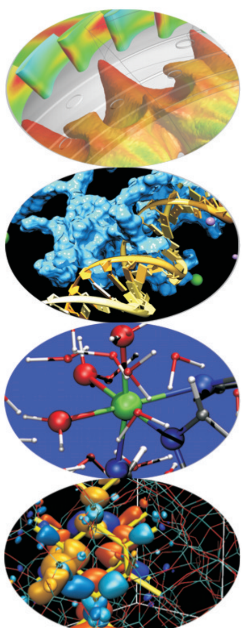
Marine CFD applications using OpenFOAM



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:

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

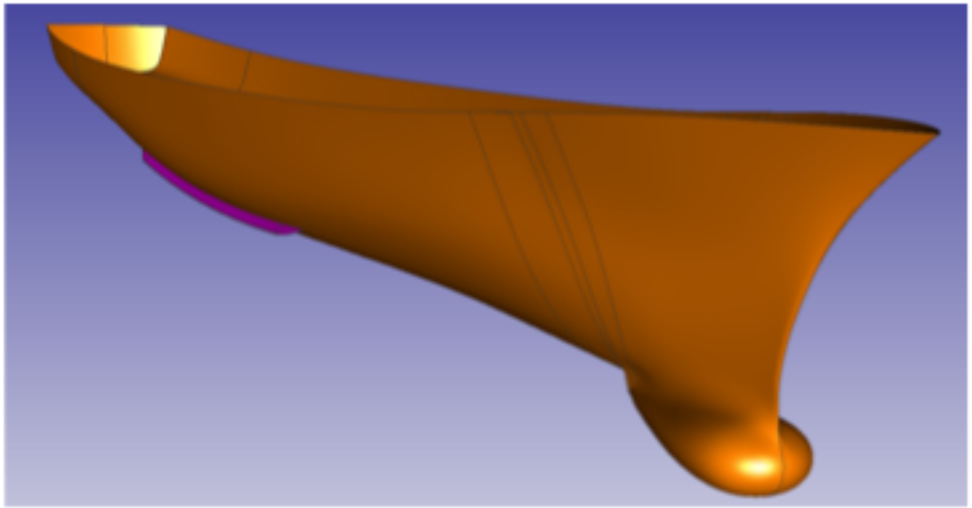
- AC72 class: high speed (u), sufficiently small y+ → **too small time-step**
- **Commercial** softwares can manage 100-1000 times larger dt
- **Unsteady** simulation in OpenFOAM results too time consuming at the moment, but used only if mandatory due to the physics of the problem
- **LTSInterFoam**: 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: $y^+ \approx 70$



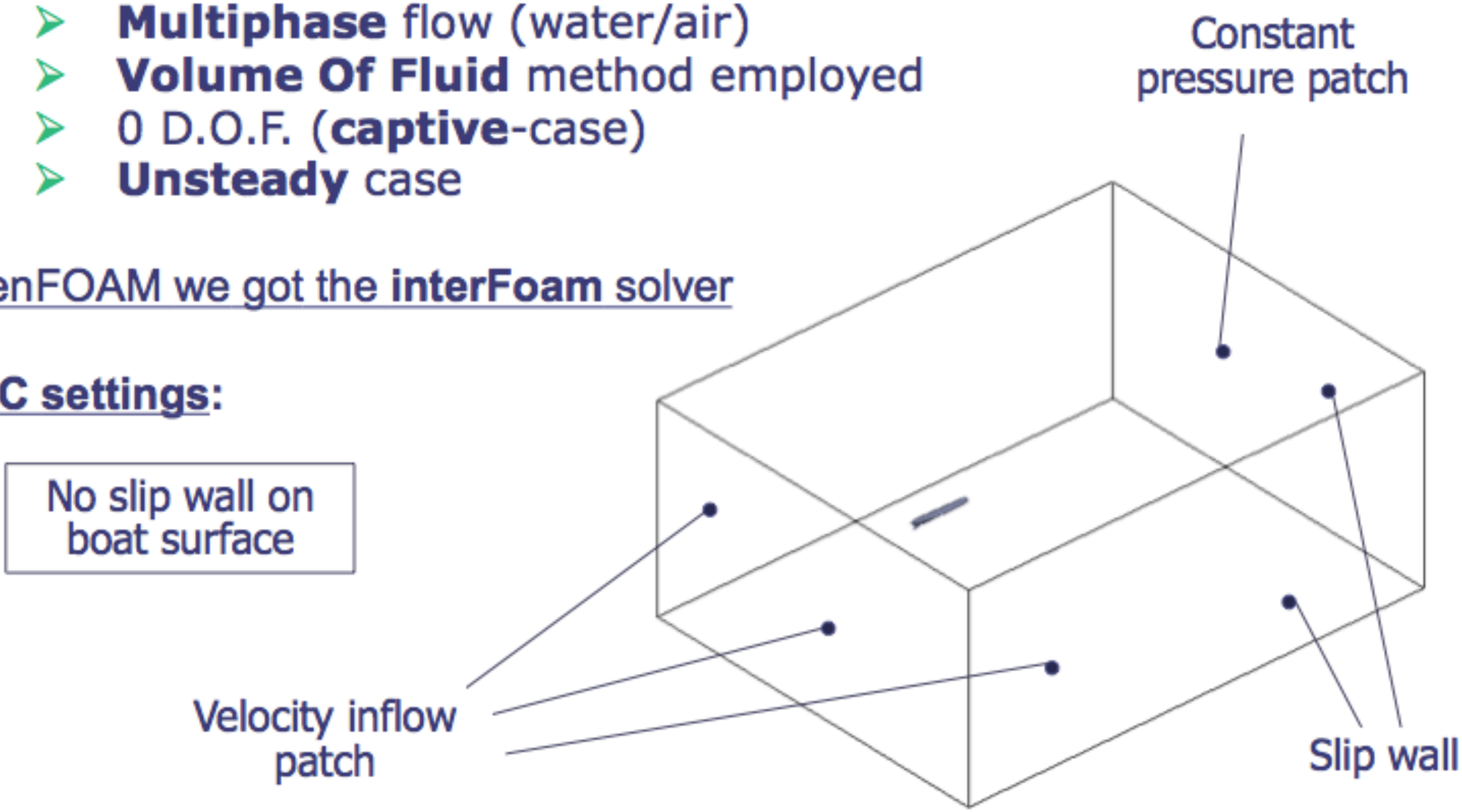
- *Standard DTMB-5415 bare hull modeled*

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:**



CFD comparison method

1. Qualitative:
   - Iso-surface of computed mass-fraction
   - Pressure on hull surface

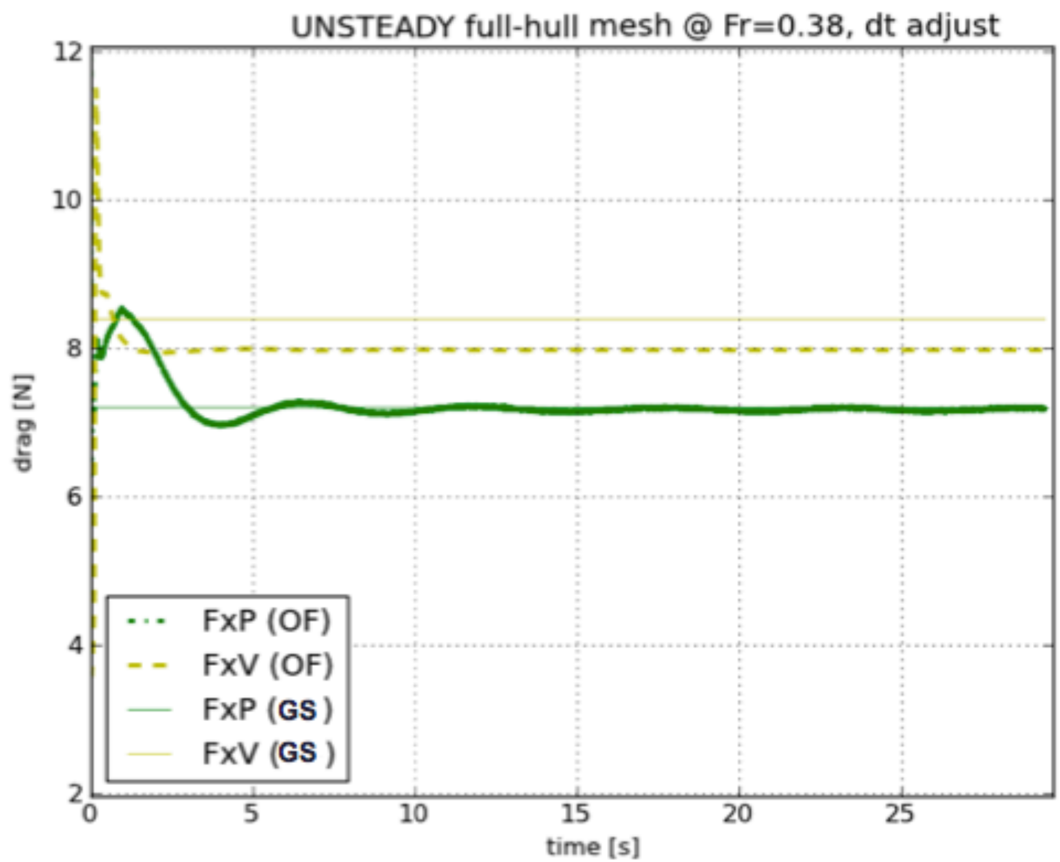
   Information about **wave shape, flow separation, stress distribution** on hull

