Title: Five Basic Classes in OpenFOAM

Instructor: Hrvoje Jasak, Wikki Ltd.

Description: The objective of this lecture is to provide a deeper insight into the structure and interaction of basic classes in OpenFOAM.

The lecture will describe the software implementation of Space and Time discretisation, including the Time class and polyMesh class, with a brief overview of polyhedral mesh handling. Field representation class hierarchy, starting from basic containers to geometric field will be presented, with a particular example of patchField virtual class hierarchy.

Matrix, Linear System and Linear Solver class hierarchy hold a sparse matrix $[A]$ and linear system $[A][x] = [b]$ represents a basis of implicit Finite Volume and Finite Element discretisation.

Discretisation Method machinery for the Finite Volume Method is encapsulated as Interpolation, Differentiation and Discretisation classes, building on previously described functionality.

This will be a thought class, consisting of a code overview, detailed look at some implementation details and a discussion on class interaction. Basic knowledge of the C++ programming language is needed to follow the developments.

Duration: 90 minutes

Date and Location: 13 June 2011, Penn State University, State College, PA, USA