Proceedings of the
XVIIIth Telemac and Mascaret User Club
19th–21st October 2011
EDF Corporate University, Chatou, France

Organised by
Laboratoire National d'Hydraulique et Environnement, EDF R&D
Laboratoire d'Hydraulique Saint-Venant, Université Paris-Est
Département SINETICS, EDF R&D

Edited by
Damien Violeau
Jean-Michel Hervouet
Emile Razafindrakoto
Christophe Denis

Credits
Cover pictures by
EDF R&D
CHC (BlueKenue®)
HR Wallingford
Foreword to the XVIIIth Telemac and Mascaret User Club

Dear delegate,

The Research and Development division of Electricité de France and The Saint-Venant Laboratory for Hydraulics are delighted to host the XVIIIth Telemac and Mascaret User Club.

The Telemac hydroinformatic system, created in 1987, has been distributed outside EDF since 1993. A small group of users then used to gather every year in order to present their results and discuss about new features and future developments. These annual meetings were first held in Chatou, then in various places all around Europe. In 2010, the key innovation was the open-source distribution of Telemac, which lead to a significant increase of the number of attendees. Therefore, this year, we decided to modify the format of the user club in order to make it more attractive and professional. Instead of an informal workshop, the 2011 Telemac and Mascaret User Club is thus organised like a small conference, with abstract submission, paper presentation and printed proceedings. I hope that this new format will meet your expectations.

The user club’s Committee received 26 abstracts of good quality, and all of them were selected to appear in the present proceedings. This demonstrates the success of Telemac in fields ranging from sediment transport to heterogeneous High Performance Computing.

For the XVIIIth Telemac and Mascaret User Club, it is with great pleasure that I welcome you all to the City of Chatou near Paris.

Thank you for your participation.

Damien Violeau
Chair of the Local Organising Committee
Commitees

Scientific Committee
Damien Violeau (EDF R&D)
Emile Razafindrakoto (EDF R&D)
Jean-Michel Hervouet (EDF R&D)
Christophe Denis (EDF R&D)
Nicole Goutal (EDF R&D)
Christophe Coulet (SOGREAH)
Rebekka Kopmann (BAW)
Sébastien Bourban (HR Wallingford)
François Hissel (CETMEF)
Dave Emerson (Daresbury Laboratory)
Pierre Lang (Ingérop)
Michel Benoit (Saint-Venant Laboratory)

Local Organising Committee
Jean-Daniel Mattei
Damien Violeau
Emile Razafindrakoto
Jean-Michel Hervouet
Christophe Denis
Mariam Drame
Christophe Kassiotis
Cédric Goeury
Contents

**Day 1** is a training day.

**Day 2**

Session 1 is dedicated to the workshop introduction.

Session 2: Sediment 1

Sediment pathways and the analysis of dredging on sediment deposition along the Norfolk and Lincolnshire coasts – UK ................................................................. 1
Nicolas CHINI, Peter STANSBY

Comparison of sediment transport formulae during a flood wave in the river: an application of Telemac teaching .................................................................................. 8
Caroline GENOUX, Philippe LARROUDE,
Lucile POULLEAU, Thomas POCHETAT

Modelling of bed forms (dunes) with Telemac 3D and Sisyphe .......................... 16
Annalena GOLL, Rebekka KOPMANN

Modelling sediment transport with hysteresis effects ..................................... 22
Michiel KNAAPEN, David KELLY

Comparison of sediment transport formulae with simulation of several storms on a Mediterranean beach and with in-situ sedimentary flux on a North Sea beach ............................................................... 28
Mehdi DAOU, Philippe LARROUDE, Adrien CARTIER, Arnaud HEQUETTE

Session 3: HPC & Interoperability

Preparing TELEMAC-2D for extremely large simulations ................................. 35
Yoann AUDOUIN, Charles MOULINEC, Robert BARBER,
Andrew SUNDERLAND, Xiao-Jun GU, David EMERSON

HPC for sensitivity studies: simulations with TOMAWAC and TELEMAC-3D .................................................. 43
Noémie DURAND, Sébastien BOURBAN, Charles MOULINEC,
Robert BARBER, Yoann AUDOUIN, Xiaojun GU, David EMERSON

Effects of the coupling between TELEMAC 2D and TOMAWAC on SISYPHE modelling in the outer Seine estuary ................................................................. 51
Nicolas GUILLOU, Georges CHAPALAIN

Session 4: Hydrodynamics

Numerical modelling of the Vaugris reservoir on the Rhône with Telemac 3D .................................................. 59
Olivier BERTRAND, Antoine LAVEDRINE, Adlane REBAI,
Eric DAVID, Gilles PIERREFEU, Pierre ROUMIEU

Modeling of a managed coastal Mediterranean wetland with TELEMAC-2D: the Vaccarès lagoons system (Rhone delta, Camargue, France) .................................................. 64
Olivier BOUTRON, Aurélien LOUBET, Olivier BERTRAND,
Patrick HÖHENER, Eric COULET, Yves CHERAIN, Philippe CHAUVELON

Three-dimensional flow patterns at a river confluence on the alluvial system of the Paraná River, Argentina ................................................................. 69
Mariana MORELL, Pablo TASSI, Carlos VIONNET
Day 3

Session 5: Sediment 2

Modelling and analysing dredging and disposing activities by use of Telemac, Sisyphe and DredgeSim ................................................................. 92
  Christian MAERKER, Andreas MALCHEREK

Bed roughness feedback in Telemac 2D and Sisyphe ................................................................. 99
  David McCANN, Alan DAVIES, James BENNELL

A continuous sediment layer concept for Sisyphe ................................................................. 105
  Uwe MERKEL, Rebekka KOPMANN

New developments and validations for the 3D sediment transport modelling within the Telemac Modelling System ................................................................. 111
  Pablo TASSI, Catherine VILLARET, Jean-Michel HERVOUET

Large scale morphodynamic modeling of the Gironde estuary ..................................................... 117
  Catherine VILLARET, Nicolas HUYBRECHTS, Lan Anh VAN

Session 6: Miscellaneous

Numerical modelling of oil spill drifts in inland waters ............................................................. 124
  Cédric GOEURY, Jean-Michel HERVOUET

Simulation for climate change and indicator of vulnerability on four French sandy beaches ............................ 132
  Philippe LARROUDE

Session 7: Numerical aspects

Adaptive vertical layering in TELEMAC-3D ........................................................................ 137
  Chris CAWTHORN, Sébastien BOURBAN, Chris MEAD,
  Matthew WOOD, Michael TURNBULL

Revisiting the Thompson boundary conditions ........................................................................ 142
  Jean-Michel HERVOUET, Christophe DENIS, Eric DAVID
Sediment pathways and the analysis of dredging on sediment deposition along the Norfolk and Lincolnshire coasts – UK

Nicolas CHINI, Peter K. STANSBY
School of Mechanical, Aerospace and Civil Engineering
The University of Manchester
Manchester, UK
Nicolas.Chini@manchester.ac.uk

Abstract— This study presents an application of the TELEMAC suite in order to assess the sediment pathways in a coastal domain where marine aggregates extraction takes place. The effect of dredging on coastal sediment deposition is analysed by considering the transport of sediment from a dredging area over an otherwise stationary bed assuming equilibrium conditions. Transport is driven by the hydrodynamics of tides, waves and surges and simulations are run for times of around a year over which the morphodynamic system is assumed to be otherwise in equilibrium. By determining the coastal sediment deposition resulting from a volume of sediment superposed on the dredging area, the nearshore-offshore linkage may be estimated. The finite-element software suite, TELEMAC system, comprising TELEMAC-2D (currents), TOMAWAC (wave action) and SISYPHE (sediment transport) is used. As well as coastal sediment deposition, large-scale, residual sediment pathways for the coastal domain are produced. The influence of waves is assessed. Case studies are undertaken for one important dredging area off the Norfolk and Lincolnshire coasts in the UK.

I. INTRODUCTION

The influence of dredging on coastal processes and erosion has been the subject of much debate. There is folklore that sand extracted offshore eventually deprives sand from beaches causing or accelerating coastal erosion. We are here concerned with dredging off the Norfolk and Lincolnshire coasts which provides about 10% of the aggregate for the UK construction industry and is thus of major strategic importance [1]. Dredged material from UK waters is also exported to the Netherlands, Belgium and France. In addition over 80 million cubic meters of marine aggregates have been used in the UK for beach replenishment since the late 1960’s, to mitigate coastal erosion [1].

This profitable activity is highly regulated. It affects areas used by marine pipelines, by offshore wind farms and for navigation. Dredging activities can modify the seabed composition up to several kilometres from the trench or the pit, impacting seabed fauna and flora [2, 3]. Since extracting sediment from the seabed reduces the volume of sediment offshore, offshore dredging may impact coastal evolution. Because of the complex nonlinear processes involved in coastal morphology, it is difficult task to isolate the influence of dredging from the background morphodynamic behaviour.

The relation between offshore dredging and coastal stability has been reported in the literature. Many studies, based on numerical modelling, focus on the effect of sandpits or trenches on tidal flows and wave propagation [4, 5, 6]. Results are site specific and tend to show that offshore dredging has minimal influence on nearshore wave height and longshore sediment transport. Idealised numerical modelling has shown that dredged, shore-connected sand ridges would recover and that sediment required to rebuild the sand ridges is provided both by offshore and nearshore zones [7]. Large scale effects using process-based numerical modelling have also been studied by [8]. Depending on the geometry of the sand pit, the extent of the area influenced by the dredging can reach 200 km². Flume and wave tank experiments show that sand extraction can affect shoreline stability depending either on the position of the trench or on the wave climate [9]. During the last decade, projects have been undertaken to tackle the issue of the impact of offshore dredging. The SANDPIT project [10] proposed guidelines for stakeholders, and detailed laboratory and field measurements were made in order to improve numerical modelling of sediment transport. Within the framework PUTMOR [11], extensive monitoring of a sand extraction pit was undertaken. A variety of numerical models developed to study the offshore morphological effect of marine aggregate extraction is reviewed in the EU MARSAND project [12].

The area of interest in the present study is the South-Western part of the North Sea (Fig. 1). The bathymetry of this area has prominent tidal sandbanks. Their interaction with the local flows and waves were studied by [13] highlighting the implications for adjacent nearshore stability. Some of these seabed features are moribund and others are mobile. [14] used a long temporal series of surveys to assess the mobility of the sandbank system of Great Yarmouth. The sediment circulation in the vicinity of moribund sandbanks was analysed by [15]. Clockwise sediment transport was shown around the sandbanks confirming numerical modelling undertaken by [16]. A general assessment of the
regional sediment transport pathways was performed during the Southern North Sea Sediment Transport Study [5]. The longshore transport estimated through this latter study was in agreement with previous studies [17, 18].

The availability of hindcasts for offshore waves and surges [19, 20] provide hydrodynamic boundary conditions for modelling within extensive coastal domains, enabling new insights to be provided by process-based morphodynamic modelling. Here, offshore boundary conditions for waves, surges and tides [21] enable the assessment of sediment pathways on the upper part of the continental shelf. The analysis and modelling strategies are presented in section 2 and 3. After analysing the sensitivity of the model to the sediment transport formulation and the relative influence of waves on sediment pathways in section 4, section 5 presents the results of the dredging impact on bed evolution. Section 6 discusses the sediment pathways followed by a source of sediment located in the Humber dredged area. Conclusions are drawn in the last section.

II. PRESENT ANALYSIS STRATEGY

Sediment transport is driven by the hydrodynamics of tidal currents and waves, and surges in extreme conditions. Most transport models determine the flux which defines bed evolution through sediment mass conservation. By running hydro-morphodynamic models for a large domain including the dredged area it is difficult to determine the effect of removing sand since the origin of sediment deposited near the shore or elsewhere is not identified.

To determine the influence of dredging we adopt an inverse approach. We consider an initial condition of a volume of sand distributed over the dredging area. This volume is then dispersed due to the hydrodynamic forcing of sediment transport over the otherwise stationary bathymetry of the coastal domain. In this way the deposition resulting only from the initial volume can be determined everywhere, including importantly the nearshore region. The flux out of the domain may also be determined. An equilibrium state may otherwise be assumed for a period of about a year. Since the dispersion of a volume from an area is now known, the sediment pathways are also known. The volume of sand deposited nearshore is defined as that deposited long a thin strip parallel to the shoreline.

