UserGuide, outline

- Available in `$WM_PROJECT_DIR/doc/Guides-a4/UserGuide.pdf`
- Introduction to OpenFOAM
- Tutorials showing how to run some *applications* (*solvers* and *utilities*)
- Descriptions of case organization
- Pre- and post-processing using blockMesh, snappyHexMesh (we will have a look at this later), paraFoam and third-party products
- General descriptions of applications and libraries in OpenFOAM.
- Lists of available applications and libraries (not necessarily complete - we will use the source code to generate a complete list)
- Information on some code structure and how to compile applications.

UserGuide, Ch.1, Introduction

- OpenFOAM is first and foremost a *C++ library*, used primarily to create executables, known as *applications*. The applications fall into two categories: *solvers*, that are each designed to solve a specific continuum mechanics problem; and *utilities*, that are designed to perform tasks that involve data manipulation.

- OpenFOAM is distributed with a large number of applications, but soon any advanced user will start developing new applications for his/ her special needs. The basic way to do this is to find and copy an application that almost does what is needed, and then to modify it by copy/paste from other applications that has some features that are needed.

- Special applications for pre- and post-processing are included in OpenFOAM. Converters to/from other pre- and post-processors are available.

UserGuide, Ch.2, Tutorials

- Set-up, simulation and post-processing of some cases.
- Some solvers and utilities are introduced.
- You will go through this chapter yourself and do the tutorials.
- We will quickly go through the basic procedure now, and you will learn how to find similar instructions in the source code.
- Also view the tutorials in Chapter 2 as examples of the tutorial you should produce in your project.
- OpenFOAM provides 'tutorials' in `$FOAM_TUTORIALS`, but most of those are just case files without explanations. Some guiding can however be found in `Allrun` scripts that are provided with the tutorials. We will compare the `icoFoam/cavity Allrun` script with the tutorials in Chapter 2 of the UserGuide. There is an alias to go to `$FOAM_TUTORIALS`, which is `tut`

UserGuide, Ch.2, Tutorials

- Some useful aliases:

```
app      cd $FOAM_APP
foam     cd $WM_PROJECT_DIR
foamfv   cd $FOAM_SRC/finiteVolume
foamsrc  cd $FOAM_SRC/$WM_PROJECT
lib      cd $FOAM_LIB
run      cd $FOAM_RUN
sol      cd $FOAM_SOLVERS
src      cd $FOAM_SRC
tut      cd $FOAM_TUTORIALS
util     cd $FOAM_UTILITIES
```

  Type `alias` for a complete list

- We will have a look at environment variables when looking at installation of OpenFOAM. Find out what an environment variable means by: `echo $FOAM_APP`. See all the environment variables by: `env`.

UserGuide, App. A, FoamX - the old GUI

- For earlier versions of OpenFOAM there was a GUI named `FoamX`.
- `FoamX` did not add anything else than installation problems, so it was removed in OpenFOAM-1.5.
- You will see that it is very easy to control OpenFOAM by file editing.
- If you want to make sure that you don't want to use `FoamX`, you have the opportunity to test version 1.4.1, which is also installed in `/chalmers/sw/unsup/OpenFOAM`. Follow the instructions in the slides from the course last year.