



성긴 격자의 결과를 초기 유동장으로 이용한 선박의 저항추정 시간단축

정광열, 오광호, 김병윤
(주) 넥스트폼

제8회 한국유체공학학술대회
2014년 8월 27~29일
롯데부여리조트



Research background

- Commercial CFD programs are commonly used for prediction of ship resistance (10 ~ 100 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

- 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

The screenshot shows the OpenFOAM website homepage. The header includes the OpenFOAM logo and the tagline "The open source CFD toolbox". The navigation menu contains links for Home, Features, Support, Training, News, and OpenFOAM Foundation. A sidebar on the left lists various features such as Overview, Standard Solvers, Mesh Generation, and Turbulence Models. The main content area is titled "Features of OpenFOAM" and contains several sections: "Solver Capabilities" (listing incompressible, multiphase, combustion, buoyancy-driven, conjugate heat transfer, compressible, and particle methods), "Library Functionality" (listing turbulence, transport/rheology, thermophysical, Lagrangian particle tracking, and reaction kinetics), "Code Customisation" (listing creating solvers and extending library functionality), "Meshing Tools" (listing mesh generation, converting meshes, and manipulation tools), "Post-processing" (listing ParaView and VTK, run-time, and third-party post-processing), and "Core Technology" (listing numerical method, linear system solvers, ODE system solvers, parallel computing, and 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- ω SST
- Free surface model
 - VOF



Object ship

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



Grid generation tool

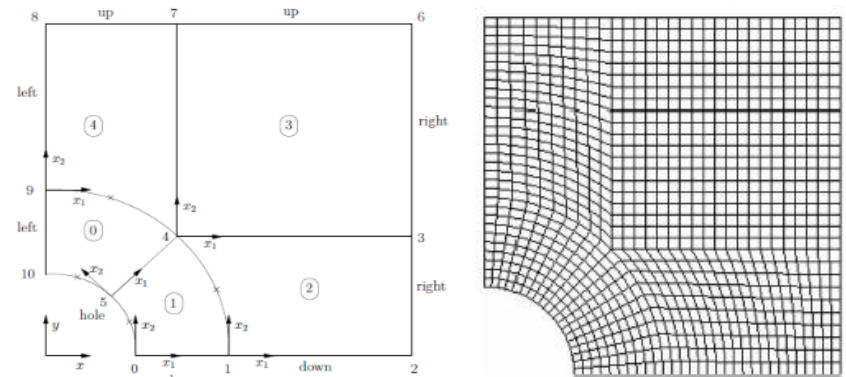
- OpenFOAM utility: blockMesh, snappyHexMesh

- blockMesh

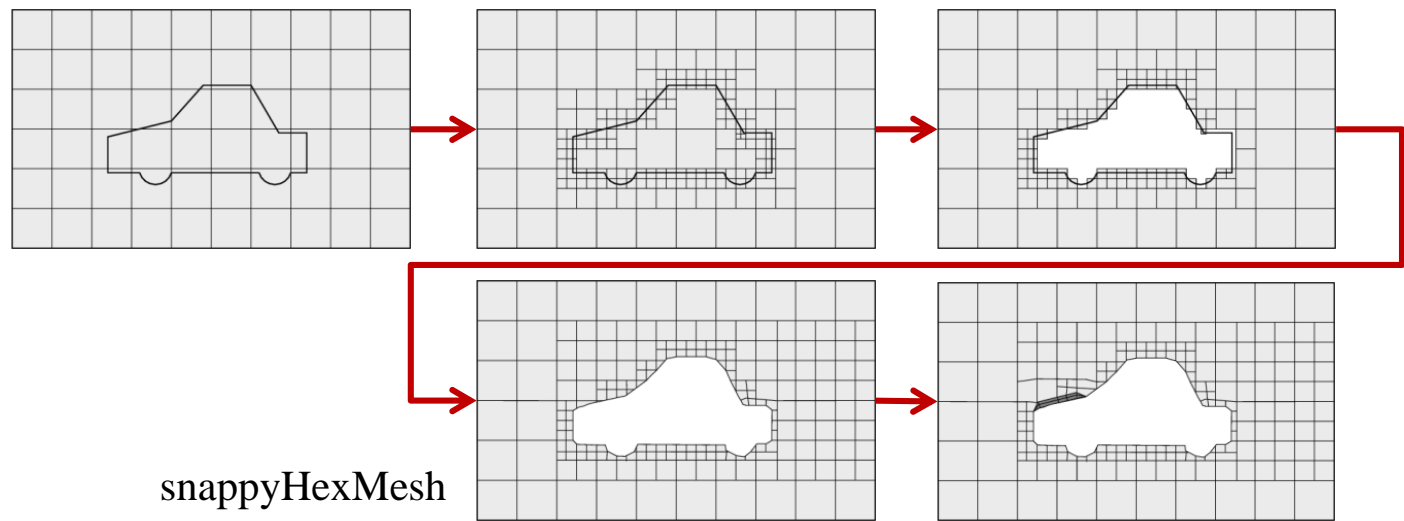
- Structured block mesh generation utility
- Usually for background mesh of snappyHexMesh