III. MODELLING STRATEGY

Numerical simulations of wind-generated waves, tides and surges propagation are made using the TELEMAC system [22]. Based on the finite element method, the system is a convenient tool enabling high spatial resolution in the zones with complex bathymetry while optimising computing time. It incorporates a wave action conservation equation solver –TOMAWAC, a Saint-Venant equation solver–TELEMAC-2D, and a sediment mass conservation equation solver –SISYPHE.

A. Computational domain

The computational domain for this study includes the coastal zone extending from the Humber Estuary in the North to Southwold in the South (Fig.1). The coastline is assumed to be at the lowest astronomical tide (LAT).

The finite element method allows flexible refinement in zones with complex bathymetry. The spatial resolution of the model varies from 250m to 4000m. Fine resolution is imposed along the coastline and the sandbanks. The number of nodes is 29762 with 58689 triangular elements.

The initial bathymetry reference is Chart Datum (CD). A conversion is performed to convert the CD reference to Ordnance Datum at Newlyn (ODN). To do so, one needs to know the difference between the mean sea level (MSL) and the LAT. The tidal model of Continental Shelf of the UK coasts, CS3X, developed by [20], has its free surface elevation reference at the MSL. From the tidal results of this model run over 40 years, the lowest tidal water level can be estimated. This level provides an estimate of the difference between the LAT and the MSL. This estimate depends on the number of tidal waves represented with the CS3X model.

According to the data at Cromer and Lowestoft, the relation between LAT and MSL is respectively –2.75m and –1.50m. Our method leads to an overestimate with these data of less than 0.15m.

B. Wave propagation module

The module, TOMAWAC, is run in non-stationary mode, solving the equation of the conservation of wave action with the following source terms: wave-wave interaction, energy dissipation by bottom friction and depth-induced breaking, and refraction due to bathymetry gradient. Water depth evolution, due to tides and surges, is taken into account. [23] found that wind stress within the domain and wave refraction by currents had little influence on wave propagation for this coastal area. Including wind stress within the coastal domain only had a noticeable effect on nearshore wave heights for speeds higher than about 20m.s⁻¹.
When waves are entering the domain, a JONSWAP spectrum is imposed at the offshore boundaries of the model. A free condition is assumed when offshore waves are travelling out from the domain. The spectral frequency and directional resolutions were set to 22 frequencies (from 0.05 to 0.37Hz) and 24 directions. The JONSWAP bottom friction coefficient of 0.038m²/s³ and the formulation for depth-induced wave breaking of Battjes and Janssen are used. The time step is set at 6min. [24] previously calibrated and validated the model.

C. Hydrodynamic tide/surge module

The propagation of barotropic tides and surges is computed using TELEMAC-2D, which solves the depth-averaged shallow water equations. Converged results were obtained with a time step of 90s. The model takes into account bottom friction, Coriolis force and wave radiation stresses. Along the offshore boundaries, water depth and current velocities are imposed. At the coastline, a free slip condition is prescribed. Bottom friction parameterisation is based on bottom roughness with a Nikuradse coefficient dependent on mean sediment size and bed forms. A spatially uniform coefficient is used here. A constant value of the Coriolis parameter is also used.

D. Sediment transport and bed evolution module

The conservation of sediment mass is solved using a finite volume scheme. Special treatment for a non-erodible bed is included within SISYPHE [25]. Along the coastline, zero sediment flux is imposed. At the outer boundaries, the condition depends on the flow. For an inflow, there is no sediment flux and no bed evolution. For an outflow, a zero gradient of sediment flux is imposed. The time step is the same as for TELEMAC-2D.

Sediment transport modelling depends on sediment grain size and semi-empirical formulae to relate flow conditions to sediment flux. Two different grain sizes representative of the area of interest are considered: median grain sizes, $d_{50}$, of 400 µm and 200 µm. Two widely used total load formulations for sediment transport rate were tested: due to Bijker and Soulsby–van Rijn [25].

E. Coupling procedure for sediment transport due to tides, surges and waves

For tides alone TELEMAC-2D and SISYPHE are internally coupled to simulate sediment transport and bed evolution [25]. In addition wave propagation generates radiation stresses which influence currents and sediment transport. Wave-integrated parameters and radiation stress were thus determined by running TOMAWAC with water elevations from a tide-only TELEMAC-2D run, usually for a period of one year. These are then input into the coupled TELEMAC-2D and SISYPHE system to give sediment transport due to currents and waves combined. Radiation stress in principle has an effect on water elevation but the resulting effect on wave propagation was found not to be significant.

F. Boundary conditions

Wave, surge and tide data imposed along the outer boundaries are provided by UKCP09 [21]. The UKCP09 outputs are hourly water elevations, depth-averaged current components and six-hourly integrated wave parameters, with a spatial resolution of approximately 12km. All these parameters are linearly interpolated both temporally and spatially along the offshore boundary of our domain.

The UKCP09 dataset is a climatic projection, set up to study the marine impact of climate change in the UK coastal waters, based on SRES scenarios [26]. However, mean and extreme statistics from 1960 to 1990 are consistent with hindcast data using re-analysed wind data ERA40 [21]. The boundary conditions correspond to the year 1989, which is an average year in terms of the number of storm events.

G. Dredging input

To study the distribution of sediment located within particular offshore areas, the model is set up with a non-erodible bed everywhere. After one semi-diurnal tidal cycle used for model spin-up, a volume of sediment is deposited uniformly within the offshore area under consideration. The thickness of the layer equals the ratio between the prescribed volume and the dredged surface area. The location of the licensed area is provided by the Crown Estate. For our application, a volume of $1.5\times10^5$ m³ is deposited, which represents a layer of 2cm.

IV. Sensitivity tests

Prior assessing the sediment pathways, some sensitivity tests are performed on the median grain size, the sediment transport formulations and on the influence of waves on sediment transport.

A. Median grain size and sediment transport formulations

Theses tests were performed to study the residual transport with only tides present. Sediment is assumed to be transported everywhere and vectors of residual sediment flux direction are presented for one neap-spring tidal cycle in Fig. 2. The results for the vector fields and hence sediment pathways are very similar for all cases, but the magnitudes of sediment transport rate were different. Higher rates are obtained for the smaller $d_{50}$. The Soulsby–van Rijn formulation gives rates higher than the Bijker formulation by a factor of about 10. This difference is partly due to the fact that the Soulsby-van Rijn formulation is implemented with a higher bottom friction coefficient, as observed in [27].

Previous research has carried out intercomparisons between the two formulae for sediment transport induced by waves and currents [27, 28]. For all conditions, the Soulsby-van Rijn formula provided greater sediment transport rates than the Bijker formula.

B. Wave influence on sediment transport

The relative influence of waves on the sediment transport is estimated in terms of residual sediment transport by defining the following ratio:
\[ r = \frac{\overline{Q}_{T,W} - \overline{Q}_T}{\overline{Q}_T} \]  

where \( \overline{Q}_{T,W} \) and \( \overline{Q}_T \) are the annual residual sediment flux vectors for combined wave and tide simulation and tide alone simulation respectively. For \( r = 0 \) waves have no effect and increasing \( |r| \) shows an increasing effect of waves relative to tides. Fig. 3a presents the spatial distribution of \( r \) with a sediment size of 400µm and using the Soulsby–van Rijn total load formula. Note \( r \) may be negative showing that waves can reduce sediment transport. Wave influence is noticeable where water is shallow, over the sandbanks and along the coastline. In deeper water, to the north, sediment flux is dominated by waves while to the east and south by tides.

The influence of waves on the sediment transport direction is also assessed through the difference in the direction between the vector for residual sediment transport due to tides and waves, \( \overline{Q}_{T,W} \), and the vector due to tides alone, \( \overline{Q}_T \). Fig. 3b shows that waves modify sediment transport direction in some very local areas, mainly around the sandbanks off Great Yarmouth.

V. TRADITIONAL ASSESSMENT OF DREDGING IMPACT

To tackle the issue of non-linearity induced by offshore material extraction, the following procedure is adopted:

- one morphological simulation with an initial non-modified seabed level,
- one morphological simulation with an initial seabed level modified by a dredging intervention,
- computation of the difference between these two simulations.
The latter computation will provide the influence of dredging. In this section, this traditional assessment is performed considering only tides. The bed evolutions over one year are estimated using the filtering input method proposed by [29] and adapted by [30]. The Soulsby–van Rijn model is here applied. A 1m depth pit is simulated in the Humber area. Fig.4 presents the difference between the non modified and modified simulations, showing the extent of the domain affected by the dredging. The maximum differences are about 5cm after one year along the Lincolnshire coast, but some differences are also noticeable further south and along the sandbanks.

VI. SEDIMENT PATHWAYS

Three different characteristics are presented. Firstly, the temporal evolution of sediment volume within the dredging area, settling on the coast and leaving the domain are computed. Then the spatial distributions of sediment are presented. Finally the distributions of volume exiting the domain are shown.

A. Temporal volume evolution

The volume of sediment, $V$, within a given area, $A$, included within the computation domain, $\Omega$, is defined as the summation of the layer thickness, $E$, above the non erodible bed at points within the given area:

$$V(t) = \sum_{i=1}^{N} E_i \chi_{A} \psi_i d\Omega$$  \hspace{1cm} (2)

where $N$ is the number of points in the computation domain, $\psi_i$ is the $i$-th point-related basic function. The characteristic function $\chi_{A}$ associated with the area, $A$, is equal to 1 for any point within $A$ and zero otherwise. The area used to compute the volume settling along the coastline is defined as the ensemble of the meshes having at least one node located along the coastline. This area is called the coastline area.

Results for the temporal evolution of volumes within the dredging area, volumes settling on the coastline and leaving the domain are presented in Fig. 5.

Both the Bijker and the Soulsby–van Rijn sediment transport formulae are used. The results are sensitive to the formulation. However, the movement of sediment from the dredging area towards the coastline is captured by both formulations. The sediment deposited along the coast is significant. For this dredging area, the model is run for 4 years. It appears that if waves are not included in the ensemble of the meshes having at least one node located along the coastline. This area is called the coastline area.
Computation, the volume of sediment within the coastline area becomes constant. This is not the case if waves are added, especially when the Soulsby-van Rijn formulation is used. The volume along the shoreline eventually starts to decrease as the sediment is exiting the domain.

Figure 5. Temporal evolution of volumes for the Soulsby-van Rijn formulation within the dredged area (black line), the coastline area (grey line) and exiting the domain (grey dashed line).

Materials extracted by offshore dredging are not only used for the construction industry but also for beach nourishment. Locations along the coastline where sediments are deposited, shown in Fig. 6, due to sediment dumped on the dredging area would be appropriate for beach replenishment to mitigate the negative impact of dredging for marine aggregates on coastline stability.

Figure 6. Spatial distribution of sediment released from the dredging area after one year, using the Soulsby van Rijn formulation without waves.

B. Spatial distribution

The sediment spatial distribution is represented as isolines of bed levels. Two isolines are chosen to represent 0.01% and 1% of the initial sediment layer. Results are presented only for the case of Soulsby-van Rijn formulation (Fig. 6).

Results show that the sediments are deposited all along the coastline from Mablethorpe in the north down to Southwold. More sediment remains along the Lincolnshire coast after one year if the waves are excluded. A large amount of sediment initially settles in the deep channel entering the Wash, the bay located between Skegness and Hunstanton.

C. Sediment exiting the domain

The spatial distribution of volume exiting the domain in one year is shown by the curves along the seaward boundaries of the domain in Fig. 7. The relative magnitude is represented by the normal to the boundary. Results are presented only for the Soulsby-van Rijn formulation. The sediment leaves the domain through the western part of the northern boundary and through the southern boundary.

Figure 7. Spatial distribution of sediment volume exiting the domain after one year shown by thick curves with relative magnitude normal to the domain boundary, using Soulsby–van Rijn formula.

