Dr. Ganbo Deng

Dr. Ganbo DENG was graduated at Tsinghua in 1984. He obtained his PHD at Ecole Centrale de Nantes (France) in 1989. Since then, he has been working in the same institute as a research scientist. His research topics cover different aspects concerning CFD simulation for incompressible flow for marine applications such as discretization scheme with finite volume as well as high order discretization scheme such as DG, HDG and spectral volume approach, fully coupled resolution for incompressible Navier-Stokes equation, fast linear solver, overset algorithm, turbulence modelization, RANSE simulation with simplified propeller model, etc. As one of the developers of a commercial flow solver, he also has an excellent expertise in RANSE simulation for marine engineering applications.

Keynote Presentation 6:
Ship maneuvering simulation with simplified propeller model

Ship maneuvering prediction with CFD has been performed with a simplified propeller model based on a open water performance curve in order to reduce CPU cost. To assess the accuracy of such prediction, comparison has been made with CFD prediction using actual propeller approach with which the rotating propeller is directly simulated, as well as with the measurement data. Both zigzag motion and turning circle motion have been simulated for two different configurations, namely the KCS and the ONRT test cases. For the ONRT test case equipped with a twin-screw propeller, as the inflow velocity to the propeller is more uniform, the simplified propeller model can simulate the action of the propeller with good accuracy. Although the side force is not taken into account in the simplified propeller model, not too much deterioration in the predicted ship motion is found due to error cancellation. For the single screw propeller configuration where the propeller operates in a non-uniform flow in the middle of the ship in the wake (the KCS test case), propeller thrust predicted by the simplified propeller model is lower. The comparison with the measurement data reveals that in addition to propeller model, one of the main sources of error in CFD prediction for ship maneuvering application comes from the prediction of rudder force. For half spade rudder, flow around the rudder is forced to separate from the gaps between the mobile part and the fixed part of the propeller due to the pressure. CFD simulation with the SST turbulence model under predicts both the axial force and the side force when flow separate around the rudder, especially when the rudder angle is high. More accurate prediction is obtained for the ONRT test case where a spade propeller is used.