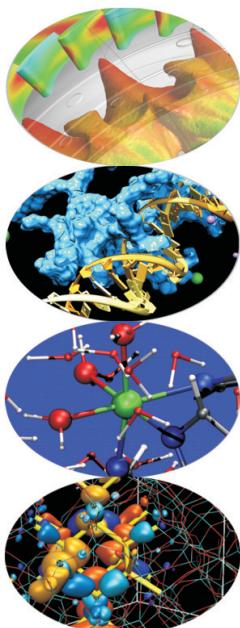
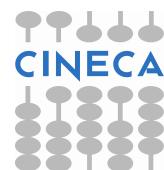


Marine CFD applications using OpenFOAM



Andrea Penza, CINECA

27/03/2014



Contents

- **Background** at CINECA: LRC experience
- **CFD** skills
- **Automatic** workflow
- **Reliability workflow**

OpenFOAM solvers for marine CFD analysis

- **6DOF/2DOF solver:**
interDyMFoam (dynamics, transient, optional wave motion)
fully explicit mules: CFL mandatory
- **Unsteady 0DOF:**
interFoam (transient captive)
- **0DOF (captive):**
LTSInterFoam (Local Time Stepping (quasi-static hypothesis),
suitable for automation and large computational campaign)

OpenFOAM: CFD mandatory

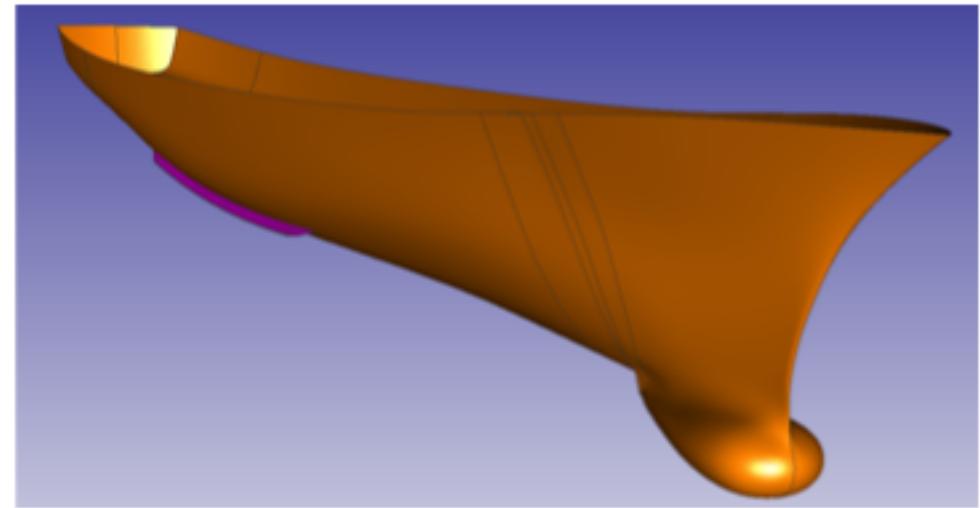
- OpenFOAM multiphase unsteady solvers have to respect the CFL condition:

$$\frac{u \Delta t}{\Delta x} \leq CFL = 1$$

- AC72 class: high speed (u), sufficiently small $y+$ → **too small time-step**
- **Commercial** softwares can manage 100-1000 times larger dt
- **Unsteady** simulation in OpenFOAM results too time consuming at the moment, but used only if mandatory due to the physics of the problem
- **LTSInterFoam**: local time stepping multiphase solver, developed "ad hoc" for Marine CFD.

CFD Model

- ① High Reynolds simulation: **RANS** model employed
- ② Turbulence model:
k- ω SST
- ③ **Wallfunction** enabled:
 $y^+ \approx 70$



➤ *Standard DTMB-5415 bare hull modeled*

CFD model for marine applications

1. Solver:

- **Multiphase** flow (water/air)
- **Volume Of Fluid** method employed
- 0 D.O.F. (**captive-case**)
- **Unsteady** case

In OpenFOAM we got the **interFoam** solver

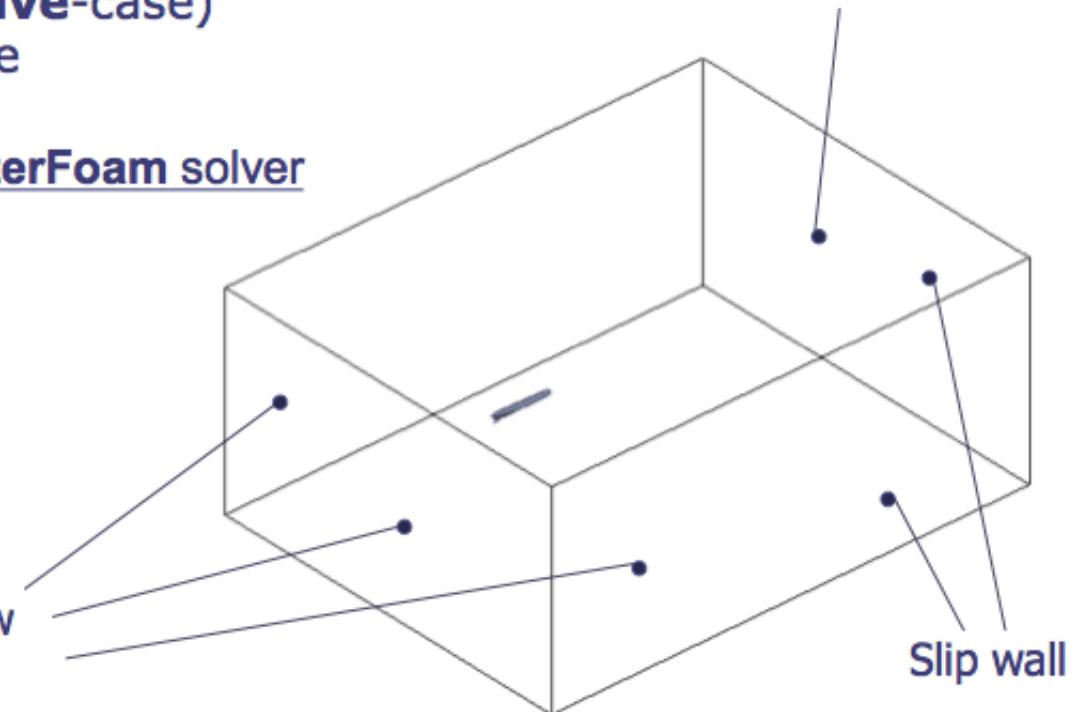
2. BC settings:

No slip wall on
boat surface

Velocity inflow
patch

Constant
pressure patch

Slip wall



CFD comparison method

1. Qualitative:

- Iso-surface of computed mass-fraction
- Pressure on hull surface

Information about **wave shape, flow separation, stress distribution** on hull

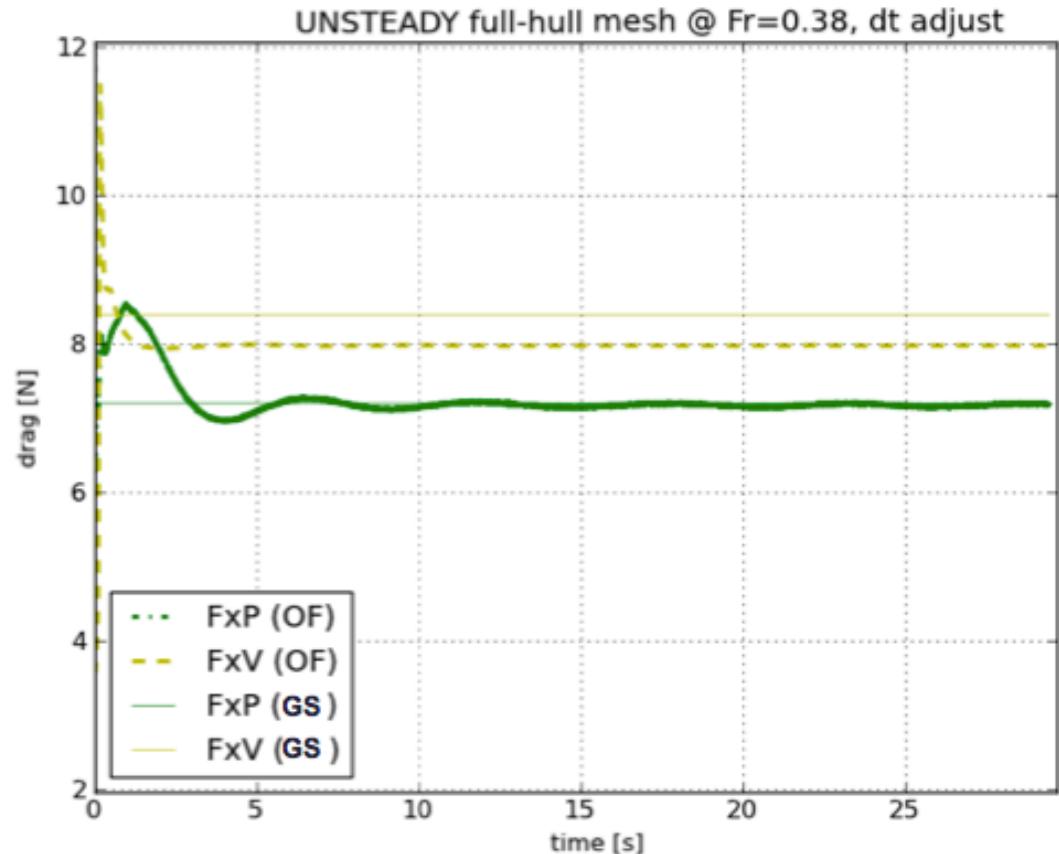
2. Quantitative:

