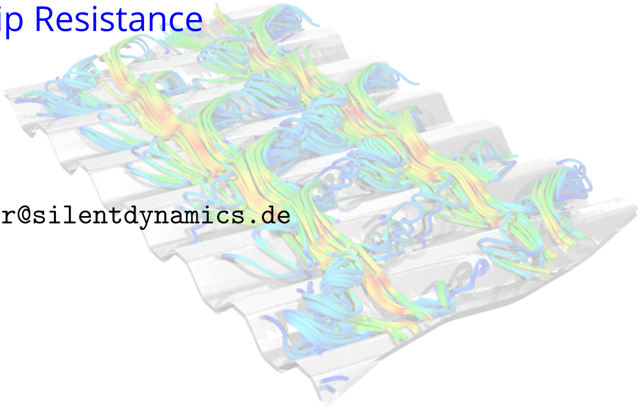
# Tutorial Ship Resistance Analysis



Hannes Kröger
hannes.kroeger@silentdynamics.de

GOFUN2017

©Copyright silentdynamics GmbH

# Contents



©Copyright silentdynamics GmbH

At small Froude Numbers and low wave making resistance:
Simulation assuming a flat water surface

- enable single phase simulation
- stationary solver
- faster than two-phase simulation

$\Rightarrow$ simpleFoam solver

© Copyright silentdynamics GmbH

Approximate consideration of free surface in single phase simulation possible:
potentialFreeSurfaceFoam

- through special boundary condition “waveSurfacePressure”

©Copyright silentdynamics GmbH

If wave making resistance cannot be neglected:
Resolution of water surface

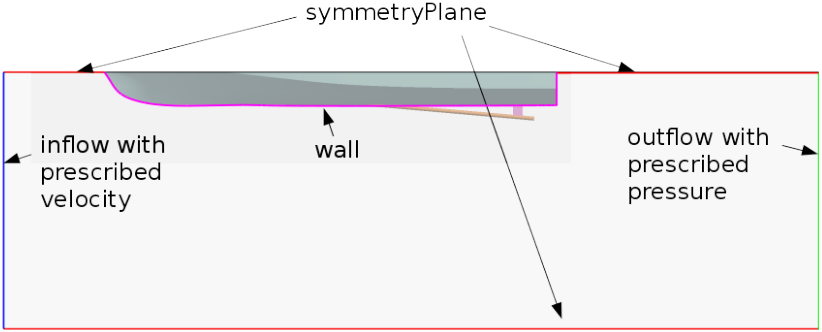
- two-phase simulation (water + air)
- only time resolved solvers in OpenFOAM
  $\Rightarrow$ interFoam
- for steady problems: quasi-steady approach with local timestepping
  $\Rightarrow$ LTSInterFoam / localEuler

©Copyright silentdynamics GmbH

If trim and sinkage is freely changing and needs to be determined:
Resolution of water surface + mesh deformation

- two-phase simulation (water + air)
- only time resolved solvers in OpenFOAM
  $\Rightarrow$ interDyMFoam

Domain for single-phase simulation




©Copyright silentdynamics GmbH

1. create template mesh using blockMesh
2. setup snappyHexMesh
3. run snappyHexMesh
4. setup solver, BCs
5. run solver
6. postprocessing

©Copyright silentdynamics GmbH

Example post processing: extract wake field for propeller design

- Method 1
  1. using tool sample
  2. star-shaped pattern of multiple line sampleSets
- Method 2
  - using ParaView
  - cutting plane

