Numerical CFD simulations – a new tool for the modelling of turbidity currents and sand dispersal in deep-water basins

Ernst W.M. Hansen[1], Wojciech Nemec[2] and Snorre Heimsund[1, 2, 3]

[1]*Complex Flow Design A.S., 7462 Trondheim, Norway*
[2]*Department of Earth Science, University of Bergen, 5007 Bergen, Norway*
[3]Present address: *Noreco A.S.A., 4005 Stavanger, Norway*

Introduction

The hydrodynamics of turbidity currents are difficult and expensive to study in the natural environments, whereas laboratory experiments are limited to small-scale, shallow-water flows carrying very fine-grained sediment. In either case, the monitoring of the current is limited to a few isolated 'stations' and merely a few measurable parameters (flow thickness, downflow velocity component, possibly sediment volumetric concentration and gouged sediment samples). Numerical simulations based on hydraulic mathematical models can be used to obviate these insurmountable difficulties, while integrating data acquired from the nature and laboratory experiments.

Computational fluid dynamics (CFD) is a numerical solution of the physical equations describing fluid flow and sediment transport. The method has been widely applied in the engineering branches of fluid mechanics, but has thus far been little used in sedimentological research and reservoir studies. The CFD software Flow-3D™ has been used to construct a three-dimensional model for the simulation of turbidity currents, including their internal hydraulic characteristics and the erosion, transport and deposition of sediment. The three-dimensional model employs a fixed Eulerian grid of rectangular finite volumes, and the turbidity current is modelled by a multi-phase, drift-flux flow method (Table 1). The sediment dispersion is treated as a continuous phase and its variable spatial volumetric concentration is calculated. The erosion, drifting and lifting of sediment particles are calculated in terms of tensors superimposed on the sediment advection with the turbulent fluid.

Table 1. General characteristics of the CFD model.

Fluid motion is described by the Navier-Stokes system of equations, combined with the Shields-Rouse equation for sediment entrainment threshold and a related equation for bed armouring effect. They are solved as a system of non-linear, transient, second-order partial differential equations.

The flow domain is divided into a dense 3D computational mesh, and the finite difference and finite volume methods are used to solve numerically the differential equations.

The numerical simulation of turbidity current involves a combination of several physical models:

- The RNG model for fluid flow turbulence.
- The drift-flux model for water-sediment mixing.
- The hindered settling model for high particle volume fraction and particle-particle interactions.
- The sediment scour model for sediment erosion and deposition.

The following mechanisms of sediment grain movement are accounted for:

- Drifting (i.e., the settling of suspended grains in fluid, modelled by a non-linear equation).
- Advection (i.e., the dragging of suspended grains by fluid flow).
- Bedload transport (modelled by the Meyer-Peter & Müller formula).
- Erosional entrainment (modelled by the Mastbergen & Van den Berg formula).

Transport of monosized or polysized sediment can be modelled (up to 45 grain-size classes may be account for).

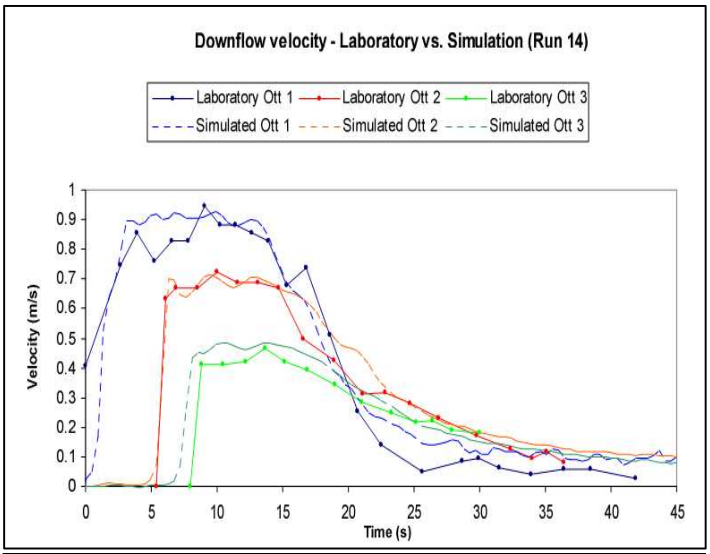
Importantly, a CFD simulation is not limited to 'stations' and allows the current behaviour to be monitored in its full 3D space and flow time. A wide range of flow characteristics can be continuously monitored, including the flow density, sediment concentration, grain-size segregation, all 3D velocity components, velocity magnitude (spatially averaged velocity field), internal shear strain rate (turbulence intensity), dynamic viscosity and bottom shear stress. Simulations imitating particular laboratory and natural flows have demonstrated that the numerical results are realistic and reliable. The Flow-3D™ model may thus serve not only as an attractive alternative to laboratory flume experiments, but also to provide unprecedented new insights in the hydrodynamics of turbidity currents. Our understanding of the turbidity currents, their hydrodynamic variability and the variability of their deposits can be greatly improved by this kind of numerical research.

