HIGH PERFORMANCE RESEARCH COMPUTING

Introduction to OpenFOAM

March 22nd 2024

Björn Windén

High Performance Research Computing
DIVISION OF RESEARCH

Agenda

- Background
- What is OpenFOAM? How is it different from other CFD packages?
- Basic Structure of an OpenFOAM case. Basic Syntax.

Hands-on

Log into Grace

- Running a basic tutorial. Step-by-step.
- Post-processing with ParaView

5 min break

- Modifying a tutorial case to suit your needs
- Implementing new methods
- Running OpenFOAM in parallel

- Q&A

My OpenFOAM Background

- Started using OpenFOAM in 2010 (OpenFOAM 1.7)
- Ship hydrodynamics
  - High Re (10^6 - 10^{10}) flows
  - Multiphase
  - Propeller/Hull/Rudder interaction
- Currently Professor in Ocean Engineering

What is OpenFOAM?

- Open **F**ield **O**peration **A**nd **M**anipulation
- Originated as the FOAM library for fluid/solid mechanics in the 1990s. For "field operation and manipulation"
- Based on C++ rather than FORTRAN -> Modular and easily expandable!
- Released under GPL in 2004 as OpenFOAM
- Now a huge, community-driven, toolbox of methods. Mainly for CFD. But also other applications that benefit from its matrix solvers (field operation/manipulation)
- Many different development tracks, commercial and open source
- Runs on UNIX-like systems (Linux, macOS)

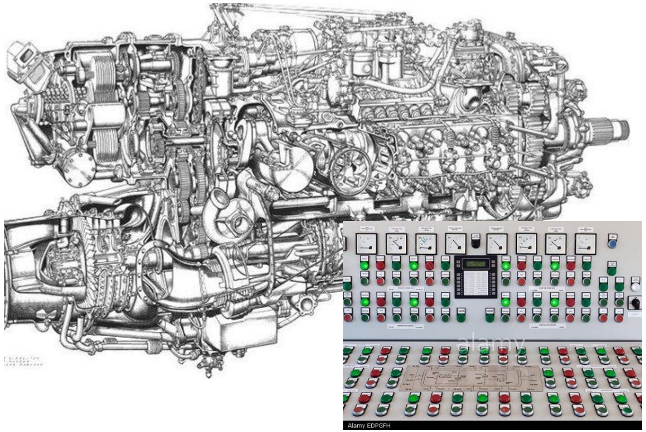
What is OpenFOAM?

OpenFOAM

OpenFOAM v.X Non-profit track, by the OpenFOAM foundation	**OpenFOAM v.YYMM** Commercial track by ESI OpenCFD	**FOAM-extend** Community project, lead by some of the original FOAM developers	**Others** In-house and other commercial/non-comme rcial ventures
Latest: **OpenFOAM v.11** https://openfoam.org	Latest: **OpenFOAM v.2312** www.openfoam.com	Latest: **foam-extend-5.0** http://foam-extend.org	BlueCFD https://bluecfd.github.io For ship hydrodynamics: NavyFOAM SHORTCUt ...

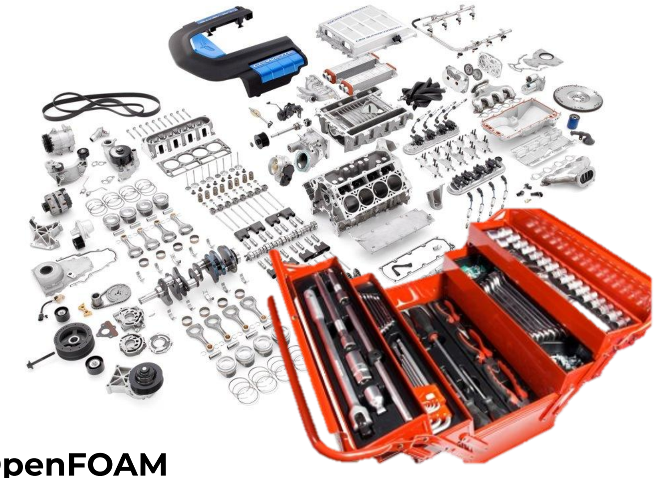
Today's course
(except we'll use OF v.10)

How is OpenFOAM Different?



Commercial Package (e.g. FLUENT, STAR etc.)