- snappyHexMesh

- Automatic Mesh generation utility
- Refine base mesh
- Remove unused cells
- Snap mesh to surface
- Add layers

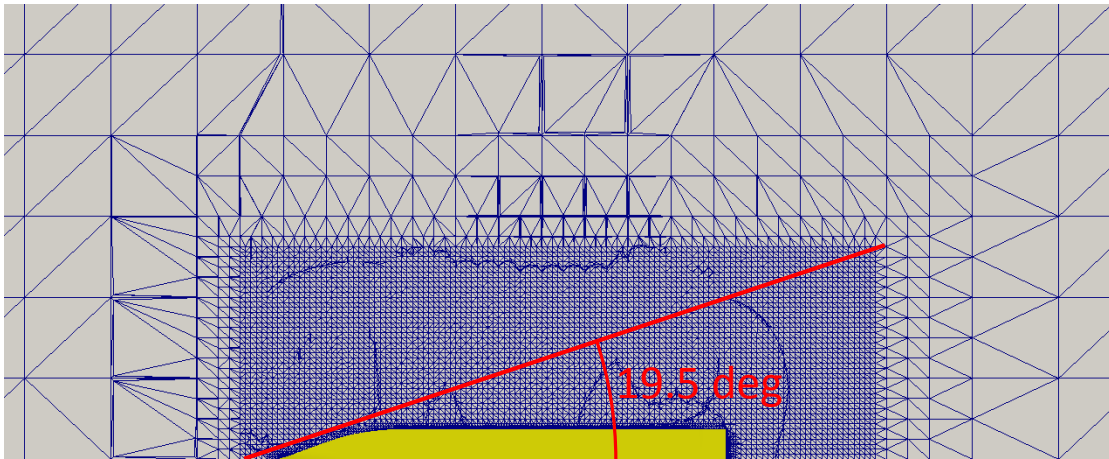
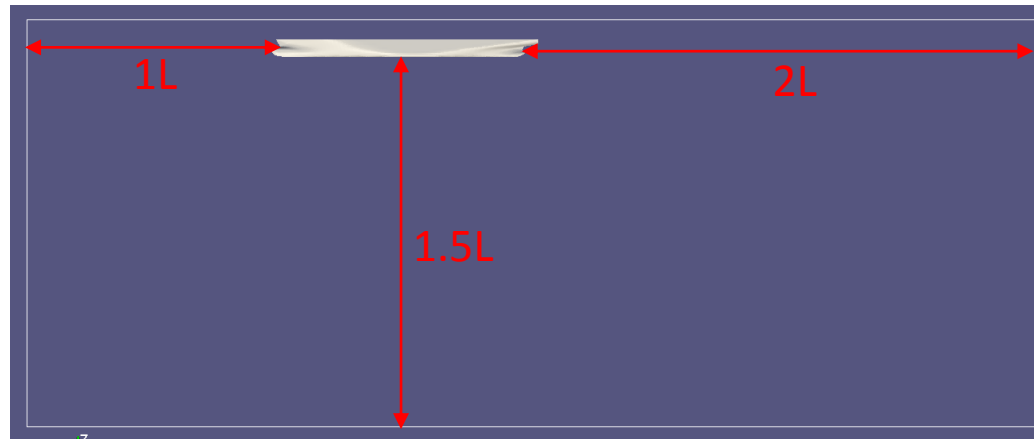


