

OpenFOAM®

in wastewater applications: *8 – Advanced topics*

nelson.marques@bluecape.com.pt

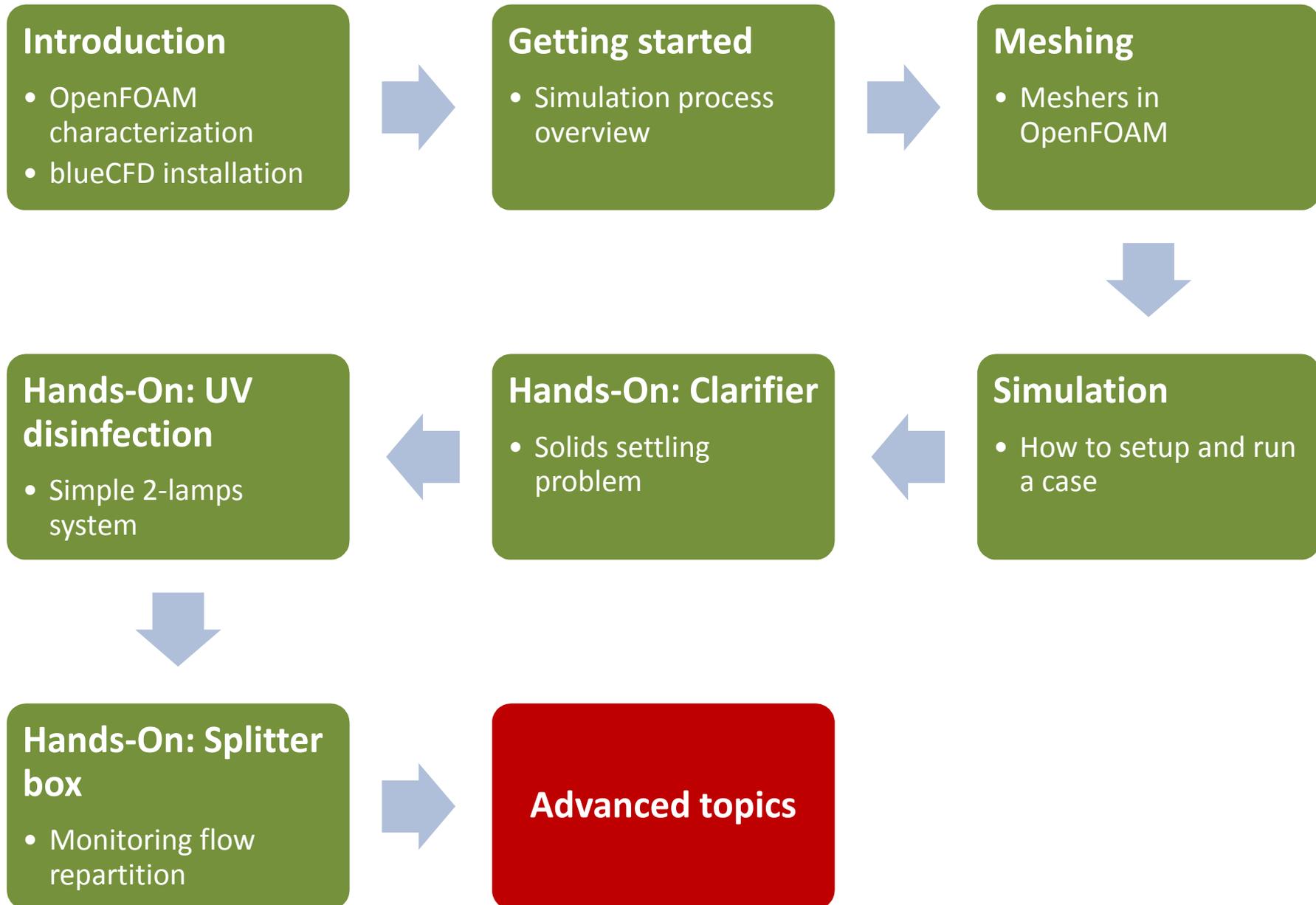
13-14th June 2015



bluecape

Computer Applications
in Science & Engineering

Progress



Contents

1. Power User: Coding is a must
2. The case for a GUI
3. Where next for training and services?
4. Large cases
5. Community involvement

Power user: Coding is a must (1/2)

When we visit the openfoam.org/features page, the first few words explicitly state:

The OpenFOAM[®] (Open Field Operation and Manipulation) CFD Toolbox is a free, open source CFD software package [...]

As a *CFD Toolbox*, it does not guaranteed that it's able to do everything you need right out-of-the-box.

It does however provide:

- The means for creating one's own solvers and libraries
- Given its open source nature, the people in the community may have already done what we need!

Power user: Coding is a must (2/2)

Overview of what you can do and learn:

- OpenFOAM User Guide and Programmer's Guide
- openfoamwiki.net/index.php/OpenFOAM_guide
- Learn C++:
 - Book: *Thinking in C++*
 - Tutorial: www.cplusplus.com/doc/tutorial/
- Study the source code (what matters to you)
 - openfoam.org/docs/cpp/
- Research the forums:
 - www.cfd-online.com/Forums/openfoam/

The case for a GUI (1/2)

Why doesn't OpenFOAM have a GUI?

It did, was named FoamX and it was reported to be akin to “keyhole surgery”^[1] – last version was in OpenFOAM 1.4.

Depending on your workflow, a GUI can end up just getting in the way, since editing text files directly can be quicker and easier to do, as well as adding some automation to the process.