- Pre-built, working out of the box (?)
- User controls the software using GUI
- User has limited knowledge/influence about what goes on behind the scenes
- <u>Professional support available</u>



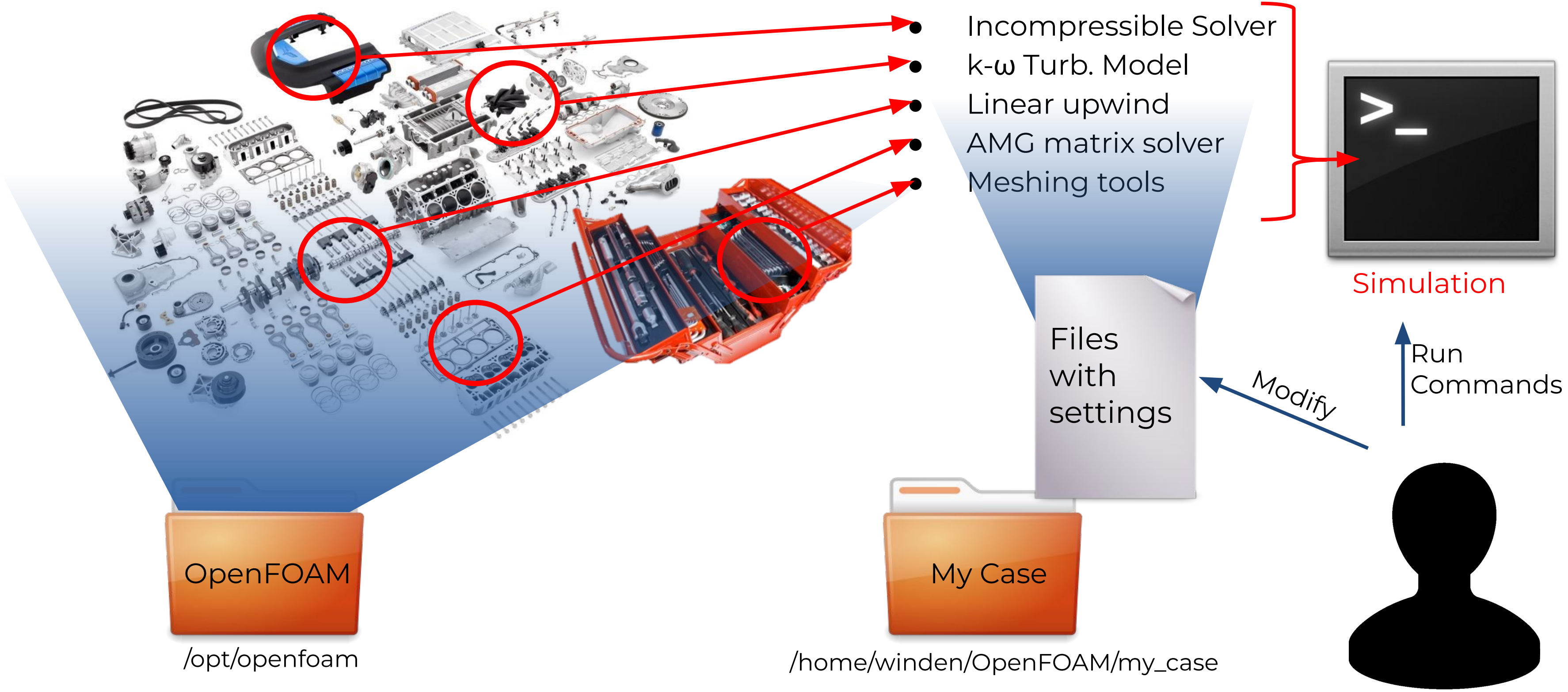
OpenFOAM

- Collection of components and tools
- User can combine appropriate components and tools to solve a particular problem
- User can add new components
- <u>Pre-assembled tutorials available</u>

How is OpenFOAM Different?

Commercial Package (e.g. FLUENT, STAR etc.)	OpenFOAM
+	**+**
• Easier to learn • Support available • Solvers are more stable • Controlled from Windows GUI	• Customizable • No license fee. No additional cost for HPC • Active user community. Especially in academia. Pre-made setups can be downloaded online • Many community-made solvers and tools
–	**–**
• Expensive. Especially for HPC • Not easy to customize • Sometimes too generous with bad user input • Novel methods may take time to implement	• Steeper learning curve. • Solvers are more unforgiving (unstable) • To take full advantage, user needs knowledge of C++ and scripting

