

Development of CFD methods for Industrial Ship Flow Applications

Takanori Hino

Yokohama National University (Japan)

Computational Fluid Dynamics (CFD) methods are now being used extensively in ship hull form designs as well as research and development in ship hydrodynamics. CFD in ship hydrodynamics emerged in Japan in the mid-1980s and from the beginning Ship Research Institute (SRI), now reorganized into National Maritime Research Institute (NMRI), has been one of the most active research organizations in numerical ship hydrodynamics. Since one of the missions of SRI/NMRI is to provide technical support to shipbuilding industry, their CFD research aims to develop numerical methods as a practical ship design tool. Various CFD codes have been developed and provided to shipyards in Japan. A review is given for the past research efforts at SRI/NMRI and the current development jointly pursued by NMRI and Yokohama National University (YNU). For the development of CFD methods, it is crucial that numerical solutions are validated using the benchmark experimental data. Series of international workshops for marine CFD have been organized to assess the state-of-the-arts of CFD and to give directions of further investigations through validations of computed results. SRI/NMRI and YNU organized three of them and provided the benchmark data to the international marine CFD community. Outcome from the past workshops are summarized together with the introduction of the recent Japan Bulk Carrier (JBC) project which has been conducted to obtain new benchmark data for a ship with an energy saving device.