Block mesh

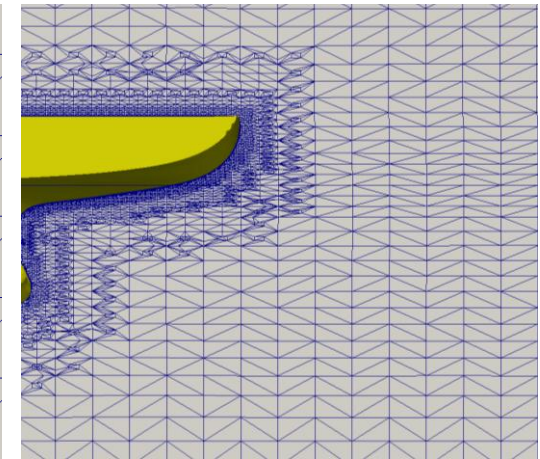


snappyHexMesh

Grid generation for KCS



For ship making waves

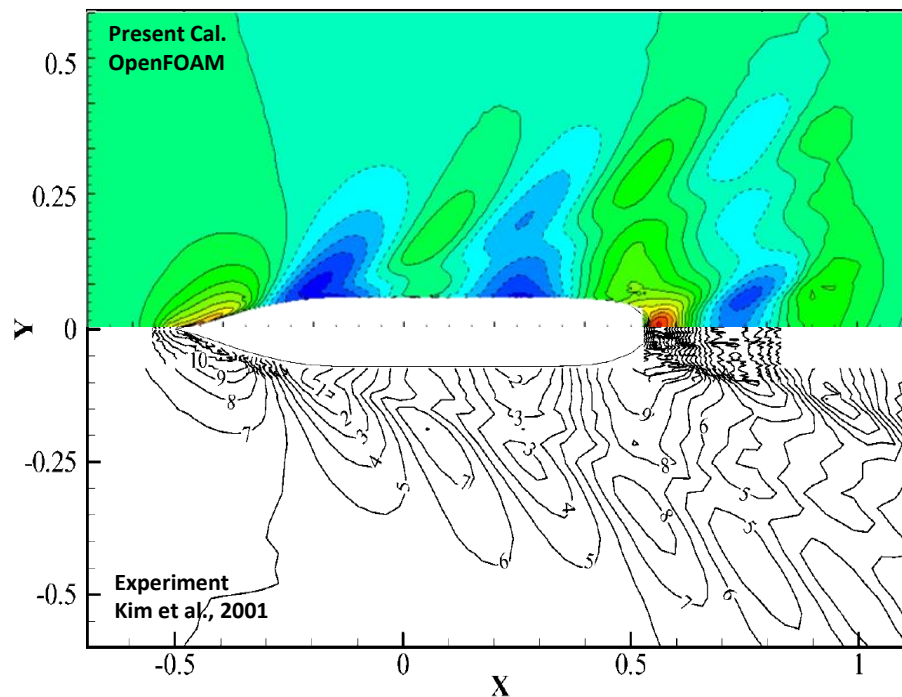


For boundary layer

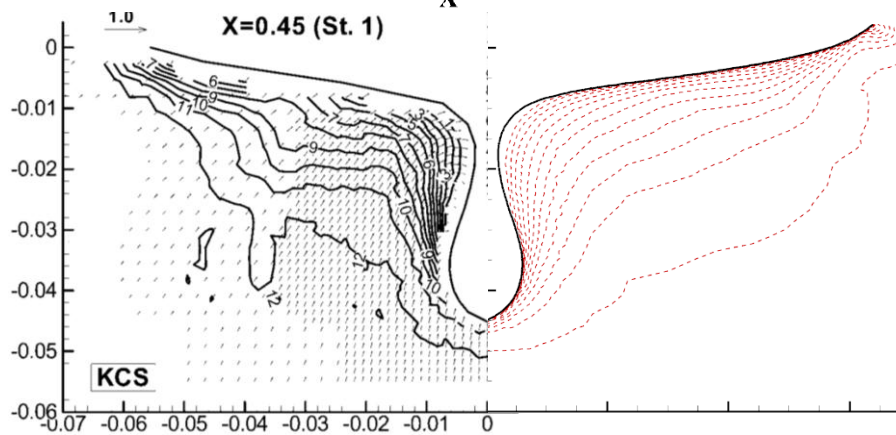
- Number of grids : 1030 k



