

CFD on HPC – OpenFOAM

03 – Advanced usage



Aleksander GRM – December, 2024



Introduction

Foil case

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).



Thank you for attention!



EuroHPC
Joint Undertaking



REPUBLIC OF SLOVENIA
MINISTRY OF HIGHER EDUCATION,
SCIENCE AND INNOVATION

This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 101101903. The JU receives support from the Digital Europe Programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Türkiye, Republic of North Macedonia, Iceland, Montenegro, Serbia.