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

Introduction to OpenFOAM: Multiphase modelling, discretization, programming

Daniel Wildt

SWARM Summer School
15 – 26 November 2021

25th November 2021

Outline I

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

GHENT UNIVERSITY

Introduction to OpenFOAM: Multiphase and Free-Surface Flows

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

- 2:45 volume fraction
- 8:50 Eulerian multi phase model
- 17:00 free-surface flow
- 31:40 Lagrangian particle tracking
- 41:15 liquid film model
- 48:25 free surface tracking
- 55:10 summary, examples VOF-solver

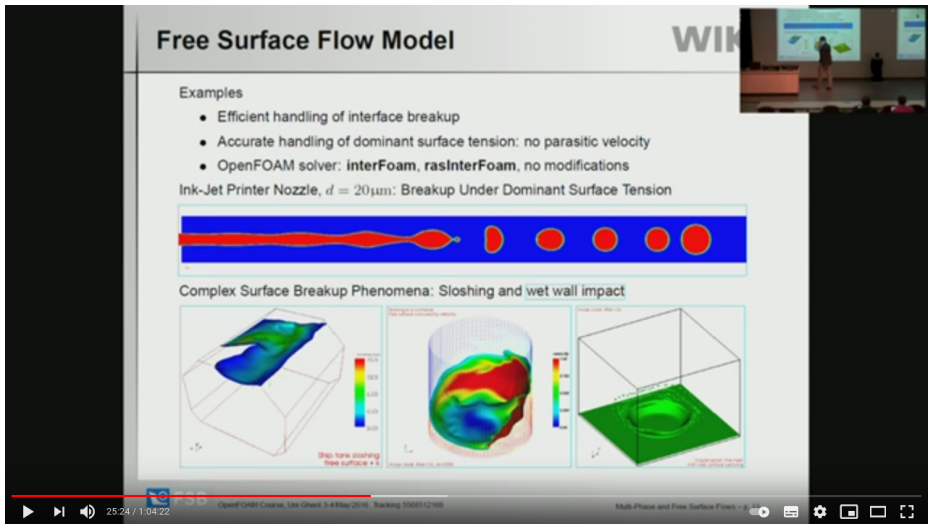

Fig.: Introduction to OpenFOAM: Multiphase and Free-Surface Flows
`https://www.youtube.com/watch?v=oPiDnB8Ibc8` (1:04:22)

Introduction to OpenFOAM: Multiphase and Free-Surface Flows

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

Rising bubble training case

- 00:00 wedge shape domain
- 08:50 initialisation `setFields`
- 15:55 domain decomposition `decomposePar`
- 25:10 run case in parallel using OpenMPI
- 37:00 `damBreak` tutorial introduction

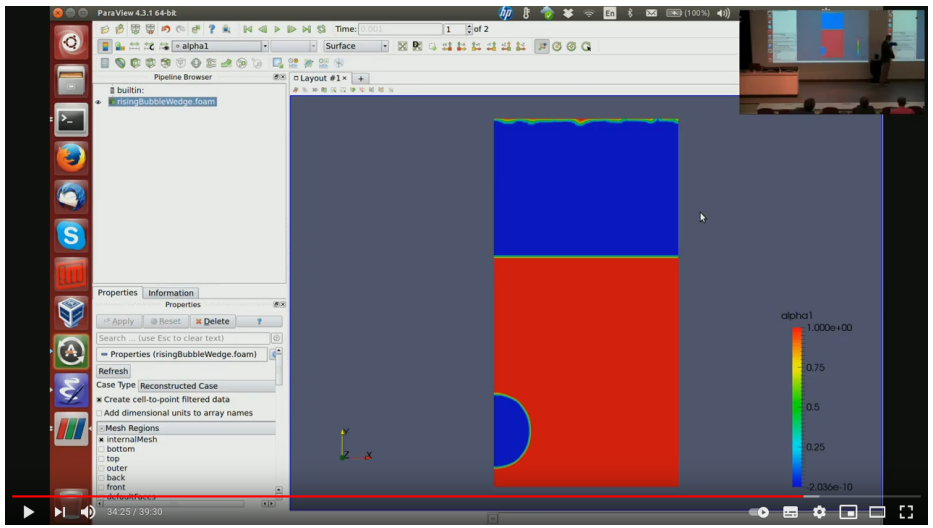

Fig.: Introduction to OpenFOAM: Rising Bubble Training Case `https://www.youtube.com/watch?v=w8G7xFFcudg` (39:30)