2. Quantitative:
   - Pressure drag
   - Viscous drag

My comparison is: **OF vs GS** (numerical)

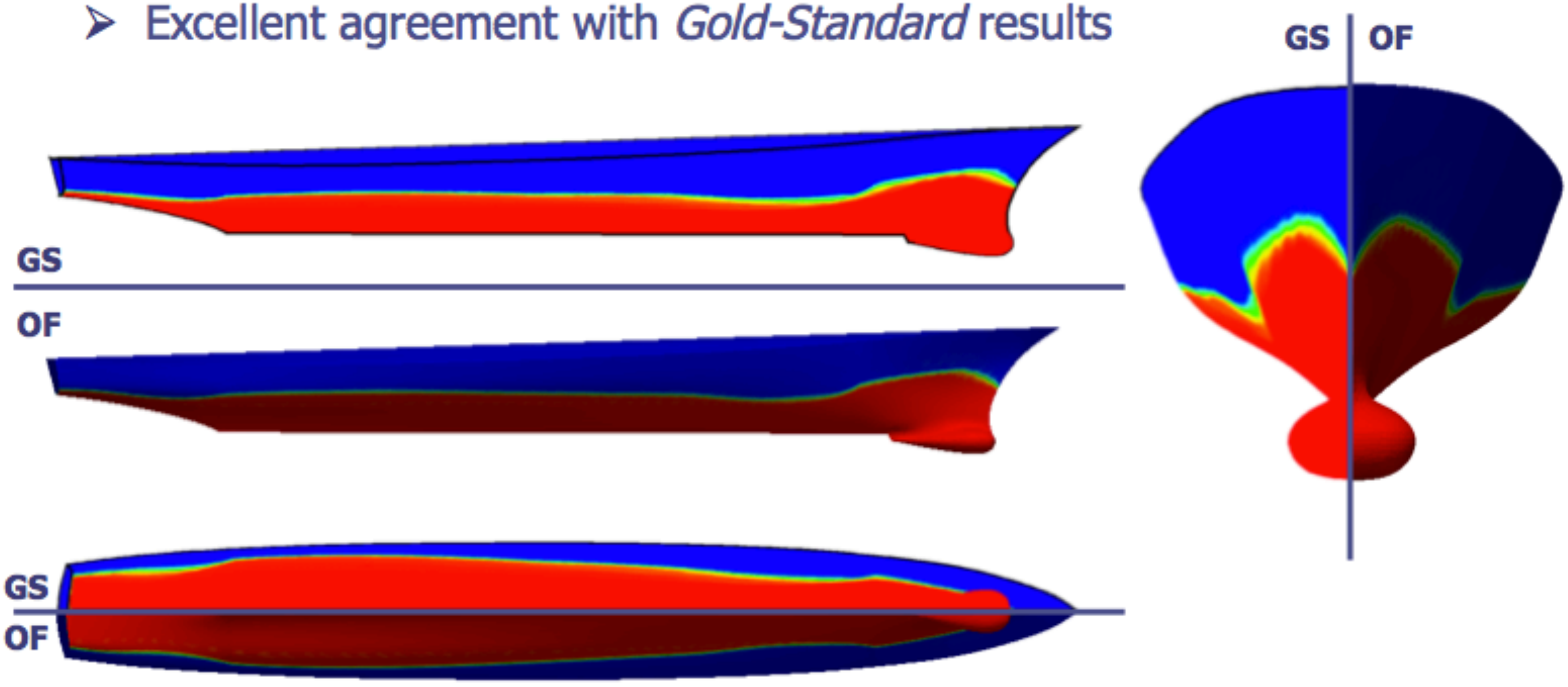
Unsteady captive - CFD Results

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

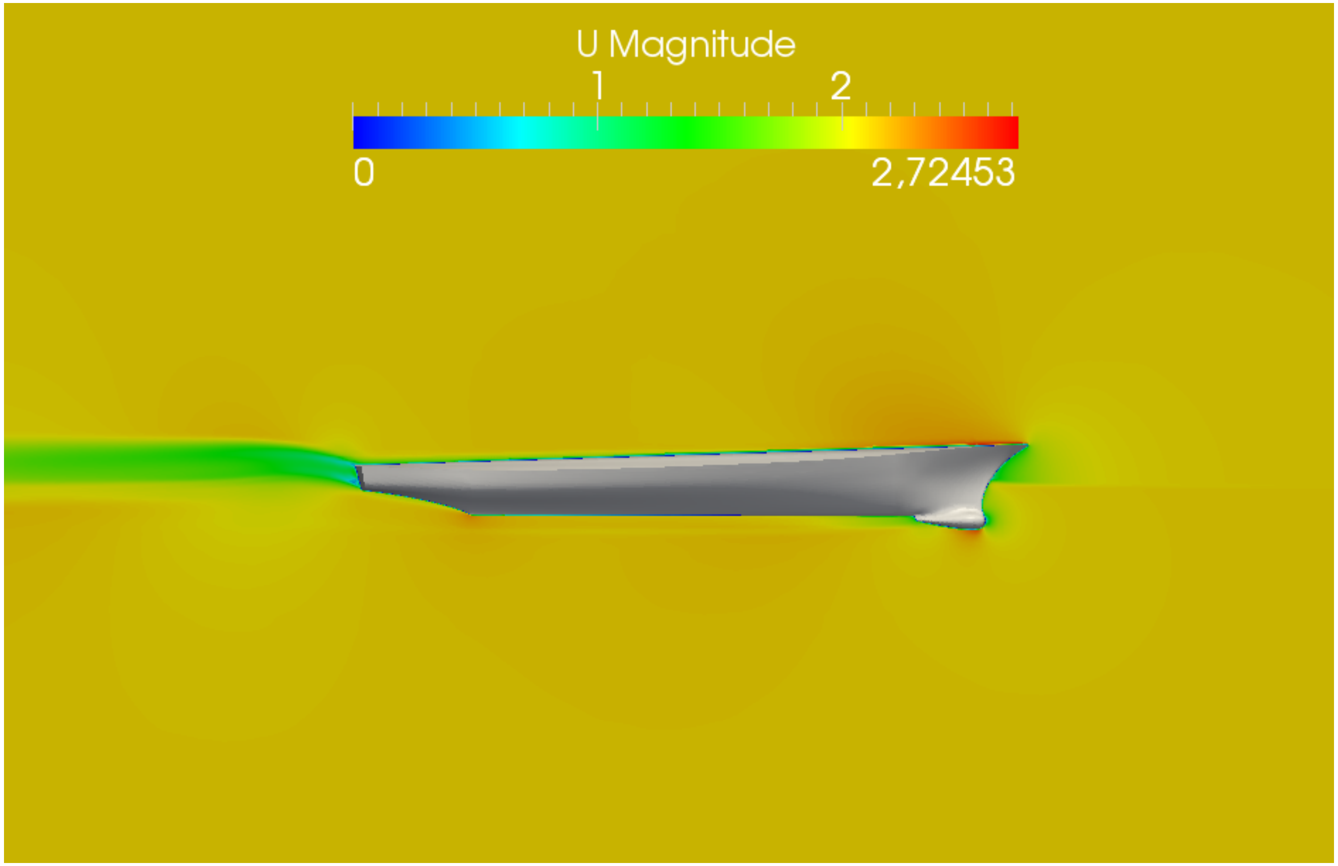


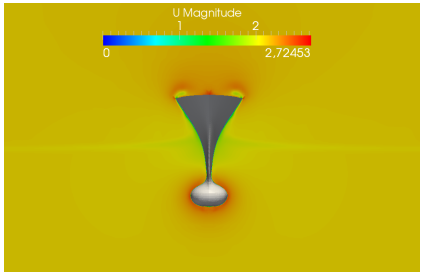
OpenFOAM vs commercial CFD (GS)