VII. CONCLUSION

Temporal sediment distributions resulting from a volume deposited on a dredging area have been simulated using the TELEMAC-2D / TOMAWAC / SISYPHE software system. A computational domain extending from Lincolnshire to Norfolk has been studied over a time span of about one year. Sediment pathways are shown and volumes deposited along a coastal strip extending about 240m from the low water spring contour have been estimated. Assuming that the bathymetry is otherwise in equilibrium, this enables the sediment pathways between the dredging area and the adjacent coastline to be determined. For the Humber dredging area results indicate that about 15% of the volume extracted offshore does not reach the nearshore strip (as it
would normally in equilibrium conditions) thus reducing beach volumes and thus reducing coastal protection.

ACKNOWLEDGEMENT

This work was funded through the Tyndall Centre for Climate Change Research and the Flood Risk Management Research Consortium. The FRMRC is supported by grant EP/F020511/1 from the Engineering and Physical Sciences Research Council, in partnership with the DEFRA/EA Joint Research Programme on Flood and Coastal Defence, UKWIR, OPW (Ireland) and the Rivers Agency (Northern Ireland). Their financial support is gratefully acknowledged.

REFERENCES

Comparison of sediment transport formulae during a flood wave in the river: an application of Telemac teaching

Caroline GENOUX
ENSE, G-INP, UJF-Grenoble Université
Grenoble, France

Philippe LARROUDE
Laboratoire des Ecoulements Géophysiques et Industriels
UJF-Grenoble Université, Grenoble, France
larroude@hmg.inpg.fr

Lucile POULLEAU
ENSE, G-INP, UJF-Grenoble Université
Grenoble, France

Thomas POCHETAT
ENSE, G-INP, UJF-Grenoble Université
Grenoble, France

Abstract—For twenty years, the state pays attention to the risks of landslides: they could create a natural dam that would obstruct the bed of a river in the Alpine valley, creating a reservoir upstream. Originally, the assumption was that the accumulation of water would eventually cause a dam failure, creating a wave of destructive flooding and widespread, could happen. More recently, studies have shown that this scenario was not realistic: in fact, the water flow is gradually eroding the dam, creating intermittent flows downstream. In 2008, the CNR (Compagnie Nationale du Rhone [7]) conducted different tests on a physical model. The overall objective of our work is the study of some of these tests and comparing the experimental results of the CNR and our morphodynamics simulation of the river bed evolution. We make comparisons between different sediment transport formulations (Rickenmann [4], Einstein-Brown [1], Grass [2], other ...) based on the study of Recking [3] to see their influence on the erosion evolution of the river bed.

I. INTRODUCTION

The Telemac chain of code is use since several years to teach the numerical approach in a project. The project (propose to the student) is during 10 weeks in the second year of the engineering school ENSE². The student work only a half day a week on the subject. The idea of this practical lecture is to teach to them how conduct a project with a numerical approach.

During the two last years, we propose to our student a comparison between numerical simulation with Telemac and data from a physical model. The experimental took place at the CNR in Lyon and the model represent a study on a possible creation of a natural dam.

The study site is located in the department of Isere. Since several years, the State gives a very detailed attention to these risks of falls of ground: they could create a natural dam which would block the bed of Romanche, creating a water reserve to the upstream. With the origin, the assumption was that the accumulation of water would end up causing the rupture of the dam, thus creating a wave of destroying immersion; more recently, studies showed that this scenario was not realistic: in fact, the dam would be eroded gradually, creating intermittent overflow rates in the downstream part of the valley.

These modifications led to the following formulation of the subject of this workshop of engineering: “Study of progressive erosion and sedimentary transport on the top of the natural dam of Séchilienne”.

In 2008, CNR (National Company of the Rhone) carried out various tests on a small-scale model of Séchilienne, in particular two tests numbered 17 and 21 (see [7]).

The objective general of our work is the study of test 21 and the comparison between the results and the data of CNR obtained with their model. We will carry out comparisons between various existing formulas of erosion and the data.

II. LOCALIZATION AND PRESENTATION

Séchilienne is a commune of Isere, localized at 30 kilometers in the south-east of Grenoble, on the road connecting Grenoble and Briançon. It belongs to the canton of Vizille and is located in the valley of the Romanche river.

Séchilienne is known for its active zone of landslide located on the Southern slope of right bank of the valley of the Romanche, at the southern end of the mountain chain of Belledonne (see Fig. 1).

This site, called “Ruins of Séchilienne”, knew many falls of blocks, in particular during years 1726, 1762, 1794, 1833 and 1906. The scree cone of the Ruins is visible from the secondary road, which serves Bourg d’Oisans.
III. PRESENTATION OF THE STUDY

A. Presentation of former studies

As indicated in the introduction, the scenarios considered up to now were very pessimistic. At the time of a rainy event of strong intensity, an important landslide of the mountain would lead to the complete obstruction of the valley and the creation of a dam of several tens of meters. That would lead to the formation of a lake of approximately 20 million m$^3$ to the upstream and the road would be cut. According to former studies, this dam will resist at the rising waters due to flooding and would be broken only when passing the peak of the hydrograph, i.e. at the most critical moment. This rupture would involve an overflow rate with the downstream, forming a wave of immersion which would reach the cities and industrial facilities located downstream from the dam. All these scenarios were deduced from digital models. On these various conclusions, several parades were installing. Many measuring apparatus were also installed in order to monitor closely the evolution of the phenomenon (contours exact of the zone of landslide, kinematics and movement of the zone…). These measurements make it possible to generate alarms in real time.

B. General presentation of the study of CNR

These data were use by CNR, which enabled it to make a more thorough study of this landslide through the construction of a small-scale physical model. The purpose of this model, on the scale 1/60e, was to study the speed of erosion of a dam by overflow of water of Romanche and to provide complementary data (flow in downstream, flow upstream, dimension of water on the level of reserve, quantity of transported materials …)

This scientific tool made it possible to model a very complicated physical process that the mathematical models could not entirely represent. Several scenarios were considered according to the characteristics below:

- The height of the dam formed by the landslide of the 3 million m$^3$;
- The materials (size, nature) constituting the dam;
- Flow of Romanche;
- Test of a second material constituting the dan different from the first testing;
- Tests relating to a 18 m height dam and to a second landslide falling into water reserve formed by a first landslide of 6 m height.

This study by the CNR on this reduced model gives the behavior of the dam formed in bottom of valley of Romanche. The principal conclusions are as follows:

- A 6 m height dam, corresponding to the landslide of a volume of 3 million cubic meters (able to occur in the 10 next years) does not present a risk of brutal rupture because of a progressive erosion. Moreover, a simultaneous hundred-year flood would increase the downstream flow only of 10%.
- A second landslide falling into water reserve formed by a first landslide does not present a real risk, neither for the downstream, nor for the upstream.
- A long term landslide (in 50 years) forming a 18 m height dam for a volume from 5 to 6 million cubic meters would present a more important risk for the downstream with an increase of the flow in the case of a hundred-year flood, in the order of 20%.

C. Presentation of our study

Thanks to the data of CNR [7], we could recover the bathymetry of Romanche. We tried to set up a digital model which will make it possible to have the results of simulation to compare with thus obtained on the small-scale physical model. We had to model the different flood that might occur, the various transformations undergone by the bottom of the river (erosion, “un-paving” …). This will calculate the over-flow in each case and to allow predictions on new models. We have two tests on the site of Séchilienne made by CNR: test 17 which was treated last year and testing 21, we will fully address this year. Both tests were carried out for a dam of 6 m above the bed of the Romanche. The same size of material was used for both tests. The results of last year on over-rates are consistent with the observations of the CNR, so we considered that the hydraulic model was correct. We have therefore chosen to focus primarily on modeling the erosion of the river bottom.

1) Test 17: During test 17, the landslide occurs out flood of the Romanche River, forming a dam in the Romanche valley. The flow of Romanche gradually fills the reserve created by the dam and flows on this and then a flood occurs later. In this scenario, the passage of a hundred-year flood which was studied. The students of last year already modeled this entire test.

We initially tried to improve modeling the un-paving and erosion by using zones polygonal rather than rectangular
(what was used previously). Then, we modeled erosion thanks to various formulas to try to take into account the phenomenon observed by CNR.

2) Test 21: During test 21, the landslide occurs for one period of the flood. In the first time, we thought that the test of CNR consisted in making fall the landslide at a given time at the maximum of the flood. However, after having looked at the provided documents more attentively, we realized that CNR used same bathymetry as in test 17.

In this case, the landslide is established in the model (what corresponds to the bathymetry of case 17), then a constant flow (corresponding to the maximum of flood) is introduced very quickly into the model. In this scenario, it is the maximum flow of the hundred-year flood which was studied. We thus modeled this scenario, by using various formulas of erosion as for test 17 (because we have same granulometry). We could thus compare our results with those of CNR. Bathymetry being identical to the preceding case, the study of this test especially allowed us to model erosion more easily.

IV. STANDARD MODELING FOR THIS PROJECT

The system TELEMAC is a whole of software of numerical simulation applied to the flows on free face, at sea or in river, in two or three dimensions.

The applicability goes from the local study of impact of the construction of works (bridges, ears, mole…) until the calculation of the current due to the waves or tides while passing by the reproduction of the flood, rupture of dams and the transport of sediments.

All the software of the system uses powerful algorithms based on the finite element method. The field of calculation is discretized with not structured grids using triangular elements, which makes it possible in particular to refine the grid in the zones of particular interest. System TELEMAC proposes a complete data processing sequence with software of simulation, “preprocessors” and “post-processors”.

The pre and post-processors are the elements of the chain which make it possible to prepare and manage a calculation, to display the results. These tools are common to all the modules of calculation, which ensures the homogeneity of the unit.

After having traced contours external and the interior lines of the field, it is necessary to define the size of the mesh. We choose here to create constant meshes. During the project, we tested several sizes of grids: 20m, 10m, 5m, 2m in the aim of evaluating the influence of the grid in calculations. We could observe that the precision of calculation increased when the mesh decreased. However, the computing times then were increased considerably. We thus found a compromise by choosing for the majority of the cases the grid of 10m.

For our study, the border upstream is with flow imposed and the border downstream on imposed height. For the side borders, they correspond to the banks and are thus regarded as solid walls.

Fig. 2 shows the hydrograph of the hundred-year flood used by CNR and that which we used in our simulations.

After having controlled the result, we use Blue Kenue to treat the data so extracting information to be analyzed and compare with the results of CNR. Thus, we could extract from the profiles height of water and bathymetry at various moments with the same sections transversely as those used by CNR. Fig. 3 represents an example of evolution of the bottom in function of time on a transversely given section obtained with Blue Kenue.

[Graphs and images as described in the text]

V. MODELING EROSION

Sediment transport can be considered as a matter production, which will be moved by the river and then sediment again: it is also called transit sediment or sediment transport. This is a complex phenomenon involving a large number of meteorological, hydrological, geological parameters, etc.

There are two types of moving materials:

- The bed load, for materials of large size, which corresponds to a transport on the bed in shifts.
- Suspension, that is to say, the finer sediment transport by the flow.
This is increased by the force of floods, the slope of the river bed and the narrowness of the channel. There are two main phases in the sediment transport: the phase of erosion and the deposition phase. For our study we only studied the phenomenon of erosion because it is the cause of the over-flow downstream (see [5] and [6]).

In our project, we are dealing with fluvial erosion, and more precisely to regressive erosion, observed by the CNR in the various tests. Regressive erosion is a mechanism for widening stream that comes after the lowering of the bed downstream. It begins with the digging at baseline also called fixed point before heading upstream (the base level of a river corresponds to the lowest level at which a river can erode bed.).

Regressive erosion is a mechanism for digging of rivers that comes after the lowering of the bed downstream. It begins with downstream at the basic level also called fixed point before rising towards upstream (the basic level of a river corresponds to the lowest level to which a river can erode bed.). This type of erosion is much faster than the traditional erosion which, it, acts in the downward direction.

This phenomenon occurs when the slope is higher than the slope of balance. This increase in slope led to an increase in the erosive power of the river: the transport capacity becomes higher than the contributions, this difference involving an erosion of the bed and banks.

Erosion is propagated then upstream to restore the initial slope of balance. In our case, the landslide is done on only one side of Romanche, which generates a nonsymmetrical deposit. An angular part called “breach” or “gap” by CNR is formed, and it is on this level that the erosion occurs mainly, which digs the dam starting from this point.

