



**swarm**



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

# Introduction to OpenFOAM: A User View by Kenneth Hoste and Hrvoje Jasak

Daniel Wildt

SWARM Summer School  
15 – 26 November 2021

**24th November 2021**

# Outline I



swarm



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

## Introduction to OpenFOAM: A User View

Recap: Advection-Diffusion equation



# Introduction to OpenFOAM: A User View



swarm



University of Natural Resources  
and Life Sciences, Vienna

Log in

- ▶ Workshop @Ghent Universtiy by Prof. Hrvoje Jasak and Kenneth Hoste in May 2016

- ▶ available online in five parts

- ▶ download slides from <https://foam-extend.fsb.hr/openfoam/tutorials/>

<https://wiki.openfoam.com/>

Introduction\_to\_OpenFOAM:

\_A\_User\_View\_by\_Kenneth\_Hoste\_and

Hrvoje\_Jasak First part of the presentation

by Prof. Hrojve Jasak (Wikki Ltd)



Main page  
Recent changes  
Random page  
Help about MediaWiki

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

Page Discussion Read View source View history Search OpenFOAM Wiki Q

## Introduction to OpenFOAM: A User View by Kenneth Hoste and Hrvoje Jasak

- **contributor:** Hrvoje Jasak and Kenneth Hoste
- **affiliation:** Wikki Ltd
- **contact:** [click here for email address](#) - [click here for email address](#)
- **OpenFOAM version:** foam-extend 3.2 but really all versions
- **published under:** CC BY license ([creative commons licenses](#))
- **additional acknowledgements:**
  - HPC-Ugent [1]#
  - Flemish Supercomputer Centre (VSC): [2]#
  - SESAMNet [3]#

Go back to Day 3#.

### Introduction to OpenFOAM: A User View#

In this video series you will listen to the lecture of Professor Hrvoje Jasak on the basics of OpenFOAM held at Ghent University in May 2016. The applied version of OpenFOAM here is foam-extend 3.2, but the information is relevant for all versions. Feel free to follow the lecture or just sit back, relax and listen to the talk.

- Part 1:
  - [screen and camera footage#](#)
  - [screen footage#](#)
- Part 2:
  - [screen and camera footage#](#)
  - [screen footage#](#)
- Part 3:
  - [screen and camera footage#](#)
  - [screen footage#](#)
- Part 4:
  - [screen and camera footage#](#)
  - [screen footage#](#)
- Part 5:
  - [screen and camera footage#](#)
  - [screen footage#](#)

Category: Basic tutorial

# Introduction to OpenFOAM: A User View (part 1/5)



swarm



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

- ▶ 08:00 first set of slides:  
“OpenFOAM: A User View”
- ▶ 47:20 slides on simpleFoam  
tutorial airFoil2D
  - ▶ dictionaries, schemes solution  
and banana trick (47:20)
  - ▶ definition of mesh files  
(1:07:30)
  - ▶ fields (1:13:00)
  - ▶ utilities (1:17:30)

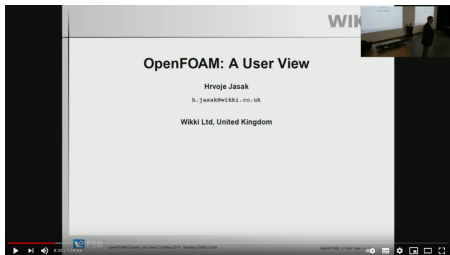


Fig.: Introduction to OpenFOAM: A  
User View (part 1/5) <https://www.youtube.com/watch?v=3iZfVmFkvB8>  
(1:18:52)

# Introduction to OpenFOAM: A User View (part 2/5)



swarm



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

- ▶ 00:00 controlDict
- ▶ 08:40 mesh, convert Fluent mesh to OpenFOAM mesh
- ▶ view mesh in Paraview (18:50)
- ▶ fields (22:10)
- ▶ checkMesh (28:00)
- ▶ simpleFoam (36:10)
- ▶ postprocessing in Paraview (44:45)
- ▶ residual plots (46:40)

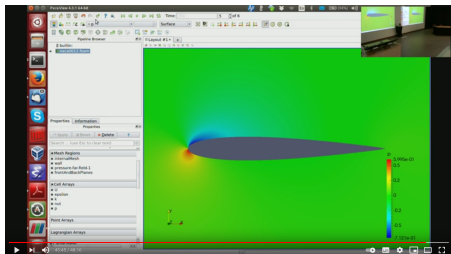


Fig.: Introduction to OpenFOAM: A User View (part 2/5) <https://www.youtube.com/watch?v=cM9RLZobnBA> (48:16)

# Introduction to OpenFOAM: A User View (part 3/5)



swarm



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

- ▶ 00:00 potentialFoam, forces function object, more postprocessing in Paraview, yPlusRAS, foamCalc
- ▶ 18:20 blockMesh
- ▶ 37:40 create new mesh on simpleFoam/pitzDaily for backward facing step
- ▶ 51:20 set-up physics for backward facing step
- ▶ 1:00:15 run backward facing step, debugging

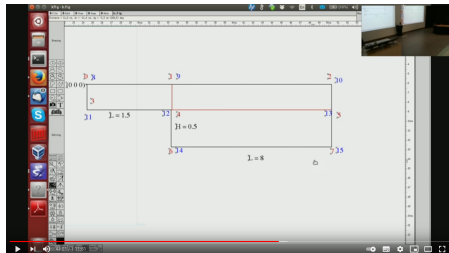


Fig.: Introduction to OpenFOAM: A User View (part 3/5) <https://www.youtube.com/watch?v=g6xttyts16s> (1:11:21)

# Recap: Advection-Diffusion equation



swarm



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

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi + \nabla \cdot (D_0 \nabla \phi) = 0 \quad (1)$$

$$\frac{\partial c}{\partial t} + u \cdot \frac{\partial c}{\partial x} + D_0 \frac{\partial^2 c}{\partial x^2} = 0 \quad (2)$$

Equation (1) is solved by the  
OpenFOAM solver  
scalarTransportFoam

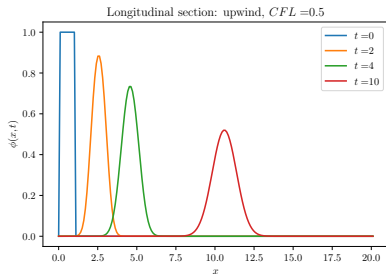


Fig.: Solution of the 1D  
Advection-Diffusion equation

# Introduction to OpenFOAM: A User View (part 4+5/5)



swarm



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

- ▶ 00:00 postprocessing engine block
- ▶ 11:15 swirl test tutorial
- ▶ 15:55 source code  
scalarTransportFoam, header  
files
- ▶ 33:50 setRootCase.H,  
createTime.H,  
creatFields.H
- ▶ 53:50 scalar transport equation  
implementation

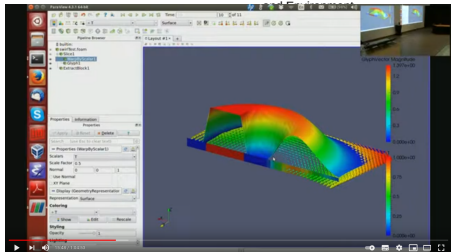


Fig.: Introduction to OpenFOAM: A User View (parts 4+5/5) [https://www.youtube.com/watch?v=IfJI\\_oMlW7o](https://www.youtube.com/watch?v=IfJI_oMlW7o) (1:04:53); <https://www.youtube.com/watch?v=LTMjPhs007g> (9:19)





**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](mailto:daniel.wildt@boku.ac.at)

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