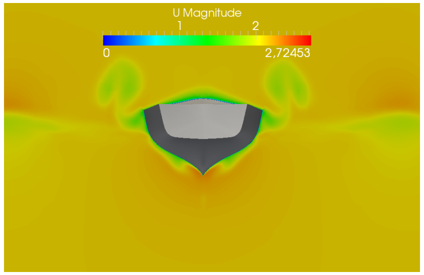
- Mass fraction visualized on hull surface: wave shape detected
- Excellent agreement with *Gold-Standard* results

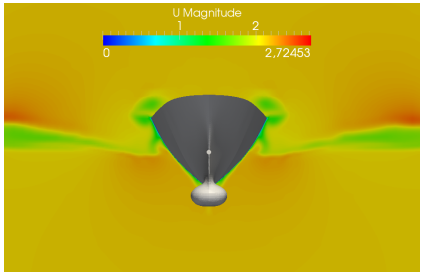


OpenFOAM velocity field



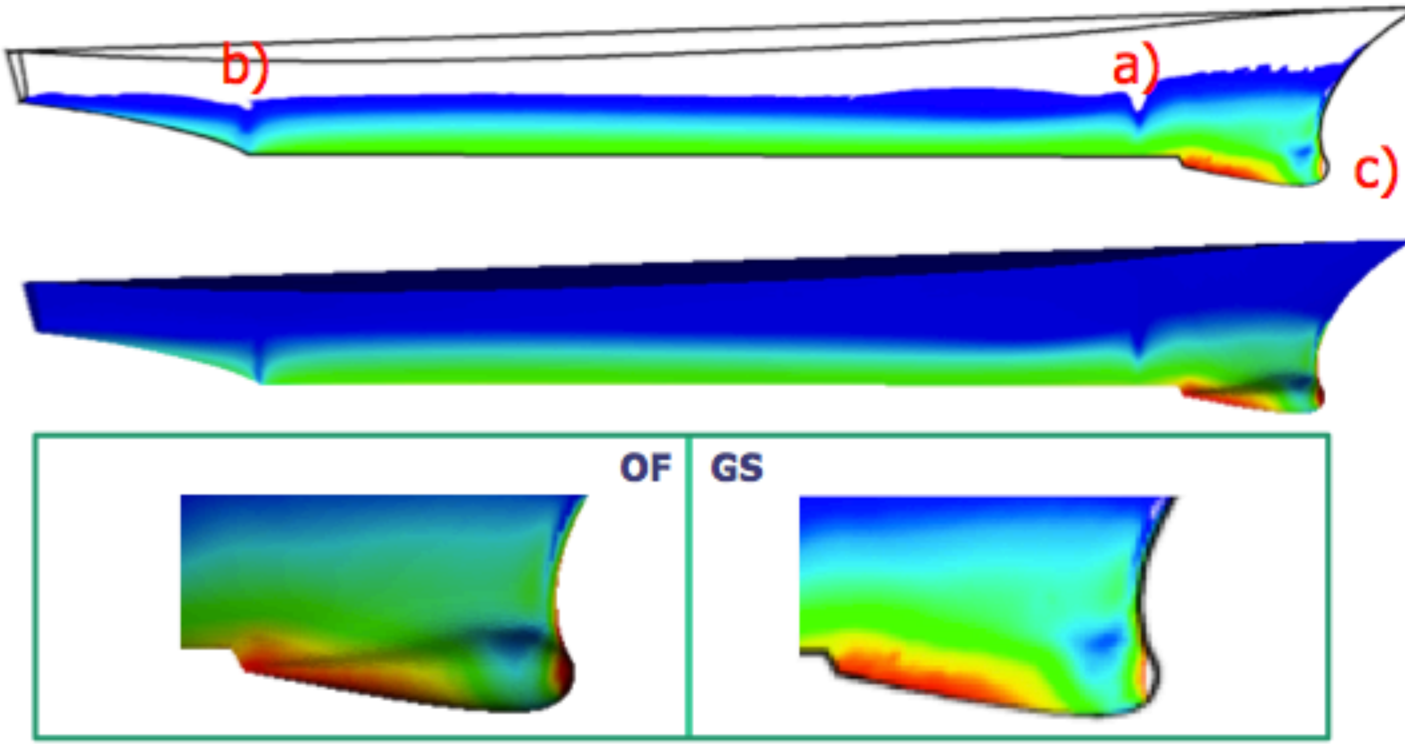






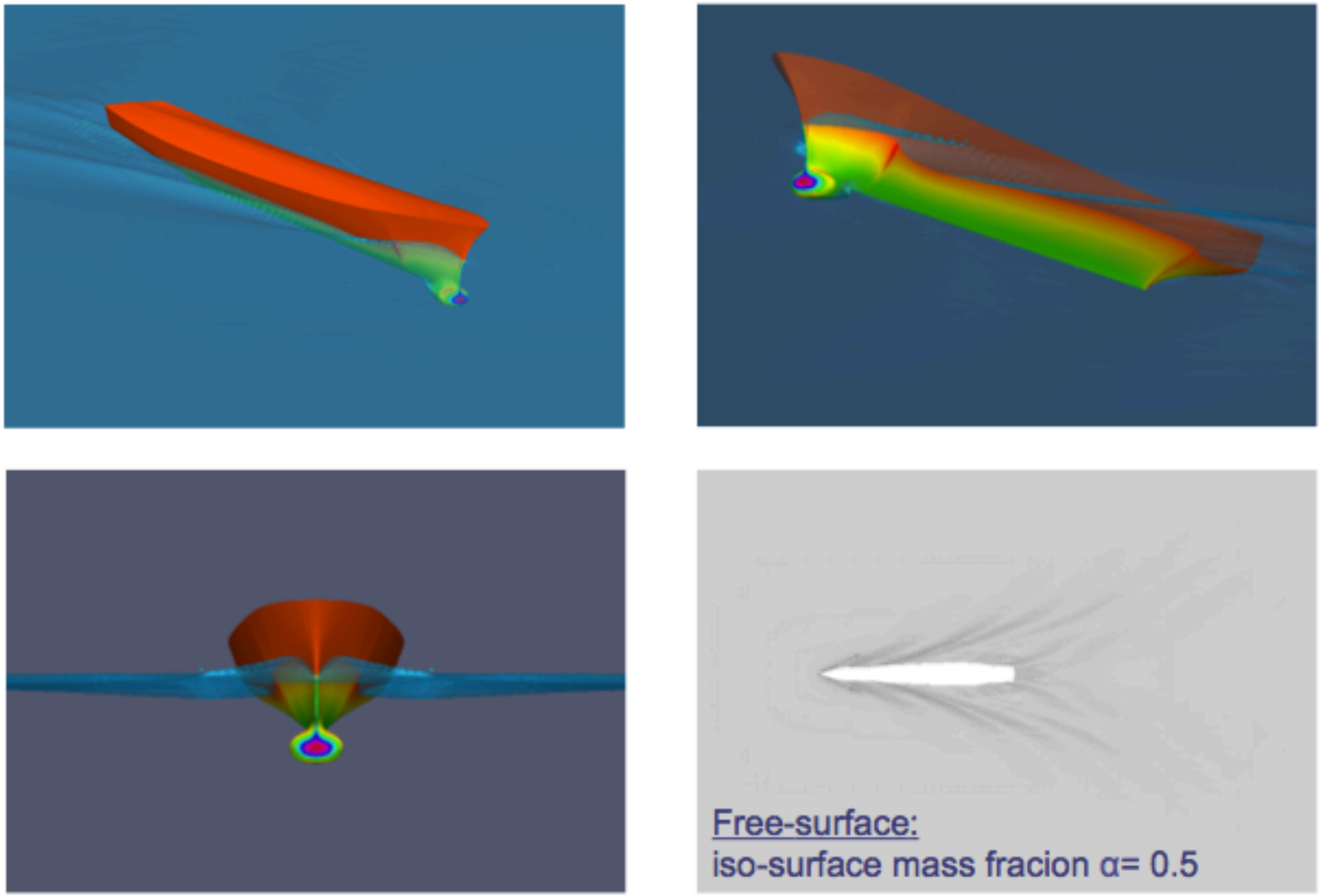
- Symmetry well caught by the solver in the velocity field computations