Calculation results



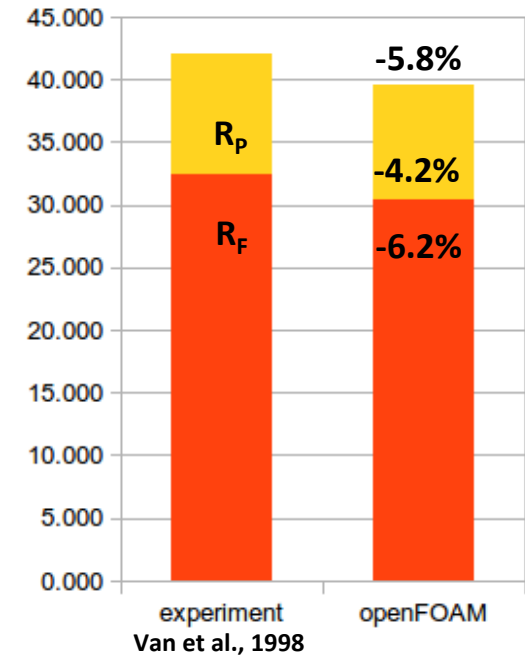
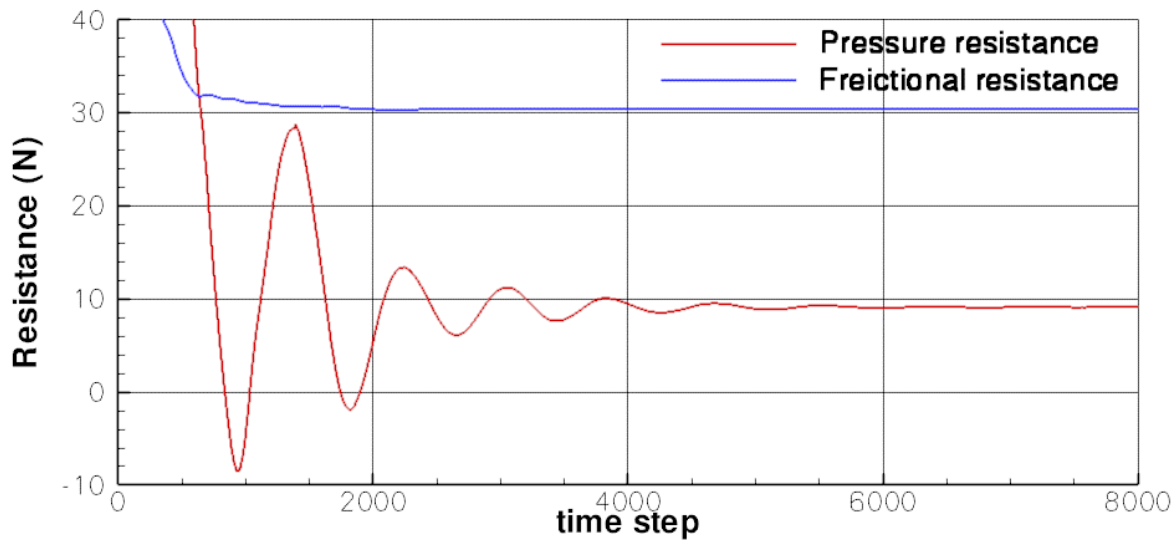
Level	H
16	0.010
15	0.009
14	0.008
13	0.007
12	0.006
11	0.005
10	0.004
9	0.003
8	0.002
7	0.001
6	0.000
5	-0.001
4	-0.002
3	-0.003
2	-0.004
1	-0.005





