



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

Introduction to OpenFOAM®

Daniel Wildt

SWARM Summer School
15 – 26 November 2021

23rd November 2021

Outline I



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

General information about OpenFOAM®

- History

- Concept

Tutorials

- Lid-driven cavity

- Dam break

- Other tutorials

Literature and resources

- Documentations by distributors

- Wikis and useful tools

Open ∇ FOAM®

General information about OpenFOAM®



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

- ▶ **Open-source Field Operation And Manipulation**
- ▶ open-source collection of CFD solvers
- ▶ includes also pre- and postprocessing tools
- ▶ no graphical user interface
- ▶ other external software used for displaying results, e. g.
 - ▶ Paraview
 - ▶ Gnuplot

```
/*----- C++ -----*/
//
//  F i e l d
//  O p e r a t i o n
//  A n d
//  M a n i p u l a t i o n
//
//  OpenFOAM: The Open Source CFD Toolbox
//  Website:  https://openfoam.org
//  Version:  6
//
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "system";
    object       controlDict;
}
// ..... //
application     pisoFoam;
startFrom       latestTime;
startTime       0;
```

 **ParaView**



General information about OpenFOAM® – histroy



swarm



University of Natural Resources
and Life Sciences, Vienna



Imperial College
London

- ▶ development started by Henry Weller in 1989 at Imperial College London
- ▶ commercially offered from 2000 to 2004 by Nabla Ltd. as FOAM
- ▶ Nabla Ltd. folded in 2004, code offered as OpenFOAM under GNU General Public License
 - ▶ Henry Weller, Chris Greenshields and Mattjis Janssens founded OpenCFD Ltd.
 - ▶ Hrvoje Jasak founded consulting company Wikki Ltd.

OpenCFD® WIKKI

Wikipedia contributors (2021)

General information about OpenFOAM® – today



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

Two parallel version today

- ▶ Foundation
 - ▶ <https://cfdirect.com>; download from <https://openfoam.org>
 - ▶ “.org-version”
 - ▶ current version: OpenFOAM 9
- ▶ ESI group
 - ▶ <https://www.openfoam.com/>
 - ▶ “.com-version”
 - ▶ current version OpenFOAM v2106



CFD Direct



General information about OpenFOAM®



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

- ▶ no support from developers
- ▶ trainings offered by different groups but expensive
- ▶ official user guide relatively short
- ▶ additional programmers and tutorials guide available
- ▶ large set of readily set-up tutorial cases covering most of OpenFOAMs' functionality

 OpenFOAM
The OpenFOAM Foundation

User Guide
version 8

22nd July 2020

<https://openfoam.org>

OpenFOAM

The Open Source CFD Toolbox

Programmer's Guide

Version: v1806
03 July 2018

General information about OpenFOAM®



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

A selection of solvers (OpenCFD, Ltd., 2020c):

- ▶ basic e. g.
 - ▶ laplacianFoam
 - ▶ scalarTransportFoam
- ▶ compressible
- ▶ incompressible
 - ▶ icoFoam
 - ▶ pimpleFoam
 - ▶ pisoFoam
 - ▶ shallowWaterFoam
 - ▶ simpleFoam
- ▶ lagrangian
- ▶ multiphase
 - ▶ stressAnalysis
 - ▶ interFoam
 - ▶ driftFluxFoam
- ▶ financial

check User Guide (OpenCFD, Ltd., 2020c) for full details and full list of solvers

General information about OpenFOAM®



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

Working with OpenFOAM

- ▶ cases set-up by defining variables in dictionaries (textfiles), e. g.:
 - ▶ p, U, ... for BCs
 - ▶ controlDict for timestep, etc.
- ▶ basic case structure includes folders
 - ▶ 0 (initial/bound. cond.)
 - ▶ constant
 - ▶ system

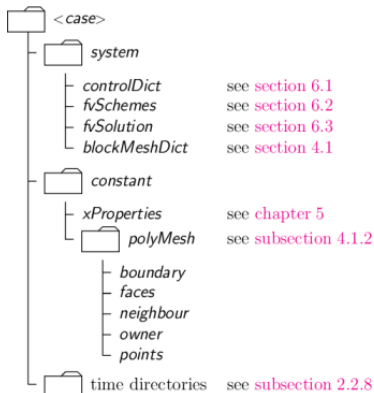


Fig.: OpenCFD, Ltd. (2020c)

