

HPC Training Series - Course 5 "Computational Fluid Dynamics (CFD) using
OpenFOAM on High Performance Computing (HPC)"

Contribution ID: 7

Type: **not specified**

**Introduction to OpenFOAM: OpenFOAM's history,
main features and case setup**

Friday, 14 June 2024 11:15 (1h 30m)

Presenter: Dr PAPOUTSIS-KIACHAGIAS, Evangelos (NTUA)