

Training course: Introduction to Computational Fluid Dynamics using OpenFOAM®, Jan 22 – Feb 5

Rita F Carvalho, DEC-FCTUC, MARE - Marine and Environmental Sciences Centre,
ARNET — Aquatic Research Network

Fernando P Bernardo, DEQ-FCTUC, CERES - Chemical Engineering and Renewable
Resources for Sustainability Centre

28 contact hours + 42 homework hours
400 EUR (academics), 800 EUR (non-academics)

Programme

January 22, 14-18h

Mathematical modelling. Physical phenomena and partial differential equations.
Introduction to the OpenFOAM® platform. Solvers and processing.

January 23, 9-13h

Pre-processing. Mesh construction. Selection of solvers. Initial and boundary
conditions. Options.

January 23, 14-18h

Post-processing.

January 29, 14-18h

Presentations by trainees and discussion.

January 30, 9-13h

Turbulence models. Multiphase flow.

January 31, 9-13h

Heat transfer, mass transfer and chemical reaction.

February 5, 14-18h

Presentations by trainees and discussion. Future developments.

