

성긴 격자의 결과를 초기 유동장으로 이용한 선박의 저항추정 시간단축

정광열, 오광호, 김병윤 (주) 넥스트폼

제8회 한국유체공학학술대회 2014년 8월 27~29일 롯데부여리조트

R&D Center. NEXTfoam CO., LTD.



- Commercial CFD programs are commonly used for prediction of ship resistance
 - $(10 \sim 100 \text{ cases are calculated for a hull design.})$
 - Large license fee
 - Huge calculation time
- Open source CFD codes are considered as an alternative
 - Free to add or modify numerical methods or functions
 - No license fee

Research purpose

- Check the applicability of OpenFOAM to ship resistance prediction
- Present a method to reduce the calculating time





- OpenFOAM
 - Open Field Operation and Manipulation
 - Open source CFD toolbox written in C++
 - Freely available and open source
 - Licensed under the GNU General Public License
 - Homepage: http://www.openfoam.org
- Solver
 - Over 80 solvers
 - Over 170 applications
- Library
 - Turbulence, Thermophysical, Chemistry
- Mesh
 - Mesh Generator, Converting
- Core Tech.
 - Numerical Method
 - Dynamic Mesh, Parallel Computing

Home Features Suppor	rt Training News OpenFOAM Foundation			
bout us Contact Jobs Legal		FOLLOW US ON Ewil		
Features	Features of OpenFOAM			
Overview	•			
standard Solvers		on) CFD Toolbox is a free, open source CFD software package		
Creating Solvers		produced by OpenCED Ltd. It has a large user base across most areas of engineering and science, from both commercial and academic organisations. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics. It includes tools for meshing, notably snappy//ext/kesh, a parallelised mesher for complex CAD geometries, and for pre- and post-processing. Atmost everything		
Aesh Generation	chemical reactions, turbulence and heat transfer, to solid of			
lesh Conversion		geometries, and for pre- and post-processing. Almost everything parallel as standard, enabling users to take full advantage of computer		
lesh Manipulation	hardware at their disposal.	, , , , , , , , , , , , , , , , , , , ,		
urbulence Models	By being open, OpenFOAM offers users complete freedor	m to customise and extend its existing functionality, either by themselve		
ransport/Rheology Models	or through support from OpenCFD. It follows a highly modular code design in which collections of functionality (e.g. numerical methods, meshing, physical models,) are each compiled into their own shared library. Executable applications are then			
hermophysical Models		OpenFOAM includes over 80 solver applications that simulate specific		
Particle Tracking Reaction Kinetics	problems in engineering mechanics and over 170 utility applications that perform pre- and post-processing tasks, e.g. meshing data visualisation. etc.			
ParaView	uata visualisation, etc.			
Run-time Post-processing	Column Conschilition	Liberto Europhiana liter		
hird-party Post-processing	Solver Capabilities	Library Functionality		
lumerical Method	 Incompressible flows Multiphase flows 	 Turbulence models Transport/rheology models 		
inear System Solvers DE Solvers	= Combustion	= Thermophysical models		
Parallel Computing	= Buoyancy-driven flows	Lagrangian particle tracking		
Dynamic Meshes	Conjugate heat transfer Compressible flows	Reaction kinetics / chemistry		
	 Particle methods (DEM, DSMC, MD) 			
	 Other (Solid dynamics, electromagnetics) 	Post-processing		
		 ParaView and VTK post-processing Run-time post-processing 		
	Code Customisation	 Third-party post-processing 		
	Creating solvers in OpenFOAM			
	Extending library functionality	Core Technology		
	Meshing Tools	= Numerical method		
	Mesh generation in OpenFOAM	 Linear system solvers ODE system solvers 		
	 Mesn generation in OpenFOAM Converting meshes into OpenFOAM format 	 ODE system solvers Parallel computing 		
	Tools to manipulate meshes	 Dynamic mesh 		



Numerical Methods

- LTSInterFoam (a standard solver of OpenFOAM)
 - Solver for steady, incompressible, turbulent, two-layer flows
- Solving algorithm
 - PIMPLE (hybrid method between PISO and SIMPLE)
- Time Integration
 - Backward Euler
 - Local time stepping
- Spatial Discretization
 - Second order central difference: linear
 - Second order upwind: linearUpwind
- Matrix Solver
 - Iterative Method (e.g. Gauss-Seidel): smoothSolver
 - Geometric and algebraic multigrid: GAMG
- Turbulence Models
 - Two equation models: $k-\omega$ SST
- Free surface model
 - VOF