The greatest advantage of the CFD approach is that it allows a laboratory flow to be upscaled to natural conditions and – with the corrections for bathymetric pressure and Coriolis force – to be used further to study the responses of turbidity currents to seafloor topography and hence also to model the effective dispersal of sand on a deep-sea basin floor. For any given seafloor topography and sediment sourcing pattern (point, multi-point or linear source), the progressive distribution of sand against the basin topography can be deterministically modelled, the location and temporal changes of depocentres can be recognized, and the seafloor zones of sediment bypass or flow spill-over can be identified. These issues are highly relevant to the majority of turbidite successions hosting hydrocarbon accumulations. The use of CFD numerical simulations in combination with the palaeogeomorphic seafloor maps provided by modern 3D seismics may offer important insights in the spatial development of hydrocarbon-hosting turbidite successions.

MODEL VERIFICATION

The three-dimensional CFD model has been validated by simulating particular laboratory flows and by comparing the simulation results with the laboratory measurements of flow parameters.

Experiment No. 1 (monosized sediment) – A numerical setup has been designed to imitate the laboratory flow reported by Baas et al. (2004) from the EuroTank facility at the Utrecht University's Department of Earth Sciences. The laboratory setup consisted of a straight channel inclined at 5^o, leading from the feeder tank to a horizontal expansion table, all submerged in freshwater. The channel was 4 m long, 0.22 m wide and 0.50 m deep. The expansion table was 3.5 m long and 3 m wide, and was located in a tank 1 m deep and 4x4 m in area (i.e., the table was free of sidewalls to minimize flow reflections). The laboratory setup had three flow-probing stations, but only a few of the flow runs provided data sufficiently 'pure' (non-averaged) and of a sufficient time span to be used for comparative purposes. Simulation results are shown in Fig. 1.



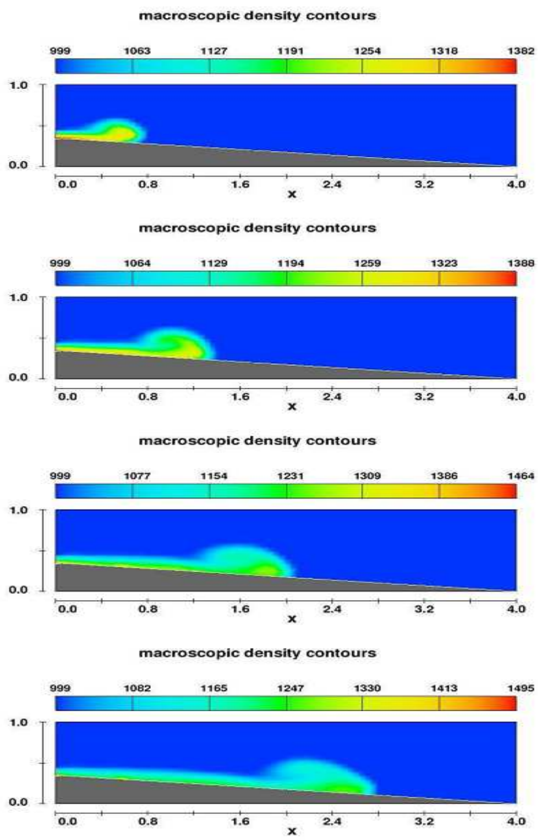


Fig. 1. Two-dimensional CFD simulation of a Utrecht laboratory flow: snapshots of the flow density (to the right, values in kg/m^3)) and a comparison of station-bound velocity time series (below). The flow initial sediment concentration was 25 vol.% (clay fraction, with a grain size of 0.15 mm). Arguably, the simulated velocity time series are more reliable, because the laboratory velocimeter propellers have recorded some mechanical 'noise' due to sediment particle entrapment.