Many formulas are proposed to model solid transport by bed load in the rivers. They use various variables, in particular data concerning the granulometry, the width of the bed, the hydraulic diameter and the slope. In this case, it was not found yet of sedimentary formula of sediment transport and erosion compatible with the bathymetry (strong slope).

Within the framework of our study with the Sisyphe code, we used several different formulas, in order to compare the numerical results with the data and to find the most adapted one:

- The formula of Bijker is normally used in the cases of bed load and suspension. It is made of two components: bed load, which is entirely empirical, and suspension, which takes as a starting point the preceding component, and which is drawn from the theory of Einstein.

- The formula of Rickenmann (see [4]) which makes it possible to obtain the bed load starting from the liquid flow of the river.

- Einstein-Brown formula [1].

- Grass formula [2]. It is build by using the morphodynamic equation of Exner.

- Empirical formula.

In addition to these theoretical formulas, we tried to propose our own formula in order to represent erosion. We tried various simple formulas, by choosing relations of proportionality between erosion and the flow per linear meter, since the flow is one of the principal parameters influencing erosion. We thus programmed relations of the form:

\[ E = Aq^n \]  \hspace{1cm} (1)

where \( A \) is a constant and \( n \) varying from 1 to 3. \( E \) represents the sea bed evolution at each time step. For \( n=3 \), erosion observed was really too strong, even with low values of \( A \). Finally, we chose \( A=1\times10^{-7} \) and \( n=1 \), which we integrated in a standard file FORTRAN for our simulations only with Telemac2d module (without the coupling with Sisyphe module).

In our study cases, we chose to look at the evolution of the bed on 3 profiles, corresponding to profiles 15, 16 and 17 of the study of CNR (see Fig. 4).

In addition, we observe some divergence of our results compared to the experimental result. For the curves representing the time above 17h, there is too much erosion, the theoretical result moves away significantly from the experimental result.

A. Comparison with test 17 data (hundred-year flood)

For this modeling, the tests are made during three days of real time in order to make entirely the hydrograph hundred-year flood. We did not model the “un-paving” (abrupt setbacks of part of the dam) observed by CNR, because we observe some divergence of our results compared to the observations on the small-scale model. On the grid of 10m, we need approximately 18 hours to carry out the simulation on a PC. All the profiles show for the results are taken looking in the upstream direction. The profiles of CNR’s data are truncated to facilitate the comparison.

1) Profile 15: with Bijker (see Fig. 6), until \( t = 2h \), we see that our curves are smoothed in contrast to those of the CNR (see Fig. 5). However, the numerical values coincide. For the curves representing the time above 17h, there is too much erosion, the theoretical result moves away significantly from the experimental result.
With the empirical formulae (see Fig. 8), the shapes of the curves are quite consistent with those of the CNR, but erosion is not large enough.

2) Profile 16: With Bijker (see Fig. 10), the results for short times are not very good (see Fig. 9): the shape is moderate and the erosion is not large enough. However, the profile obtained for the curve \( t = 18h \) is closer to the data, but the following simulations are incorrect (the erosion is too great).

With Einstein formula (see Fig. 7), we obtain similar results to those obtained for Bijker, except that the results are good up to \( t = 5h40 \). There is still a significant erosion of the profiles at the final time.

With Einstein (see Fig. 11), the results at short times are not correct. On the other hand at very long times (\( t = 29h \)), the behavior and the values are correct.
With the empirical formula (see Fig. 12), the behavior is correct for all times, but on the other hand the erosion is for each time step too weak.

3) **Profile 17**: with Bijker (see Fig. 14), the behavior is quite correct. In contrast the values of erosion are not very good at long times (about one meter gap between simulate and experimental test (see Fig. 13)).

With Einstein (see Fig. 15), the curves at short times are incoherent because we observe identical profile from \( t = 2h \) to \( t = 18h \). On the other hand, the shape of the last curve is good, but erosion is too weak. With the empirical formula (see Fig. 16): The results are wrong. The shape is correct but the erosion is quasi non-existent.

**B. Comparison with test 21 data (ten-year flood)**

For this modeling, the tests are made during 15h to simulate the maximum of the ten-year flood \( (Q = 300m^3/s) \). We either did not model the “un-paving” observed by CNR. On the grid of 10m, we need approximately 6 hours to carry out the simulation on a PC.

1) **Profile 15**: with Bijker (see Fig. 17), the results are good for very short times. On the other hand, it seems according to the figure of CNR that an “un-paving” occurs after 30mn so our results after thi time are wrong because of the gap create by the “un-paving” on the river bed. With a translation of our curves, the shape seems to be correct.

With Einstein (see Fig. 17), the depths of erosion are correct along the entire simulation in time. On the other hand, the digging of the channel is done over a width much more reduced than in the results of CNR. That can be due to that the “un-paving” occurred after 30mn changes the shape of the breach in the dam.

With Grass formula (see Fig. 17): There still, the phenomenon of “un-paving” is not modeled, which distorts interpretation. If not, the profiles are overall good, with a broader channel which corresponds well to the results provided by CNR.

1) **Profile 16**: with Bijker (see Fig. 18), the depths of erosion are not good. But temporal data of the CNR are missing on their graph, so the interpretations are less easy. The shape is relatively correct but the erosion is too weak (only 2m in 15 hours).

With Einstein (see Fig. 18), the shapes and the values are good. But, we observe on the profile of CNR at \( t = 11h40 \) an
increase of the river bed elevation, which can correspond to a deposit of sediment which is difficult to simulate.

With Grass model (see Fig. 18), the results are not very realistic. Erosion is progressive but does not correspond at all to the results observed on the small-scale model.

Figure 17. Test Case 21: profile 15, the CNR data and simulated data with Bijker, Einstein and Grass formulae.

3) Profile 17: with Bijker and Grass models, erosion is nonexistent in this part upstream of the dam, which is in clear disagreement with experimental data from the CNR. With Einstein, there is an erosion that has generally the same shape as the experimental profiles, but the values are not very good (too much erosion before \( t = 6h \) and not enough after).

Figure 18. Test Case 21: profile 15, the CNR data and simulated data with Bijker, Einstein and Grass formulae.
VII. CONCLUSIONS

This project of Workshop of Engineering on the topic of the Ruins of Séchilienne allowed us to extend our knowledge as regards numerical modeling of flow in river and to control the bases of the software Matisse, Telemac and Bluekenue. We are now able, starting from data of bathymetry of a river bed and hydrological data, to model a flood and to extract from the results information necessary to their good comprehension.

This study was the occasion for us to measure the difficulty in following objectives: many data-processing problems forced us to reorganize the aim to achieve. In addition, of new elements of comprehension reached us during the project and led us to adapt our study.

For test 17, the simulation which approaches more the tests carried out by CNR is the one using the formula of Bijker. In addition, the phenomenon of “un-paving” is not taken into account in our study, which can explain why erosions obtained numerically are often too weak. Our empirical formulae seem correct but remain to be improved. It would be necessary to continue to make evolve/move the coefficients to obtain erosion a little more important and to better follow the results of CNR.

For test 21, the simulation which approaches more of the tests carried out by CNR is the one using the formula of Einstein.

We decided this year to look further into the modeling of erosion; we hope that our results will make it possible to make progress for the study in the future projects of Workshop of Engineering. Modeling must still be improved, in particular with the “un-paving” phenomenon.

ACKNOWLEDGEMENT

Thank to the CNR and Sylvain Reynaud for the data. We also acknowledge Daniel Verdel and Olivier Manin from the Symbhi who authorized the CNR to give us the data and the results of the experiment. This work is the continuation of the work of the previous year by Amoroso Lauranne, Lambert Marine, De Bortoli Romain and Saidon Simon.

REFERENCES

Numerical modelling of bed formes (dunes) with TELEMAC3D and SISYPHE

Annalena GOLL, Rebekka KOPMANN and Thomas BRUDY-ZIPPELIUS
Hydraulic Engineering Department
Federal Waterways Engineering and Research Institute (BAW)
Karlsruhe, Germany
Annalena.Goll@baw.de

Abstract— This paper presents simulations of dunes in an open channel flume with TELEMAC3D and SISYPHE. The simulations are compared to results of a laboratory flume situated at the BAW (German Federal Waterways Engineering and Research Institute), Karlsruhe, where three-dimensional sand dunes were produced and studied. The behaviour of the simulated dunes, their origin and movement has been studied. It shows that successful dune modelling can be done by using a calibrated parameter set. The distribution of the vertical layers has proved to be a crucial parameter when simulating dunes with TELEMAC3D and SISYPHE. Very fine spacing near the bottom is needed, scaling with the grain size of the sediment or the friction coefficient respectively. The distribution of bed shear stress influencing the bed forms is changed by the choice of number and spacing of the vertical layers. So far the produced bed forms move too fast and the bed load discharge is too high, even with a reduced pre-factor of the bed load transport formula. As this research is ongoing further insights are expected.

I. INTRODUCTION

Recently there has been an immense advance in seizing the dynamics that lead to and govern the complex processes of river dune morphology. New measuring instruments allowed better and thorough experimental studies, e.g. [1], [2], [3] and [4]. High performance computer made detailed simulations possible, e.g. [5], [6], [7], [8] and [9]. High resolution simulations enabled to investigate long known phenomena like flow separation behind dunes [10], [11], and to give a realistic picture of dune movements [12], [9].

But still there remain questions concerning the challenge to adequately connect bed morphologies to the turbulent flow field and to find proper dune dimension definitions [13]. For example it has been shown that the form of the dunes effects the amount of turbulence of the flow [1], [14]. On the other hand there have been studies which name the choice of the turbulence model ambiguous concerning the morphodynamic computation [6]. Studies that try to evaluate computational models that are used to simulate the turbulent flows which govern the dune movement process found significant differences between the models [11].

Even so the achievements named above are immense it is so far not possible to predict dune movements for large river sections or forecast long-term developments. In general the aim therefore has to be to use data obtained from experimental flumes to calibrate models that can be used for large scale simulations and project work.

This paper presents calculations of morphodynamic and flow of a open channel laboratory flume situated at the BAW, Karlsruhe, where three-dimensional sand dunes were produced and studied. Morphodynamic simulations of the dune bed with TELEMAC3D and SISYPHE are compared to the flume data obtained by photogrammetry. This work is supposed to be a first step on the way to fit and validate the models and find parameter sets that can be used for long term and large scale predictions of dune movements.

II. EXPERIMENTAL SETUP

A. Experimental Flume

The experimental flume is situated at the German Federal Waterways Engineering and Research Institute (BAW), Karlsruhe. Details can be taken from [15].

The experimental open channel flume is 32m long and 2m wide. It has a rectangular cross section with one side-wall of glass with metal bars and one side-wall of plastic material. The bottom is constantly covered with sediment which has a steep grading curve (with mean diameters 1% 0.4275mm, 3% 0.605mm, 64% 0.855mm, 32% 1.2mm) and a total mean diameter of 0.94mm. The slope of the bed is 0.6‰. The water depth is regulated by a flow-governed weir at the outlet. For the presented calculations only runs with \( Q = 140 \text{ l/s} \) and corresponding sediment input of 37 kg/h are considered. The water depth at the outlet is then 0.141m and the mean water depth is 0.175m.

B. Measuring Device

The following description of the measuring device is taken from [16]. For further details please refer to this paper.

The used measurement system has been developed by the BAW. It is a laboratory based three-dimensional photogrammetric measurement system for the monitoring of alluvial bed topography allowing for repeated, instantaneous recording of dune beds during water flow. It was initially developed to measure dry channel surfaces but has been further developed for subaqueous surfaces. The system consists of an automated camera orientation unit and is based on bar coded markers and...
a grid projection unit for identification of bed topography. Achievable vertical accuracies for bed elevation measurements are 1 mm for subaqueous measurements and 0.1 mm for dry bed model measurements.

The advantage over existing measuring devices is the possibility to record continuously and instantaneously through the water surface, instead of emptying the flume to be able to record the dry bed topography. For the presented flume results a picture was taken every 20s of a selected area of the flume. After each run of 6h the complete flume was recorded as well.