- Pressure drag
- Viscous drag

My comparison is: OF vs GS (numerical)

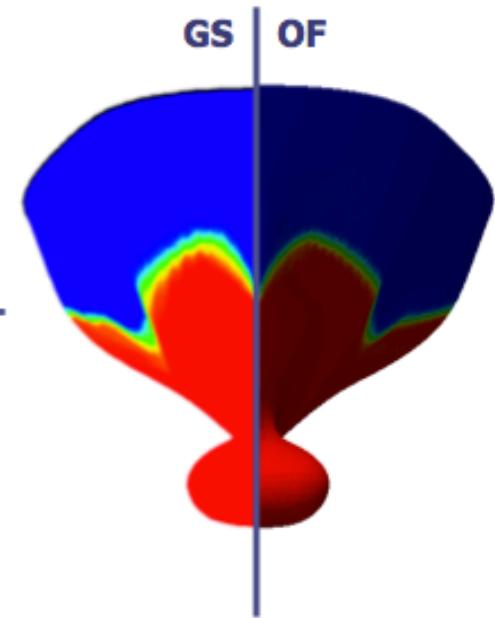
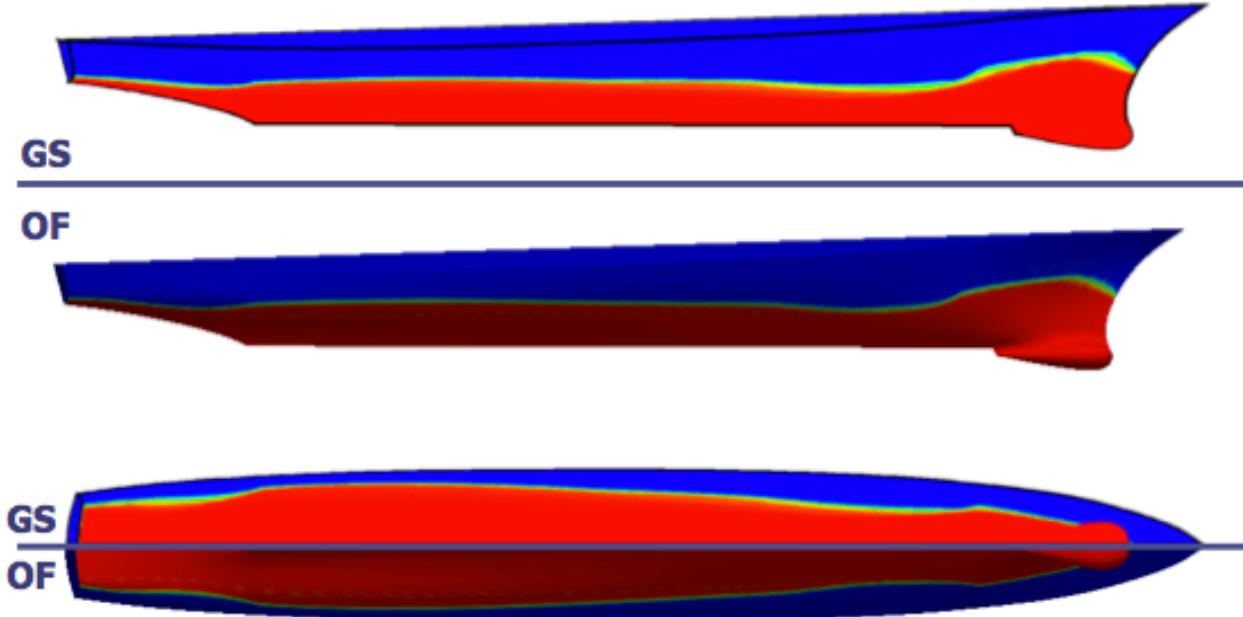
Unsteady captive - CFD Results

- Convergence reached in 10 s
- OpenFOAM vs Gold-Standard
Drag values :
 - FxP: -0.34 %
 - FxV: -4.95 %
- Quite good agreement of results

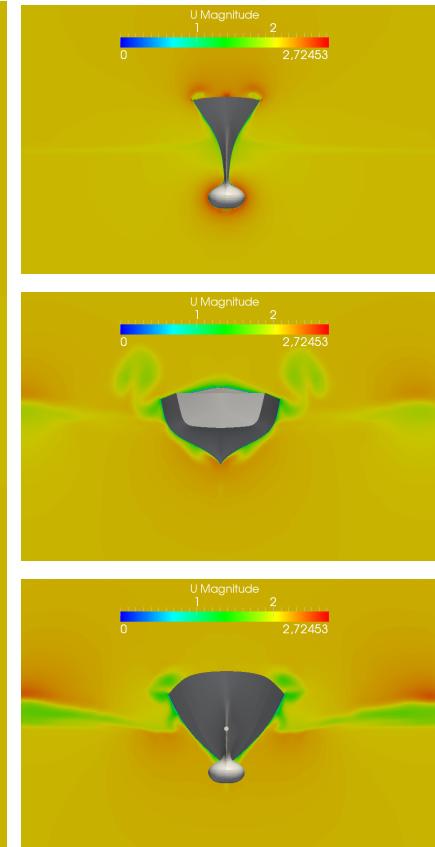
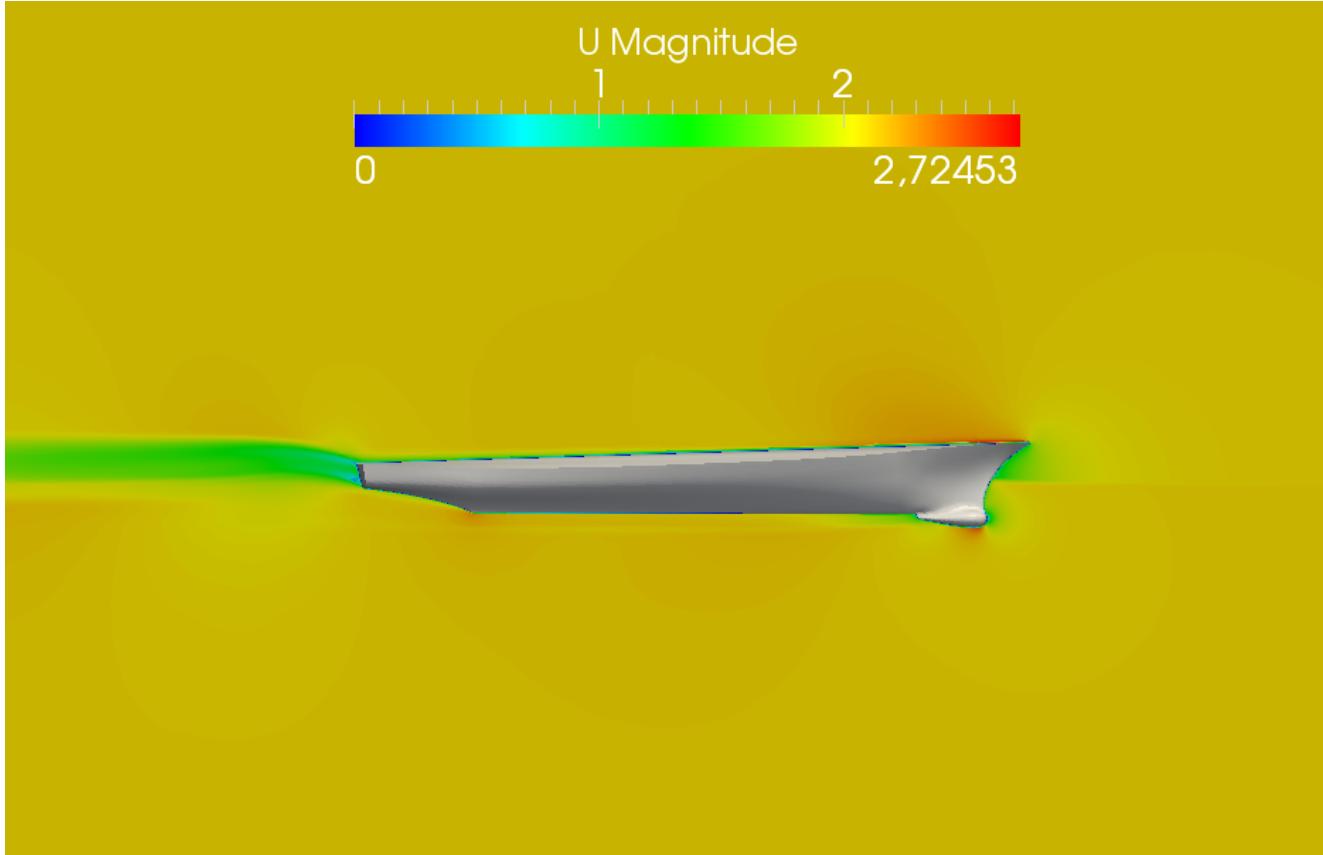


