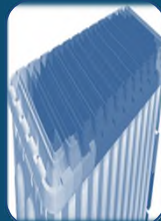
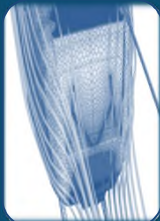


# OpenFOAM®

## in wastewater applications: *0 – Overview*

*nelson.marques@bluecape.com.pt*

13-14<sup>th</sup> June 2015



**bluecape**

Computer Applications  
in Science & Engineering

# Objectives and Outcomes

## Objectives

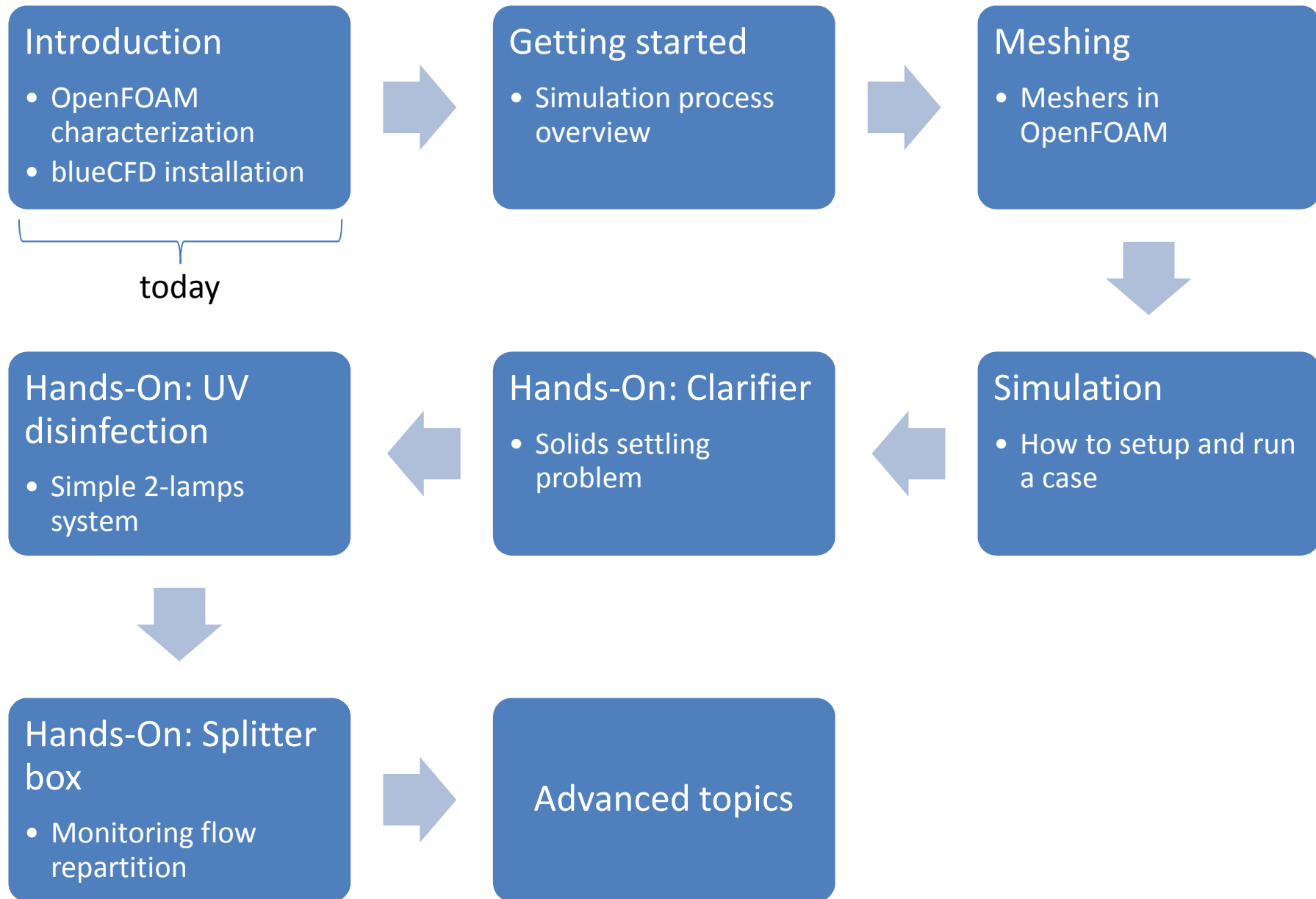
- Make the case that OpenFOAM is a viable platform for CFD work in water and wastewater treatment
- Explain what OpenFOAM really is and how it is “built”
- Provide instructions on how to efficiently setup and use OpenFOAM
- Show what is available out-of-box and is relevant to WW community

## Outcomes

Attendants will:

- Have access to OpenFOAM on their computers
- Know their way around the installation
- Become familiar with OpenFOAM’s case structure (i.e., files)
- Know what steps to take to setup and run a case
- Know the most important parameters to adjust in a case setup
- Have access to four cases relevant to WWT world

# Contents



# Drowning fish?...

At some point during the workshop, there is a chance you may relate a bit to this little fella!

Relax, OpenFOAM is known for being though. Specially first timer users if that is your case!

On our side we have breaks and I will be around on Monday as well. Please feel free to ask to your heart's content.

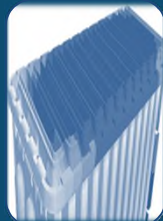
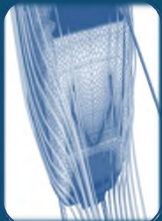
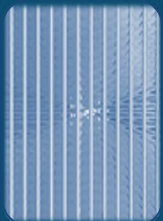
Going forward: practice makes perfect!



# Thank you for your time.

Next:

*1 - Introduction*



bluecaPe

Computer Applications  
in Science & Engineering