III. NUMERICAL MODEL

For the numerical computation, TELEMAC3D and SISYPHE (version v6p0) were used (for a detailed model description, please refer to http://www.opentelemac.org and http://docs.opentelemac.org).

The computational grid spans the complete experimental flume except the starting and ending part (flume meter 2 until 29.97) in x-direction and over the full 2m in y-direction. The mean horizontal mesh size of the unstructured sigma-mesh is 15cm. The vertical discretisation was subject to calibration and thus several discretisations were used: 10 respectively 20 vertical layers were unevenly distributed with fine spacing near the bed, coarsening towards the free surface according to the log-law. Additionally an extremely fine bottommost layer was added. The mean water depth of the flume is 0.175m.

At the inlet constant discharge and no sediment is given at the boundary. The sediment input is realised with the dredging and disposal module DredgeSim [17] coupled to SISYPHE to reproduce the conditions of the physical model, where the sediment input is alternated every hour from the left to the right hand side on the first half square meter of the flume input. Like in the experimental flume the water level at the outlet is kept constant.

Both pure hydrodynamic and coupled hydrodynamic and morphodynamic simulations were done. For the hydrodynamic computations without morphodynamics a preformed dune bed is chosen. Therefore a fully developed dune bed morphology is taken from photogrammetric measurements of the flume after 6h (see Fig. 1). For coupled morphodynamic simulations the calculations start from a plane bed.

A. Numerical Configuration of TELEMAC3D

Several configurations were tested [18]. As the turbulence modelling is of major importance when simulating bed forms, the choice of the turbulence model (and corresponding parameters) has been given special attention. The best results could be found with the mixing length model of Prandtl as the vertical turbulence model and the Smagorinsky model as the horizontal one.

The molecular viscosities (coefficients for diffusion of velocities) were set to $10^{-4}$ m$^2$/s to stable the simulations, as it seems that in case of flume experiments the turbulence models create too less turbulent viscosity. The scheme for advection of velocities is the explicit scheme with SUPG (Streamline Upwind Petrov Galerkin) in connection with the use of the wave equation.

B. Numerical Configuration of SISYPHE

The bed load transport formula was Meyer-Peter & Müller [19] with a pre-factor of 3. The slope effect formula of Soulsby [20] and the deviation formula of Talmon [21] were used, using parameter for deviation = 2. Bed load sediment was a four fraction grain mixture (with mean diameters 1% 0.4275mm, 3% 0.605mm, 64% 0.855mm, 32% 1.2mm) that yielded an overall mean diameter of 0.96mm. The friction coefficient was chosen according to $3D_{so}$ to 2.82mm.

IV. RESULTS

Calibration and numerous test simulations have shown that dune modelling with TELEMAC3D and SISYPHE is very sensitive to the distribution of the vertical layers. With uniform distribution no dune formation or movement could be obtained. Only with unevenly distributed vertical layers which have fine spacing near the bed and coarse towards the free surface according to the log-law a dune-like formation could be produced. This experience has been stated before [6]. It has also shown that the correct dune height and the formation of dunes in general further depend on the choice of the distance of the first vertical layer.

This becomes obvious as the bed shear stress $\tau$ in SISYPHE is calculated from the velocities of first node above the bed surface according to:

$$\tau = \rho u^2 = \rho \left[ \frac{\kappa}{\log(30\Delta/k)} \right] u^2 + v^2$$ \hspace{1cm} (1)

where $\Delta$ is the distance between the bed and the first vertical layer. This is why the bed shear stress depends strongly on the ratio of this distance and the friction coefficient $k$.

Fig. 2 shows the flume bottom of 4 runs with different vertical layer distributions. The first 6–8m of each flume are influenced by the bed load input. Only two runs (10 layer fine. 20 layer fine) yield a dune formation with comparable to the dune heights of the physical model. Both runs have a very small first vertical layer (0.00138 times the water depth). The figure shows that a bigger first vertical layer results in a less dominant dune formation.

Simulations over a preformed dune bed (Fig. 1) show the different bed shear stress that is produced due to the different vertical layer distributions (Fig. 3). Four different dunes of the preformed bed and the corresponding bed shear stress can be seen. The two vertical distributions where the first bottommost layers were refined (_fine) yield a bigger amount of bed shear stress. It seems that a higher bed shear stress is needed to form a distinctive dune bed. Decisive is the fact that not the simulation with the by trend highest bed shear stress (10layer_fine) produces the most similar bed forms to the flume experiment, but the run with slightly less bed shear stress (20layer_fine) (see Fig. 2 as well). The 20–layer run has only selective peaks of higher bed shear stress. Obviously dune formation and dune shape in TELEMAC3D and SISYPHE not only depend on the amount of produced shear stress but also on its distribution (Figs. 3a) and 3b). This coincides well with existing research, e.g. [22], [23], [24] and [25], [26], [27], [28].

The height, length and the movement of the dunes in the physical model has been compared to the movement of the...
The questions remains why the distribution of the vertical layers has such an immense influence on the bed forms and the bottom shear stress. Over the course of a dune the velocity profiles are stretched and compressed and are no longer logarithmic [25]. The calculation of the shear stress in TELEMAC3D follows Eq. (1). This logarithmic equation should provide the same bottom shear stress no matter where the velocity is extracted, but only if the velocity profile is logarithmic. As this is not the case over the dunes, it becomes crucial where on the profile the effective velocity is extracted. An additional finer discretisation further changes the course of the profile, as the profile is reproduced in more detail. As the extraction point turns out to be decisive in case of non-logarithmic profiles, the identification of this point with the actual correct velocity is of major interest. The presented results confirm this.

Even though an adequately simulation of dune height and length could be done the produced bed forms move too fast and the bed load discharge is too high. The reduction up to a very low value of the pre-factor in the Meyer-Peter & Müller bed load transport formula did not solve that problem. It can be reasoned that the dune movement is not bound strictly to the right dune geometry, at least in the numerical model. This is why calibration parameters must be found which affect only the dune movement but not the geometric dimensions. More investigation is needed to understand the connection between the different calibration parameters in order to get an optimal parameter set.

VI. OUTLOOK

Further investigations are needed to extract a case independent parameter set for dune origination and movement. Especially a rule for a proper distribution of vertical layers would be a big step in direction of dune prediction. As this research is ongoing further insights are expected.

So far the numerical model only considers bed load transport. This assumption was made as dune movement mainly happens through bed load transport, not in suspension. Observations of the physical model show that part of the transport is happening through bed load that is thrown into the flow by turbulence “bursts” hitting the dune. This is a long known effect that has been studied intensively before, e.g. [29], [30], [31] and [32]. Even so this transport method is the minor one, the disregard of it might be crucial. The necessity of a proper choice of a correct bed load transport model in connection with a matching turbulence model becomes obvious. Further research in this direction is in progress.

Moreover it is necessary to understand the complex processes of dune movement to successfully simulate dune movement with numerical models. Through the close cooperation between the research of the flume experiment and the numerical simulations at the BAW a very fruitful exchange of knowledge, understanding and discussion is possible. One next step are detailed measurements of the flow field above fixed (and prior to this naturally formed) dunes. This will help to specify the validation of the hydrodynamic part of the simulations which is essential for a successful morphodynamic simulation.
ACKNOWLEDGEMENT

We thank Martin Henning for fruitful discussions and for providing the data of the flume experiments.

REFERENCES


![Figure 1. Pre-formed bed obtained from photogrammetric measurements of the flume after 6h, used for numerical simulations.](image-url)
Figure 2. Calculated bed forms after 6h with different vertical layer distributions, start from level bed.

Figure 3. Simulated bed shear stress distributions over 4 different dunes of the pre-formed bed (Fig.1) for different vertical layer distributions.
Figure 4. a) Dune movement in the physical model. b) Dune movement in the numerical model after 5 (blue), 10 (red), 15 (green) and 20 minutes (black).

Figure 5. Comparison of measured (crosses) and simulated (square and triangle) a) dune length and b) dune heights.
Modelling sediment transport with hysteresis effects

Michiel A. F. KNAAPEN  
Coasts & Estuaries Group  
HR Wallingford  
Wallingford, UK  
m.knaapen@hrwallingford.com

David M. KELLY  
Coasts & Estuaries Group  
HR Wallingford  
Wallingford, UK  
d.kelly@hrwallingford.com

Abstract—This paper details the extension of the sediment transport and morphology model SISYPHE 2D to include a lag term within the bed exchange source term of the, depth-averaged, continuity of sediment concentration equation. This lag term represents the time it takes for a sediment concentration profile to adapt to spatial or temporal changes in the flow. The inclusion of a lag term means that the settling velocity is no longer the only scaling factor for the exchange of sediment between the water and the bed. The newly modified SISYPHE 2D is tested against field data from the Thames estuary (UK), flume experiments on a dredged trench and a hypothetical channel widening. It is illustrated that the lag factor introduced into SISYPHE 2D is essential to model the sediment transport and morphodynamics, especially when considering engineered situations, where the bed is out of equilibrium with the flow conditions. Moreover, with this lag factor included, there is evidence that SISYPHE 2D can be used for (short term) morphodynamic modeling of engineered situations.

I. INTRODUCTION

In the sediment transport and morphology model SISYPHE 2D, erosion deposition mechanism assumes equilibrium conditions; it assumes that the sediment concentration profile in the vertical instantaneously adapts to any spatial or temporal variations in the flow. This means that the sediment exchange rate between the bed and the water column is governed by the difference between the amount of sediment in the water column and the equilibrium sediment concentration, scaled solely by the settling velocity of the sediment under consideration. This assumption of an instantaneous response of the sediment concentration profile to variations in the flow is invalid for a large range of sediments, but especially for fine grained sediments.

In reality, the due to inertia effects, it takes time for the sediment concentration profile to adjust to the new flow velocity. The actual sediment concentration profile differs from the equilibrium sediment concentration profile. This introduces a hysteresis effect in the sediment concentrations during a tidal cycle. For equal flow conditions, the observed sediment concentrations are higher in a decelerating flow than in an accelerating flow.

This difference between actual and equilibrium concentrations creates a lag effect: the actual exchange between the bed and the water column is lower than predicted using the assumption of equilibrium conditions. This also implies that the bed changes occur more slowly than is predicted assuming equilibrium conditions. The errors made using the assumption of equilibrium sediment concentrations are most apparent in the case of engineering problems, which often introduce rapid changes in the local flow velocities.

In 1981, Miles [1] derived a solution of the 1D suspended sediment concentration equation, taking inertia effect of the sediment into account, utilizing the bottom boundary conditions proposed by Lean [2]. This paper reports on the introduction of this lag effect in the sediment transport to SISYPHE 2D in order to parameterize the effect of settling lag on suspended sediment concentration and the associated morphological evolution based on Miles’ work [1].

The paper is split into four distinct sections. In section II we discuss the sediment concentration equation in SISYPHE 2D with particular attention given over to the bed exchange source term including the lag effect. Section III introduces a saturated reference concentration based on the suspended load transport predictor of Soulsby-van Rijn [3]. In section IV, we discuss the effect of the lag term for three test cases including channel widening (A), a comparison of model results with field data from the of the outer Thames estuary in the UK (B) and a morphodynamic test against flume experiments of trench infill (C). Finally, in section IV we draw conclusions and suggest potential extensions to improve the realism of sediment transport within SISYPHE 2D.

