

How to use free-surface flows in OpenFOAM® 3.0

Brief of a training track

Due to several changes made in OpenFOAM 2.3 and newer versions, this session aims to teach you how to setup your cases accordingly to run with `interFoam`.

This will be a hybrid type of session: the presentation will be presented as a lecture with examples, which will have the main information and respective steps for the updated Spillway tutorial, allowing for attendees to do the steps themselves along with the session.

Training track "How to use free-surface flows in OpenFOAM® 3.0" consists of two parts. The first part is the lecture about the `interFoam` solver and correct boundary conditions in different versions of OpenFOAM. The second part is the hands-on training: listeners try to solve the Spillway case step-by-step together with the speaker.

Training course plan

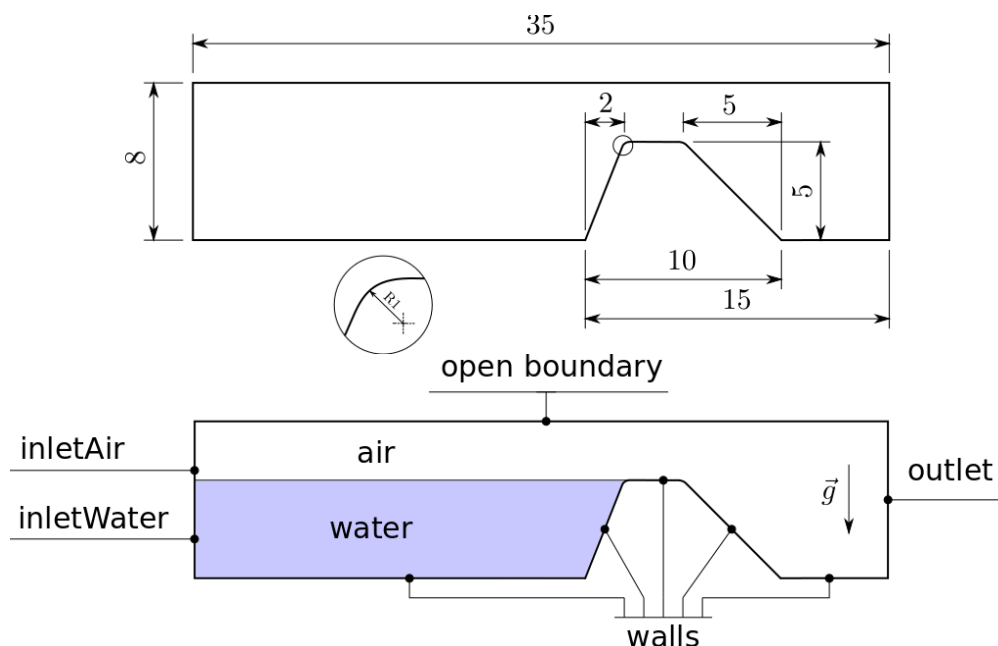
- Introduction.
- Key points of training course.
- `interFoam` solver: how it works:
 - Governing equations.
 - Volume of Fluid method.
 - Solution process.
 - **Boundary conditions.**
- Spillway tutorial:
 - Model setup.
 - Physical properties setup.
 - Mesh generation.
 - **Boundary conditions setup.**
 - Numerical schemes and time advancement settings. Running.
 - Results.
- Conclusions and discussion.

Description of materials

1. The folder `cases/` contains the pack of different versions of Spillway case.
 - a. The folder `spillway_2p2/` is the Spillway case for OpenFOAM v.2.2.x. This is the material only for demonstration of differences in case settings between this version of OpenFOAM and actual version (v.3.0.x).
 - b. The folder `spillway_3p0/` is the actual version of Spillway case which will be used on the practical part of the training course. In this folder you can find two cases — with correct boundary conditions (the subfolder `correct/`) and with wrong BC (the subfolder `wrong/`). Therefore you have a possibility to compare these cases and see that correct boundary conditions for pressure are critically important for correct computations.
2. The video `spillway_pm.mp4` contains results from the folder `spillway_3p0/` to compare them in one video.
3. The presentation of the training course `How to use free-surface flows.pdf` will be demonstrated to listeners on the OpenFOAM Workshop 2016. It contains all needed information and helps you to understand the speaker better.

Spillway: model setup

Let's consider the water flow over a spillway. The water will enter the domain with a fixed velocity along the left face, flow over the dam and out through the right boundary. A spillway width dimensions are shown in the image below (all dimensions are in meters). Velocity of water on the inlet is 0.6 m/s.



Physical properties of water and air are in the table below.

| | Water | Air |
|--|-------|---------|
| Density, kg/m ³ | 1000 | 1 |
| Kinematic viscosity, m ² /s | 1e-6 | 1.48e-5 |
| Surface tension, N/m | 0.07 | |

The case based on the *damBreak* tutorial supplied with OpenFOAM. For reference, link to the wiki page of the original Spillway tutorial:

<https://www.hpc.ntnu.no/display/hpc/OpenFOAM+-+Spillway+Tutorial>