Results of KCS

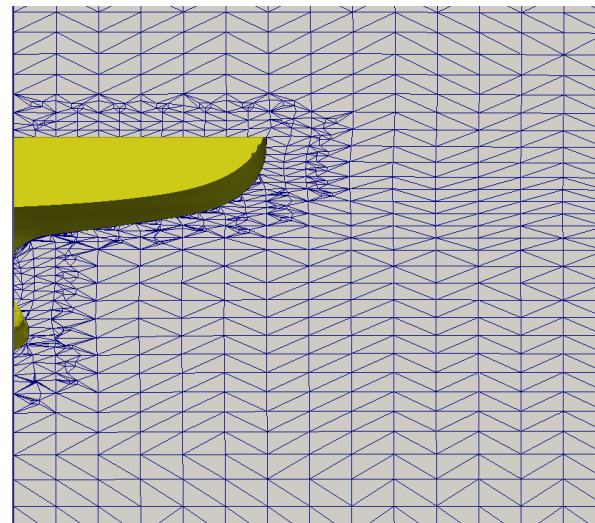
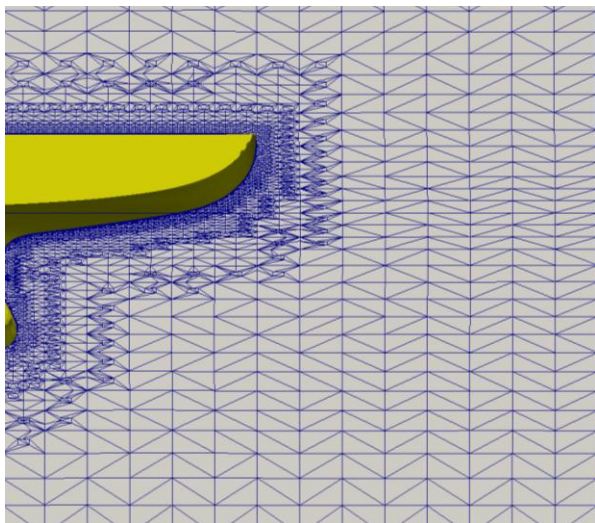
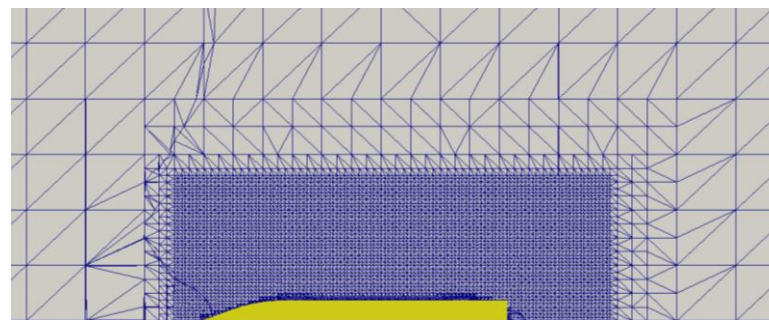
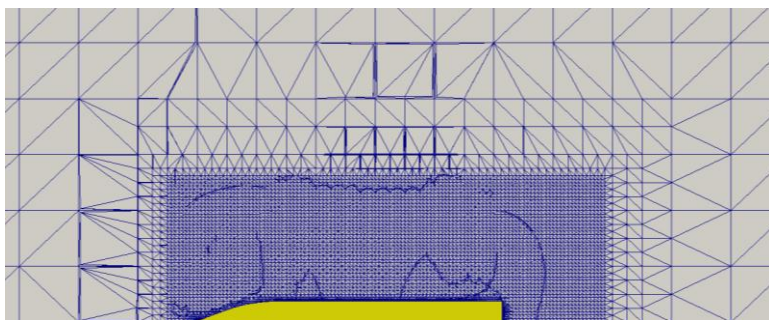
- Grid generation and calculating time : **4 hour 10 min** (with 16 cores)
 - Calculation time depends on wave generation time





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

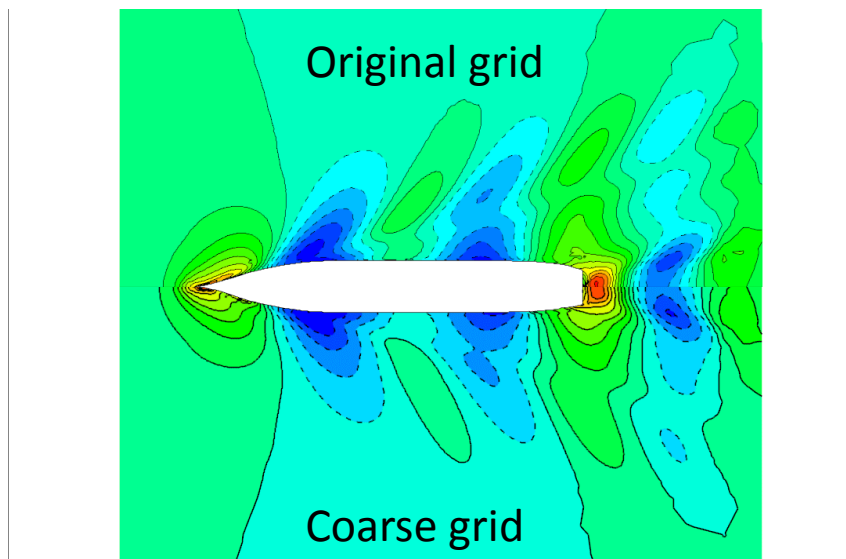
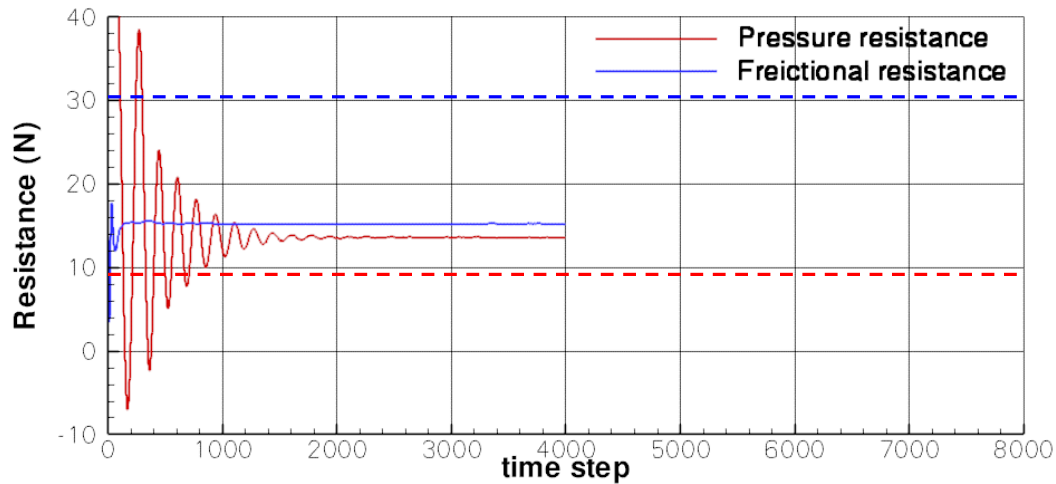


