

# How to use free-surface flows in OpenFOAM® 3.0

---

## Structure of a training track

Training track "How to use free-surface flows in OpenFOAM® 3.0" consist of two parts. The first part is the lecture about the view to the `interFoam` solver inside and about correct boundary conditions in different versions of OpenFOAM. In this file you can see the short description of files for the training track.

- 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 only material 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 (subfolder `correct/`) and with wrong BC (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 it in one video.
- 3) The presentation of the training course `How to use free-surface flows.pdf` will be demonstrated to people on the OpenFOAM Workshop 2016. It contains all needed information and helps you to understand the speaker better.