Title: **Standalone OpenFOAM bundles - OpenFOAM for Windows and OpenFOAM in Box.**

Instructor: Jakub Benda
Co-instructor: Jiri Simonek

Summary:
Workshop will be an introduction of standalone OpenFOAM® bundles - OpenFOAM for Windows and OpenFOAM® in Box (for Linux).

Abstract:
The aim of the workshop is to introduce to its participants "OpenFOAM® for Windows" and "OpenFOAM® in Box". Show the enormous benefits and simplicity of its installation and operation.

OpenFOAM® for Windows is free, natively compiled package for running OpenFOAM® in Windows. The package includes the newest OpenFOAM® 3.0 source code and binaries - moreover it contains also Turbo Blade Post in Paraview 5, CFMesh and SWAK for FOAM. OpenFOAM® for Windows works without any limitations - all the libraries, utilities and solvers are natively compiled in Windows and it allows all users to develop, compile and use new features for OpenFOAM® directly in Windows. The newest version OpenFOAM® for Windows 16.06 is available for free download at http://www.cfdsupport.com/download-openfoam-for-windows.html .

OpenFOAM® in Box is a free and unique software package for any Linux distribution. It is ready to use. No compilation. No dependencies. All the software components are already installed locally inside the package. It includes the newest OpenFOAM® 3.0 source code and binaries - moreover it contains also Turbo Blade Post in Paraview 5, CFMesh and SWAK for FOAM. The newest version OpenFOAM® in Box 16.06 is available for free download at http://www.cfdsupport.com/download-openfoam-in-box.html .

Session type:
Introduction - Lecture with examples
But it could be also hands-on with those who will have OpenFOAM® for Windows 16.06, OpenFOAM® in Box 16.06 in his or hers computer.

Software stack:
OpenFOAM® for Windows 16.06, OpenFOAM® in Box 16.06 (both based on OpenFOAM® 3.0)