Introduction to OpenFOAM

March 22\textsuperscript{nd} 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 Field Operation And Manipulation**
- 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 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-commercial 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...
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
- Professional support available

OpenFOAM

- Collection of components and tools
- User can combine appropriate components and tools to solve a particular problem
- User can add new components
- Pre-assembled tutorials available
# How is OpenFOAM Different?

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

- Incompressible Solver
- k-ω Turb. Model
- Linear upwind
- AMG matrix solver
- Meshing tools

Simulation

Files with settings

Modify

Run Commands

OpenFOAM

/home/winden/OpenFOAM/my_case

My Case

/opt/openfoam
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

**my_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
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.
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.