OpenFOAM vs commercial CFD (GS)

- Mass fraction visualized on hull surface: wave shape detected
- Excellent agreement with *Gold-Standard* results

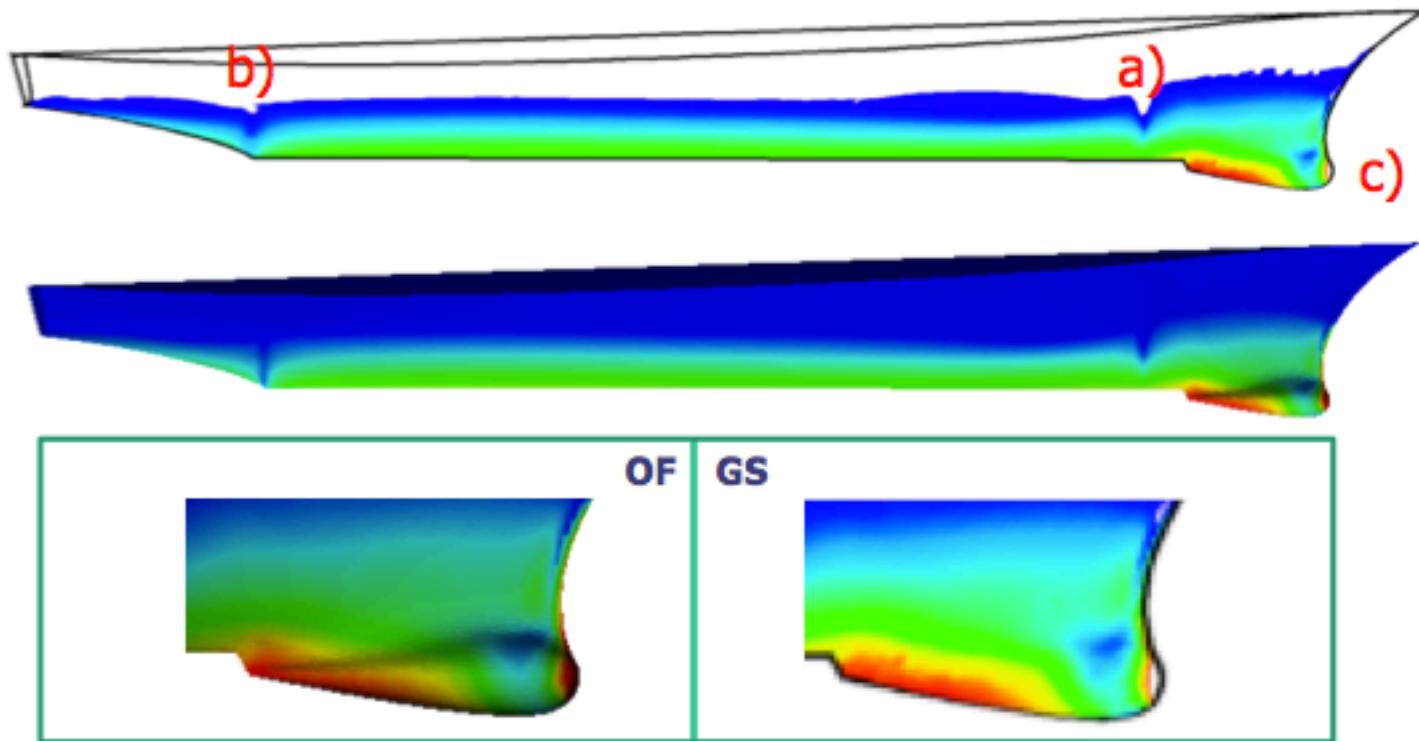


OpenFOAM velocity field



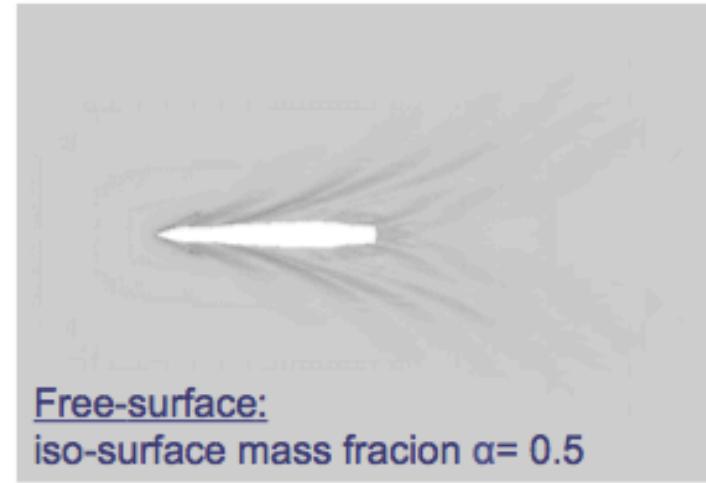
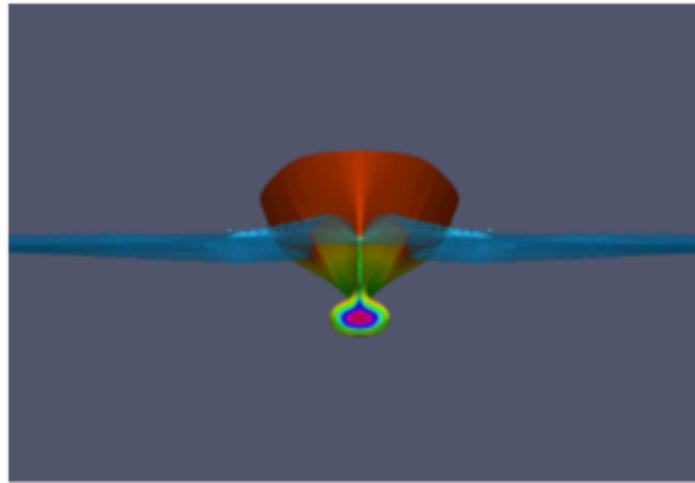
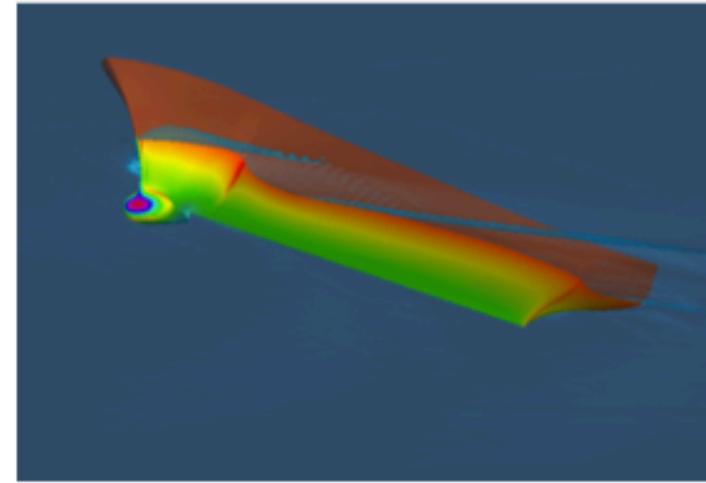
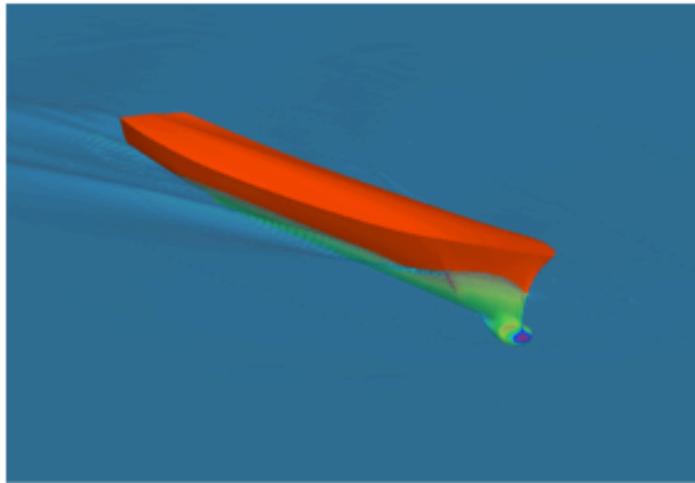
- Symmetry well caught by the solver in the velocity field computations

