Numerical modelling of ships navigating through mud

Stefano Lovato[1], Geert Keetels[1], Cees van Rhee[1]

[1] Section Offshore & Dredging, Department of Maritime and Transport Technology, Delft University of Technology

Safe navigation in shallow water areas such as ports and waterways is ensured by setting a minimum distance between the ship's keel and the bottom, known as the under keel clearance (UKC). However, in such areas the bottom is often covered by a layer of mud. Under these circumstances, the UKC is no longer unequivocally determined and the prediction of the ship's controllability becomes more challenging. Selecting the solid bottom as reference would not ensure safe navigation because ships would frequently navigate through mud. On the other hand, selecting the water-mud interface as reference bottom would be too conservative since a contact between the ship's keel and the water-mud interface would hardly cause any damage to the ship.

An alternative approach for selecting the reference bottom is the so called "nautical bottom" approach. The nautical bottom is defined by PIANC [1] as the level where physical characteristics of the bottom reach a critical limit beyond which contact with a ship's keel causes either damage or unacceptable effects on controllability and manoeuvrability.

However, little is known about the link between ship's behaviour and mud characteristics, so in practice the nautical bottom is often defined as the level where mud reaches a critical density (e.g. 1200 kg/m3 for many harbours).

Model-scale experiments were carried out in the past to link mud properties and ship's manoeuvrability [2]. However, the large number of parameters to be addressed and the complex time-dependent non-Newtonian behaviour of mud make it very difficult both to obtain conclusive results and to apply model-scale results to full-scale. In this research we use CFD as the viable alternative to overcome these obstacles (Figure 1).

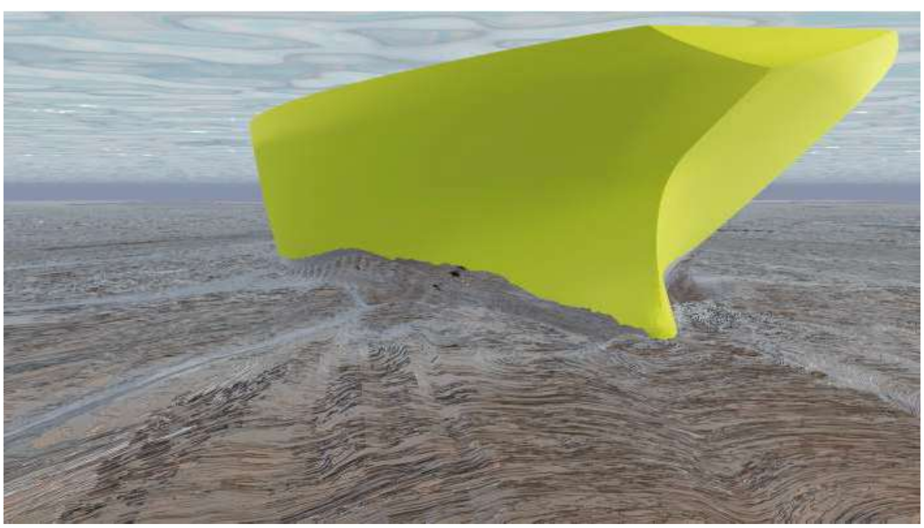

Figure 1: Rendering of a numerical simulation of a KVLCC2 hull navigating through a mud layer using ReFRESCO.

As already mentioned, mud is typically a non-Newtonian fluid. Rheological measurements showed that the Herschel-Bulkley model is suitable to describe the flow of mud (see e.g. [3,4]). Therefore the Herschel-Bulkley model has been implemented in a viscous-flow multiphase CFD code called ReFRESCO (www.refresco.org), developed by the Maritime Research Institute Netherlands (MARIN).

In practice, the Herschel-Bulkley model has been implemented as modification of the molecular viscosity, which becomes a function of the local shear rate. With zero or low deformation, structures at molecular level are formed and the fluid appears highly viscous. In contrast, when the fluid is stirred, the structure breaks down and viscosity decreases.

Before applying the code to simulate the flow around a ship, the code has been tested both to ensure that there are no coding mistakes and that the code can handle a flow dependent viscosity with strong gradients across the fluid domain.

Firstly, the code has been rigorously verified both for laminar single- and two-phase flows using the Method of Manufactured Solutions. Secondly, the code was tested on simple benchmark problems such as Poiseuille flow and the flow around a sphere (Figure 2). Comparison with results from literature revealed that the code is able to reproduce the drag force on objects in Herschel-Bulkley fluids.