Tutorials: Lid-driven cavity



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

The “Hallo World” case of OpenFOAM

- ▶ create mesh: `blockMesh`
- ▶ view mesh in Paraview:
`paraFoam`
- ▶ check mesh: `checkMesh`
- ▶ run solver: `icoFoam`
- ▶ post-process in Paraview

check Tutorial Guide (OpenCFD,
Ltd., 2020b) for details

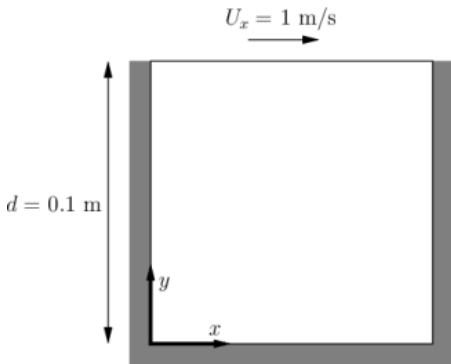


Fig.: OpenCFD, Ltd. (2020b)

Tutorials: Lid-driven cavity



swarm



- ▶ Meshing using blockMesh
- ▶ mesh and geometry defined in system/blockMeshDict

for details check User Guide (OpenCFD, Ltd., 2020c) section “Mesh generation and conversion”

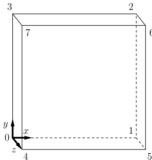


Fig.: OpenCFD, Ltd. (2020b)

```
----- C++ -----
//      F i e l d      | OpenFOAM: The Open Source CFD Toolbox
//      O p e r a t i o n | Website: https://openfoam.org
//      A n d           | Version: 6
//      M a n i p u l a t i o n
//
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       blockMeshDict;
}
// .....

convertToMeters 0.1;

vertices
(
    (0 0 0)
    (1 0 0)
    (1 1 0)
    (0 1 0)
    (0 0 0.1)
    (1 0 0.1)
    (1 1 0.1)
    (0 1 0.1)
);

blocks
(
    hex (0 1 2 3 4 5 6 7) (20 20 1) simpleGrading (1 1 1)
);

edges
(
);

boundary
(
    movingWall
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }
    fixedWalls
    {

```

Tutorials: Lid-driven cavity



**University of Natural Resources
and Life Sciences, Vienna**
Department of Water, Atmosphere
and Environment

Other important dictionaries:

- ▶ boundary and initial conditions `0/p`, `0/U`
 - ▶ physical properties
`constant/transportProperties`
 - ▶ discretization `system/fvScheme`
 - ▶ solver settings `system/fvSolution`
 - ▶ control `system/controlDict`
 - ▶ transient solvers set time step `deltaT`
 - ▶ steady state solvers time step corresponds to solver iteration \Rightarrow `set deltaT 1`
-
- Daniel Wildt Summer School

[illegible]

Tutorials: break

Dam



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

A multiphase case, using the Volume of Fluid method (Euler-Euler)

- ▶ phase variable α
 - ▶ air: $\alpha = 0$; water: $\alpha = 1$
- ▶ initial field defined in `system/setFieldsDict`
- ▶ initialise using `setFields`

check Tutorial Guide (OpenCFD,
Ltd., 2020b) for details
possible tasks: refine mesh, extend
simulation time

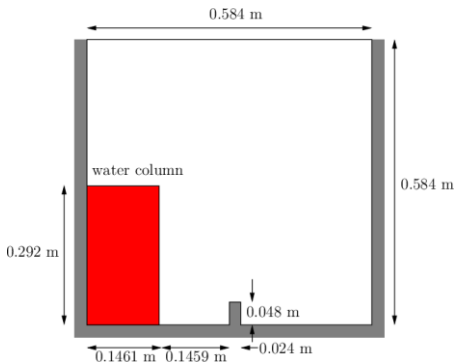


Fig.: OpenCFD, Ltd. (2020b)

Tutorials: Other tutorials



swarm



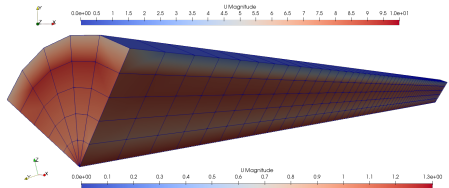
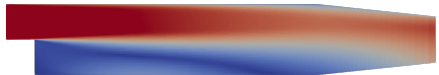
University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

- ▶ steady state turbulent flow
incompressible/simpleFoam
/

- ▶ pipe flow: pipeCyclic
- ▶ backward facing step:
pitzDaily

- ▶ transient turbulent flow:
incompressible/pisoFoam/
 - ▶ cavity: cavity

see Tutorial Wiki by OpenFOAM Community (2021) at
https://wiki.openfoam.com/Main_Page



