Objectives

In the practical ship design procedure, experiment in towing tank which needs large costs or resources must be carried out to clarify the quantitative ship performance. If large scale CFD can be replaced to experiment, ship performance will be obtained quickly, or inexpensively. Furthermore, it is expected that the ship design will be upgraded by using of detailed flow field information which is impossible or difficult to obtain from actual flow measurement in experiment. The scope of this project is to investigate the feasibility of large scale CFD with LES model in case of flow field around marine propeller by using of "K-Computer", considering CFD may take place of towing tank test in the near future.

Outline of Results

In this project, the flow around marine-propeller is investigated by using of "FrontFlow/Red" with 6.4 Billion fine grid and LES model on "K-Computer". By using 6.4B fine grid, it is possible to obtain a result closer to the experimental result than in the case of using less fine grids used past year of which number of grids were 0.8B or 0.1B. This result suggests that the accuracy of calculation depends on grid resolution, and can be improved due to refinement of grid. However less fine grids provide equivalent flow to fine grid in laminar region, fine grid can provide more detailed flow structure which is impossible to clarify by using of less fine grid in turbulent region. These results on influence between calculation accuracy and grid resolution are very important information to decide grid resolution. In next year, the influence of Reynolds number will be investigated.