Several GUIs do already exist, which are listed here:

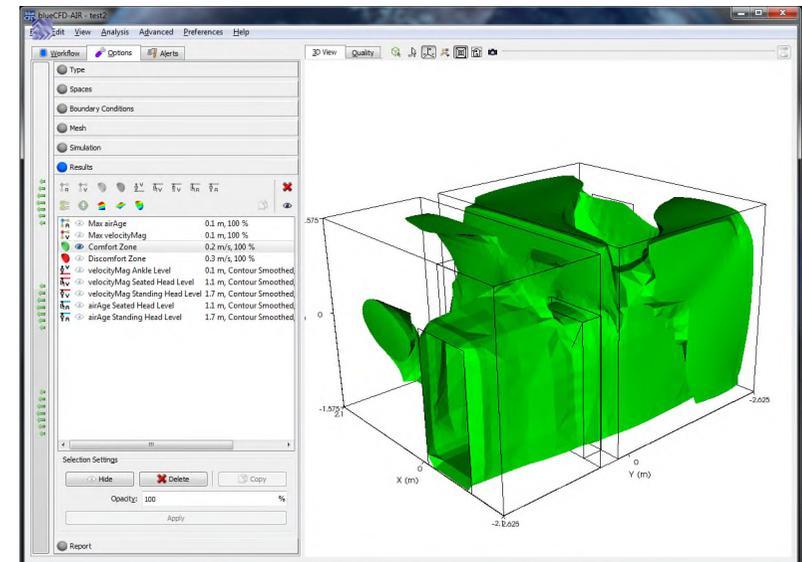
- openfoamwiki.net/index.php/GUI

[1] [openfoamwiki.net/index.php/Main_UserGuideAddendum#Appendix A: The FoamX case manager](http://openfoamwiki.net/index.php/Main_UserGuideAddendum#Appendix_A:_The_FoamX_case_manager)

The case for a GUI (2/2)

Pointing out a few from the list:

- CastNet by DHCAE
 - www.dhcae-tools.com/CastNet.html
- HELYX by ENGYS
 - engys.com/products/helix
- DICE (open source)
 - dicehub.net
- HELYX-OS (open source)
 - engys.com/products/helyx-os
- blueCFD-AIR (yes, we have one!)
 - bluecape.com.pt/blueCFD-AIR



Where next for training and services?

- OpenCFD (ESI Group) – official support providers
- CFD Direct (original developer Henry Weller, circa 1989)
- Wikki (early developer Hrvoje Jasak, circa 1993)
- Chalmers Professional Education
 - Håkan Nilsson's PhD course in CFD with OpenSource software

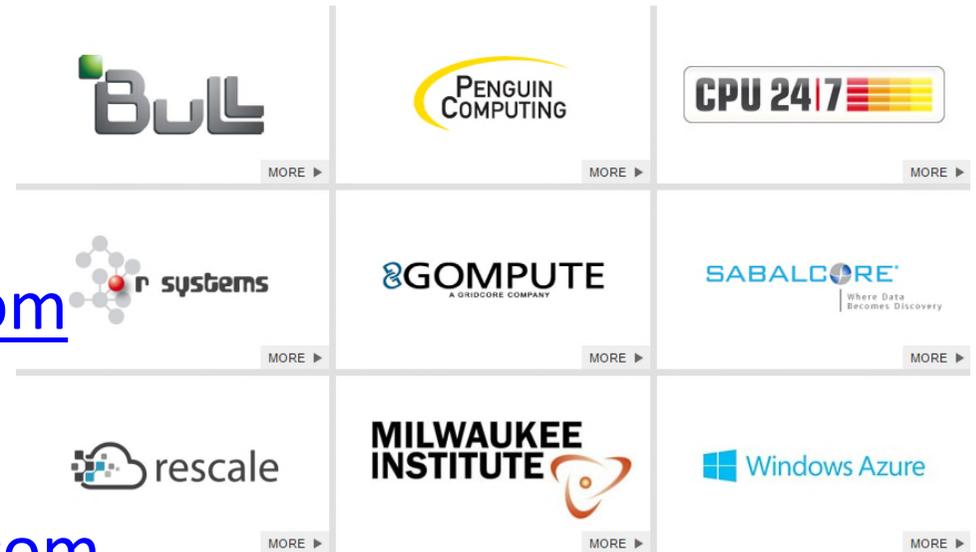
Complete lists:

- openfoamwiki.net/index.php/Main_Courses
- openfoamwiki.net/index.php/Template:Frontpage_Links

Large cases

OpenFOAM on the cloud:

- Rescale: www.rescale.com
- Gompute: www.gompute.com
- SimScale: simscale.com
- SabalCore: www.sabalcore.com
- UberCloud: www.theubercloud.com



For the DIY cloud crowd:

- Amazon EC2: aws.amazon.com/ec2
- Google Cloud: cloud.google.com
- Microsoft Azure: azure.microsoft.com

Community involvement

- Forums:
 - www.cfd-online.com/Forums/openfoam/
 - extend-project.de
- Wiki: openfoamwiki.net
- Contributions:
 - openfoamwiki.net/index.php/Extend-bazaar
 - openfoamwiki.net/index.php/Main_ContribOther
- Bug reports:
 - www.openfoam.org/bugs/
 - sf.net/p/openfoam-extend/ticketsfoamextendrelease/

Thank you for your time.

Next:
Happy “CFDing”!!



bluecaPe

Computer Applications
in Science & Engineering