must be edited, see the `icoFoam/cavityClipped` tutorial.
- The flags `-parallelSource` and `-parallelTarget` are used if any, or both, of the cases are decomposed for parallel simulations.

The sample utility

- This is used to produce graphs for publication or to extract surfaces. You will use this utility when you do the tutorials of the UserGuide, and when you do the ercoftacConicalDiffuser case-study.
- Usage:

```
Usage: sample [-noZero] [-case dir] [-parallel] [-constant]
    [-latestTime] [-time time] [-help] [-doc] [-srcDoc]
```

- Copy and modify `sampleDict` in the `solidDisplacementFoam plateHole` tutorial. In this case a line with 100 sample points is defined along the `leftPatch`, between `(0 0.5 0.25);` and `(0 2 0.25);`
- An example of a gnuplot command that uses the output and compares it to an analytical solution:

```
plot [0.5:2] '<datafile>', 1e4*(1+(0.125/(x**2))+(0.09375/(x**4)))
```

- `sample` can also be used to sample surfaces.
- See the UserGuide for more details on `sample`

The foamCalc utility

- This utility calculates new fields from existing ones. This replaces some of the utilities in previous versions of OpenFOAM, such as `magU` and `Ucomponents`.
- Usage:
  `foamCalc <calcType> <fieldName1 ... fieldNameN>`
- To get a list of available `<calcType>`s, write:
  `foamCalc xxx` and get the following list:
  `div, components, mag, magGrad, magSqr`
- Examples:
  `foamCalc div U`
  `foamCalc components U`
- The new fields are written in the time directories.

The setFields utility

- The `setFields` utility is used to set values to the fields in specific regions. You will use this when you do the tutorials in the UserGuide.
- Usage:
```
setFields [-noZero] [-case dir] [-parallel] [-constant]
    [-latestTime] [-time time]  [-help] [-doc] [-srcDoc]
```
- A `setFieldsDict` dictionary is used. Find an example in `$FOAM_TUTORIALS/interFoam/damBreak/system/setFieldsDict`.
  The `defaultFieldValues` sets the default values of the fields. A `boxToCell` bounding box is used to define a set of cells where the `fieldValues` should be different than the `defaultFieldValues`.

The funkySetFields utility

- The `funkySetFields` utility is a user contributed utility available in the OpenFOAM Wiki:
  `http://openfoamwiki.net`
  Click Contributions/Utilities/preProcessing
- `funkySetFields` is a development of the `setFields` utility, and it includes the option of specifying mathematical expressions etc.

Probes

- Probes the development of the results during a simulation, writing to a file. This is an example of `functionObjects`, which was introduced in OpenFOAM-1.4. With `functionObjects` you can add libraries with more functionality to existing solvers without recompiling the solver. You can find some more `functionObjects` in OpenFOAM-1.5, and also in the OpenFOAM Wiki (`www.openfoamwiki.net`), and in the OpenFOAM-extend project (`www.sourceforge.net`).
- Copy and modify the `functions` part at the end of
  `$FOAM_TUTORIALS/oodles/pitzDaily/system/controlDict`
  to your case and run it.
- Visualize the output file in Matlab after removing the first line in the output file. It is probably easy to visualize also with gnuplot and xmgrace.

decomposePar

- `decomposePar` makes a domain decomposition for parallel computations. This is described in the UserGuide.

- Usage:

```
decomposePar [-ifRequired] [-force] [-cellDist] [-case dir]
     [-fields] [-filterPatches] [-copyUniform]  [-help]
     [-doc] [-srcDoc]
```

- A `decomposeParDict` specifies how the grid should be decomposed. An example can be found in `$FOAM_TUTORIALS/interFoam/damBreak/system/decomposeParDict`.

- There are some different alternatives for which method to use for the decomposition. See the UserGuide. `numberOfSubdomains` specifies the number of subdomains the grid should be decomposed into. Make sure that you specify the same number of subdomains in the specific decomposition method you will use, otherwise your simulation might not run optimal.

- We will discuss parallel processing later on.

reconstructPar

- `reconstructPar` is the reverse of `decomposePar`, reassembling the grid and the results.
- Usage:

```
reconstructPar [-noZero] [-region name] [-case dir] [-constant]
     [-latestTime] [-time time]  [-help] [-doc] [-srcDoc]
```

- This is usually done for post-processing, although it is also possible to post-process each domain separately by treating an individual processor directory as a separate case when starting `paraFoam`.
- We will discuss parallel processing later on.

Specifying a maximum Courant number and varying time steps

- Some solvers, like the `interFoam` solver allows a varying time step, based on a maximum Courant number. Some extra entries should then be added to the controlDict dictionary:

```
adjustTimeStep  yes;
maxCo           0.5;
maxDeltaT       1;
```

- The solver is told to adjust the time step so that the output still occurs at specific times using:

```
writeControl    adjustableRunTime;
writeInterval   0.05;
```

How to search for other tutorials

- Type `tut` to go to the `$FOAM_TUTORIALS` directory. Here you find many case-setups for the solvers in OpenFOAM. Unfortunately, they are not well-described. Describing these tutorials may be part of your project.