Introduction to OpenFOAM: Finite Volume Discretization in OpenFOAM

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

see also PhD thesis by Jasak (1996)

- 11:00 Gradient scheme
- 21:30 Convection Discretization
- 26:20 Face interpolation
- 31:00 Diffusion discretization, non-orthogonality
- 39:45 Discretization settings
- 44:45 `airFoil2D` discretization
- 54:30 bounding messages
- 56:35 function object `minMaxField`

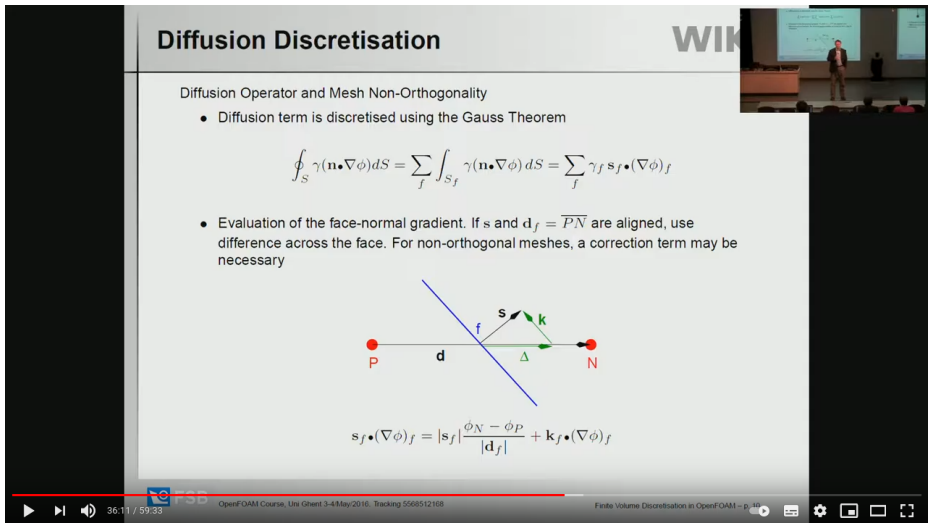

Fig.: Introduction to OpenFOAM: Finite Volume Discretization in OpenFOAM `https://www.youtube.com/watch?v=a4B_oXR5Kzs` (59:33)

Introduction to OpenFOAM: Programming in OpenFOAM

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

- 00:00 code organization, environment variables, build system
- 12:20 programming guidelines
- 17:35 makefiles
- 23:00 building with `wmake`
- 46:05 new function object, new application
- 1:10:40 reading additional input variables

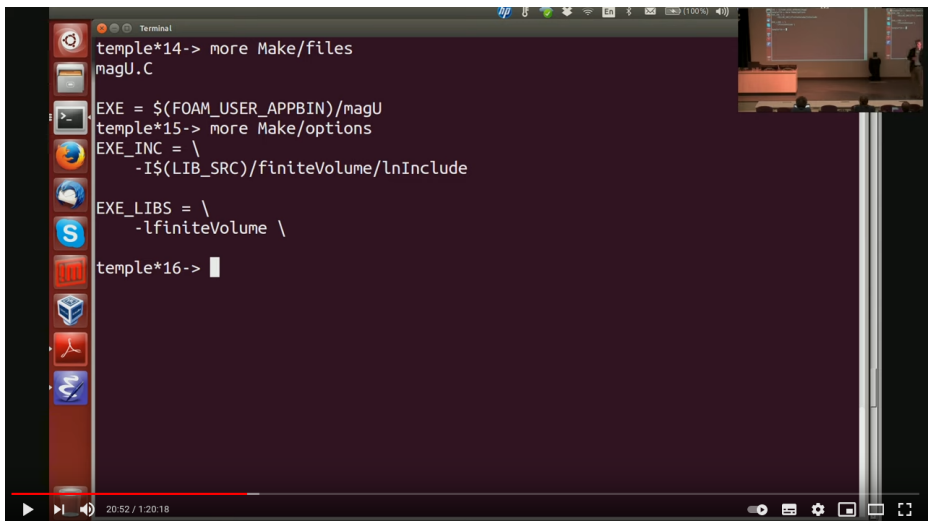

Fig.: Introduction to OpenFOAM: Programming in OpenFOAM `https://www.youtube.com/watch?v=ScXa5W8IES4` (1:20:18)

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/`

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

Literature I

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

Jasak, H. (1996). 'Error analysis and estimation for the finite volume method with applications to fluid flows'. Thesis of Dissertation. Imperial College London. URL: https://spiral.imperial.ac.uk/handle/10044/1/8335.