Learning how to use free surface flows in OpenFOAM® 3.0

Instructor: Victoria Korchagova, ISP RAS

Training type: Intermediate

Session type: Lecture with examples

Software stack:

- OpenFOAM 3.0.x

Development team website: https://github.com/unicfdlab

Full description

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 course plan:

1. Introduction: why free-surface flows are so interesting for researchers and engineers; plan of training course; key points (boundary conditions in different versions of OpenFOAM).
2. Example for the training course – Spillway. Model setup.
3. Steps to run Spillway tutorial:
    (a) **blockMesh** utility and “blockMeshDict” structure;
    (b) **snappyHexMesh** utility;
    (c) boundary conditions setup (volume fraction/velocity/pressure);
    (d) “fvSchemes” and “fvSolution” setup;
    (e) running **interFoam**; watching to results in different versions.
4. Conclusions and discussion.

For reference, link to the wiki page of the original Spillway tutorial:

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