Pressure field over hull

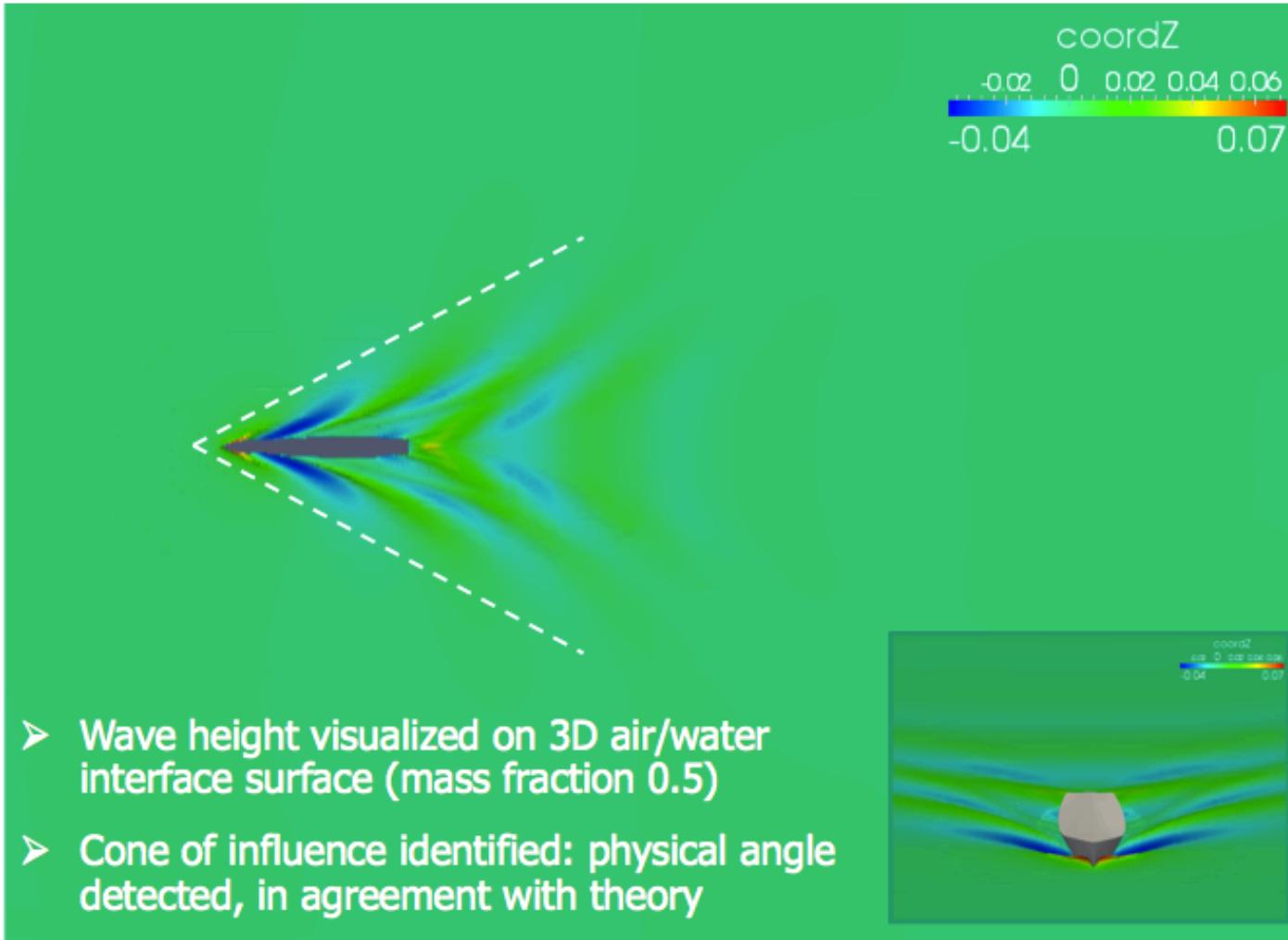


- a) Zero pressure distribution well caught
- b) Zero pressure transom well caught
- c) Bulb pressure distribution to be further investigated

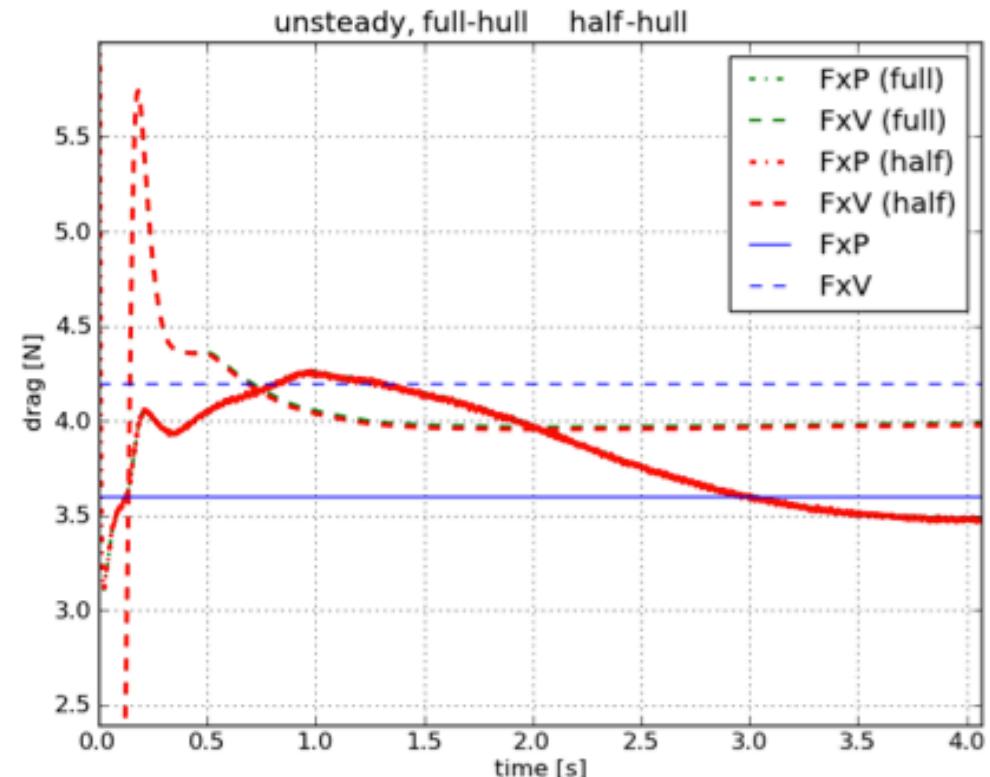
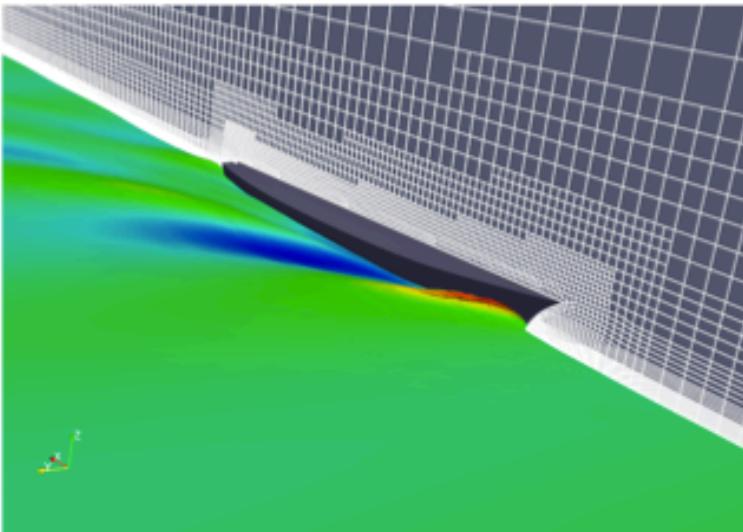
Free surface visualization



CFD results: agreement with theory

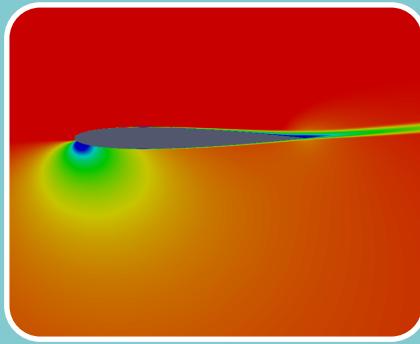
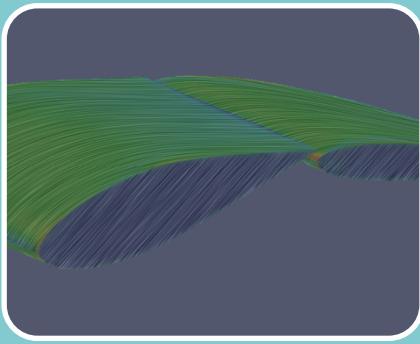
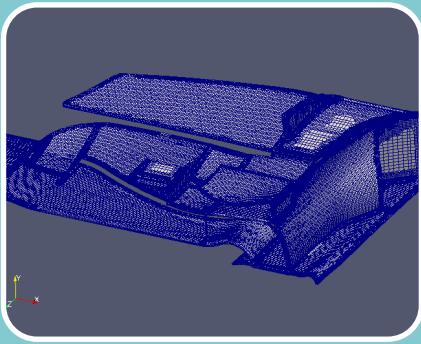
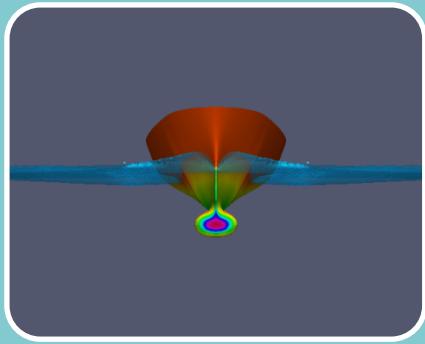


DTMB-5414: half hull simulations



➤ save 43.5% computational time

CFD skills applied to AC72 issues



Two phase High Reynolds RANS CFD analysis

Free surface simulation of high performance boat (AC72 kat) and appendages

3D complex geometries meshing

Highly automated meshing process of 3D complex shapes; fully-structured, hybrid or unstructured mesh on problem demand.