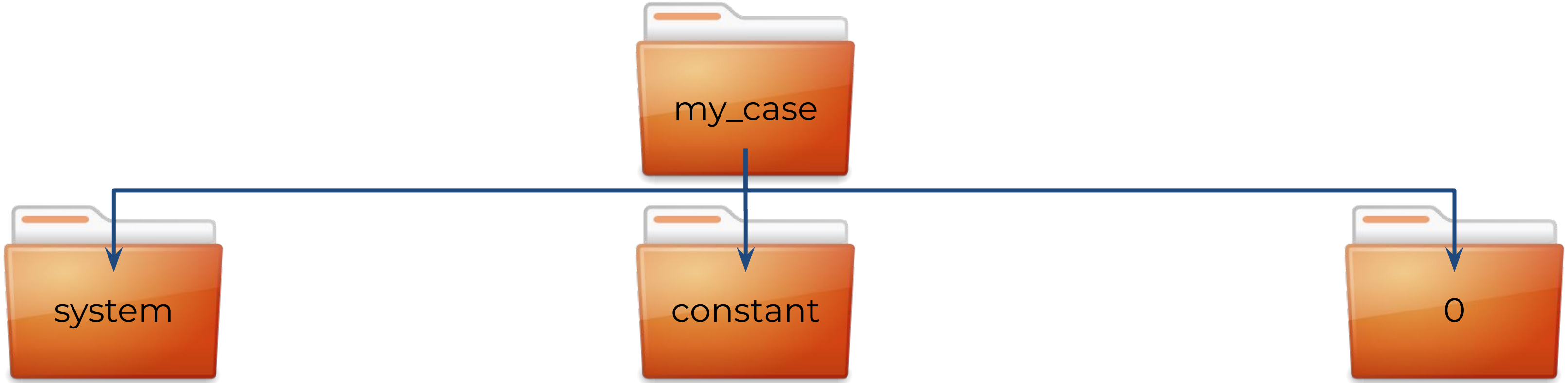
Basic Structure of an OpenFOAM case



Basic Structure of an OpenFOAM case

- Solvers and tools are stored in a central location (where OpenFOAM was compiled).
- These are made available by sourcing the bashrc file for the appropriate version, or using module load
- Multiple versions can be installed on the same system. The sourced/loaded one will be used
- “Case” files are stored on the users profile e.g. /home/<user>/OpenFOAM/my_case

Basic Structure of an OpenFOAM case



Main settings controlling the solver etc. (Run-time modifiable)

Start/End Time, Time Step
Solver Settings, Numerical Schemes
Mesh refinement settings

Things that are constant

Physical Properties, environmental constants
Turbulence Model
Mesh

Time step data
(0 is default start time)

Boundary conditions
Initial conditions

Basic Structure of an OpenFOAM case

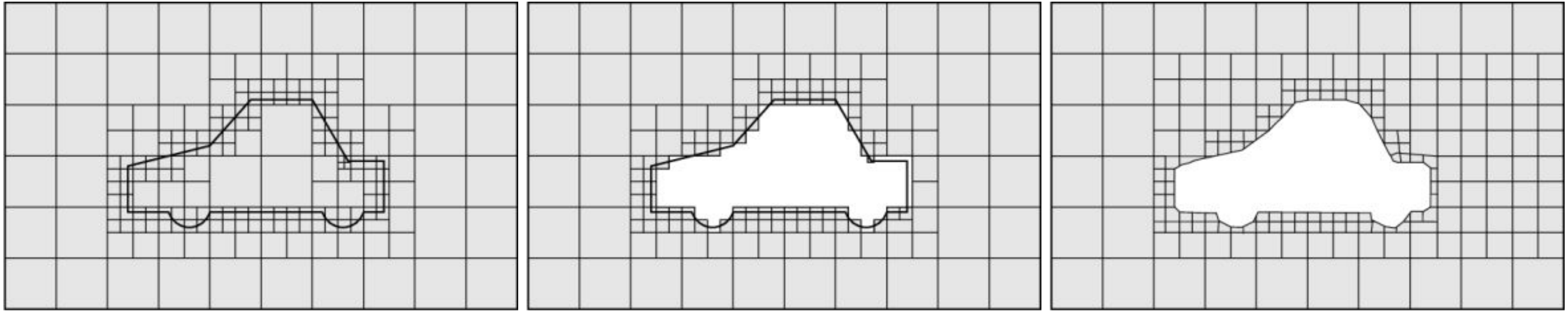
- Tutorials usually contain an “Allrun” script that contains all the commands required to run the tutorial; typed out in sequence. Just run that script to complete the tutorial.
- In the hands-on section we will run these manually and discuss what they do.
- In essence, to run an OpenFOAM case, cd to the case file (that contains 0,constant,system), type the name of the command. Look at allrun script to get clues what commands need to be run.
- Let’s try it!

Start Hands-on Session

snappyHexMesh

- Parallel and automatic quality control. Semi-automatic meshing.
- [A Comprehensive Guide To snappyHexMesh]()

Refine Split Snap



simpleFoam

- Run to end time in system/controlDict

OR

- Stop based on residual control in system/fvSolution

Acknowledgements

This work was supported by

- The National Science Foundation (NSF), award number:
  - 1925764 - SWEETER - SouthWest Expertise in Expanding, Training, Education and Research

- Staff and students at Texas A&M High-Performance Research Computing.

High Performance
Research Computing
DIVISION OF RESEARCH

https://hprc.tamu.edu

HPRC Helpdesk:

help@hprc.tamu.edu
Phone: 979-845-0219

Help us help you. Please include details in your request for support, such as, Cluster (Faster, Grace, Terra, ViDaL), NetID (UserID), Job information (Job id(s), Location of your jobfile, input/output files, Application, Module(s) loaded, Error messages, etc), and Steps you have taken, so we can reproduce the problem.