



PARTNERSHIP FOR ADVANCED COMPUTING IN EUROPE

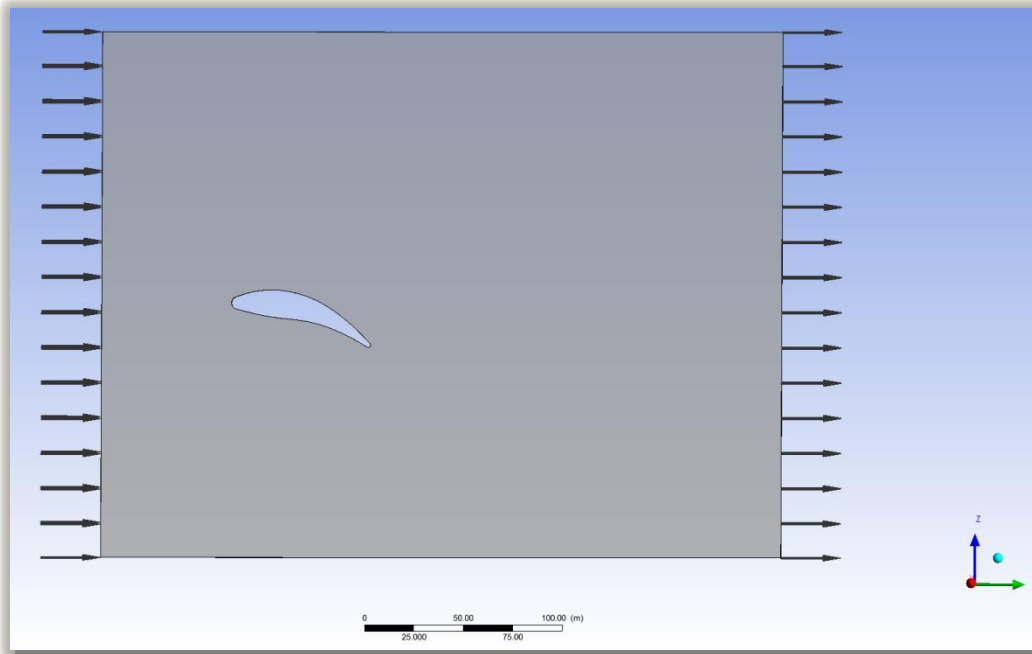
Simulations – parallel calculation in Ansys

Primož Drešar, UL FS



Flow around aerofoil (CFD)

- 2-D profile (geometry and BCs)



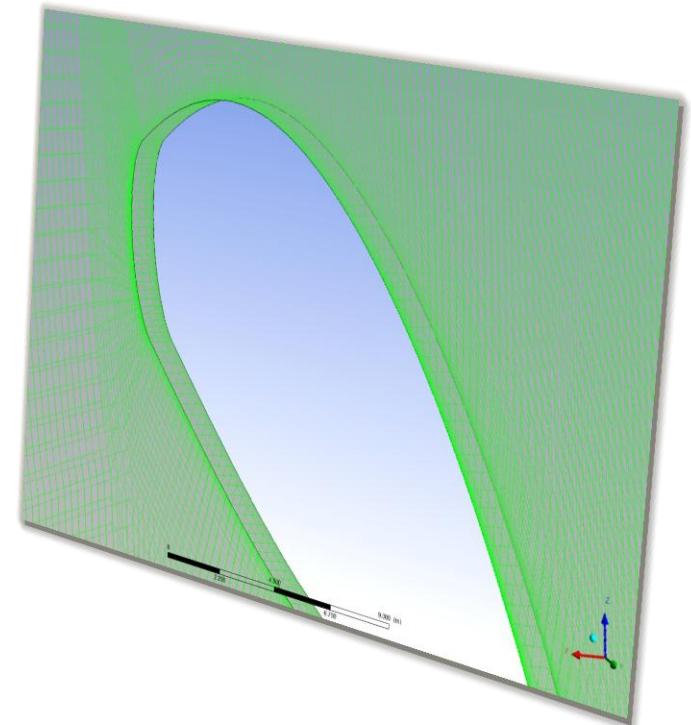
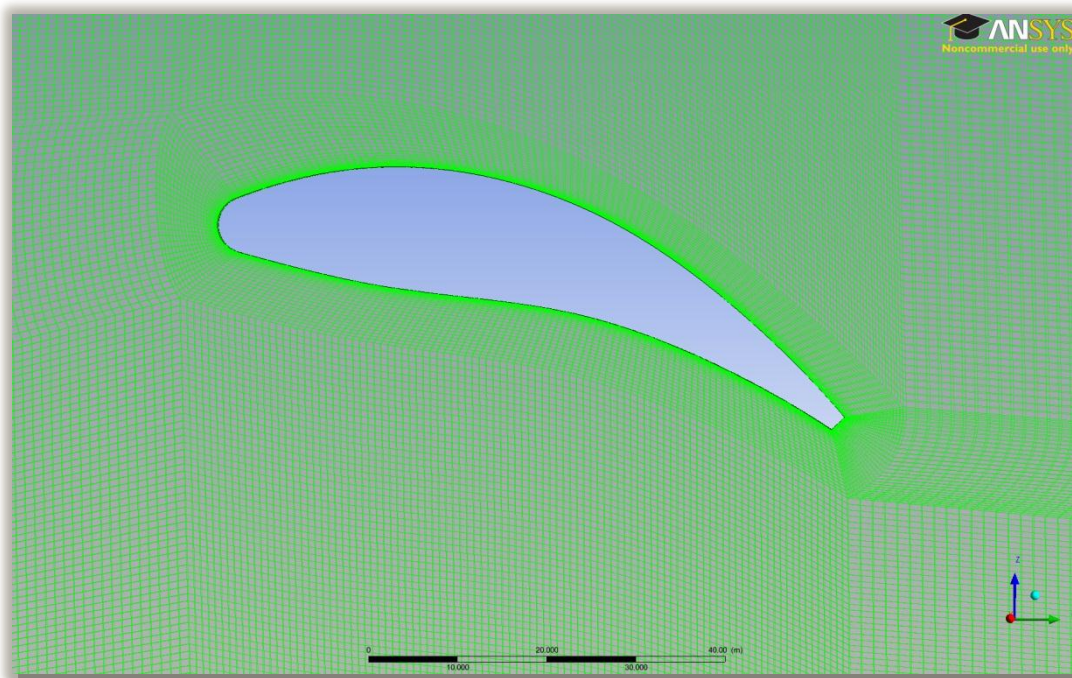
Inlet velocity :
5 m/s

Relative pressure on outlet:
 $p = 0$ Pa

Static flow.

Flow around aerofoil (CFD)

- Mesh topology is given
 - Ansys CFX module does not support 2-D analysis
 - We build thin 3-D mesh 3 elements thick. Built out of hexahedra.



Flow around aerofoil (CFD)

- Setting up a calculation on HPC Prelog
- Batch mode line in Linux
 - `bsub -n 8 -J 2Dprofil 'cfx5solve -double -def profil2D.def -par-dist `echo $LSB_HOSTS|tr " " ", "` -start-method "Platform MPI Distributed Parallel",`
- Opening residual monitor while performing a calculation
 - `cfx5solve -interactive`

Flow around aerofoil (CFD)

- Showing results
- Results file : <job name>_001.res