Literature and resources: Documentations by distributors



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

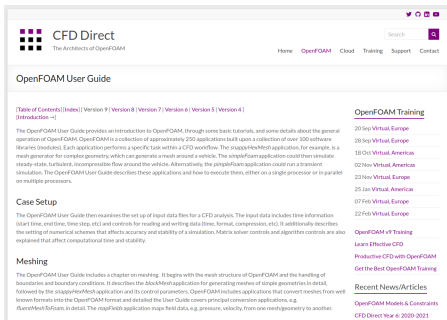
OpenFOAM v9 by The OpenFOAM Foundation (www.openfoam.org and <http://cfd.direct/>)

The OpenFOAM Foundation (2021[c]). *OpenFOAM v9 User Guide*.

URL: <https://cfd.direct/openfoam/user-guide/> (visited on 28/07/2021)

The OpenFOAM Foundation (2021b). *OpenFOAM v9 C++ Source Code Guide*. URL:

<https://cpp.openfoam.org/v9/> (visited on 28/07/2021)



Literature and resources: Documentations by distributors



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment



OpenFOAM Technical Guides

Below is a collection of guides and articles about computational fluid dynamics with OpenFOAM from its core maintainers and contributors.

Fluid Dynamics

- Computational Fluid Dynamics
- Tensor Mathematics
- Energy Equation in OpenFOAM

Multiphase Flows and Particles

- OpenFOAM Barycentric Tracking
- Water Waves in OpenFOAM
- Interface Capturing in OpenFOAM
- Guide to CFD for Polydisperse Flows

Computers and Software

- OpenFOAM Linux Guide
- Parallel I/O in OpenFOAM
- Redesigning OpenFOAM for the Future

LATEST NEWS

OpenFOAM 9 Released
20th July 2021

Download v9 | Ubuntu
20th July 2021

Download v9 | Linux
20th July 2021

Download v9 | macOS
20th July 2021

Download v9 | Source Pack
20th July 2021

Funding OpenFOAM in 2021
2nd November 2020

ARTICLES

Learn Effective CFD

Getting the Best OpenFOAM
Training

Productive CFD in OpenFOAM

OpenFOAM User Guide

OpenFOAM Open Day 2018

Technical Guides by The OpenFOAM Foundation

The OpenFOAM Foundation
(2021a). *OpenFOAM Technical
Guides*. URL: <https://openfoam.org/guides/>
(visited on 28/07/2021)

Literature and resources: Documentations by distributors



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

OpenFOAM v2012 by ESI Group
(www.openfoam.com)

OpenCFD, Ltd. (2020c). *OpenFOAM User Guide*.

URL: [https:](https://www.openfoam.com/documentation/user-guide)

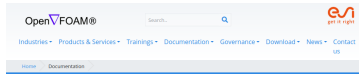
[//www.openfoam.com/documentation/user-guide](https://www.openfoam.com/documentation/user-guide)
(visited on 26/07/2021)

OpenCFD, Ltd. (2020b). *OpenFOAM Tutorial Guide*. URL: [https:](https://www.openfoam.com/documentation/tutorial-guide)

[//www.openfoam.com/documentation/tutorial-guide](https://www.openfoam.com/documentation/tutorial-guide)
(visited on 26/07/2021)

OpenCFD Ltd. (2021). *OpenFOAM Programmer's Guide*.

OpenCFD, Ltd. (2020a). *OpenFOAM Extended Code Guide*. URL: <https://www.openfoam.com/documentation/guides/latest/doc/> (visited on 28/07/2021)



OpenFOAM® Documentation

User Guide

Gain understanding of how OpenFOAM cases are assembled and evaluated in the OpenFOAM user guide:

- [Download PDF !\[\]\(32d80ecefdca755418ad4c38cd582e9c_img.jpg\)](#)
- [View on-line](#)

Tutorial Guide

A collection of tutorials to help users get started with OpenFOAM covering a range of topics, including incompressible, compressible and multiphase flows, and stress analysis

- [Download PDF !\[\]\(fd3cba180102feeedb735de5a505e32c_img.jpg\)](#)
- [View on-line](#)

Extended Code Guide

Browse the [extended code guide !\[\]\(346f5b9c8222e44e815e44b5dc7c53e5_img.jpg\)](#) to see how OpenFOAM operates under-the-hood. As an open source code, users can directly see how the code is written and learn how the functionality is implemented. The extended documentation provides descriptions for many aspects of the code, including:

Literature and re-sources: Wikkis and useful tools



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

OpenFOAM tutorials
collection and a “3 weeks
self-learning course”:

OpenFOAM
Community (2021).
Tutorial Wiki.
Ed. by OpenCFD,
Ltd. URL: https://wiki.openfoam.com/Main_Page
(visited on
28/07/2021)



Main page
Recent changes
Random page
Help about MediaWiki

Tools
What links here
Related changes
Special pages
Permanent link
Page information
Cite this page

Log in

Read View source View history Search OpenFOAM Wiki

Main Page

Welcome to the OpenFOAM Wiki!

