CFD simulations using OpenFOAM (PTC course)

Friday, 15 December 2017

<u>Physical modelling and numerical simulations: initial and boundary condition; incompressible simulations -</u> <u>SIMPLE/PISO/PIMPLE solvers, hands-on session; transport and turbulent models; solution monitoring and control; linear</u> <u>solvers setting</u> - 207 (09:00 - 10:30)