Experiment No. 2 (two-sized sediment) – Constant-volume laboratory flows with two-sized (bidisperse) sediment were performed by Gladstone et al. (1998) in a glass-walled flume 5.7 m long and 0.2 m wide. The flume was filled with tap water to a height of 0.4 m. A well-mixed watery sediment suspension (0.016 m^3 volume) was released from a lockgate to generate the flow. Silicon carbide particles were used to create an excess density, with a varied proportion of fine (0.025 mm) and coarse particles (0.069 mm). The flow frontal velocity and the 'deposit density' (sediment mass per unit bed area) were measured along the tank for each of the six experimental flows (Fig. 2A,B). CFD simulations have satisfactorily imitated the laboratory results (Fig. 2C,D)).

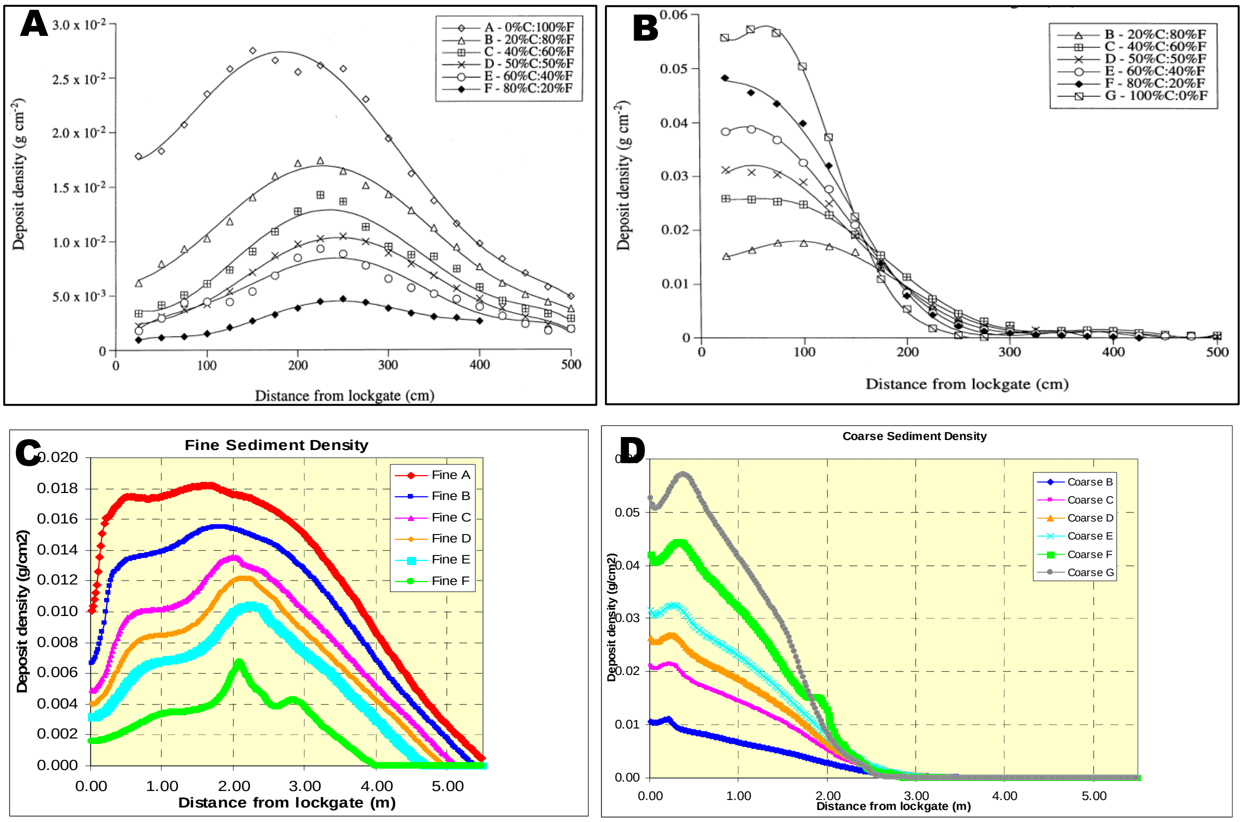


Fig. 2. (**A** & **B**) Laboratory measurements of 'deposit density' along the flume floor, for flows with different percentages of coarse (C) and fine particles (F). (**C** & **D**) The corresponding results from numerical simulated flows.

PRELIMINARY APPLICATIONS

The Ormen Lange field – The CFD model has been used to simulate turbidite deposition in the Ormen Lange field, offshore Norway. The basin-floor topography used was the basal surface of the reservoir's Egga Unit, derived from 3D seismics combined with back-stripping and decompaction. The deformation induced by the Recent Storegga Slide has also been removed. The aim of this simulation study was to assess the influence of basin-floor topography on the spatial pattern of sediment dispersal by turbidity currents in the Ormen Lange area, assuming a point-source scenario developed by previous researchers. A few tens of identical large-scale turbidity currents have been run over the topographic surface to monitor their changing velocity field and sediment dispersal pattern (Fig. 3). The result show that turbidity currents are highly sensitive to subtle variations in the seafloor topography and confirm further the notion that the Ormen Lange sub-basin was supplied with sediment from a single main source in the southeast.

The Monterey Canyon System – The December 2002 and November 2003 turbidity currents recorded by the USGS/NPS three mooring stations in the Monterey Canyon System (Xu et al., 2004) have been simulated on a trial-and-error basis, with the results verified through comparisons with the station-bound flow velocity profiles. The USGS detailed bathymetric map has been used for the canyon topography. The two reported flows were relatively small (ca. 30 m in thickness) and dissipated with the middle to lower canyon reaches. A 10-times larger flow was then simulated to monitor and predict the behaviour of currents that actually supply sediment to the Monterey submarine fan.