- Note that it is recommended to copy the tutorials to your `$WM_PROJECT_USER_DIR/run` directory before running them or making any modification to them, so that you always have a clean version of the tutorials. If you use the pre-installed version at Chalmers you actually cannot modify files in the installation anyway.

- There are no specific tutorials for the utilities, but some of the solver tutorials also show how to use the utilities. We will later learn how to find those using Linux search commands.

- All the solver tutorials have `Allrun` scripts that describe the use of those tutorials. We will now have a look at the `Allrun` script of the `$FOAM_TUTORIALS/icoFoam` tutorials. This is actually what you will do manually when you do the `cavity` tutorials in the UserGuide. In other words, you can use the `Allrun` script as a short summary of the description in the UserGuide.

Run the icoFoam cavity tutorials using the Allrun script (1/5)

(Note that I did not have time to make sure that the description below is without any errors after I converted it from 1.4.1 to 1.5. The principle is however correct.)

In the `icoFoam` tutorial directory there is an `Allrun` script.
When running this script it is preferred to copy the entire `tutorials` directory.

If you run the `Allrun` script for the icoFoam cavity tutorials you actually
first run the cavity case

```
#Running blockMesh on cavity:
blockMesh
#Running icoFoam on cavity:
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (2/5)

If you run the `Allrun` script for the icoFoam cavity tutorials you actually **then run the cavityFine case**:

```
#Cloning cavityFine case from cavity:
mkdir cavityFine
cp -r cavity/{0,system,constant} cavityFine
    [change "20 20 1" in blockMeshDict to "41 41 1"]
    [set startTime in controlDict to 0.5]
    [set endTime in controlDict to 0.7]
    [set deltaT in controlDict to 0.0025]
    [set writeControl in controlDict to runTime]
    [set writeInterval in controlDict to 0.1]
#Running blockMesh on cavityFine
blockMesh
#Running mapFields from cavity to cavityFine
mapFields -case cavity -sourceTime latestTime -consistent
#Running icoFoam on cavityFine
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (3/5)

If you run the `Allrun` script for the icoFoam cavity tutorials you actually
then run the cavityGrade case:

```
#Running blockMesh on cavityGrade
blockMesh
#Running mapFields from cavityFine to cavityGrade
mapFields -case cavityFine -sourceTime latestTime -consistent
#Running icoFoam on cavityGrade
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (4/5)

If you run the `Allrun` script for the icoFoam cavity tutorials you actually **then run the cavityHighRe case**:

```
#Cloning cavityHighRe case from cavity
mkdir cavityHighRe
cp -r cavity/{0,system,constant} cavityHighRe
#Setting cavityHighRe to generate a secondary vortex
    [set startFrom in controlDict to latestTime;]
    [set endTime in controlDict to 2.0;]
    [change 0.01 in transportProperties to 0.001]
#Copying cavity/0* directory to cavityHighRe
cp -r cavity/0* cavityHighRe
#Running blockMesh on cavityHighRe
blockMesh
#Running icoFoam on cavityHighRe
icoFoam
```

Run the icoFoam cavity tutorials using the Allrun script (5/5)

If you run the `Allrun` **script for the icoFoam cavity tutorials you actually then run the cavityClipped case**:

```
#Running blockMesh on cavityClipped
blockMesh
#Running mapFields from cavity to cavityClipped
cp -r cavityClipped/0 cavityClipped/0.5
mapFields -case cavity -sourceTime latestTime
    [Reset the boundary condition for fixedWalls to:]
    [        type              fixedValue;          ]
    [        value             uniform (0 0 0);     ]
    [      We do this since the fixedWalls got      ]
    [   interpolated values by cutting the domain   ]
#Running icoFoam on cavityClipped
icoFoam
```

Run all the tutorials using the Allrun scripts

- You can run a similar script, located in the tutorials directory, and also named `Allrun`. This script will run through all the tutorials (calls `Allrun` in each solver directory).
- You can use this script as a tutorial of how to generate the meshes, how to run the solvers, how to clone cases, how to map the results between different cases etc.

Finding tutorials for the utilities in OpenFOAM

- There are no tutorials for the utilities, but we can search for examples:

```
find $WM_PROJECT_DIR -name \*Dict | grep -v blockMeshDict | \
                     grep -v controlDict
```

You will get a list of example dictionaries for some of the utilities.

Now you should be ready to go on exploring the applications by yourself.

More tutorials can be found in

- The Programmer's guide, chapter 3
- The OpenFOAM Wiki
- The OpenFOAM Forum
- The Internet