II. CONCENTRATION EQUATION WITH LAG EFFECTS

In order to compute the time evolution of suspended sediment concentration SISYPHE 2D solves the primitive variable form of the 2D transport equation, i.e.:

\[
\frac{\partial C}{\partial t} + \left[ \left( \frac{u_{\text{con}}}{h} \right) \frac{\partial C}{\partial x} + \left( \frac{v_{\text{con}}}{h} \right) \frac{\partial C}{\partial y} \right] = F(C) + \frac{1}{h} \left[ \frac{\partial}{\partial x} \left( \frac{h c}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{h c}{\partial y} \right) + \beta_s \left( C - C_s \right) \right] \tag{1}
\]

where \( C \) and \( C_s \) are the depth-averaged concentration and depth-averaged saturated concentration respectively, \( h \) is water depth. The convection velocities \( u_{\text{con}} \) and \( v_{\text{con}} \) are obtained by multiplying the depth-averaged flow velocities by a convection factor \( (F_{\text{con}} < 1) \), \( c \) is a dispersion coefficient and \( F \) is a scaling factor that includes the fall velocity and a profile parameter relating depth averaged and reference level concentrations:

\[
\beta_s = C \tilde{C}^{-1}
\]
According to the accepted theory of sediment suspension turbulence opposes gravity and ensures that the sediment is distributed vertically throughout the water column. The continuity of sediment concentration equation in one, vertical, dimension is

$$\frac{\partial C}{\partial t} = W_i \frac{\partial C}{\partial z} + \frac{\partial}{\partial z} \left( D_s \frac{\partial C}{\partial z} \right) \quad (2)$$

where \( W_i \) is fall velocity. The vertical diffusivity \( D_s \) is approximated from the horizontal parabolic eddy viscosity assuming a logarithmic velocity profile, which describes the time rate of change of sediment in the vertical, \( z \), direction for uniform flow conditions [4]:

$$D_s = \frac{\nu}{\kappa u^*}$$

where \( u^* \) is the friction velocity and \( \kappa = 0.4 \) is von Karman’s constant. Solutions to equation (2) can be found by employing suitable boundary conditions. The free surface boundary is trivially defined as, at the free surface, there must be zero flux of sediment. At the bed there are a number of options and various assumptions have been made to describe the exchange of sediment between the water and the bed. Mei [4] assumes that the concentration at the bed responds instantaneously to changes in the flow; such an assumption is, however, unrealistic as it requires that the rate of exchange of sediment is infinite at some initial time. Lean [2] argued that it is the sediment entrainment rate that responds most rapidly to changes in the flow leading to bottom boundary conditions that can be expressed mathematically as

$$D_s \frac{\partial C_s}{\partial z} = 0 \quad (3)$$

where the saturated concentration \( C_s \) is the concentration that is in equilibrium with the flow and fulfills

$$W_i \frac{\partial C}{\partial z} + \frac{\partial}{\partial z} \left( D_s \frac{\partial C}{\partial z} \right) = 0 \quad (4)$$

Boundary condition (3) is physically more plausible than that suggested by Mei [4].

Using the approach suggested by Mei [4], Miles [1] found a similarity solution to equation (2) for the bottom boundary conditions given in Lean [2]. This solution provides an approximate explicit analytical solution for \( C \). Using this solution Miles [1] shows that the erosion deposition source term can be written as

$$W_s (C_s - C_i)_{z=0} = W_i \left[ \left( 1 + 2\tau^2 \right) \text{erfc}(\tau) - \frac{2}{\sqrt{\pi}} \tau e^{-\tau^2} \right]$$

$$\times (C_s - C_i)_{z=0} \quad (5)$$

where \( C_i \) is the initial near bed concentration and

$$\tau = \sqrt{\frac{W_i t}{4D_s}}$$

is a dimensionless, time-like, variable. This modified source term now incorporates a scaling factor that accounts for both the settling velocity and the lag time required for the concentration profile to adjust to changes in the flow.

### III. SATURATED CONCENTRATION

As well as modifying the bed exchange source term we also modified SISYPHE 2D by introducing a suspended load transport formula based on the Soulsby-van Rijn formulation for suspended load only [3]. The (depth-averaged) sediment transport rate, \( q_s \) of Soulsby-van Rijn [3] is converted into a (depth-averaged) saturated concentration under the assumption that \( C_s = q_s (hU)^{-1} \). This is then converted in a saturated concentration at the reference level:

$$C_s = \frac{C_s F}{W_s h U} = \frac{q_s F}{W_s h U} \quad (5)$$

This approach allows the model to calculate the Soulsby-van Rijn transport formula, while treating the suspended transport rates through the concentration equation. The former is useful as the transport formula of Soulsby-van Rijn has been well calibrated and has been shown to predict reliable sediment transport rates under equilibrium conditions [3]. The latter is important as it allows us to use the concentration equation, which should improve the model accuracy when lag effects are important.

### IV. RESULTS AND DISCUSSION

In this section we present the results to three test cases selected in order to illustrate the importance of including a lag term when using SISYPHE 2D for a variety of applications. Both tests 1 and 2 are designed to show the effect of including the lag term in SISYPHE 2D when compared to standard SISYPHE 2D whereas test 3 provides a comparison of simulated results with data observed in the field.

#### A. Channel widening

This first test case was undertaken in order to visualise the impact of the newly introduced lag term on suspended sediment concentrations. A 500 m straight channel is assumed 50 m wide, but with a wider section (70 m) in the middle. The water is 5 m deep along the whole channel. At either end, a constant flow velocity is assumed of about 0.75 m/s from right to left, which reduces to approximately 0.55 m/s in the centre and even lower near the sides. The bed is assumed to be covered with well sorted, uniform fine sand with a 0.1mm median diameter. Due to the flow deceleration, sediment should settle out reducing the suspended concentrations. However, as this settling takes time, there is a gradual decrease in the concentrations. At the contraction further downstream, the concentrations will increase gradually again. To show the initial concentration pattern along the channel, the sediment transport is simulated without bed updates.

With the standard version of SISYPHE 2D, using the Soulsby-van Rijn total load transport formula without the settling and erosion lag, the predicted concentrations show a very sharp drop when the flow velocities reduce due to the channel widening (right half of Fig. 1). Similarly, as one would expect, the re-suspension is almost instantaneous at the contraction (left half of Fig. 1).
When we calculate the same situation using SISYPHE 2D using the lag term introduced in section the sediment concentrations adapt much slower to the spatial changes in the flow velocity imposed by the channel widening and contracting (Fig. 2). There is a distinct sediment plume travelling into the wider section of the channel, and only at the downstream end of this wider section does the sediment concentration reach equilibrium conditions again. Similarly, the simulated concentration after the channel contraction (far left of Fig. 2) does not return to equilibrium conditions before the boundary of the model domain.

**B. Outer Thames estuary**

During 1971 and 1972, HR Wallingford undertook a study of the potential infilling of an approach channel at Foulness in the outer Thames estuary, UK (Fig. 3). Simultaneous sediment concentration and velocity profile data was collected in the deep water channels of the estuary. The data is characteristic of the transport of fine sand by strong tidal flows in deep water without any influence from wave stirring (average wind speed was 5 knots and the wave stirring was negligible at depths of 3–7 m). Refer to [5] for an in-depth summary of the data.

Modified SISYPHE 2D was run employing a simple numerical flume with a horizontal bed. Currents and water depths observed at location FM1 (5.2 m tidal range, 1 m/s maximum ebb current and 1.1 m/s maximum flood current; refer to [5] for more details) were applied uniformly across the channel. Boundary conditions at either end of the flume were provided by forcing saturated sediment concentrations there. Thus, the effect of the non-uniform bottom topography on the flow, present at the data site, is not accounted for. The particle distribution of the sediment collected at FM1 consisted of silt and fine sand. The bed material was shown to consist of fine sand (median grain size 160 µm) with a long tail of fine material. The measured suspended material was finer (median grain size 100 µm). To cover both fine-sand and silt dynamics, a simulation employing a mixed sediment bed (75 µm [33%], 125 µm [33%], 150 µm [29%] and 200 µm [5%]) was run in order to best represent the measured bed composition simply.

Fig. 4 shows the computed depth-averaged saturated and the computed actual concentrations for the fraction with $D_{50} = 150$ µm, as computed in the mixed sediment case. The figure illustrates the lag introduced by modifying the entrainment-deposition source term; one can see that the actual lags behind the saturated concentration in both the entrainment and deposition phases of the tidal cycle. With smaller grain sizes, this lag becomes longer in the deposition phase, but slightly shorter in the erosion phase.
To further investigate the difference in prediction error for the ebb and flood phases, we modelled the transport with separate fractions of 75 µm, 100 µm, and 150 µm.

It was found that the transport rates for the ebb stage are well reproduced using sediment with a 75 µm diameter (Fig. 7), but that these simulations overestimate the transport during the flood phase. In contrast, simulations using sediment with a 100 µm diameter give accurate predictions of the transport during flood, but under-predict transport during the ebb. The simulations using a sediment with a 150 µm diameter under-predict the transport rates in both the ebb and flood phase.

C. Trench infill

The flume experiments carried out at Delft Hydraulics [6] are the third test for the new version of SISYPHE 2D. The experiments were performed in a small flume with a length of 17 m, a width of 0.3 m and a depth of 0.5 m. Sediment was used with $D_{50} = 0.1$ mm and $D_{90} = 0.13$ mm. Sand was supplied at constant rate at upstream section of flume to maintain equilibrium conditions. The channel had side slopes of 1 to 12 and a depth of 0.125 m.

Regular waves with a period of 1.5 s and height of 0.08 m were generated and a steady current following the waves was imposed. The water depth was 0.255 m and the current velocity was 0.18 m/s. The mobile bed consisted of well sorted sediment with 0.1 mm median diameter ($D_{90} = 0.13$ mm) and density 2650 kg/m$^3$. The mean fall velocity of the suspended sediment was 0.07 m/s.

To maintain equilibrium bed conditions away from the channel, 0.0167 kg/s/m sediment was fed into the flume at the inflow boundary. Velocities (acoustic-Doppler) and suspended concentration (siphon system) profiles were measured at five stations near the trench at the initial stage of the experiment, when morphodynamic change was negligible.

Using Soulsby–van Rijn at these scales is impossible, as this formula is invalid for water depths smaller than 1 m. This limitation is circumvented by scaling the experiment up to field dimensions, multiplying the domain lengths by 10 and the time by $\sqrt{10}$. Assuming the morphology is bed-load dominated, the sediment grain size has not been altered. Assuming the morphology is suspended-load dominated, the sediment grain size is also multiplied by 10.
simulation, the time and spatial dimensions are rescaled back to the scales of the flume experiment.

The sediment transport and morphodynamics of the experiment were simulated using the existing option in SISYPHE 2D of Soulsby–van Rijn as a total load predictor (referred to as ‘total load’ option), using the modified Soulsby-van Rijn method described in section III, (referred to as SISYPHE 2D-HR no lag) and the full modified Soulsby–van Rijn including lag (referred to as SISYPHE-HR).

Fig. 8 shows the results of the three options under the assumption that the morphodynamics are dominated by bedload sediment transport. With the lag factor included, modified SISYPHE 2D-HR predicts the location of the trench accurately and reproduces both the slopes correctly, even if the measured upstream slope has progressed slightly further than the modelled one. There are some minor errors, however, as the downstream bed level is eroding slightly (error 6.5 mm) and the infill in the centre is slightly under-predicted (error 6.5 mm; or < 10%).

Both simulations without the lag factor contain considerable errors, even if the total load option predicts the change in depth of the trench quite well. The centre of the channel migrated 4.5 m (total load) too far. The upstream slope of the trench is too steep and the downstream slope too gentle in the total load option. Moreover, boundary issues this option in SISYPHE 2D to deposit significant amounts of sediment upstream of the trench.

SISYPHE 2D-HR no lag over-predicts the infill of the channel and the centre of the channel migrates 5 m (SISYPHE 2D-HR no lag) too far.

![Figure 8](image)

**Figure 8.** Trench profiles measured at the start of the experiment (dashed black) and after 10 hours (solid black) compared to model simulations using the different versions of the Soulsby–van Rijn transport predictor existing in SISYPHE 2D (solid red), SISYPHE 2D-HR without lag effects (dashed red) and SISYPHE 2D-HR (solid blue). The modelling grain size is 0.1 mm.

To check that the assumption of bedload dominance is correct, the models have been run with the scaled grain sizes (e.g. 1mm diameter instead of 0.1 mm). Fig. 9 then shows that all models underestimate the infill rate and slightly overestimate the downstream migration of the trench.

The simulation with the modified version of SISYPHE 2D predicts slightly more infill than both simulations without lag effect. The simulations without lag effects are virtually identical, apart from some boundary effects.

![Figure 9](image)

**Figure 9.** Trench profiles measured at the start of the experiment (dashed black) and after 10 hours (solid black) compared to model simulations using the different versions of Soulsby–van Rijn’s transport predictor existing in SISYPHE 2D (solid red), SISYPHE 2D-HR without lag effects (dashed red) and SISYPHE 2D-HR (solid blue). The modelling grain size is 1 mm.

