Turbulence Modeling for Locally-Refined Free-Surface Flow Simulations in Offshore Applications

Arthur E.P. Veldman1, Roel Luppes1, Henri J.L. van der Heiden1, Peter van der Plas1, Joop Helder2 and Tim Bunnik2

1 Institute for Mathematics and Computer Science, University of Groningen, Groningen, The Netherlands
2 MARIN, Wageningen, The Netherlands

ABSTRACT

To study extreme hydrodynamic wave impact in offshore and coastal engineering, the VOF-based CFD simulation tool ComFLOW is being developed. Recently, much attention has been paid to turbulence modeling and local grid refinement. In particular, a blend of a QR-model and a regularization model has been designed. The QR-model belongs to a class of modern eddy-viscosity models, where the amount of turbulent eddy viscosity is kept minimal. For validation, experiments have been carried out at MARIN.

KEY WORDS: CFD; VOF-method; turbulence model; run-up; wave loading; semi-submersible; moonpool

INTRODUCTION

Extreme hydrodynamic wave impact on rigid and floating structures is of high industrial interest in offshore and coastal engineering. To study the occurring phenomena the CFD VOF-based simulation tool has been developed; see e.g. Kleefsman et al. (2005), Veldman et al. (2011) and Veldman et al. (2014). In the early phase of the ComFLOW development, emphasis has been on simulating momentum-dominated phenomena, such as the impact of extreme waves (e.g. green water loading (Fekken et al. 1999) and wave run-up) and on sloshing (e.g. in LNG tanks (Wemmenhove et al. 2008)). In these applications viscous effects can be mostly neglected. Later, the application area has been extended to flows where the influence of viscosity is becoming noticeable, like in side-by-side mooring or inside moonpools. Thus, recently much attention has been paid to turbulence modeling.

Numerical simulation of turbulent flow has to face the challenge of the very small spatial and temporal scales present in turbulence, requiring computational grids and time steps that resolve these small scales, which is currently not affordable. Thus strategies have been developed to model the effects of the subgrid scales onto the resolved scales: turbulence modeling through RANS or LES models. Most eddy-viscosity models (like k-ε or Smagorinski) add an excessive amount of ‘turbulent’ diffusion to model the dissipative effect of turbulence. Doing so, in laminar and transitional flow regions they seriously disturb the physical flow phenomena.

Therefore, in modern turbulence models the amount of turbulent diffusion is better controlled. One such model is the QR-model by Verstappen (2011). Based on functional-analytic arguments, it estimates the unresolved subgrid-scale details, and minimizes the amount of turbulent diffusion that is added. These estimates can be described in terms of the second and third invariants, Q and R, of the rate-of-strain tensor; hence the name. This method not only recognizes laminar parts of the flow, but also whether the turbulent flow field is more or less two-dimensional (relevant near free surfaces). In regions of backscatter, the QR-model is extended with a non-diffusive regularization model that reduces the production of the smaller scales. Irregular geometries will be treated with an immersed boundary cut-cell method, combined with a local-refinement strategy.

The behavior of the new model will be demonstrated on simulations of the water motion inside moonpools, and on extreme wave impact against a semi-submersible offshore platform. For validation purposes, MARIN has performed a series of experiments.

THE NAVIER-STOKES EQUATIONS

The incompressible, turbulent fluid flow is modeled by means of the Navier-Stokes equations

\[ Mu=0, \quad \partial \mathbf{u} / \partial t + C(\mathbf{u}) \mathbf{u} + G p - D \mathbf{u} = f. \]  

Here M is the divergence operator describing conservation of mass. Conservation of momentum is based on the convection operator \( C(\mathbf{u}) \mathbf{v} = \nabla \cdot (\mathbf{u} \mathbf{v}) \), the pressure gradient operator \( G \nabla p \), the diffusion operator \( D \nabla \cdot \nabla \mathbf{u} \) and the forcing term \( f \). The kinematic viscosity is denoted by \( v \).

The Navier-Stokes equations (Eq. 1) are discretized on an Arakawa C-grid. The second-order finite-volume discretization of the continuity equation at the 'new' time level \((n+1)\) is given by

\[ M^0 u_h^{(n+1)} = -M^f u_h^{(n+1)}, \]  

where \( M^0 \) acts on the interior of the domain and \( M^f \) acts on the boundaries of the domain. In the discretized momentum equation, convection \( C(\mathbf{u}) \mathbf{u} \) and diffusion \( D \) are discretized explicitly in time. The pressure gradient is discretized at the new time level. In this exposition, for simplicity reasons the first-order forward Euler time integration will be used. In the actual calculations, the second-order Adams-Bashforth method is being applied.

Taking the diagonal matrix \( \Omega \) to denote the matrix containing the volumes of the control volumes, gives the discretized momentum equation as

\[ \Omega (u_h^{(n+1)} - u_h^{(n)}) / \partial t = -C(\mathbf{u}) u_h^{(n)} + D u_h^{(n)} - G p_h^{(n+1)} + f. \]  

\[ (n+1) - (n) = \]