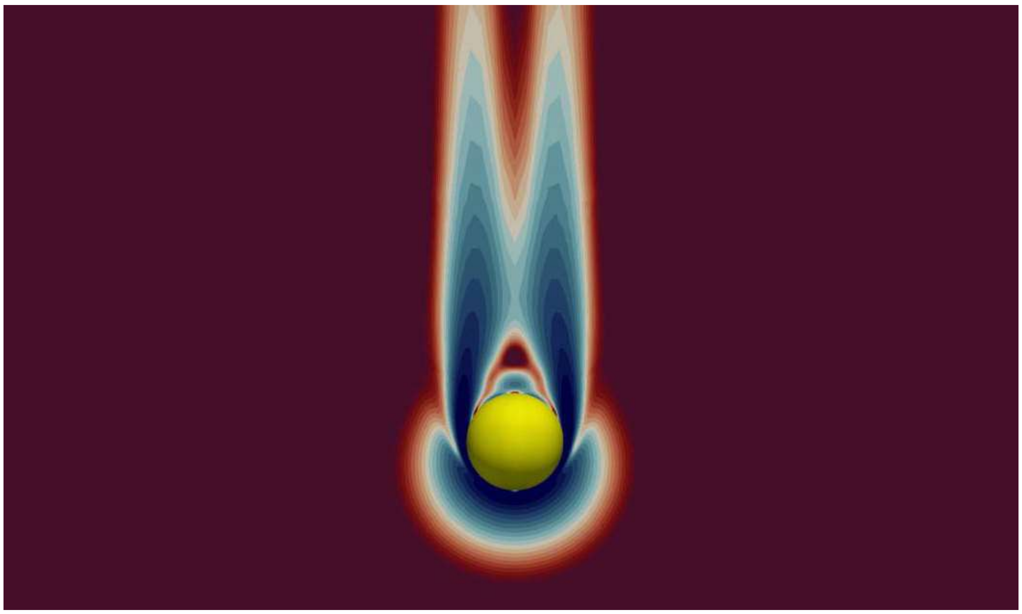

Figure 2: Contour diagram of viscosity for a sphere moving through a homogeneous Herschel-Bulkley fluid. Dark red and blue represent high and low viscosity, respectively. Flow is from bottom to top. The upstream Herschel-Bulkley fluid is undisturbed and presents large viscosity. When the fluid meets the sphere it is subjected to shear deformation which induces a structure break-down in the fluid. Viscosity is therefore lower near the sphere and in the wake region.

Current research is focussing on assessing how well standard RANS models such as k-omega and Spalar-Allmaras perform when applied to turbulent non-Newtonian flows. Preliminary simulations of the flow of Herschel-Bulkley fluids over a flat plate at high Reynolds number showed that different RANS models can produce completely different results. Current efforts are thus being made to study how standard RANS can be modified to account for non-Newtonian rheology.

Simultaneously, the CFD code is being applied to simulate the flow around a ship both in homogeneous mud (Figure 3) and in two-layer system with water on top and mud on the bottom (Figure 4). The goal of these simulation is to study the effects of mud rheology and UKC on the horizontal and vertical forces acting on the ship. Further research will be carried out to include such effects in a manoeuvring model.

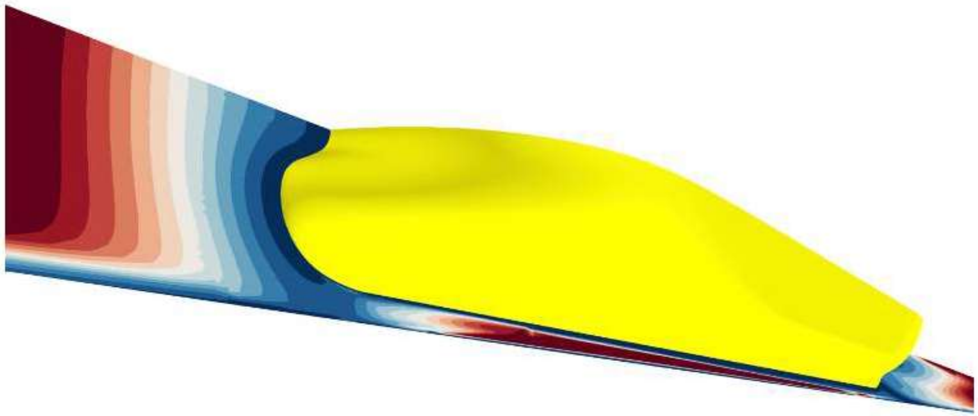

Figure 3: Contour diagram of viscosity for a ship moving through a homogeneous mud layer. As for the sphere in Figure 2, the high deformation induced by the presence of the ship tends to reduce the effective viscosity of mud. However, the presence of the solid boundary near the ship's bottom seems to induce the formation of a solid "plug" region, i.e. a region of nearly undeformed fluid with high viscosity.

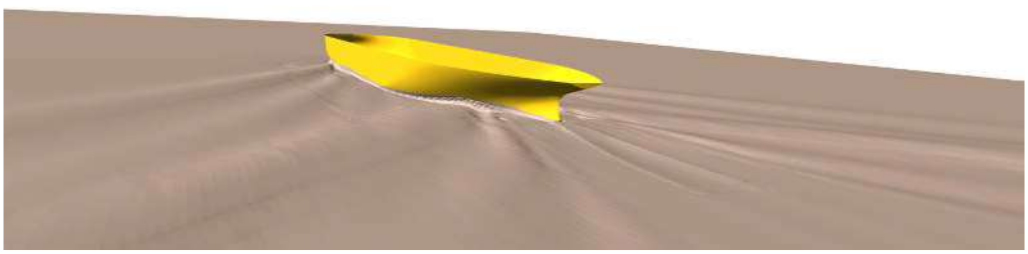

Figure 4: Wave pattern on the water-mud interface due to the passing of a ship with negative UKC. The wave pattern has a wide angle typical of shallow water navigation in subcritical regime

References

1. PIANC (2014) Report MarCom WG 121: Harbour Approach Channels - Design Guidelines, Tech. Rep. (2014).
2. Delefortrie, G., Vantorre, M. and Eloot, K. (2005) Modelling navigation in muddy areas through captive model tests. Journal of Marine Science and Technology 10, 188–202. doi:10.1007/s00773-005-0210-5.
3. Wurpts, R.W. 15 Years experience with fluid mud: Definition of the nautical bottom with rheological parameters. Terra et Aqua (2005).
4. Coussot,P. and Piau, J.M. (1994) On the behavior of fine mud suspensions. Rheologica Acta 33, 175–184. doi:10.1007/BF00437302.