V. DISCUSSION AND CONCLUDING REMARKS

An algorithm to model sediment transport based on the Soulsby van Rijn transport predictor [3] has been added to SISYPHE 2D. This algorithm converts the formula for the suspended load transport rate of Soulsby–van Rijn into a reference concentration that can then be transported using advection diffusion.

Furthermore, a lag factor for the erosion and deposition rate is developed for situations where the sediment concentrations are not in equilibrium with the flow conditions due to temporal and spatial variations in the flow. This lag factor is based on Miles’ solution of the 1D suspended sediment concentration equation [1], taking inertia effect of the sediment into account, utilizing the bottom boundary conditions proposed by Lean [2].

This new version of SISYPHE 2D has been tested for three distinct test cases. The modified code has been applied to simulate the transport rates measured in the outer Thames estuary UK (Foulness). The transport rates were calculated using a mixed sediment bed with a grain distribution similar to the measured distribution. The simulated transport rates agreed well with observed transport rates during the flood tide, including the hysteresis effects; the concentration differences between accelerating and decelerating tides.

However, during the ebb tide, the transport rates were under-estimated in the simulations. Additional runs with single
grain bed material showed a good agreement between the observed transport rates during ebb if the grain size was 75 µm. This is the observed grain size of the suspended sediment [5].

Based on these results, we conclude that the main differences between the predicted and measured concentrations for the mixed sediment simulation are caused by the simplification of the modelling domain. Where the simulation is using a straight flume with uniform sediment, in reality the Foulness measurement are taken in the Thames.

A possible explanation might be that the suspended transport during the ebb is dominated by the finer material from further up the estuary, but no confirmation for this explanation has been found in the experimental data.

The modified version of SISYPHE 2D is also able to simulate the infill of a trench over time as measured in the flume by van Rijn [6]. Both the location of the trench and the infill rate were estimated accurately (the depth difference was less than 10% of the trench depth). To achieve this accuracy, the lag factor is essential as it enhances the infill rates and reduces the migration of the trench.

In the third test case, where a channel with a widening section in the middle is modelled, the lag effect creates a smooth transition of the sediment concentrations between the wide and the narrow sections.

In conclusion, it has been illustrated that the lag factor introduced into SISYPHE 2D is essential to model both sediment transport and morphodynamics. Moreover, with this lag factor included, there is evidence that SISYPHE 2D can be used for (short term) morphodynamic modelling of engineered situations, where the bed is out of equilibrium with the local flow conditions.

REFERENCES


Comparison of sediment transport formulae with simulation of several storms on a Mediterranean beach and with in-situ sedimentary flux on a North Sea beach

Mehdi DAOU, Philippe LARROUDE
Laboratoire des Ecoulements Géophysiques et Industriels
UJF-Grenoble Université, Grenoble, France
larroude@hmg.inpg.fr
Adrien CARTIER, Arnaud HEQUETTE
LOG, Université du Littoral Côte d’Opale
Dunkerque, France

Abstract—This paper discusses the abilities of numerical models to predict the morphodynamics over loose and rigid beds. In the first part the sediment transport model is presented which solves the bed evolution equation in conjunction with sediment transport formulas. The flow field and the water depth are calculated using the depth-averaged hydrodynamic model TELEMAC-2D developed by Électricité de France.

I. INTRODUCTION

The work consisted in setting up the methodology of calculation (De Vriend (1987) [9], De Vriend and Stive (1987) [10], Smit et al., 2008 [19]). The principle is to make an external coupling of three codes. This coupling consists in enchained Artemis for swells, Telemac2d for the currents and Sisyphe for the morphodynamic evolution (Hervouet, 2007[12]). The basic principle of this external coupling is to make this loop on the codes with a step of morphodynamic time depending essentially on weather conditions and on the environment hydrodynamics of the studied beach. These models were used in the framework of a simulated meteorological cycle describing the seasonal evolution of hydrodynamic factors.

This paper discusses the abilities of numerical models to predict the morphodynamics over sandy and rigid beds. In the first part the sediment transport model is presented which solves the bed evolution equation in conjunction with sediment transport formulas. The flow field and the water depth are calculated using the depth-averaged hydrodynamic model TELEMAC-2D and simplified model called Multi1dh. This model was already used and tested in Camenen and Larroudé, (2003b) [3].

The objectives which we want to reach during this study are multiple. First, we are going to set up a procedure for linking the three codes to be able to simulate realistic climates. This procedure is validated from the point of view of the hydrodynamics and morphodynamic evolution (Larroudé, 2008 [14]). This technique of simulation will then be used to compare and to study the contribution of various sediment transport formulae (as in Camenen, 2002 [1], on the site of Sète during two specific storm (see Robin et al., 2010 [17]).

We improve this methodology to simulate the Rising-Apex-Waning of a Storm event. We also look at the comparison of the current during these different periods of the storm. To calibrate all these sediment formulae, we also compare our simulations with in-situ data of longshore and cross-shore sediment transport measured on several beaches of the North sea and of the English Channel (Cartier and Héquette, 2011 [4] and 2011b [5]).

II. DESCRIPTION OF THE STUDIED SITES

A. Site 1

The “Plage de la Corniche”, located near Sète on the Mediterranean coast, France, was selected as the first study area (Fig. 1). Located in a microtidal, swell-dominated coastal environment, the “Plage de la Corniche” is a linear beach of about 2.5 km length. The mean near shore bed slope is 0.04, while the median grain size in the surf zone is 0.25mm.

The mean significant offshore wave height is about 1.5 m increasing to 3–6 m during storms, while the predominant wave direction is from SSE with occasional SE swells. There is no significant seasonal variation in the offshore wave climate.

Certain and Barusseau (2006) [7] showed that the morphodynamic evolution of offshore bars in a microtidal environment and bimodal moderate wave regime follows two different conceptual models, the main one being a seasonal pattern in line with the observed cycle of hydrodynamic conditions (see also Certain, 2002 [6]).

B. Site 2

The second studied area consists of three intertidal sandy beaches on the coast of Nord-Pas-de-Calais (northern France) located at Zuydcoote, Wissant and Hardelot Plage (Fig. 2). Zuydcoote site, located east of Dunkirk, is characterized by a beach of fine sand ($D_{50}$ = 0.2 mm), 350 to 400 m wide, with
an average slope of about 0.014. The tidal range varies from 3.4 m during neap tide to 5.2 m during spring tides on average, and this site can therefore be considered as meso-to macrotidal. The coast, oriented NE-SW, facing the North Sea, is characterized by fetch-limited short waves.

The Wissant site is located in a bay that extends over 6 km, bordered on the south by the Cap Gris Nez and the north by the Cap Blanc Nez. The hydrodynamics is more powerful due to the exchange of water mass between the North Sea and The English Channel which is particularly intense. The test site is located in the eastern part of the bay, characterized by a beach of fine sand (D50 = 0.22 mm) and an average slope of 0.012. The coast is subject to a tidal range from 4.2 m to 6.7 m for neap and spring tides.

The third site is located at Hardelot beach, at the Dune du Mont St-Frieux. The beach consists of fine sand (D50 = 0.23 mm) and has an average slope of 0.026. The tidal range reaches 4.8 m at neap tide on average and 8.0 m in times of great water. Oriented N-S, the coast is facing the English Channel.

Tidal currents on the three beaches flowing parallel to the shore and are characterized by a flood-dominated asymmetry. This dominance of flood currents, combined with a system of winds and swells from the SW, generates a hydrodynamic circulation and sediment transport directed eastward on the coast of the North Sea and northward on the shores of the Channel (Sipka and Anthony, 1999 [18]).

The purpose of the study is to estimate the longshore flow in the surf zone (and sometimes in the shoaling zone). Sediment transport rates were estimated using streamer traps following the method proposed by Kraus (1987) [13], allowing to measure suspended and near bed transport.

Kraus structures capture the sediment in suspension over a depth range of about 1m, they are composed of five nets with a mesh size of 63 microns to trap sediment at 0.05, 0.26, 0.46, 0.66 and 0.86 m above the bed. During high wave energy conditions, the sediment trap has to be deployed in shallower water and the two upper nets (0.66 and 0.86 m) are then removed. The structures are placed for 10 minutes, facing the mainstream, which is determined visually by the operator. The sampling time may vary depending on the conditions of agitation and/or the rise/fall more or less quickly of the sea level so this sampling time could be from 5 to 10 minutes.

Current meter devices are also deployed on the foreshore. The instruments are routinely placed on the outer side of intertidal bars. Three devices were used, ADCP, S4 and ADW Valeport (electromagnetic current meter). It saves data to hydrodynamic 2Hz, a burst of all the 9min and 15min. Only Valeport S4 allows us to have data at 2 Hz.

Morphological monitoring of each zone was carried out each sampling day using a high precision DGPS, for detail see Cartier and Héquette (2011b [5]).

Sediment fluxes are obtained at a given point in the littoral zone and at given time of a tidal cycle. The Flux is integrated in the water column (kg.s\(^{-1}\).m\(^{-1}\)). Trapping was carried out in different directions in order to measure longshore, onshore and offshore sediment flux.
III. MODEL AND METHODOLOGY

The sedimentary evolution is modeled under the action of the oblique incident waves and is coupled with different numerical tools dedicated to the other processes involved in the near shore zone. We can mention the following modules:

The wave module takes into account the surge energy dissipation (hyperbolic equation of extended Berkhoff), (LNHE, Artemis, 2002). The Artemis code (Agitation and Refraction with Telemac2d on a Mild Slope) solves the Berkhoff equation taken from Navier-Stokes equations with some other hypotheses (little camber of the surface wave, little slope, etc.).

The main results are, for every node of the mesh, the height, the phase and the incidence of the waves. Artemis can take into account the reflection and the refraction of waves on an obstacle, the bottom friction and the breakers. One of the difficulties with Artemis is that a fine mesh must be used to have good results whereas Telemac2d does not need such a fine mesh.

The hydrodynamic module calculates currents induced by means of the surge of the waves, from the concept of radiation constraints obtained according to the module of waves, (LNHE, Telemac2d, 2002). Telemac2d is designed to simulate the free surface flow of water in coastal areas or in rivers. This code solves the Saint-Venant equations vertically averaged.

Then, the main results are, for every node of the mesh, the water depth and the velocity averaged over the depth. Telemac2d is able to represent the following physical phenomena: propagation of long periodic waves, including non-linear effects, wetting and drying of intertidal zone, bed friction, turbulence, etc.

The sedimentary module integrates the combined actions of waves and currents (2D or 3D) on the transport of sediment. The Sisyphe code solves the bottom evolution equation which expresses the conservation of matter by directly using a current field result file given by Telemac2d. (Fig. 4). Several of the most currently empirical or semi-empirical formulas are already integrated in Sisyphe. In this paper we show only the simulations with the Bijker formulas. The main results are, for every node of the mesh, the bottom evolution and the solid transport. The equations of the three modules are detailed in Hervouet, 2007 [12].

A hydrodynamic simplified model (called Multi1DH) uses the following assumptions: a random wave approach, in a 1DH (cross-shore) direction. An offshore wave model ( shoaling + bottom friction + wave asymmetry) is used with the break point estimation. The waves in the surf zone are modeled with the classic model of Svendsen (1984) [20] with an undertow model (roller effect, Svendsen, 1984 [20], Dally et al. 1984 [8]). The longshore current model is the Longuet-Higgins’s model (1970) [16]. The model is included in the Sisyphe code to calculate the sea bed evolution with several sediment transports formulas.

IV. RESULTS

We set up a procedure to use the coupled codes Artemis-Telemac2d-Sisyphe and more particularly we improved the treatment of the boundary conditions in order to be able to work on fields of calculations close to the coastal zone and equivalents in dimension for the three codes. The wave module grid is equal to the flow and morphodynamic grid. The waves are incidents on both the lateral and seaward boundaries of the grid. The lateral boundaries of the flow model are defined as zero water levels.

The morphological evolution in the near shore region, including its large-scale features, was first investigated using a combination of a commercial 2DH model (Camenen and Larroudé, 2003 [2], 2003b [3]). Simulation of the wave-driven currents was carried out with Telemac, a finite elements model, and the Sisyphe sand transport module served to compute sediment transport rates and bed evolution. This methodology of morphodynamic modeling for sandy beaches was already improved in terms of mesh, time step and convergence in Camenen (2002) [1], Larroudé and Camenen (2004) [15] and in Falquès et al. (2008) [11] and Larroudé (2008) [14].