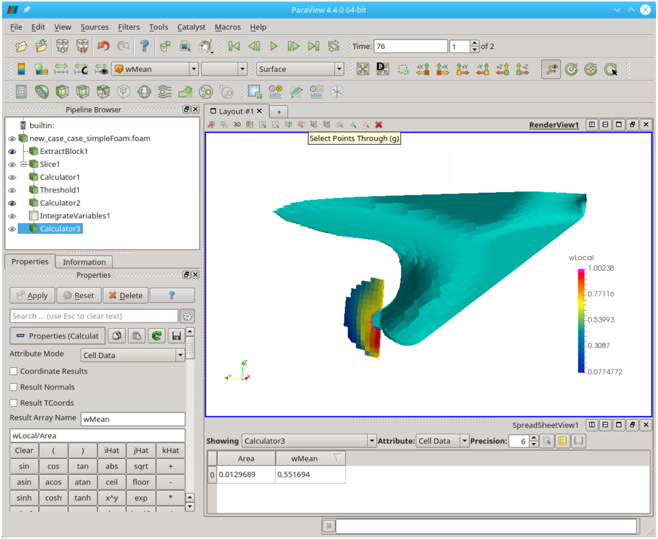
©Copyright silentdynamics GmbH

Propeller location: $\vec{p} = (0.17 \;\; 0 \;\; -0.21)^T$, $R = 0.1$

1. Slice Filter: Origin $\vec{p}$, Normal $\vec{e}_x$
2. Calculator Filter: Result "R", Expression "`sqrt((coordsX-0.17)^2+coordsY^2+(coordsZ+0.21)^2)`"
3. Threshold Filter: "R" between 0 and 0.1
4. Calculator Filter: Mode "Cell Data", Result "wLocal", Expression "`(2.196+U_X)/2.196`"
5. Integrate Variables Filter: Field Association "Cell Data"
6. Calculator Filter: Mode "Cell Data", Result "wMean", Expression "`wLocal/Area`"

©Copyright silentdynamics GmbH

# Wake Field




©Copyright silentdynamics GmbH

Volume-of-Fluid method (VOF)

- volume fraction $\alpha$

$$\rho = \alpha\rho_w + (1-\alpha)\rho_L$$

$$\nu = \alpha\nu_w + (1-\alpha)\nu_L$$

- transport of volume fraction

$$\frac{\partial\alpha}{\partial t} + \nabla\cdot(\alpha\vec{u}) = 0$$

©Copyright silentdynamics GmbH

- volume fraction is not continuous
- problem: numerical simulation, such that
  - interface stays sharp and is not excessively smeared
  - solution remains stable
  - spurious velocities at boundary are avoided
  - mass conservation is fulfilled

©Copyright silentdynamics GmbH

Solution of VOF equation in OpenFOAM

- sharpening of interface by additional compressive term

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{u}) + \nabla \cdot (\alpha \vec{u}_c) = 0$$

- time integration my MULES

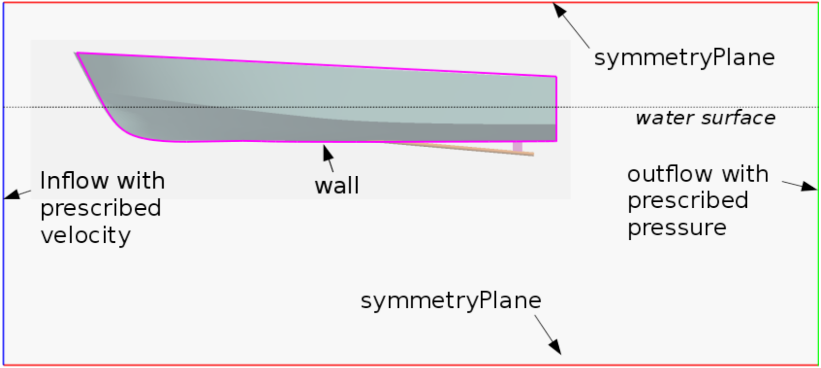
©Copyright silentdynamics GmbH

For steady state problems:
local time stepping approach

- spatially varying time step size
- as large as possible according to stability criterion
- unphysical, but convergent at infinite time
- requires spatial smoothing of time step size

©Copyright silentdynamics GmbH

Domain for two-phase simulation




©Copyright silentdynamics GmbH

- walls:
  `type zeroGradient;`
- inlet: specification if incoming water level necessary
  `type fixedValue;`
- outlet: extrapolation from interior
  `type zeroGradient;`