Pressure field over hull

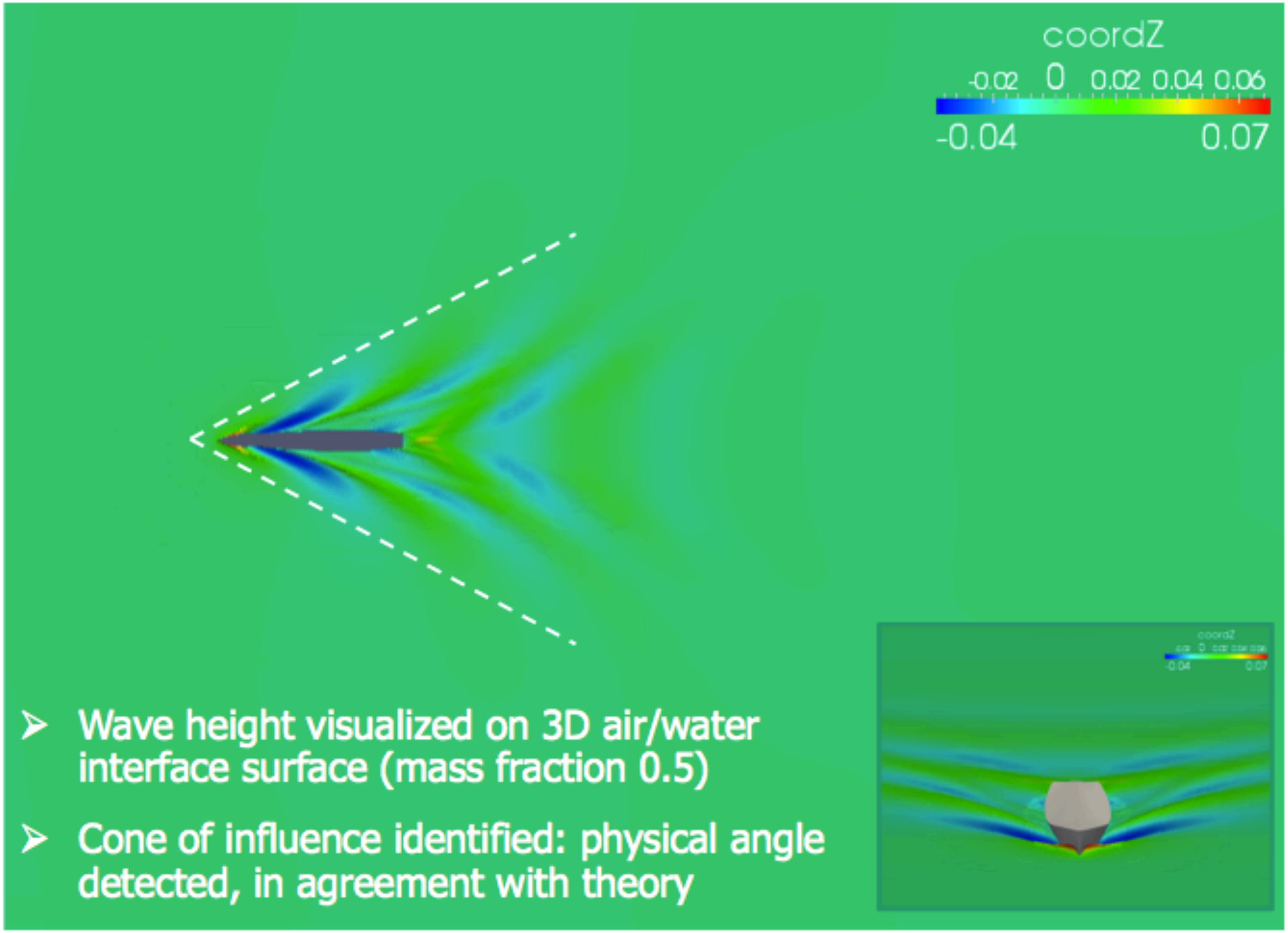


a) Zero pressure distribution well caught
b) Zero pressure transom well caught
c) Bulb pressure distribution to be further investigated

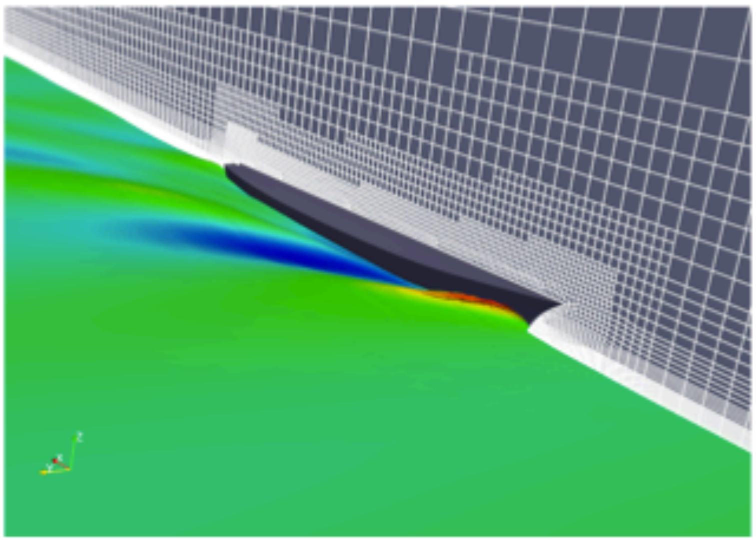
Free surface visualization

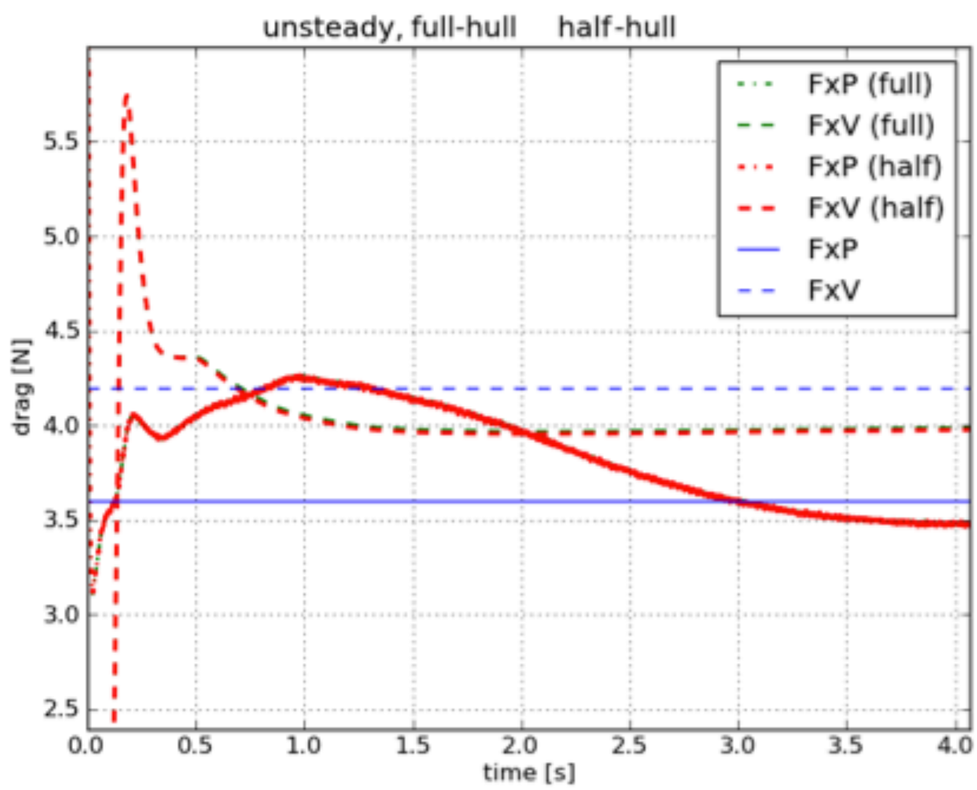


CFD results: agreement with theory



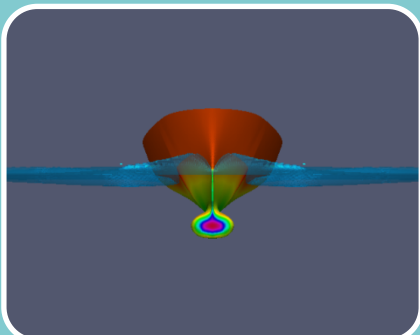
DTMB-5414: half hull simulations





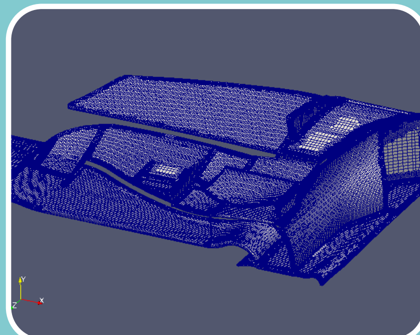
- save 43.5% computational time

CFD skills applied to AC72 issues



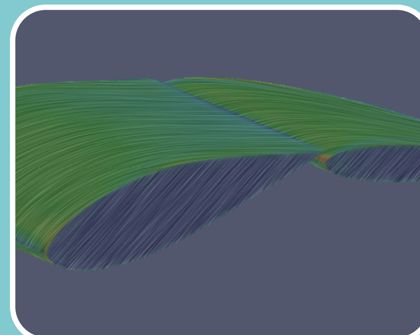
Two phase High Reynolds RANS CFD analysis

