B02. Numerical investigation of a ship propeller jet using computational fluid dynamics (CFD)

Yung-Jeh Chu
Wei-Haur Lam

Department of Civil Engineering, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia

Seabed scouring occurs due to the direct impingement of ship's propeller jet to the seabed. Numerous efforts had been carried out to preserve the seabed ecosystem. Limited researches are proposed to investigate the seabed scour due to the ship's propeller jet. However, this awareness does exist and some parties had proposed various theories and method to predict the degree of damages induced by ship's propeller jet. The ability to predict the velocity profile produced by a ship's propeller jet enhances the design of a new harbour in order to reduce the risk of induced seabed scouring.

The investigation on the velocity and the turbulence behaviors of a ship's propeller jet using computational fluid dynamics (CFD) is presented in this study. The CFD modeling includes propeller creation, grid generation, boundary condition setting and the suitable turbulence model selection. The influences of propeller geometry to the velocity distribution within the jet are also been investigated using computational fluid dynamics. The research is expected to provide a modeling technique which is able to be used to predict the velocity and turbulent behavior of a ship's propeller jet.

There are several ways to predict the fluid velocity profile produced by the ship propeller jet. Among them are the experimental method, site investigation models (Bollard pull), existing analytical method and also CFD modelling. The study will concentrateon CFD modelling, which includes 1) to simulate the propeller wash induced by ship propeller jet by using commercial software, Fluent as solver and post-processor, Gambit as pre-processor; 2) to simulate the propeller wash induced by ship propeller jet by using an open source program, OpenFOAM 2.1.x as solver, ParaView as post-processor; 3) to compare and comment on the results analysed from the commercial solver, Fluent and the open source solver OpenFOAM backed up with analytical solutions.