CFD simulations using OpenFOAM @IT4I

Thursday 14 December 2017 - Friday 15 December 2017

VŠB - Technical University Ostrava, IT4Innovations building

Programme

Course introduction

• course outline and introduction to OpenFOAM

• Linux background, preparation for hands-on sessions

• my first OpenFOAM case, hands-on session

• OpenFOAM case structure

Geometry and meshing

• meshing strategies in OpenFOAM

• blockMesh, snappyHexMesh, hands-on session

• mesh conversion from external packages (ANSYS Fluent example), hands-on session

Mesh manipulations

• mesh manipulation, hands-on session

• mesh partitioning for parallel computing, hands-on session

• parallel meshing, parallel redistributing, hands-on session

Physical modelling and numerical simulations

• initial and boundary condition

• incompressible simulations - SIMPLE/PISO/PIMPLE solvers, hands-on session

• transport and turbulent models

• solution monitoring and control

• linear solvers setting

Parallel computing

• run in parallel on workstation/laptop/supercomputer

• programming boundary conditions

Post-processing

• data Vizualization, ParaView, EnSight

• OpenFOAM format conversions

• plotting graphs of force coefficients, etc.

- utilities for post-processing
- hands-on session