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



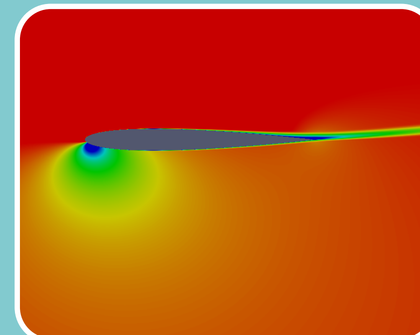
3D complex geometries meshing

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



Aerodynamics

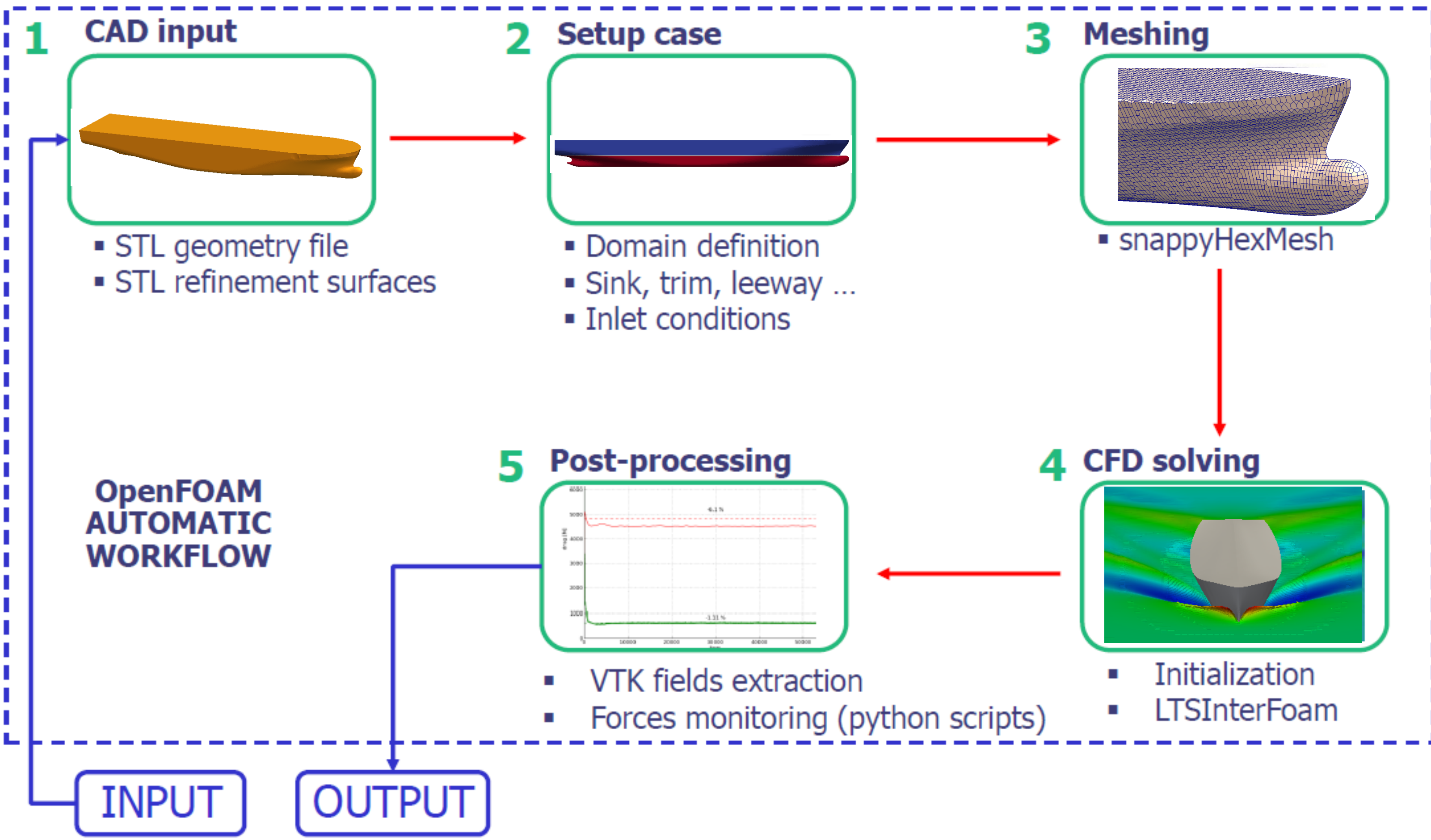
Aerodynamic of high Reynolds number RANS simulation of 3D bodies; high parallel CFD computations