Original grid system

Coarse grid system

Results from coarse grid

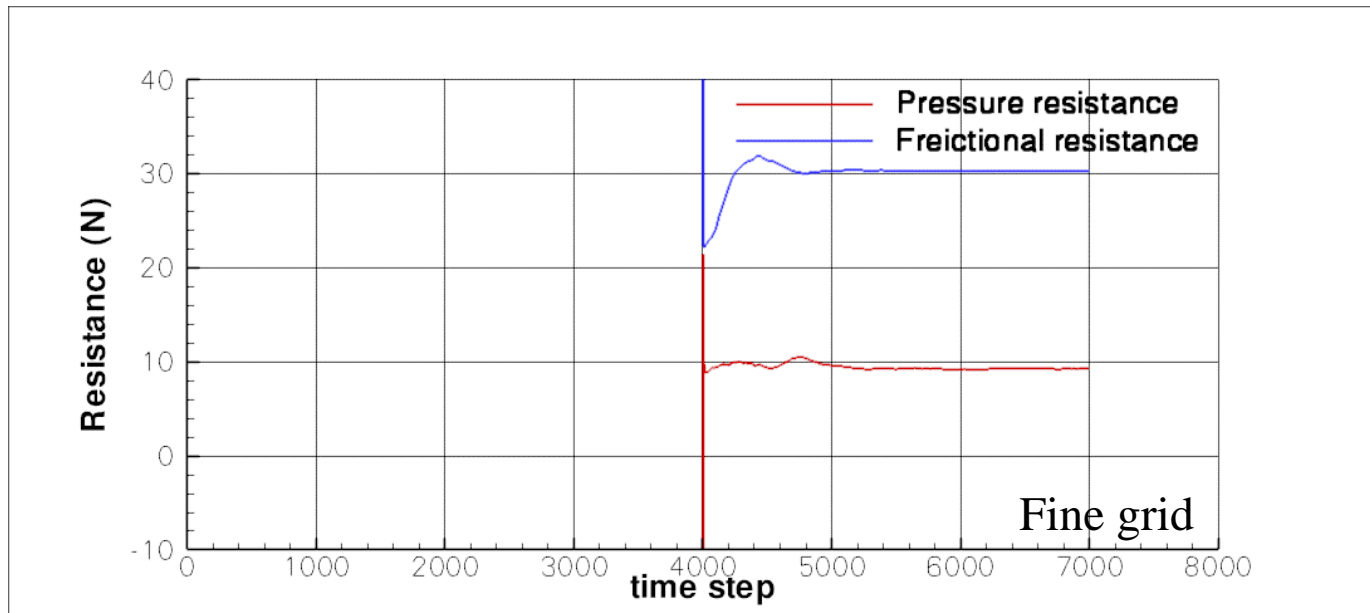
- Grid generation and calculating time : **18 min** (with 16 cores)





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





Remarks

- 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 → 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