Advanced usage of OpenFOAM user defined functions - how to create simple FSI program with functionObject API

Instructor: Ilia Marchevsky, PhD, ass. prof., BMSTU

Training type: Intermediate

Session type: Hands-on

Software stack:

- OpenFOAM 3.0.x

Developers of this session:

- I. Marchevsky, Russia
- M. Kraposhin, Russia
- S. Strijhak , Russia

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

Full description

Module begins with short introduction to fluid-structure interaction (FSI) problems, overview and classification of cases which arises in real world. One of the most simple and common case of FSI is selected for demonstrations of OpenFOAM's capability to create complex numerical models - weakly coupled motion of solid body with limited number of degrees of freedom in flow. For the demonstration purposes particular real world example is considered - flow-induced vibration of cylinder, connected to spring which is fixed on another of its side. Mathematical model is formulated for this case.

With respect to this example case different approaches for solution of this problem are discussed. Difference between custom solver, *functionObject* and *fvOption* classes is shown in discussion. For the current case selection of standard OpenFOAM solver **pimpleDyMFoam** and usage of *functionObject* is argued. Procedure of creating user *functionObject* is discussed, including: setup of custom dynamic library, creation of class which inherits standard OpenFOAM library "forces", working with *objectRegistry* class, mesh motion issues and etc.

During the track we will compile our FSI class and will run simulation with new library. Results of FSI simulation will be analyzed, compared to experimental observations and to results of simulation from in-house code, implementing vortex method.

The attendees will require good knowledge of setting up cases, running/modifying tutorial cases as well as a basic understanding of programming/compiling source code. This module will be hands-on. The attendees will require a laptop with a current OpenFOAM installation or - preferably - be able to boot the conference USB stick. Materials of this tutorial are located at git archive and can be downloaded from github https://github.com/unicfdlab/TrainingTracks/tree/master/OpenFOAM/simpleFsi-OF3.0.0 for OpenFOAM version 3.0.0.