Aerodynamics

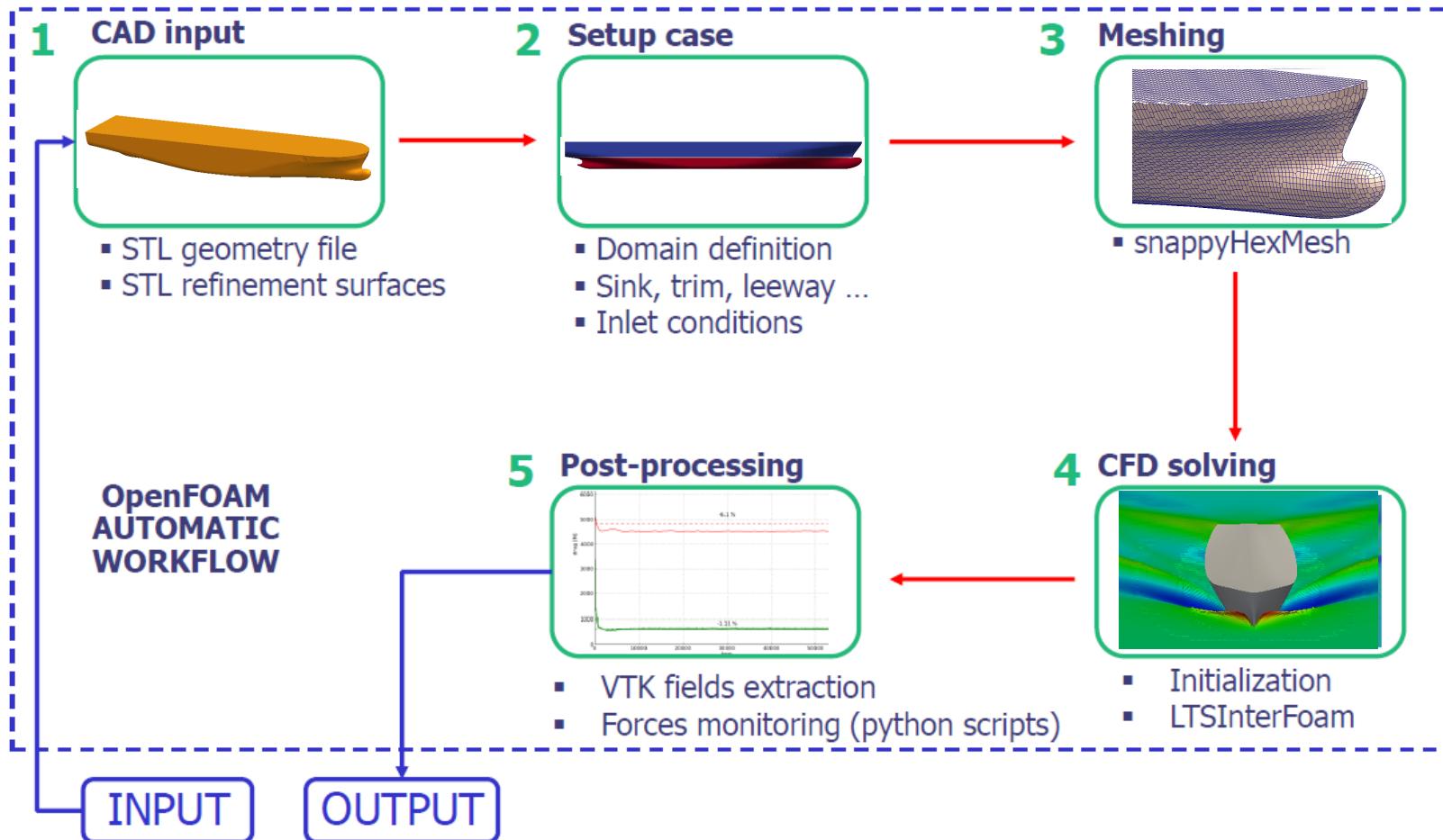
Aerodynamic of high Reynolds number RANS simulation of 3D bodies; high parallel CFD computations

2D airfoil design

Wing section efficient RANS simulation. Airfoil design optimization based on RANS code data



Marine CFD automatic workflow



OpenFOAM automatic workflow evaluation

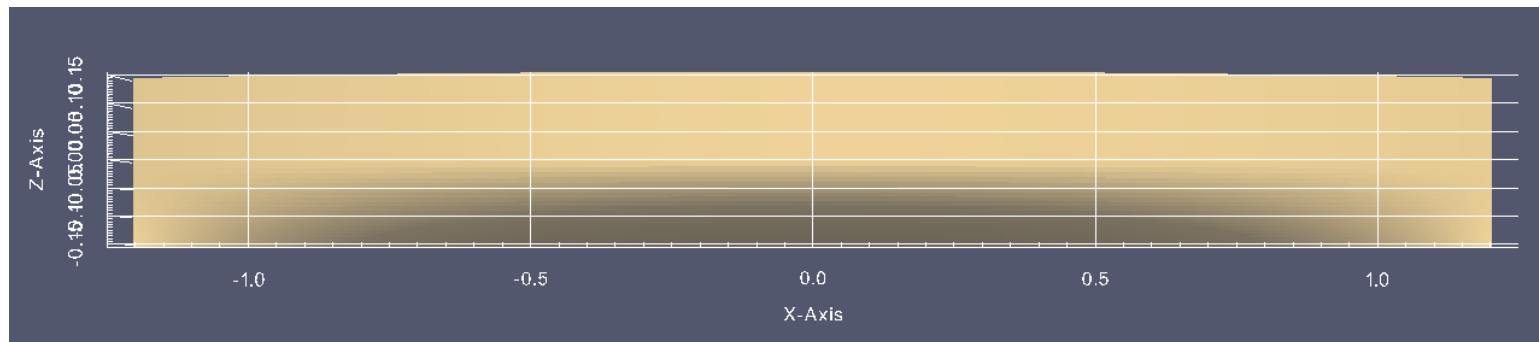
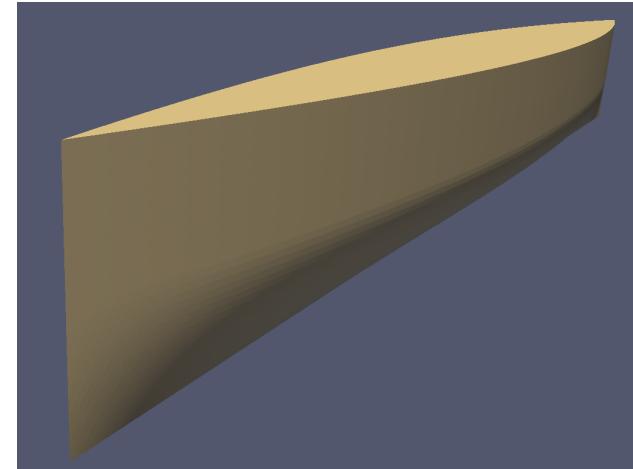
Automation