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

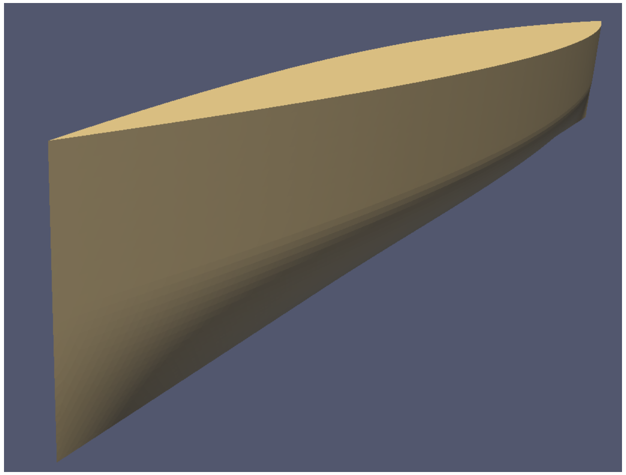
② Scalability

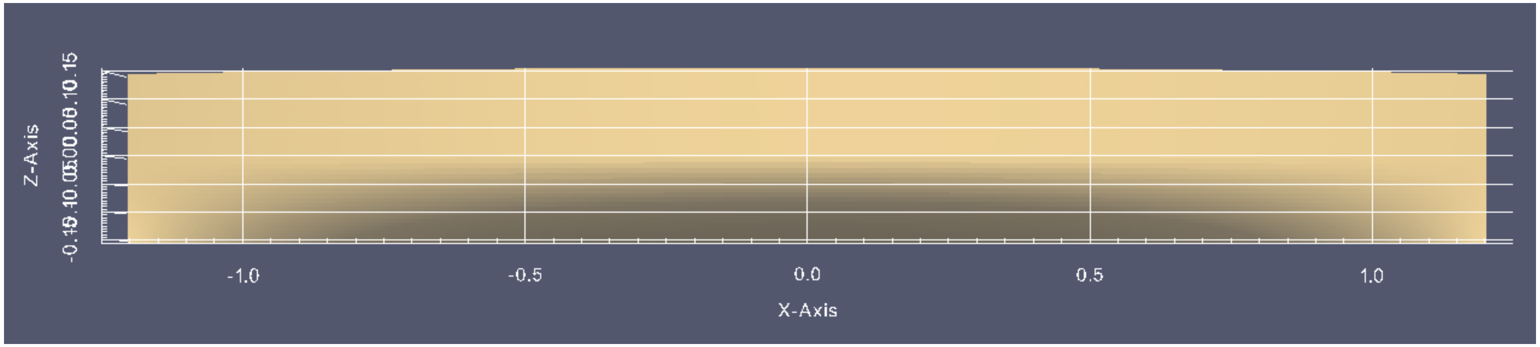
③ Reliability

Ready to CFD production on HPC cluster

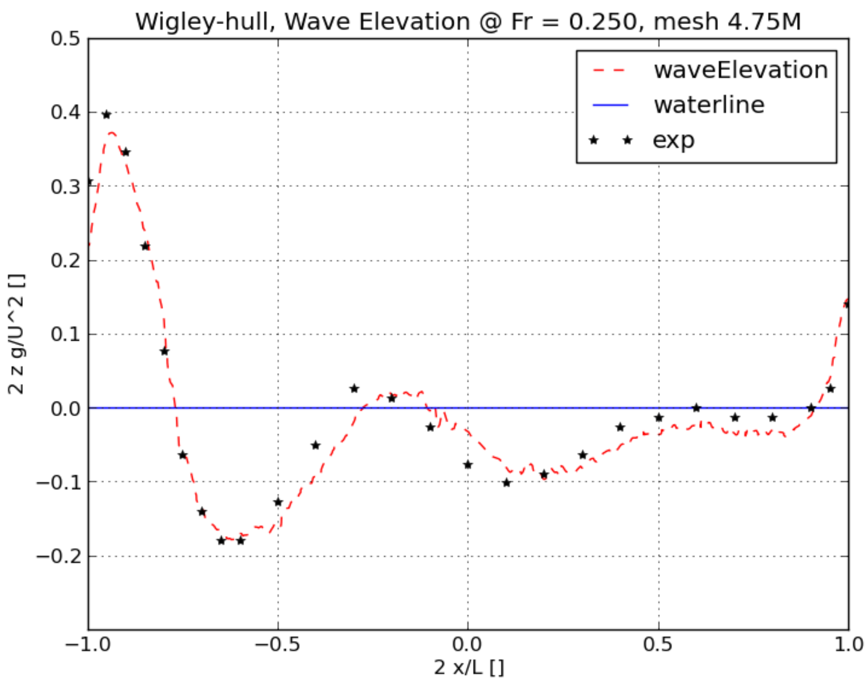
Wigley-hull

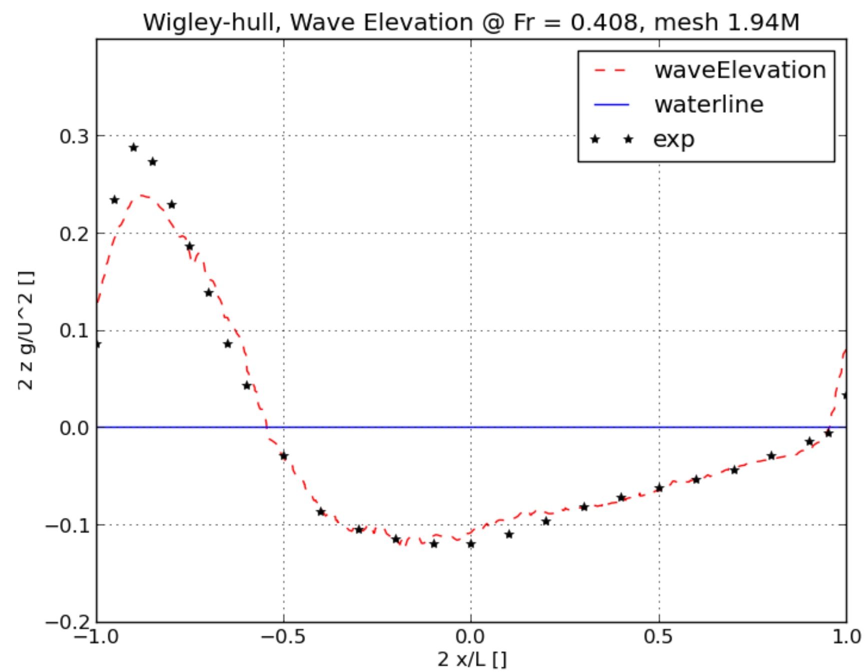
- **Description**: widely used in marine engineering for validation of measures
- Standard reference





Accuracy: CFD vs experimental





- Wigley-hull wave elevation @ different Froude number

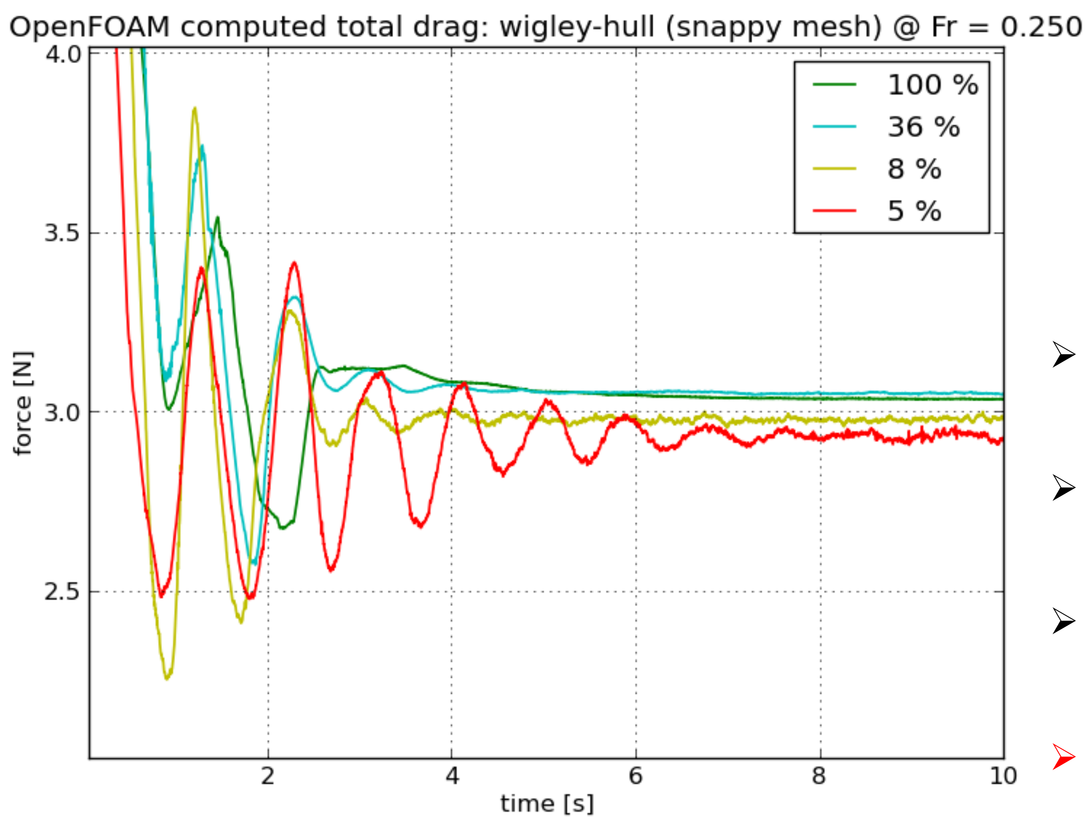
Accuracy: mesh sensitivity

- 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

CINECA SCAI SuperComputing Applications and Innovation

CINECA COURSES High Performance Computing 2014

Accuracy: mesh sensitivity



Mesh-size	F value [N]	F diff%
100%	3,03	Used-as-GS
36%	3,05	0,6%
8%	2,98	1,6%
5%	2,93	3,3%

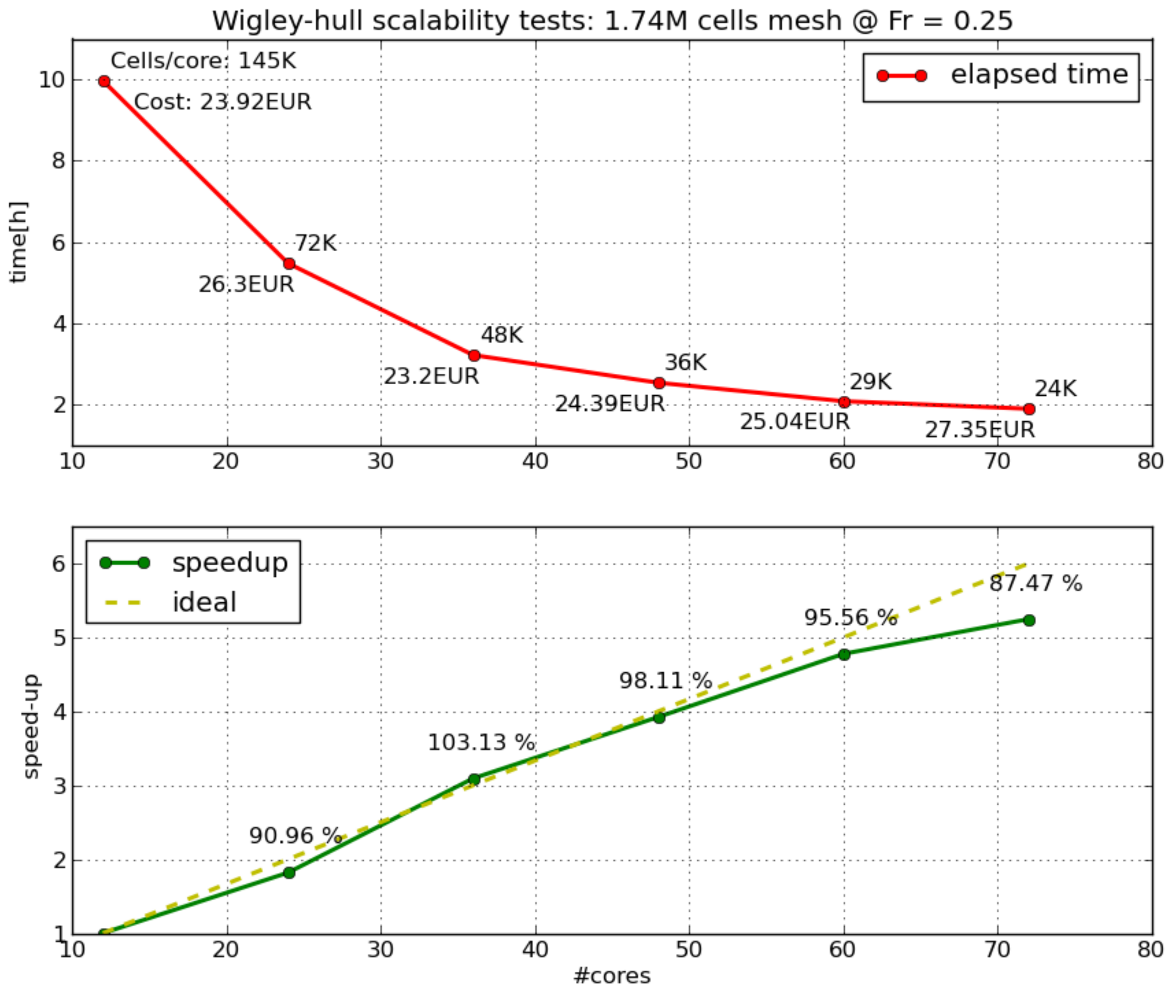
- Reducing mesh size in not critical for the absolute convergence but just delays it.
- 5% size mesh respect to GS produces a 3% discrepancy in the total computed drag
- 5% size mesh respect to GS requires just 2h @ 12 cpu to reach convergence
- **User choice: different accuracy, different mesh size,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