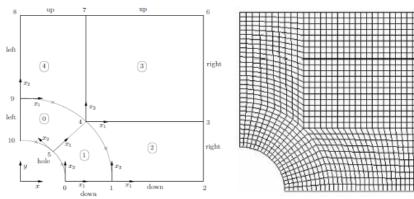


- KCS (KRISO Container Ship)
 - Scale factor : 31.6 (model LBP : 7.2786 m)
 - Speed : 2.1964 m/s (design speed)
 - $Rn : 1.4 \times 10^7$
 - -Fn: 0.26
 - Draft : 0.3418 m (design draft)

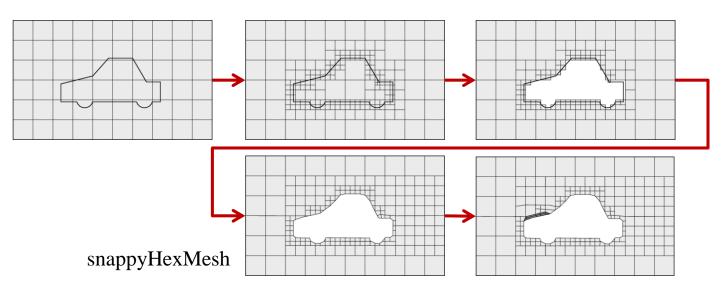


Grid generation tool

- OpenFOAM utility: blockMesh, snappyHexMesh
 - blockMesh
 - Structured block mesh generation utility
 - Usually for background mesh of snappyHexMesh
 - snappyHexMesh
 - Automatic Mesh generation utility
 - Refine base mesh
 - Remove unused cells
 - Snap mesh to surface
 - Add layers



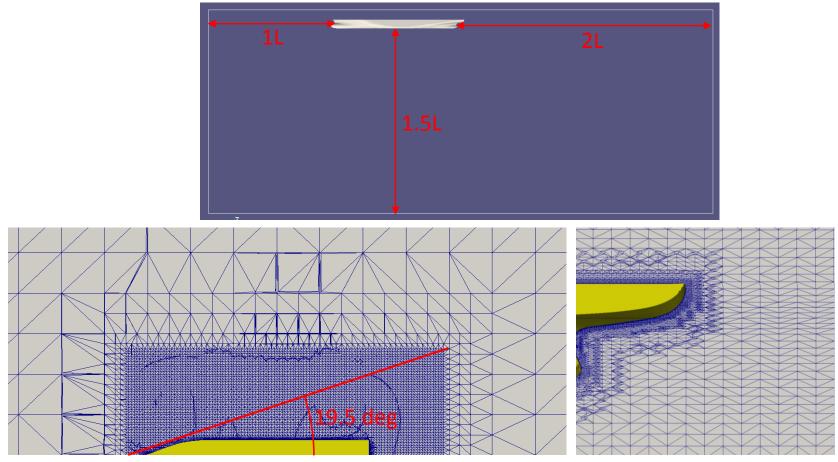
Block mesh



R&D Center. NEXTfoam CO., LTD.



Grid generation for KCS



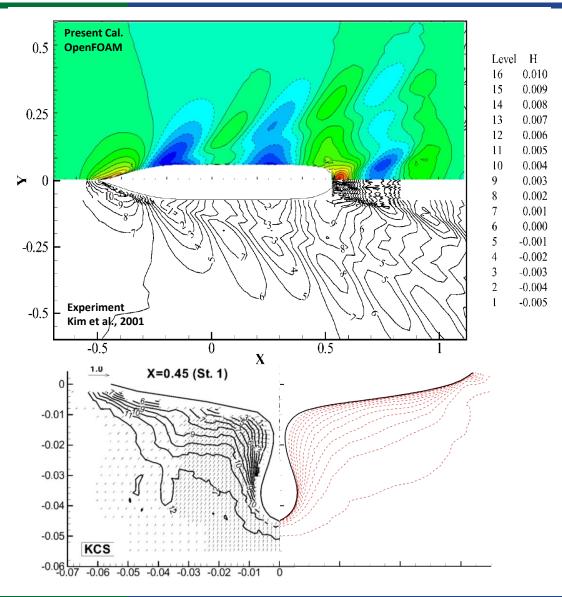
For ship making waves

For boundary layer

• Number of grids : 1030 k



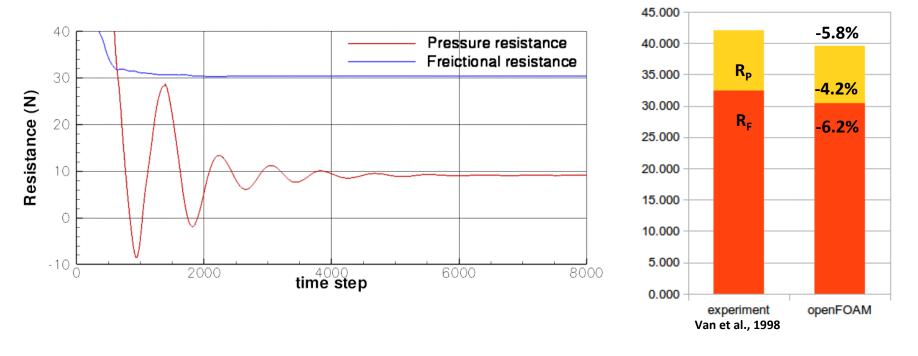
Calculation results



R&D Center. NEXTfoam CO., LTD.