- ① Accuracy
- ② Scalability
- ③ Reliability

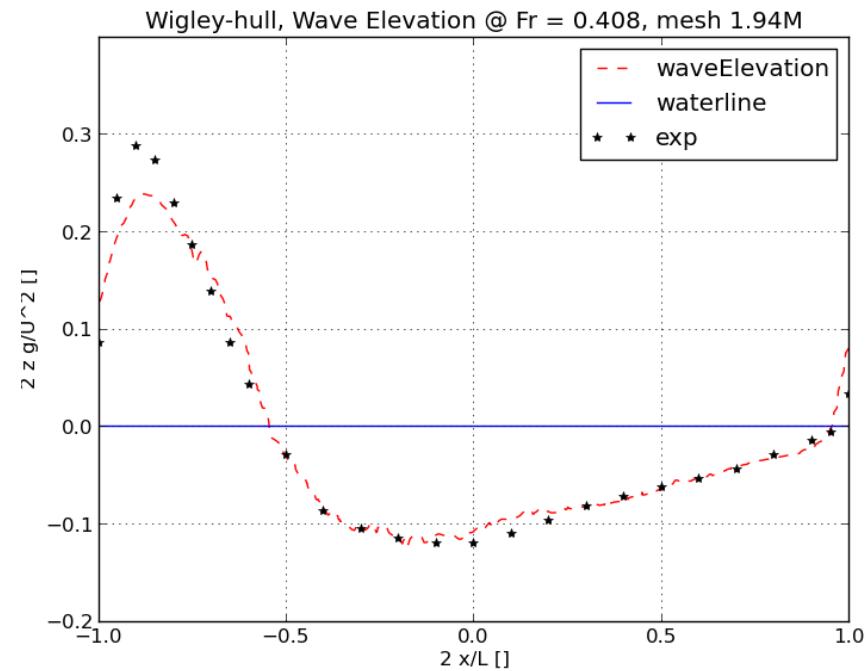
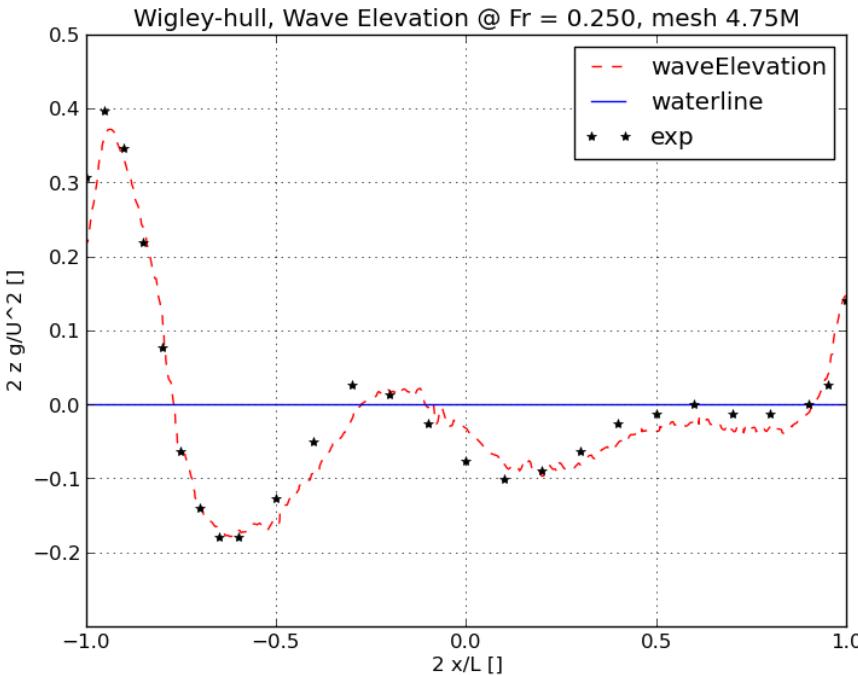
**Ready to CFD
production on
HPC cluster**

Wigley-hull

- **Description:** widely used in marine engineering for validation of measures
- Standard reference



Accuracy: CFD vs experimental

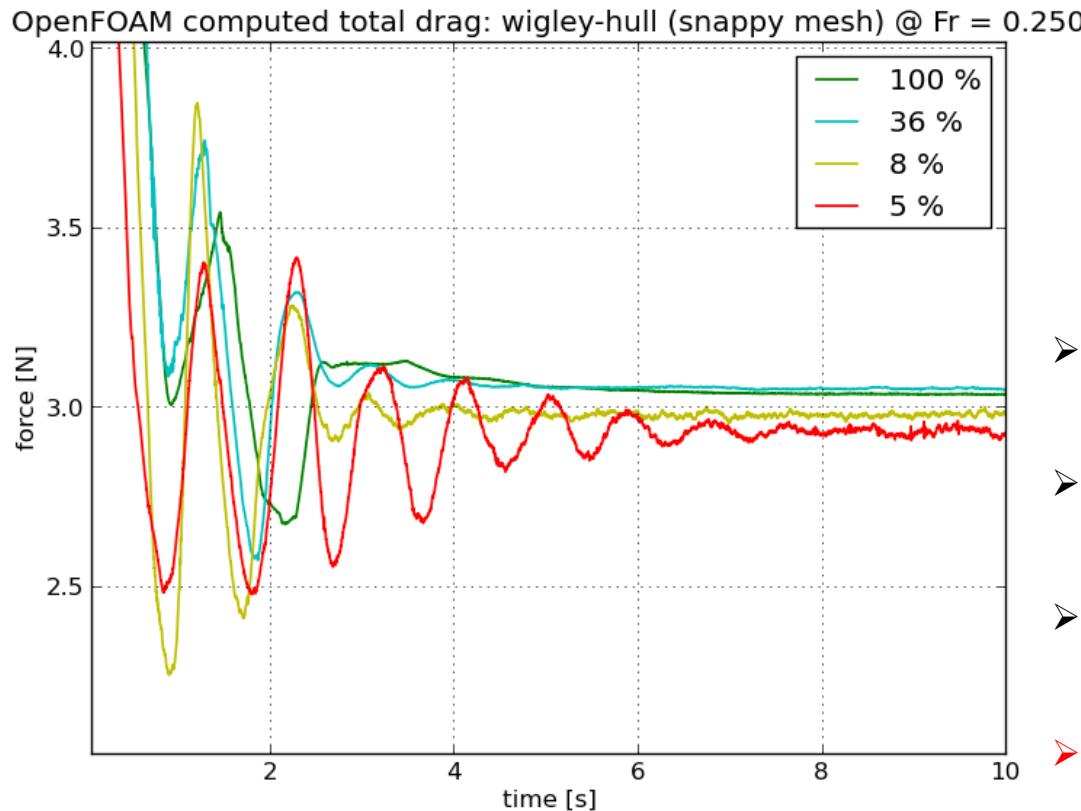


- Wigley-hull **wave elevation** @ different Froude number

Accuracy: mesh sensitivity

- Fixed Froude number. On purpose degradation of mesh reducing number of cells to investigate how total computed forces become (in)accurate
- Considerable advantages in elapsed time required
- Mesh size range [% cells respect to gold-standard mesh]:
5.0% - 8.0% - 36.% - **100.% (gold-standard)**
- Cores range: 12 – 24 @ PLX, CINECA cluster

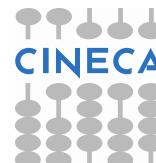
Accuracy: mesh sensitivity



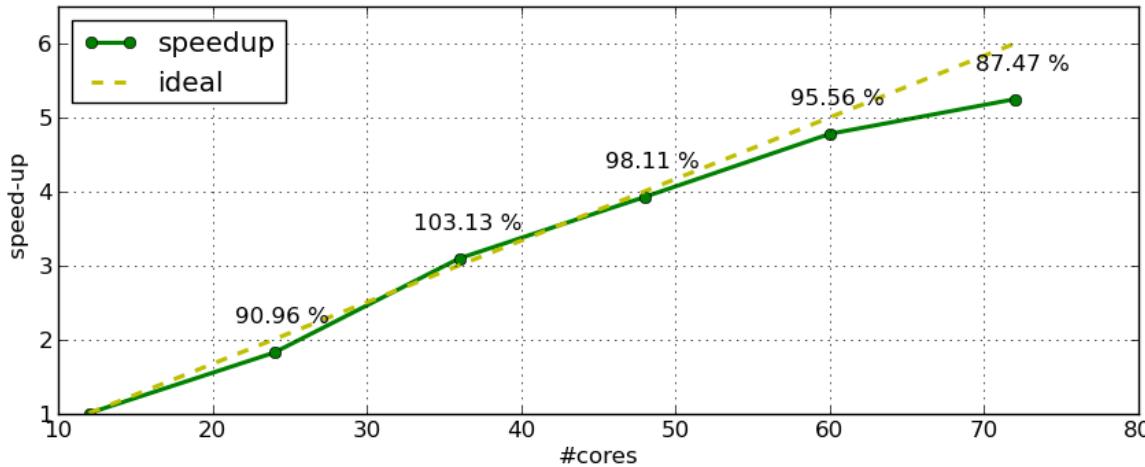
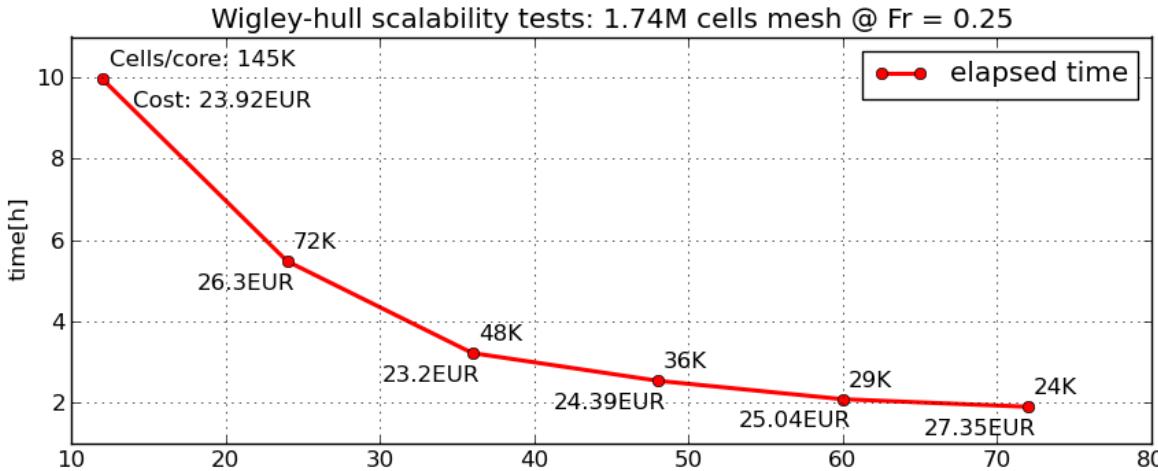
- Reducing mesh size is not critical for the absolute convergence but just delays it.
- 5% size mesh respect to GS produces a 3% discrepancy in the total computed drag
- 5% size mesh respect to GS requires just 2h @ 12 cpu to reach convergence
- **User choice: different accuracy, different mesh size, different cost.**