PRACE CINECA

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:

$$S = \frac{et_s}{et_p}$$

- Efficiency:

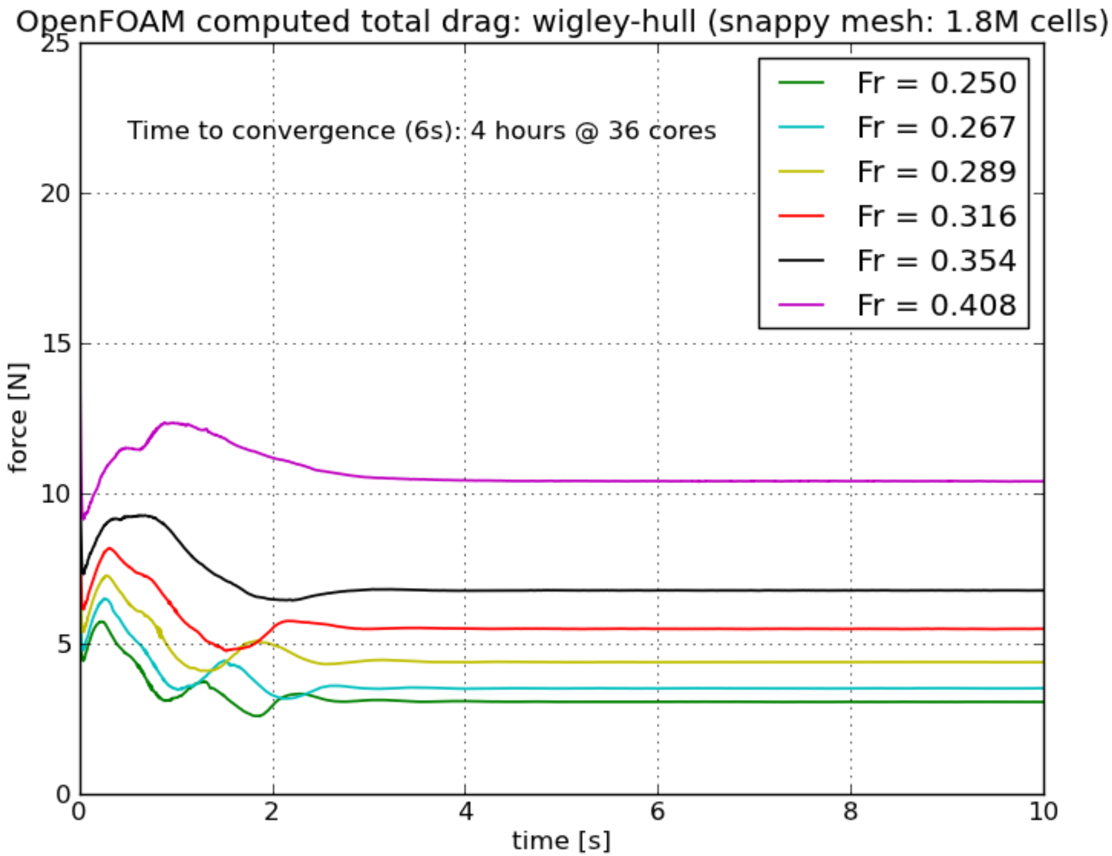
$$E = \frac{S}{cores}$$

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

Reliability

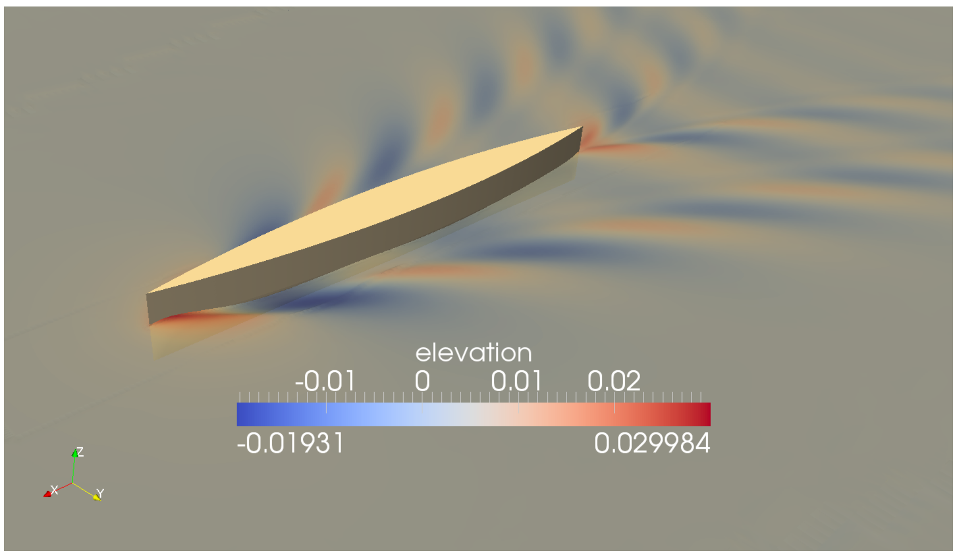
- Different computed forces due to different Froude number e.g. inlet velocity
- Fixed mesh size: 1.7 mln cells
- Fixed number of cores: 36 @ PLX, CINECA cluster
- Froude number range: 0.250 0.267 0.289 0.316 0.354 0.408
- Key value indices: total forces, viscous forces, pressure forces, wave height