©Copyright silentdynamics GmbH

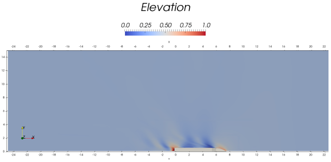
interFoam calculates with modified pressure $p_{rgh} = p - \rho g h$ (with $h = \vec{x} \cdot \vec{g} / |\vec{g}| - h_{ref}$)

- walls:
  ```
  type fixedFluxPressure;
  ```
- inlet with prescribed velocity
  ```
  type fixedFluxPressure;
  ```
- outlet (water surface needs to be aligned with h=0)
  ```
  type fixedValue;
  value uniform 0;
  ```

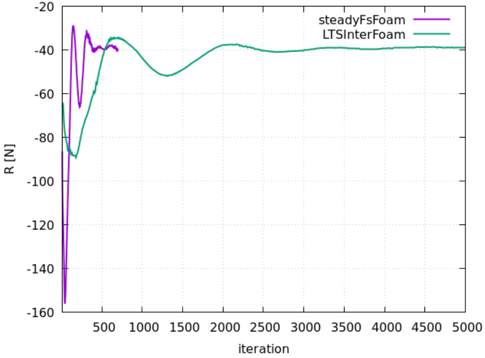
©Copyright silentdynamics GmbH

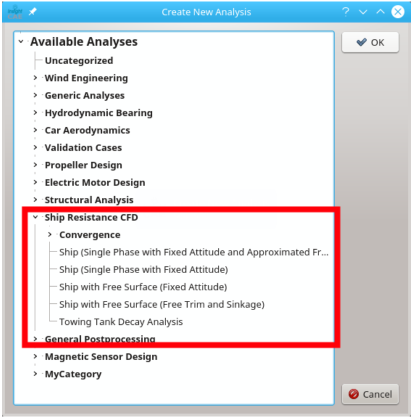
- $\alpha$ fields needs to be initialised
  1 in water
  0 in air
- tool `setFields`

# Current development: steady-state VOF solver



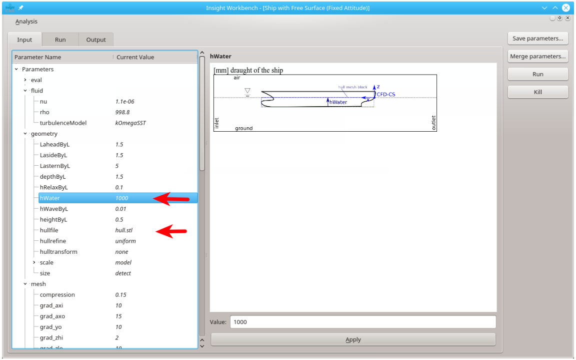

- considerable faster convergence
- same accuracy





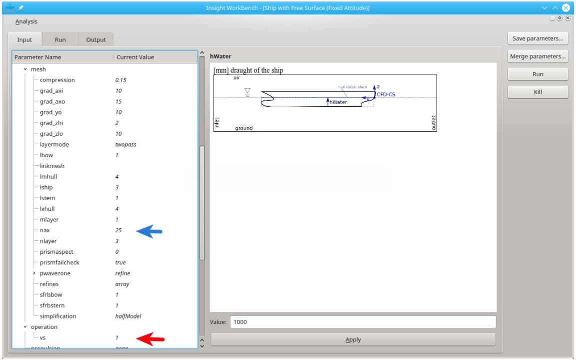

- different automated workflows available
- single-phase
- two-phase
- dynamic mesh


©Copyright silentdynamics GmbH




- most parameters need not to be touched
- only some required (red)

©Copyright silentdynamics GmbH




- most parameters need not to be touched
- only some required (red)


©Copyright silentdynamics GmbH

Thank you for your attention!

Dr.-Ing. Hannes Kröger
Email: hannes.kroeger@silentdynamics.de
Tel.: +49 381 36 77 98 53

http://silentdynamics.de
http://sourceforge.net/projects/insightcae

©Copyright silentdynamics GmbH