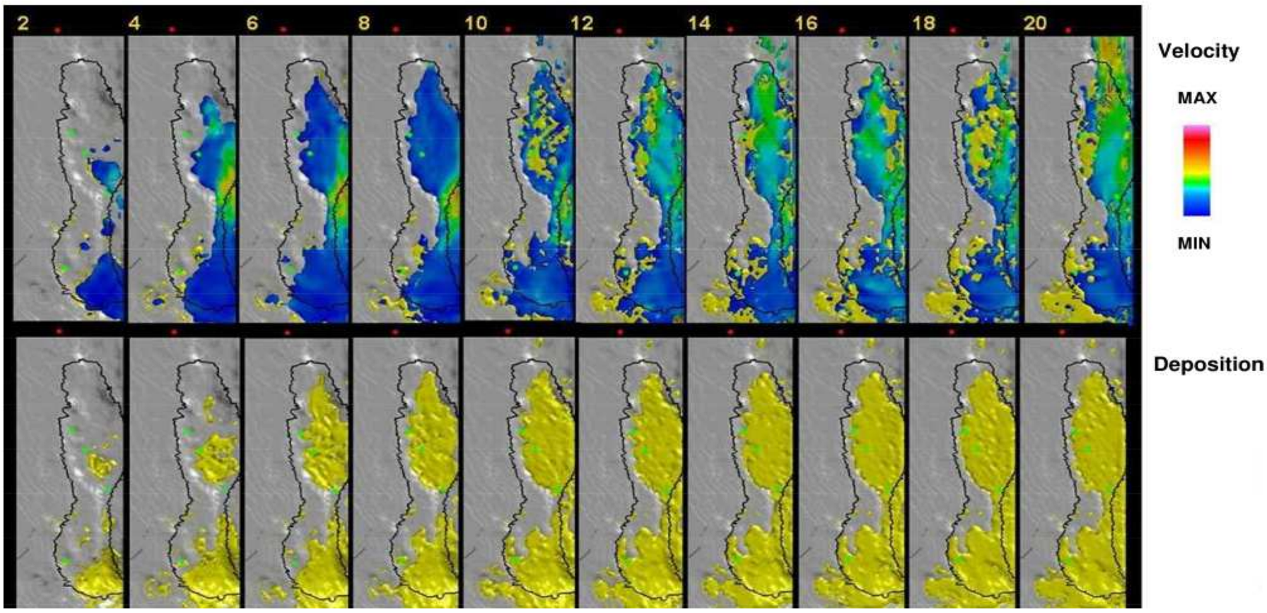


Fig. 3. Plan-view snapshots of the velocity field of successive flows (top) and the corresponding pattern of sand accumulation in the Ormen Lange sub-basin.

The simulation results confirm that the velocity field of turbidity current is highly sensitive to seafloor topography. All the flows appear to have experienced several phases of acceleration and deceleration, and large parts of the flows were ponded by arriving into 'dead-end' arms of the canyon system (Fig. 4). None of these phenomena was recognized by the