Scalability

- Different elapsed time due to different used computational cores
- Fixed **mesh** size: 1.7 M cells
- **Cores** range: 12 – 24 – 36 – 48 – 72 @ PLX, CINECA cluster
- Fixed number of iterations: **5000** (up to convergence)
- Key value **indices**: elapsed-time, speedup, efficiency



Scalability results



- Speed-up:

$$S = \frac{ets}{et_p}$$
- Efficiency:

$$E = \frac{S}{cores}$$

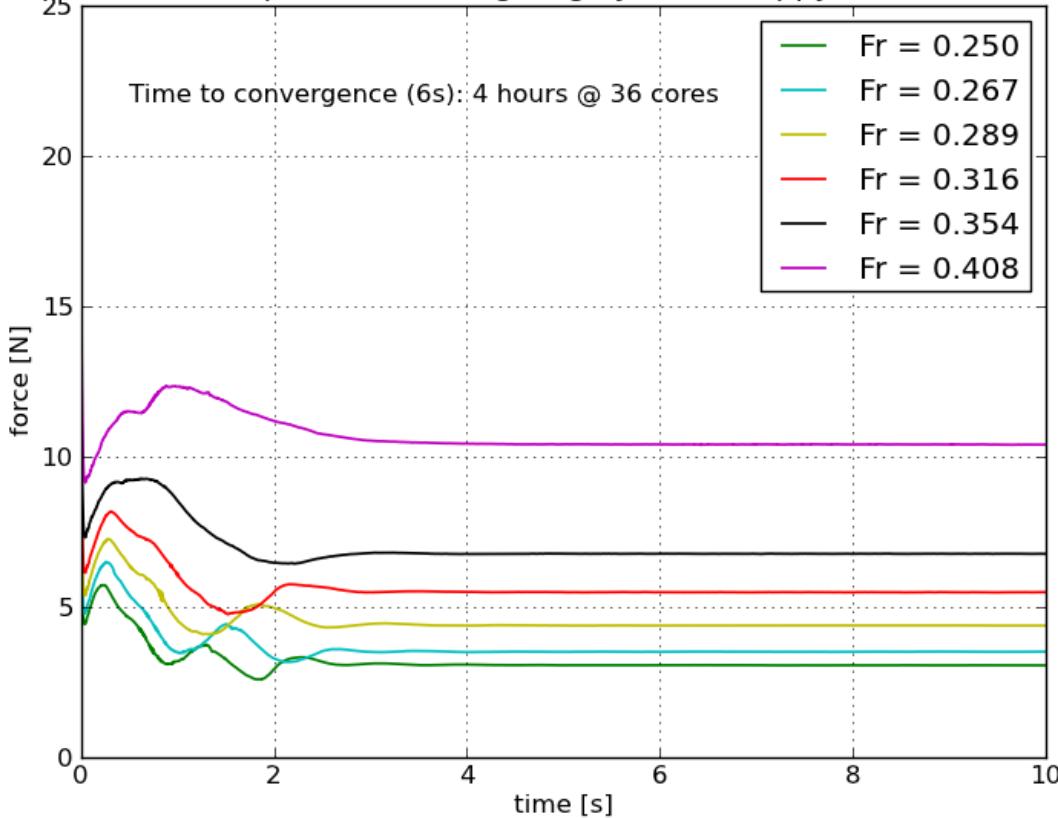
- Convergence reached in **2h**
- High efficiency up to **24k cells/core**

Reliability

- Different computed forces due to different Froude number e.g. inlet velocity
- Fixed **mesh** size: 1.7 mln cells
- Fixed number of **cores**: 36 @ PLX, CINECA cluster
- **Froude** number range: 0.250 0.267 0.289 0.316 0.354 0.408
- Key value **indices**: total forces, viscous forces, pressure forces, wave height

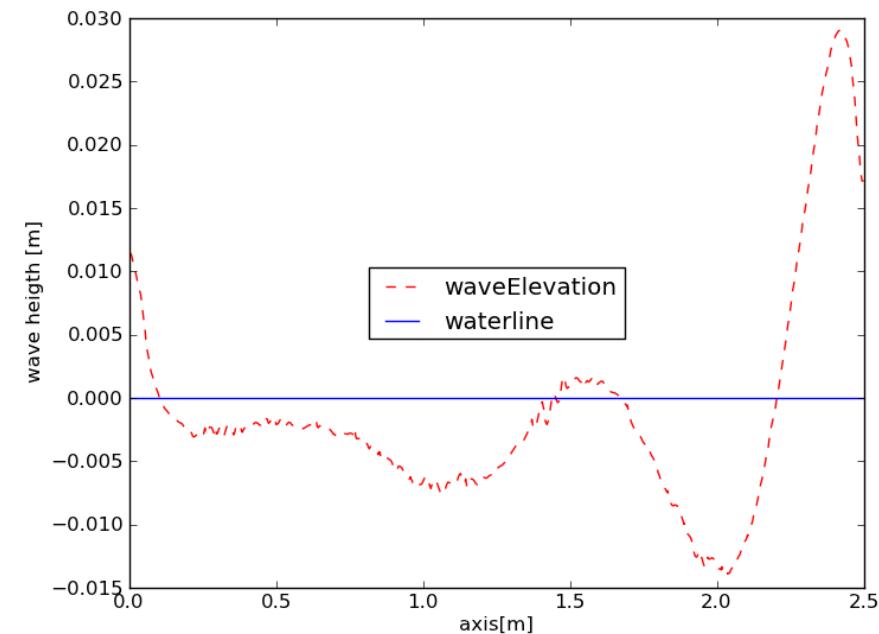
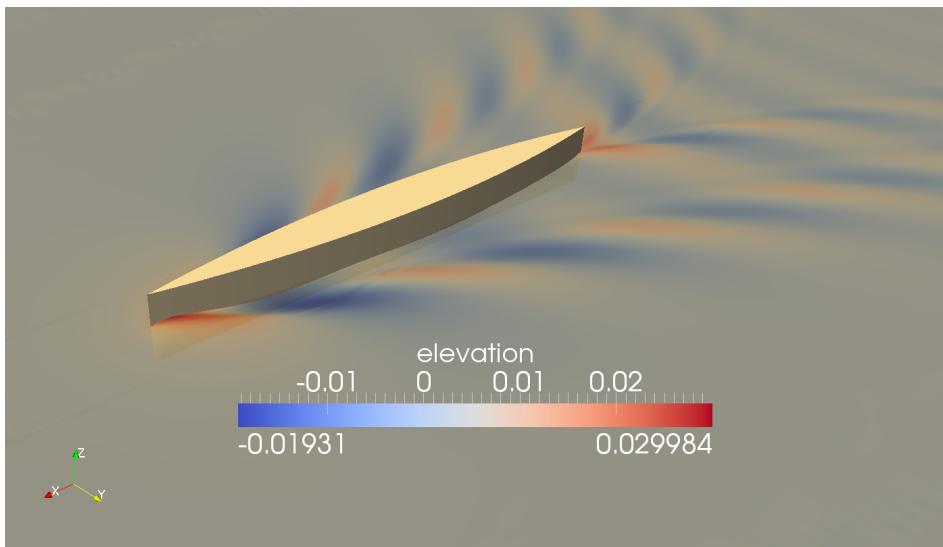
Reliability: results

OpenFOAM computed total drag: wigley-hull (snappy mesh: 1.8M cells)