This wiki is sponsored and managed by [OpenFOAM.com/](https://www.openfoam.com/) and members of the OpenFOAM community. To sponsor this wiki, please [contact us](#).

All proceeds are guaranteed towards the future maintenance and development of OpenFOAM.

Contents (hide)

- 1 Getting started
- 2 OpenFOAM Governance
- 3 Tutorials
- 4 Literature

Getting started

When getting started with OpenFOAM coding and installation, the following locations are useful:

- The [OpenFOAM Code Wiki](#), which provides [build instructions](#) and [upgrade information](#) as well as [migration information](#).
- The [OpenFOAM Code README](#), which provides general information and cross-links to [build requirements](#) etc.
- The [OpenFOAM repository](#) ([issue tracker](#)) and the [ThirdParty repository](#) ([issue tracker](#))

OpenFOAM Governance

ESI/OpenCFD and its partners launched the [OpenFOAM Governance](#) initiative in 2018 to bring the OpenFOAM Community together and participate within a welcoming, co-operative framework, to ensure the project's longevity and maintain its core values of being freely-available and open-source. Follow the links to find out more about the Technical Committees and their current projects.

- [Technical Committees](#)
- [Special Interest Groups](#)

Tutorials

If you want to get suggestions, how to start with OpenFOAM we can offer you a [collection of tutorials](#), which were gathered in a community project by several community members.

Literature and re-sources: Wikkis and useful tools



University of Natural Resources
and Life Sciences, Vienna

Unofficial OpenFOAM Wikki:
[OpenFOAM Wiki \(2021\).](http://www.openfoamwiki.net)
Unofficial OpenFOAM wiki.
URL:
www.openfoamwiki.net
(visited on 06/08/2021)

The screenshot shows the homepage of the Unofficial OpenFOAM Wiki. At the top, there is a navigation bar with links to 'Main page', 'Recent changes', 'FAQ', and 'Page Tools'. A search bar is located on the right. The main content area features a large heading 'Unofficial OpenFOAM wiki' with a subtext '810 pages have been created'. Below this, a list of recent releases and extensions is provided, including OpenFOAM versions 9, 10, and 11, and various release notes. A section titled 'Future events in the OpenFOAM-World' includes a timeline link. The page is divided into several colored boxes: a green box for '1 Refer' with links to User Guide, Programmer's Guide, and Source Code; a blue box for '7 The page of the hour' featuring a random selection of articles and a link to Blender; an orange box for '12 News' announcing the 1st International preCICE Workshop at the Technical University of Munich; a green box for '2 Events' with a list of all events and an RSS feed; and a blue box for '8 Community' with a link to users. The footer of the page lists '5 Parks'.

Literature and re-sources: Wikkis and useful tools



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

OpenFOAM Forum at
CFD Online:

CFD Online
(2021[a]).
OpenFOAM Forum.

URL:
<https://www.cfd-online.com/Forums/openfoam/>
(visited on
06/08/2021)

CFD Online
www.cfd-online.com

Home > Forums > Software User Forums > OpenFOAM

OpenFOAM

Forum	Sub-Forum	Threads	Posts
OpenFOAM News & Announcements	OpenFOAM News & Announcements	475	2,890
	OpenFOAM Announcements from OpenFOAM Foundation	2,748	20,021
OpenFOAM Installation	OpenFOAM Installation	5,232	24,477
	OpenFOAM Meshing & Mesh Conversion	3,345	12,589
OpenFOAM Pre-Processing	OpenFOAM Pre-Processing	30,094	84,444
	OpenFOAM Running, Solving & CFD	2,473	10,006
OpenFOAM Post Processing	OpenFOAM Post Processing	6,407	29,285
	OpenFOAM Programming & Development	332	597
OpenFOAM Verification & Validation	OpenFOAM Verification & Validation	3,839	20,081
	OpenFOAM Community Contributions	805	4,802
OpenFOAM Bugs	OpenFOAM Bugs		

Literature and resources: Wikkis and useful tools



swarm



University of Natural Resources

A tool for optimizing blockMesh grading:

OpenFOAMWiki (2020).