three mooring stations, and the present study thus gives an unprecedented new insight into the behaviour of turbidity currents in one of the best-known modern submarine canyon systems.

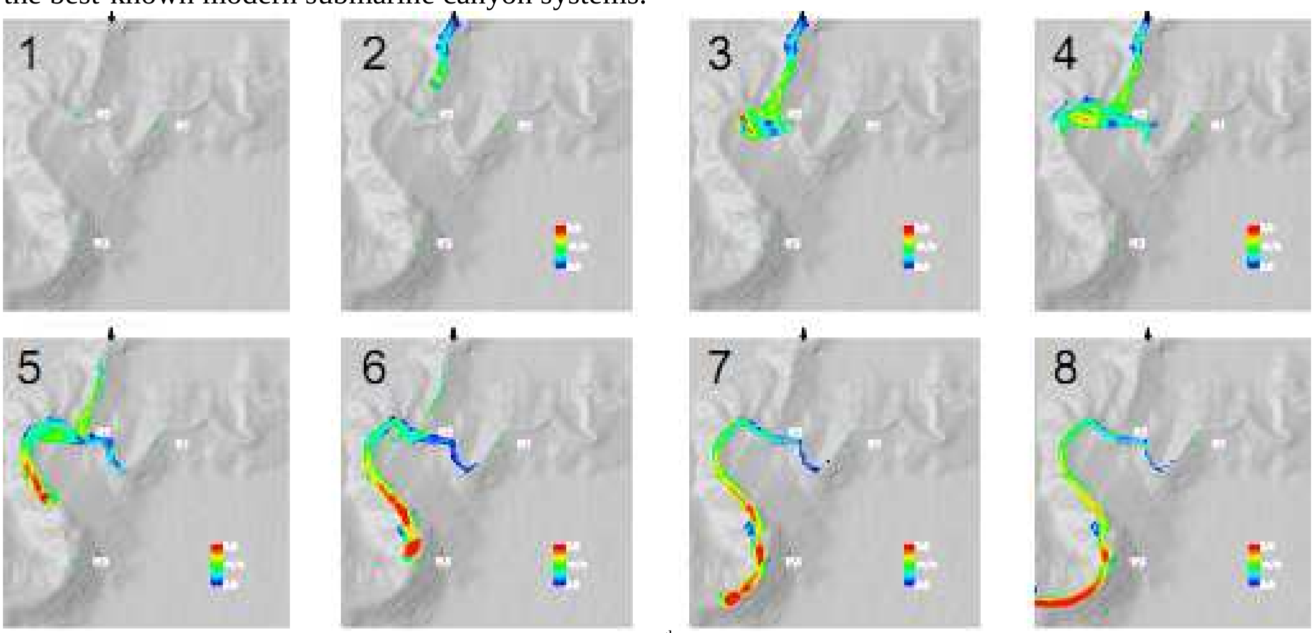


Fig. 4. Simulation snapshots of the December 17th, 2002, flow in the Monterey Canyon System (red = highest velocity, dark blue = lowest, near-zero velocity).

ACKNOWLEDGEMENTS: We kindly thank the Research Council of Norway (NFR), StatoilHydro, Norske Shell, Det Norske, DONG, Gas de France and Flow Science for supporting our research.