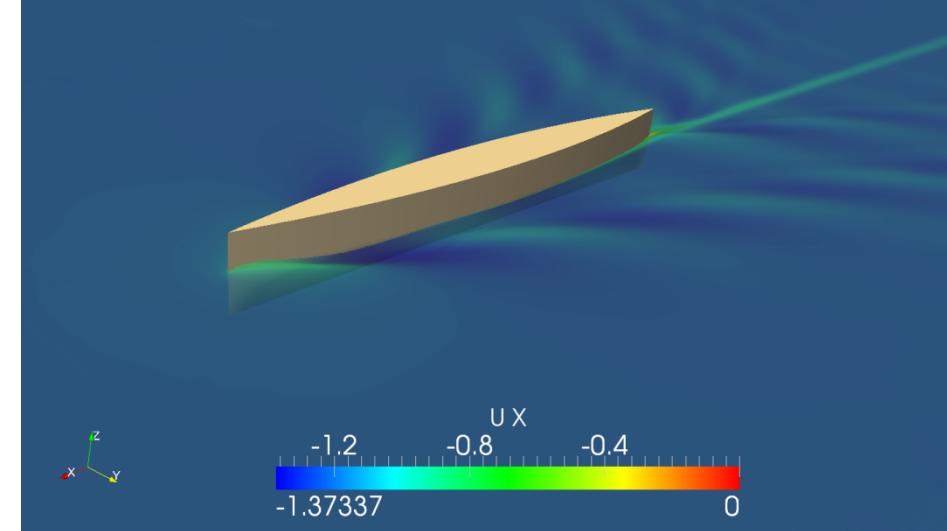
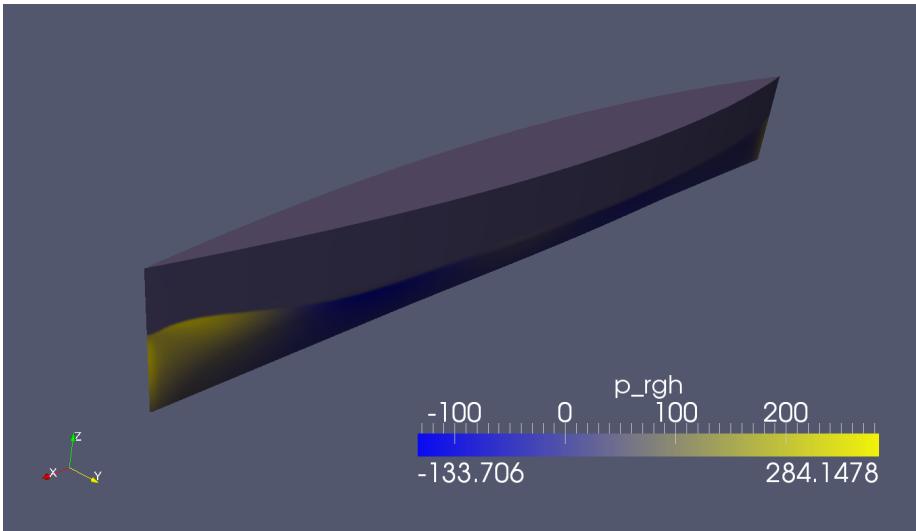


- Stable solution reached within **4s** (4k iteration for LTS solver)
- Fixed **cut-off** at 6s.
- Stable **means** are computed in the selected range 4s – 6s, so 4h @ 36 cpu exploiting best scalability

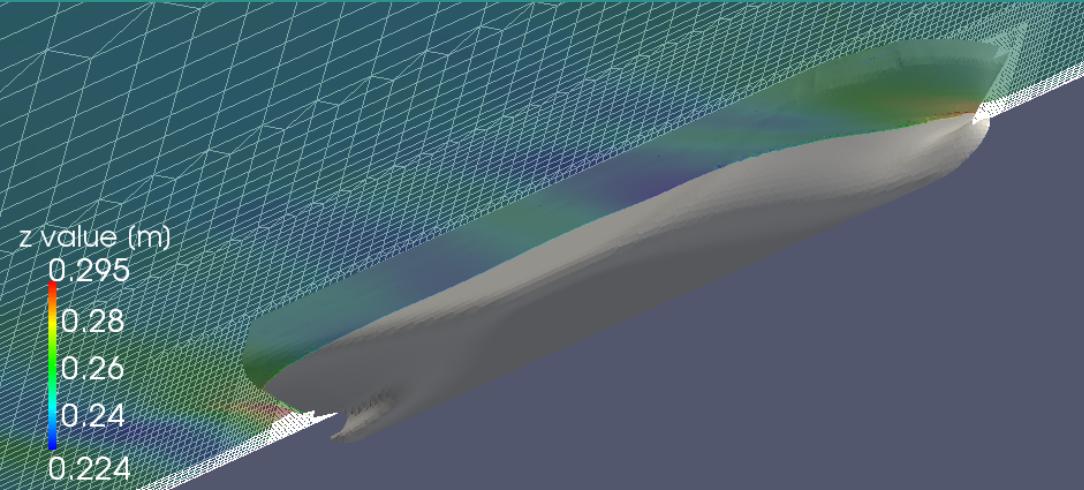
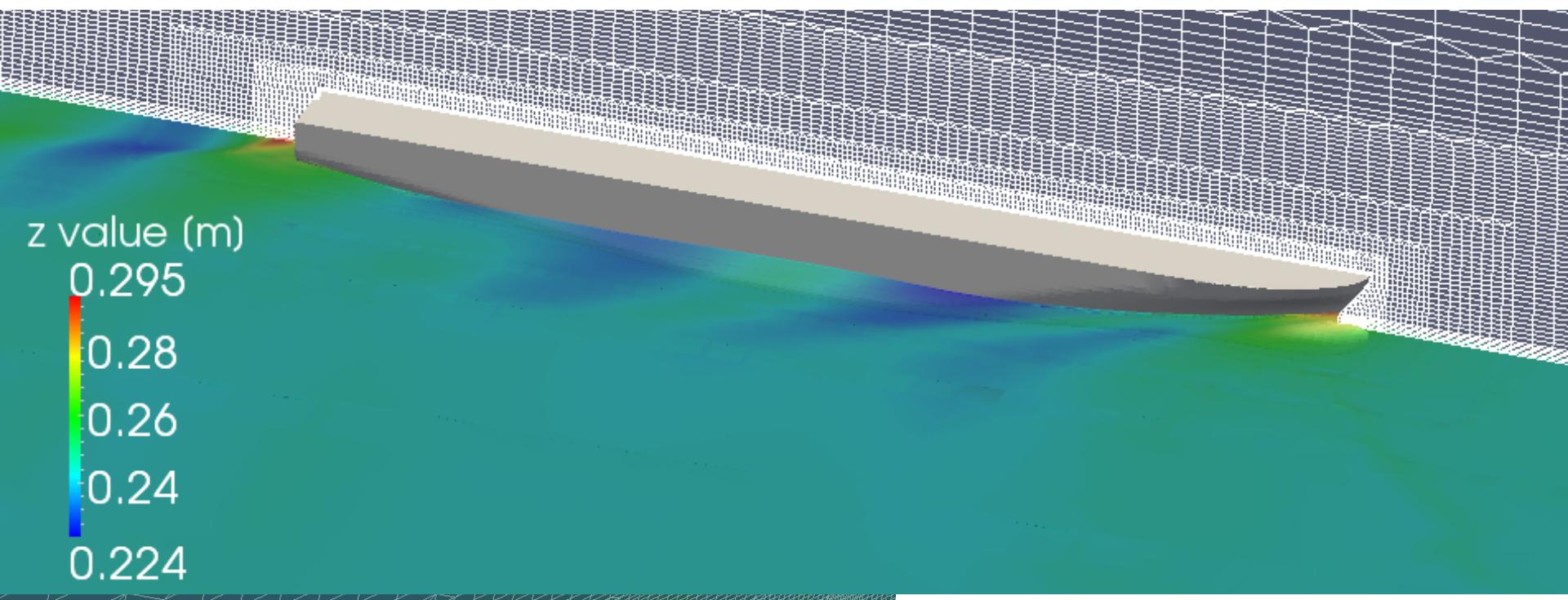
Wave elevation ($\alpha = 0.5$)



Pressure & axial velocity

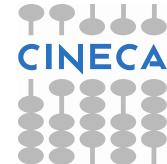


Hands-on: CFD result



DTC Hull tutorial:

\$FOAM_TUTORIALS/multiphase/
LTSInterFoam/DTCHull



Hands-on: OpenFOAM commands

① CAD transformation: scaling, trim, sink

- surfaceTransformPoints –scale
 - yawPitchRoll
 - translate

② Setup constants:

- Edit constant/transportProperties
- Edit constant/RASProperties

③ Setup BCs:

- Edit 0.org files

④ Setup free surface initial position:

- Edit system/setFieldDict
- Run setFields

⑤ Decompose domain:

- Edit system/decomposeParDict
- Run decomposePar

⑥ Run solver:

- mpirun –np ... LTSInterFoam -parallel

⑦ Reconstruct domain

- Run reconstructPar