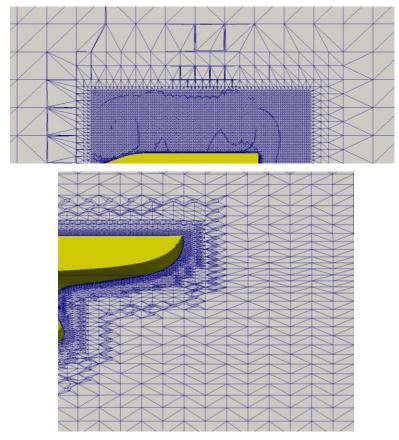
- Grid generation and calculating time : 4 hour 10 min (with16 cores)
 - Calculation time depends on wave generation time

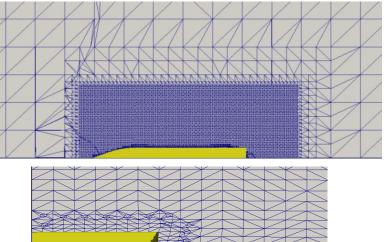


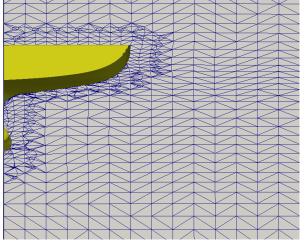


Coarse grid generation for waves

- Almost of grids for boundary layer are removed $(1030 \text{ k} \rightarrow 201 \text{ k})$
 - The same calculation domain with original grid system
 - The grid resolution for waves are not changed







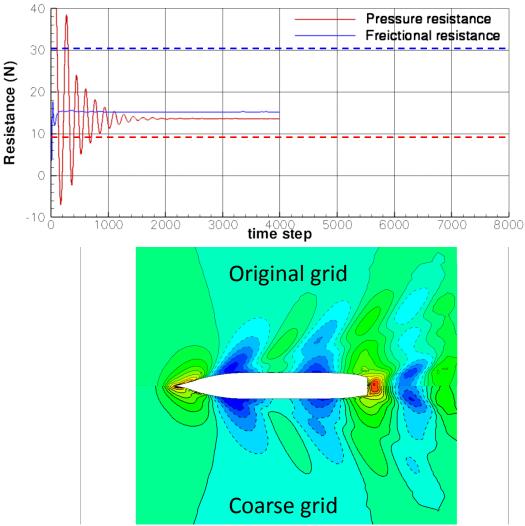
Coarse grid system

Original grid system

R&D Center. NEXTfoam CO., LTD.



• Grid generation and calculating time : 18 min (with16 cores)

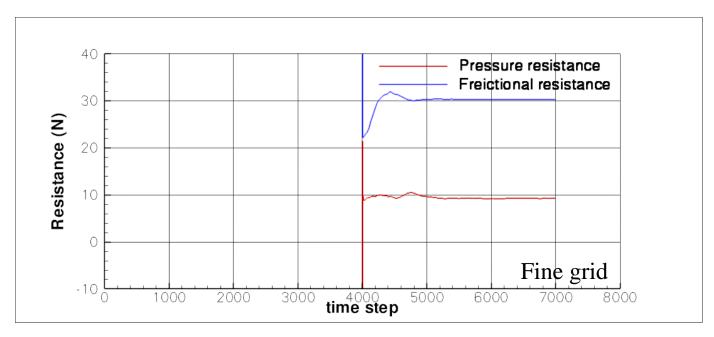


R&D Center. NEXTfoam CO., LTD.



Initialization with coarse grid result

- Mapping VOF function, velocity and pressure to original grid system
 - Initialization of turbulence kinematic energy, eddy viscosity and specific dissipation
- Time for mapping and calculation : 1 hour 40 min





- Open source CFD code (OpenFOAM) is applicable to prediction of ship resistance.
 - More verifications are required for practical use
- By mapping the results of coarse grids to fine grids, the calculation time is reduced
 - about 4 hours \rightarrow about 2 hours
 - Grid refinement test required for more accurate assessment.
 - The effects of boundary layer on waves is not significant.
- Further research will be conducted to improve the accuracy of the solver.
 - Wall function, turbulence model

Thank you for your attention

R&D Center. NEXTfoam CO., LTD.