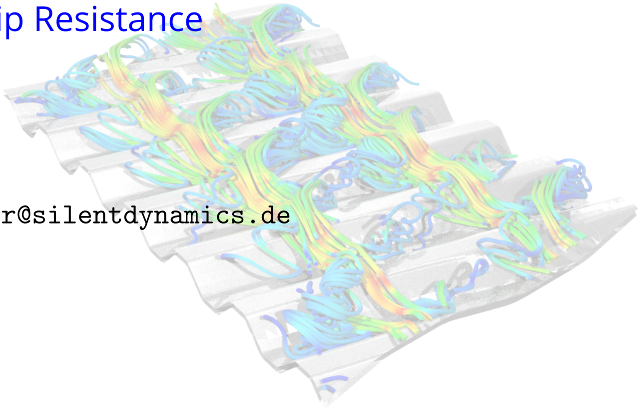
silentdynamics

Tutorial Ship Resistance Analysis



Hannes Kröger
hannes.kroeger@silentdynamics.de

GOFUN2018

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

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

- through special boundary condition “waveSurfacePressure”

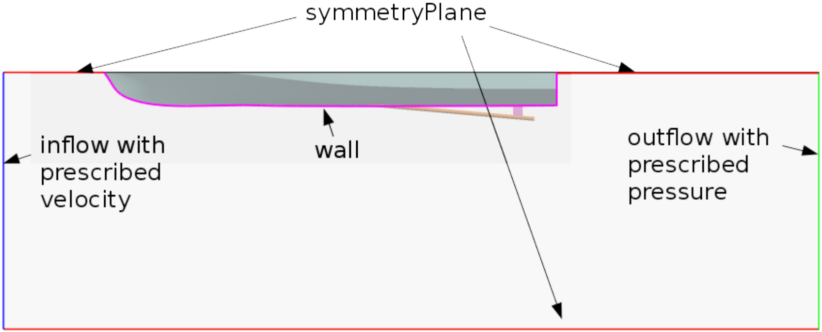
If wave making resistance cannot be neglected:
Resolution of water surface

- 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

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



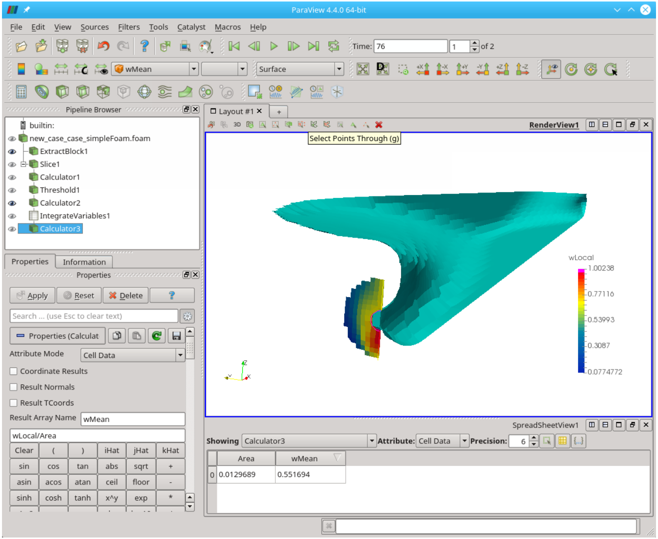
1. create template mesh using blockMesh
2. setup snappyHexMesh
3. run snappyHexMesh
4. setup solver, BCs
5. run solver
6. postprocessing

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

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`"


ParaView 4.4.0 64-bit
File Edit View Sources Filters Tools Catalyst Macros Help
Time: 76
1
of 2
wMean
Surface
Pipeline Browser
builtin:
new_case_case_simpleFoam.foam
ExtractBlock1
Slice1
Calculator1
Threshold1
Calculator2
IntegrateVariables1
Calculator3
Properties
Information
Apply
Reset
Delete
Search ... (use Esc to clear text)
Properties (Calculat
Attribute Mode
Cell Data
Coordinate Results
Result Normals
Result TCoords
Result Array Name
wMean
wLocal/Area
Clear ( ) iHat jHat kHat
sin cos tan abs sqrt +
asin acos atan ceil floor -
sinh cosh tanh x^y exp *
Layout #1
RenderView1
Select Points Through (g)
wLocal
1.00238
0.77116
0.53993
0.3087
0.0774772
SpreadSheetView1
Showing Calculator3
Attribute: Cell Data
Precision: 6
Area wMean
0 0.0129689 0.551694

Volume-of-Fluid method (VOF)

