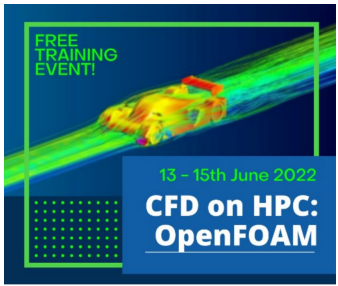
Literature and Tools



EuroCC workshop

Aleksander GRM

13-15, June 2022

Literature

There are two main web pages with links to literature:

- openfoam.org (on PC workstations)
- cfd.direct (on HPC systems)

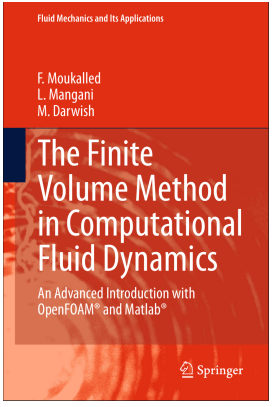
User guides:

- User Guide #1
- User Guide #2
- Programming Guide – Learn c++ code!

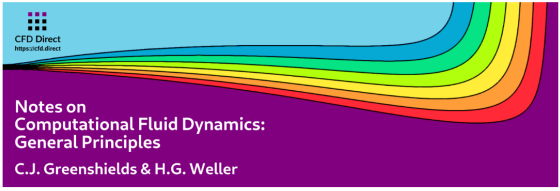
My GitHub repository:

- OpenFOAM_School@github

Link to my GoogleDrive location with books for OpenFOAM



link to the book



About the Book

Notes on Computational Fluid Dynamics (CFD) was written for people who use CFD in their work, research or study, providing essential *knowledge* to perform CFD analysis with confidence. It offers a modern perspective on CFD with the finite volume method, as implemented in OpenFOAM and other popular general-purpose CFD software. Fluid dynamics, turbulence modelling and boundary conditions are presented alongside the numerical methods and algorithms in a series of short, digestible topics, or *notes*, that contain complete, concise and relevant information. The book benefits from the experience of the authors: Henry Weller, core developer of OpenFOAM since writing its first lines in 1989; and, Chris Greenshields, who has delivered over 650 days of CFD training with OpenFOAM.

ISBN 978-1-3999-2078-0, 291 pages.

link to the book

Tools

Set up Python environment with GMSH

To be able to run advanced GMSH examples we need to set up Python environment

```
1. load module python:
  $> ml av python (check target version)
  $> ml python-version

2. Create new env:
  $> python3 -m venv local

3. Activate new env:
  $> source local/bin/activate

4. Install new packages (active env local):
  $(local)> pip install numpy scipy sympy matplotlib
  $(local)> pip install --upgrade gmsh
```

Load **only** Python environment

To use Python environment we need only to load it

```
1. load module python:
  $> ml av python (check target version)
  $> ml python-version

2. Activate new env:
  $> source local/bin/activate
```

Load OpenFOAM environment

To use OpenFOAM environment we need to load

```
List available modules:
    $> module avail openfoam
    $> ml av openfoam (equvalent with upper command)

Load specific module:
    $> module load gnuplot/5.0.5-foss-2016b
    $> ml gnuplot/5.0.5-foss-2016b (equvalent with upper command)

For OpenFOAM to be running on HPC@ULFS we need to load this modules:
    $> ml openfoam-2112-gcc-11.2.0-lhrpyq4
    $> ml ml qt-5.15.3-gcc-8.5.0-scmeit7 (graphics libs for gnuplot)
    $> ml gnuplot/5.0.5-foss-2016b (for foamMonitor application)

    and load you Python evn with
    $> source work/Python/local/bin/activate (or path where your local
    Python is)
```

Add the following part at the end in `system/controlDict`

```
functions
{
    #includeFunc  residuals
}
```

Create residual dictionary file `system/residuals` and include

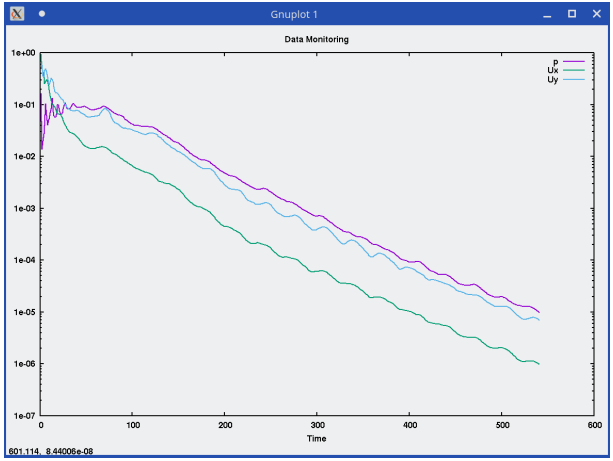
```
#includeEtc "caseDicts/postProcessing/numerical/solverInfo.cfg"

fields  (p U);
```

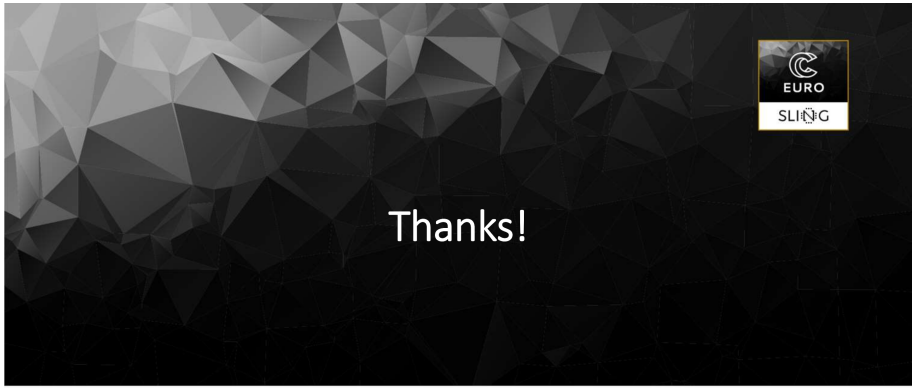
Run monitor command

```
foamMonitor -l -r 1 postProcessing/residuals/0/solverInfo.dat
```

```
foamMonitor -l -r 1 postProcessing/residuals/0/solverInfo.dat
```



Sponsorship



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951732. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, United Kingdom, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Switzerland, Turkey, Republic of North Macedonia, Iceland, Montenegro

EuroHPC
Joint Undertaking