Scripts/blockMesh grading calculation. URL:

https://openfoamwiki.net/index.php/Scripts/blockMesh_grading_calculation (visited on 28/07/2021)

Tool for estimating required wall resolution:

CFD Online (2021[b]). *y+ Wall Distance Estimation.* URL:

<https://www.cfd-online.com/Tools/yplus.php> (visited on 28/07/2021)



Literature and resources: Wikkis and useful tools



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere

Prof. Håkan Nillson is offering a free course on OpenFOAM for PhD students and provides his course materials online. In the course of “The 3rd UCL OpenFOAM Workshop”, 24th February 2021 he received the “OpenFOAM community contribution award” for this course.

H. Nilsson, ed. (2020). *Proceedings of CFD with OpenSource Software*.
Chalmers University of Technology.
URL: http://dx.doi.org/10.17196/OS_CFD#YEAR_2020

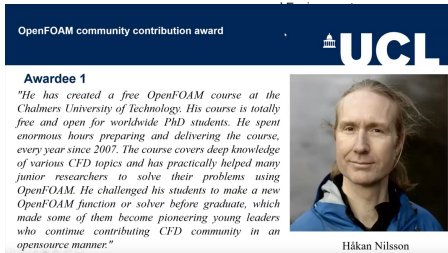


Fig.: Screenshot from the award ceremony for “The OpenFOAM community contribution award” at “The 3rd UCL OpenFOAM Workshop” on 24th February 2021



swarm



**University of Natural Resources
and Life Sciences, Vienna**
Department of Water, Atmosphere
and Environment

**University of Natural Resources and Life Science,
Vienna**

Department of Water, Atmosphere and Environment
Institut of Hydraulic Engineering and River Research

Daniel Wildt, MSc

Muthgasse 107, A - 1190 Wien

Tel.: 01-47654-81935

daniel.wildt@boku.ac.at

<http://www.wau.boku.ac.at/iwa/>



CFD Online (2021[a]). *OpenFOAM Forum*. URL:

<https://www.cfd-online.com/Forums/openfoam/> (visited on 06/08/2021).

CFD Online (2021[b]). *y+ Wall Distance Estimation*. URL:

<https://www.cfd-online.com/Tools/yplus.php> (visited on 28/07/2021).

Nilsson, H., ed. (2020). *Proceedings of CFD with OpenSource Software*. Chalmers

University of Technology. URL: http://dx.doi.org/10.17196/OS_CFD#YEAR_2020.

OpenCFD, Ltd. (2020a). *OpenFOAM Extended Code Guide*. URL:

<https://www.openfoam.com/documentation/guides/latest/doc/> (visited on 28/07/2021).

OpenCFD, Ltd. (2020b). *OpenFOAM Tutorial Guide*. URL:

<https://www.openfoam.com/documentation/tutorial-guide> (visited on 26/07/2021).



OpenCFD, Ltd. (2020c). *OpenFOAM User Guide*. URL:
<https://www.openfoam.com/documentation/user-guide> (visited on 26/07/2021).

OpenCFD Ltd. (2021). *OpenFOAM Programmer's Guide*.

OpenFOAM Community (2021). *Tutorial Wiki*. Ed. by OpenCFD, Ltd. URL:
https://wiki.openfoam.com/Main_Page (visited on 28/07/2021).

OpenFOAM Wiki (2021). *Unofficial OpenFOAM wiki*. URL: www.openfoamwiki.net
(visited on 06/08/2021).

OpenFOAMWiki (2020). *Scripts/blockMesh grading calculation*. URL: https://openfoamwiki.net/index.php/Scripts/blockMesh_grading_calculation
(visited on 28/07/2021).

Literature III



swarm



University of Natural Resources
and Life Sciences, Vienna
Department of Water, Atmosphere
and Environment

The OpenFOAM Foundation (2021a). *OpenFOAM Technical Guides*. URL:
<https://openfoam.org/guides/> (visited on 28/07/2021).

The OpenFOAM Foundation (2021b). *OpenFOAM v9 C++ Source Code Guide*. URL:
<https://cpp.openfoam.org/v9/> (visited on 28/07/2021).

The OpenFOAM Foundation (2021[c]). *OpenFOAM v9 User Guide*. URL:
<https://cfd.direct/openfoam/user-guide/> (visited on 28/07/2021).

Wikipedia contributors (2021). *OpenFOAM — Wikipedia, The Free Encyclopedia*. URL:
<https://en.wikipedia.org/w/index.php?title=OpenFOAM&oldid=1034750560>
(visited on 26/07/2021).