- volume fraction α

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

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

- transport of volume fraction

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

- 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

Solution of VOF equation in OpenFOAM

- sharpening of interface by additional compressive term

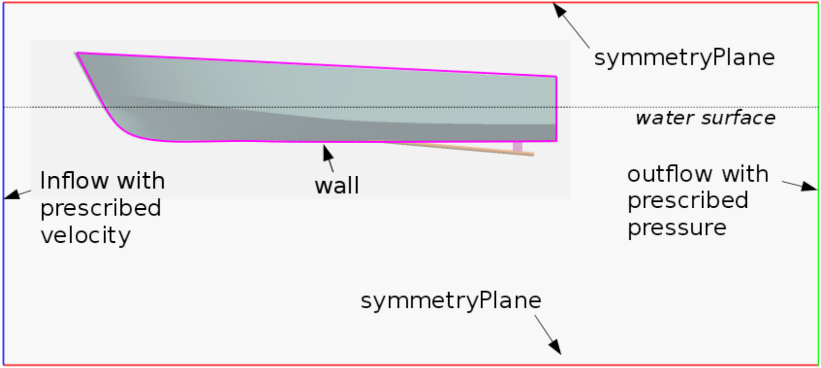
$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{u}) + \nabla \cdot (\alpha \vec{u}_c) = 0$$

- time integration my MULES

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

Domain for two-phase simulation



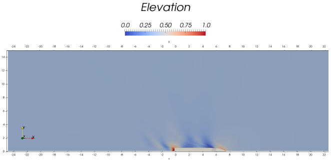
- walls:
  `type zeroGradient;`
- inlet: specification if incoming water level necessary
  `type fixedValue;`
- outlet: extrapolation from interior
  `type zeroGradient;`

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

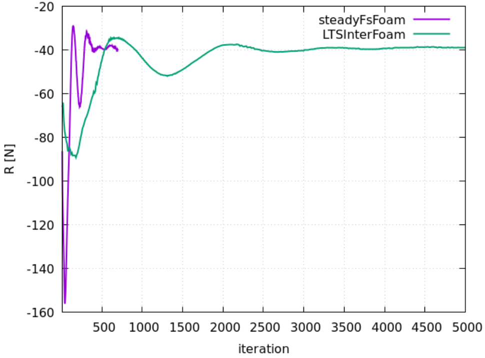
- walls:
  ```
  type fixedFluxPressure;
  ```
- inlet with prescribed velocity
  ```
  type fixedFluxPressure;
  ```
- outlet (water surface needs to be aligned with h=0)
  ```
  type fixedValue;
  value uniform 0;
  ```

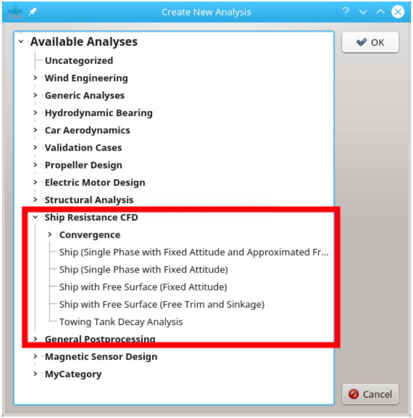
- α fields needs to be initialised
  1 in water
  0 in air
- tool `setFields`

Current development: steady-state VOF solver

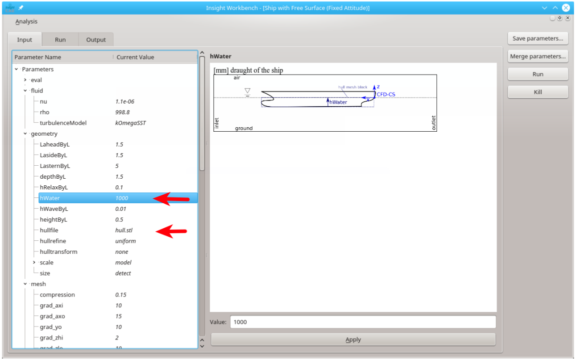


- considerable faster convergence
- same accuracy

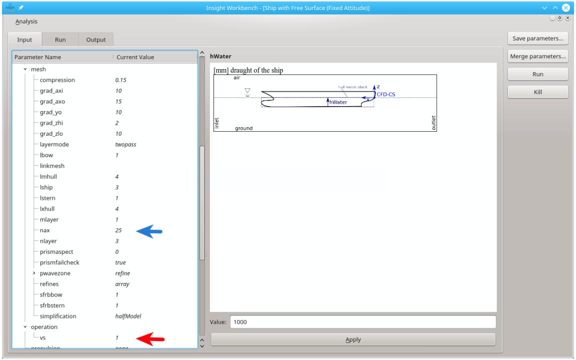




- different automated workflows available
- single-phase
- two-phase
- dynamic mesh



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



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

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