SIMULATING
THE FUTURE

WORKSHOP ON COMPUTATIONAL SIMULATION USING ANSYS

INITIATIVE

An ESSS Institute and University of Coimbra initiative

WORKSHOP

Introduction to Computational Fluid Dynamics
June 4th - 3PM - 4:30PM

EVENT

Free event
Online (live)

PROGRAM

Introduction to Computational Fluid Dynamics

- Introduction to Computational Fluid Dynamics;
- CFD simulation steps;
- Industrial and academic applications;
- Fundamentals of HPC Concepts in CFD;
- Hands on: CFD Workshop with Ansys Fluent

ENROLLMENT

Point your camera at the QR Code and sign up now