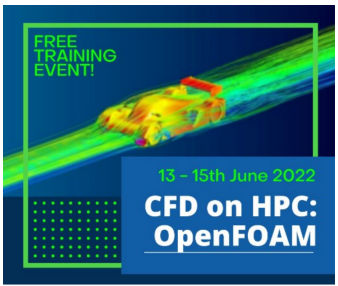
OpenFOAM School – Module 03

Advanced usage



EuroCC workshop

Aleksander GRM – May, 2024

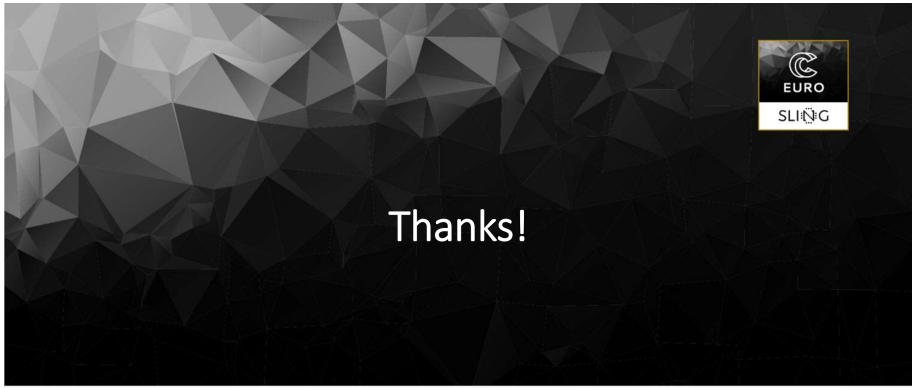
Content

Introduction

Foil case

- describe basic case philosophy
- describe two different meshing approaches
  - OpenFOAM foil `blockMesh`: change angle of attack with velocity vector rotation
  - GMSH foil `mesh`: change angle of attack with geometry rotation
- Describe two different solutions (OF module `foamRun`)
  - steady state – `SIMPLE` algorithm
  - transient – `PIMPLE` algorithm
- show mesh generation for different angles of attack (compare with fixed mesh),
- solution initialization with `potentialFoam` integrated in `foamRun`,
- compare solutions of the case for angle of attack 20° (turbulent and laminar solver).

Sponsorship



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951732. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, United Kingdom, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Switzerland, Turkey, Republic of North Macedonia, Iceland, Montenegro

EuroHPC
Joint Undertaking