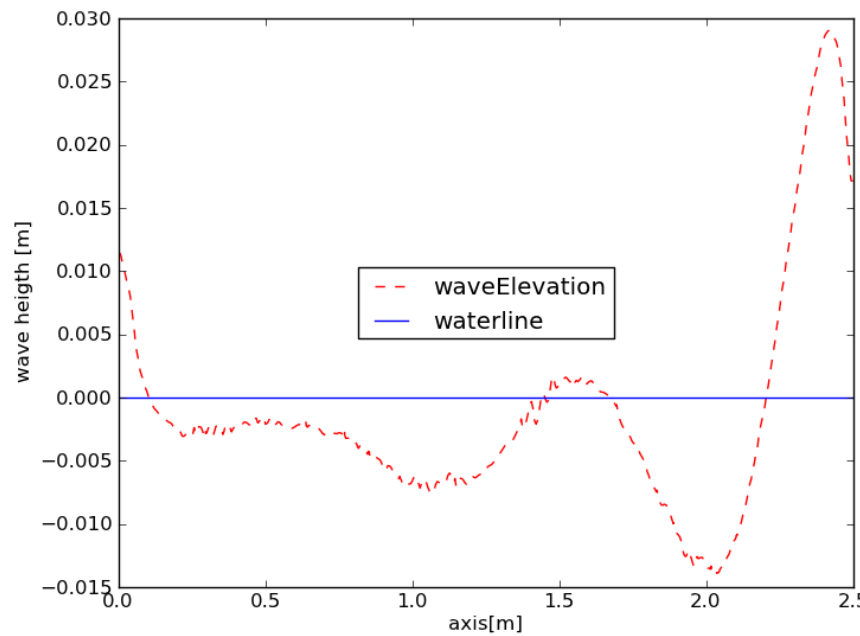
Reliability: results



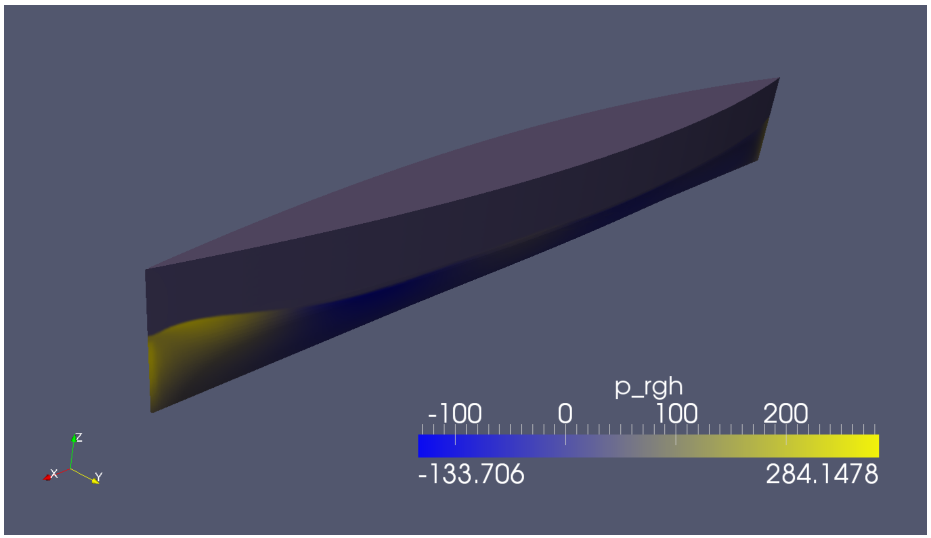
- Stable solution reached within 4s (4k iteration for LTS solver)
- Fixed cut-off at 6s.
- Stable means are computed in the selected range 4s – 6s, so 4h @ 36 cpu exploiting best scalability

Wave elevation (α = 0.5)

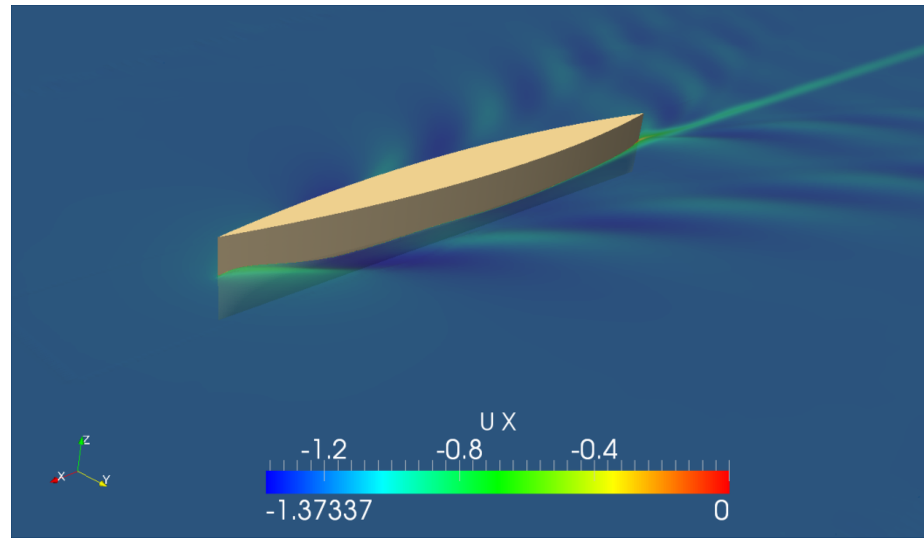




Pressure & axial velocity

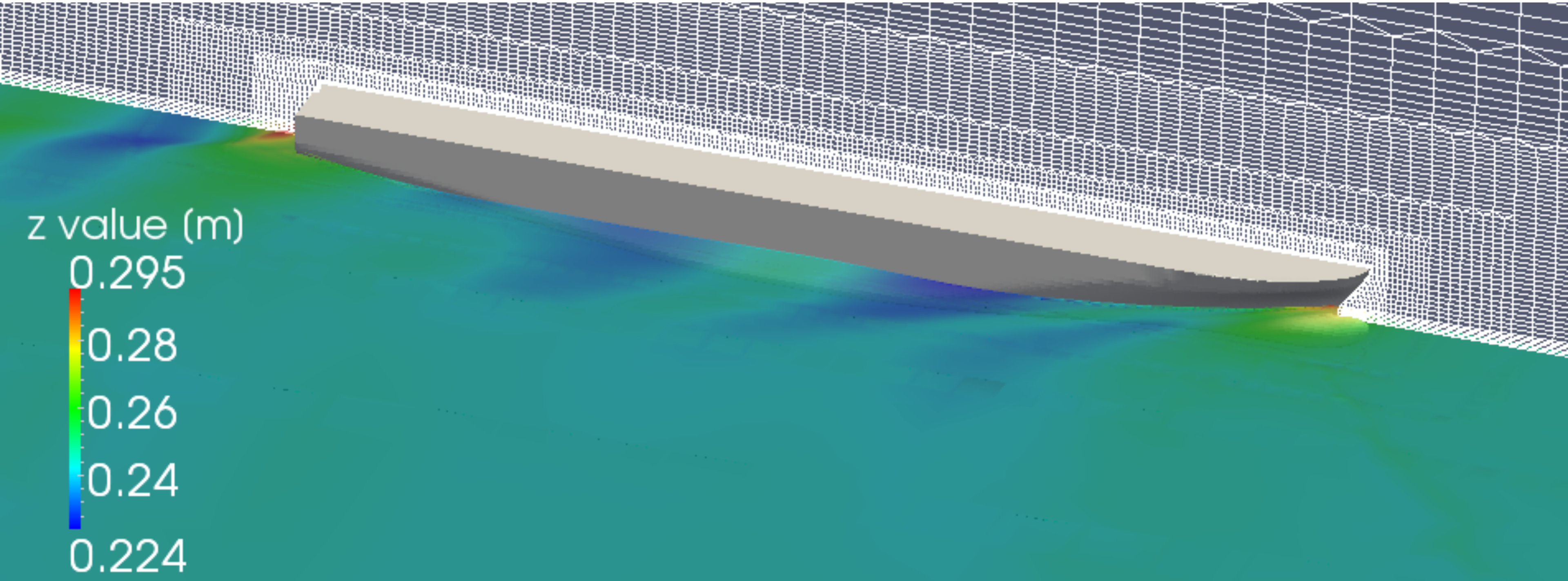


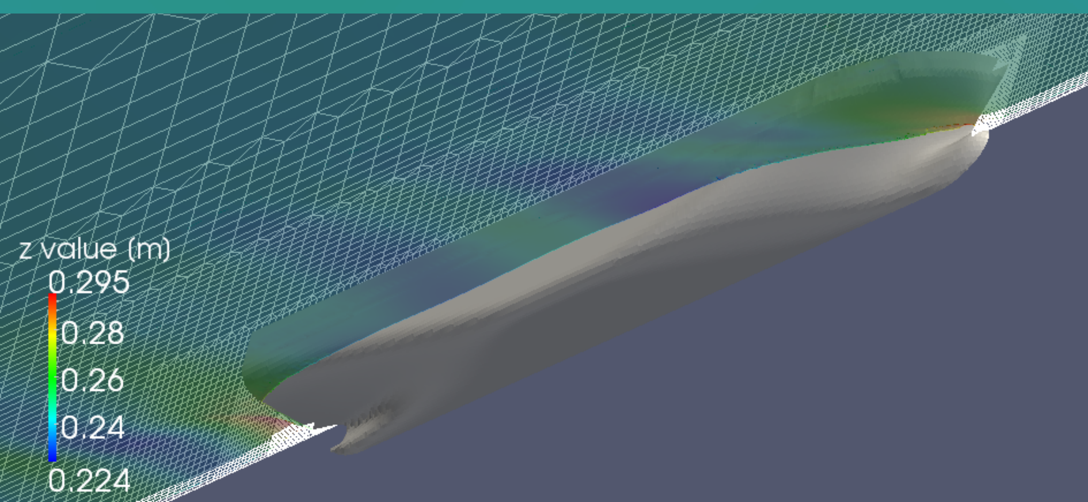
Pressure over boat hull



Axial (x) velocity over wave 3D surface

Hands-on: CFD result





<u>DTC Hull tutorial:</u>

$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