

This course material is free: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation. It is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.

More details about the GNU General Public License can be seen at:  
<<http://www.gnu.org/licenses/>>.

The essence of today's course is to help users to understand the difference and move from local machines (desktop, laptop, workstation) to one of the supercomputers of SCC (UC3=bwUniCluster or HoreKa).

You need separately a program to copy files from/to the supercomputer. For this you may use WinSCP (windows), MobaXterm (windows), filezilla (linux) etc. You also need a tool to type the commands (called terminal) – this can be e.g. PuTTY, or a local linux partition on your local computer. There are many different ways of doing those tasks, we cannot list all of them here. Our computers can be accessed by:

uc3.scc.kit.edu (bwUniCluster) or horeka.scc.kit.edu (HoreKa)

Local computer:  
Serial computations

1. Install openFOAM

2. Use local HDD

3. Start directly:  
pimpleFOAM

4. View in ParaView

Supercomputer:  
parallel simulations

1. „module load ...“ + foamInit

2. Use workspaces

3A. decomposePar  
(for parallel jobs)

3B. Edit a job-file and submit a job  
to one of the queues

3C. **Do not use the Allrun-script** in  
case of large parallel runs! Grid  
generation is not parallelized and  
**Allrun is very bad for large cases.**  
Use the steps we teach you today!

4A. reconstructPar  
(optional but recommended)

4B. Copy result-files  
locally and view in  
ParaView

A typical OpenFOAM case in the **run-directory** (or, **case-directory**, or **main-directory**) is organized into a few standard folders that separate setup, properties, and initial data. The **system directory** contains the files that control how the simulation is run, such as controlDict, fvSchemes, and fvSolution, which define time control, discretization, and solver settings. The **constant directory** stores information that stays fixed during the run, for example the mesh in **polyMesh subdirectory** and physical properties such as transportProperties or turbulence-model data. The **0.orig directory** usually serves as a template for the initial time folder: it contains the starting field files, such as U for velocity and p for pressure, together with their boundary conditions, and is often copied or renamed to 0 before a case is started.

**Summary:** system = run control, constant = case properties, 0.orig = initial fields.

