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 \(\gamma^+\)  →  too small time-step

- **Commercial** softwares can manage 100-1000 times larger \(\Delta t\)

- **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.
① High Reynolds simulation: **RANS** model employed

② Turbulence model: \( k-\omega \) 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
   - 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
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

Free-surface:
iso-surface mass fraction $\alpha = 0.5$
CFD results: agreement with theory

- Wave height visualized on 3D air/water interface surface (mass fraction 0.5)
- Cone of influence identified: physical angle detected, in agreement with theory
DTMB-5414: half hull simulations

- save 43.5% computational time
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

1. **CAD input**
   - STL geometry file
   - STL refinement surfaces

2. **Setup case**
   - Domain definition
   - Sink, trim, leeway ...
   - Inlet conditions

3. **Meshing**
   - snappyHexMesh

4. **CFD solving**
   - Initialization
   - LTSInterFoam

5. **Post-processing**
   - VTK fields extraction
   - Forces monitoring (python scripts)

OpenFOAM AUTOMATIC WORKFLOW

INPUT

OUTPUT
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 @ Fr = 0.250, mesh 4.75M

Wigley-hull, Wave Elevation @ Fr = 0.408, mesh 1.94M

- waveElevation
- waterline
- exp

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.
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

- **Wigley-hull scalability tests:** 1.74M cells mesh @ Fr = 0.25

  - **Cells/core:** 145K
  - **Cost:** 23.92EUR

  - **Elapsed time**

  - **Speed-up:**
    \[ S = \frac{e_{ts}}{e_{tp}} \]
  
  - **Efficiency:**
    \[ E = \frac{S}{\text{cores}} \]

- 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 ($\alpha = 0.5$)
Pressure & axial velocity

Pressure over boat hull

Axial (x) velocity over wave 3D surface
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