We first present results for Site 1 (“Plage de la Corniche”), focussing on the month of December 2008 and February 2009 for the validation and the first test of the different sediment transport formulas. During these months, we had two similar storms in term of significant wave height, period but in December the outer bar moved offshore...
and during the storm of February this outer bar moved onshore (see Figs. 5a and 5b).

In the case of February, the Multi1dh model reproduces very well the onshore migration of the bar with all the sediment transport formulas (Figs. 5b, 5d). On the opposite, the off shore migration is not so well simulated but the results seem to be acceptable (Figs. 5a, 5c). For the 2DH model ATS, in these cases of cross shore migration of these sand bodies, the modeling of the cross shore current (undertow) is missing. The results for both storms are thus not well representatives (Fig. 6).

Our results show that the different formulas of transport did not correctly reproduce what was observed in reality. During the storm of February, the onshore displacement of the inner bar was not represented by the majority of formulas. We can see through this case that the Bijker formula and Soulsby-Van Rijn overestimates sediment transport, while the other formulas underestimate. The modeling of the December storm, however, shows that some formulas are more robust than others, the formulas of Ribberink, of Soulsby-Van Rijn and Egelund Hansen being the least robust. For the same storm, the Bijker formula seems to overestimate the sediment transport, while the Watanabe & Dibajinia formulae coded in the Sisyphie code has a tendency to overestimate the sediment flux when there are strong velocities.

The second part of the study is to compare the solution obtained with different sediment transport formulas directly with in situ measurements of sediment transport. We use a set of data obtain on the North Sea and the English Channel beaches. There are three calculations per site with the formulas of Bijker, Einstein and Van Rijn with the first approach of simulation and there are two calculations per site with the formulas of Bijker and Dibajinia-Watanabe for the second numerical approach.

<table>
<thead>
<tr>
<th>DATE and TIME</th>
<th>Measured Longshore Flux (kg.s⁻¹.m⁻¹)</th>
<th>Calculated Longshore Flux (kg.s⁻¹.m⁻¹)</th>
</tr>
</thead>
<tbody>
<tr>
<td>13/11/2008 15:50</td>
<td>4.5x10⁻⁴</td>
<td>3.6x10⁻⁵</td>
</tr>
<tr>
<td>14/11/2008 10:45</td>
<td>4.8x10⁻⁴</td>
<td>1.1x10⁻⁵</td>
</tr>
<tr>
<td>17/11/2008 13:17</td>
<td>9.9x10⁻⁴</td>
<td>1.3x10⁻⁵</td>
</tr>
<tr>
<td>13/11/2008 15:50</td>
<td>4.7x10⁻⁴</td>
<td>0.0</td>
</tr>
<tr>
<td>14/11/2008 10:45</td>
<td>6.2x10⁻⁵</td>
<td>0.0</td>
</tr>
<tr>
<td>17/11/2008 13:17</td>
<td>1.2x10⁻⁴</td>
<td>0.0</td>
</tr>
<tr>
<td>24/11/2009 14:22</td>
<td>7.8x10⁻²</td>
<td>2.4x10⁻²</td>
</tr>
<tr>
<td>27/11/2009 12:07</td>
<td>8.2x10⁻⁴</td>
<td>6.3x10⁻⁴</td>
</tr>
<tr>
<td>30/11/2009 14:37</td>
<td>4.2x10⁻⁴</td>
<td>5.7x10⁻⁵</td>
</tr>
<tr>
<td>03/12/2009 11:22</td>
<td>6.0x10⁻⁴</td>
<td>9.4x10⁻⁴</td>
</tr>
<tr>
<td>06/12/2009 13:07</td>
<td>2.7x10⁻³</td>
<td>3.9x10⁻⁵</td>
</tr>
</tbody>
</table>
The difference between the first and second numerical approach is the way of taking into account the boundary conditions in term of velocity in the telemac2d code and using wave simulation result directly into some sediment transport formulae in Sisyphe code.

For the Hardelot and Wissant data, the comparison between measured and simulated sediment fluxes show that the first approach gives better results, but for Zuydcoote it seems that the second approach is more appropriate (see Table I and Fig. 8).

The results obtained with the other formulas show that there is often an order of magnitude between in-situ and computed data which demonstrate that although these early tests are interesting further work is still needed to improve sediment transport modeling in these macrotidal environments (see Figs. 9 and 10).

All the numerical results are taken in the middle of the simulation domain between the beach and the offshore boundary. But we can see that there is no variation in the calculated flux with Bijker or Van Rijn on respectively Hardelot and Wissant while the sediment flux measured vary greatly (see Figs. 9 and 10). To avoid this problem, our numerical results were extracted at the same location in which the water depth over the measurements is (for all case test on a site) in the same order of the average water depth of the simulated data (see Figs. 12 and 14, respectively for Hardelot and Wissant site). In these figures, the value noted case-figure 11 and case-figure 13 correspond to those measured during data collection in-situ.

We can see on Figs. 11 and 13 a better consistency between the predicted and the measured transport rate. The next step of our study will be to calibrate correctly the way of extracting the numerical fluxes and compare the results with the all formulas tested against the in-situ data.

The sediment traps mainly collecting sediments transported in suspension, the next step of this study will be to de-couple the suspended and bed load transport calculation in the different sediments transport formulas to compare only the suspended fluxes in the North sea and English Channel
Beaches data. The second improvement will be to reach a better precision on the velocity field.

V. CONCLUSIONS

We used a 2DH morphodynamic model to simulate the evolution of the linear sandy beaches, these investigations being aimed to better define the vulnerability of these coastal landforms to storm event. We have calibrated our numerical methodology of simulation against in situ measurements. The first results are good in terms of comparison with in-situ data regarding the hydrodynamic and morphodynamic parameters. This work was based on different scenarios of wave classes, storm occurrence frequency, etc… We have had to simulate all these configurations to identify the sensitivity of rising-apex-waning of the storm. Our methodology of simulation and the complementarily of both models allowed us to test the various configuration of storms to understand the results derived from the in-situ data.

Figure 12. Water depth for all the run and data on Hardelot used in Figs. 9 and 11.

Figure 13. In-situ data Qsm (Wissant) and simulated Qsc (first approach, with Van Rijn) longshore sediment fluxes (depth average around 1m).

ACKNOWLEDGEMENT

This work has been supported by French Research National Agency (ANR) through the Vulnerability Milieu and Climate program (project VULSACO, n° ANR-06-VMC-009), RELIEF MICROLIT and the CNRS PLAMAR project. We highly appreciate the constructive discussions with D. Idier, (BRGM, Orléans, France), Raphaël Certain and Nicolas Robin from the University of Perpignan.

REFERENCES


[15] Larroudé, Ph., Camenen, B., 2004, 2DH and multi1DH morphological model for medium term evolution of large scale features and nourishment in the nearshore region: application to TrucVert and Corniche beach (France) and la Barrosa beach (Spain). 29th International Conference on Coastal Engineering. ASCE, Lisbon.


Abstract—This paper describes the latest developments that have been carried out to prepare TELEMAC-2D for simulations using grids composed of hundreds of millions of elements. Even running modest-sized simulations involving around 2 to 10 million grid elements highlights some critical issues concerning both the grid generation and the subsequent grid pre-processing which is currently handled by the PARTEL TELEMAC system tool. A serial accelerated global mesh refinement technique is presented which allows the generation of a 425-million element grid from an existing 106-million element grid in less than an hour on a fat node of an IBM POWER7 cluster. The current version of PARTEL (version 6.0) relies on METIS 4.0 as the partitioner and has two main drawbacks for extremely large simulations; namely, METIS 4.0 is highly memory consuming, and secondly, PARTEL is extremely time-consuming when performing the rest of the pre-processing stage. Four alternative partitioners are tested on large grids, and a new parallel pre-processing tool, PARTEL_P, has been designed with the aim of optimising memory consumption. This new tool allows the pre-processing of a 200-million element grid on up to 32,768 sub-domains and its output has successfully been used to evaluate the scaling performance of TELEMAC-2D on an IBM Blue Gene/P.

I. INTRODUCTION

The TELEMAC system [1,2] is a multi-scale hydrodynamics free-surface suite that can solve either the two-dimensional shallow water equations (TELEMAC-2D) or the Navier-Stokes equations (TELEMAC-3D) depending on the approximation made in the calculation of the velocity component in the vertical direction. The system relies on the BIEF (Bibliothèque d’Eléments Finis) finite-element library which contains the subroutines to perform the fundamental operations on scalars, vectors and matrices, the iterative solvers, and the discretisation schemes used by the hydrodynamic solvers. The present study has focused specifically on the computational properties of the shallow water equation solver, TELEMAC-2D. The various computational stages necessary to perform a simulation with TELEMAC-2D proceed as follows:

1. Generation of a grid of triangular elements with a mesh generator, and the generation of the bathymetry of the flow domain. This stage is currently performed in serial using the mesh generation tool supplied with the TELEMAC suite. However, third-party mesh generators can also be used to create the finite-element mesh. It is also possible to globally refine an existing mesh to increase the spatial resolution of the simulation; this is also performed in serial using a tool that has recently been optimised.

2. The pre-processing stage: this includes mesh partitioning, calculation of the mesh connectivity, assignment of the boundary conditions, identification of the halo cells, and pre-processing for the method of characteristics for advection. The mesh partitioning and all other pre-processing tasks are currently performed using a serial utility called PARTEL. Serial mesh partitioning is limited by memory availability whereas the rest of the pre-processing tasks are limited by both memory and time constraints.

3. Solution of the shallow water equations using TELEMAC-2D: the equations are solved either in a fully-coupled mode or with the help of a wave equation, depending on the option chosen. The spatial discretisation, in general, is linear and several advection schemes are available depending on the type of flow. Options include the method of characteristics, the streamline-upwind Petrov-Galerkin scheme (SUPG) and residual distributive schemes (such as the N-scheme and PSI-scheme). The matrix-storage in TELEMAC is edge-based and several linear solvers are available in the BIEF library, including conjugate gradient, conjugate residual, CGSTAB and GMRES solvers. TELEMAC-2D is fully parallelised using MPI.

The current work has focused on stages 1 and 2 of the solution procedure. Section II of the paper provides an overview of the computer hardware used in the present study, Section III provides a brief description of the selected test cases and Section IV explains how the global mesh refinement is carried out. Section V then describes the four partitioners considered in the study whilst Section VI details the new parallel pre-preprocessor, PARTEL_P. Finally, Section VII of the paper illustrates the scaling performance of TELEMAC-2D using a 200-million element grid.

II. HARDWARE

A. IBM POWER7 cluster [3]

The POWER7 cluster of large memory nodes is composed of four 'POWER 750 Express' nodes, each with
256 GB of memory and 32 processor cores clocked at 3.55 GHz. The cluster is currently linked using an InfiniBand interconnect. A single node has been demonstrated to give 674 Gflop/s Linpack using the new VSX instructions implemented on the POWER7 giving a theoretical peak floating point performance of 8 operations per clock cycle.

B. IBM Blue Gene/P [4]

A single rack of the Blue Gene system contains 1024 chips with four processor cores per chip in the P system, giving 4096 cores per rack in total. Memory is provided at 512 MB per core. The BlueGene/P system uses a processor from the Power 450 family running at 850 MHz. A single rack has a theoretical peak floating point performance of 13.9 Tflop/s.

C. Local PC

A local PC has also been used for some of the test cases. The local PC is a Linux machine with an Intel(R) Core(TM)2 Duo CPU processor clocked at 2.66 GHz. The system has 4 GB of RAM.

III. DESCRIPTION OF THE TEST CASES: THE GIRONDE ESTUARY AND THE MALPASSET DAM-BREAK

Two separate test cases have been considered in this study; the first considers tidal propagation in the Gironde Estuary [5] while the second simulates the catastrophic Malpasset dam-break flood event [6,7].

A. The Gironde Estuary

The first test case considers the simulation of tidal propagation in the Gironde Estuary in the south-west of France. The Gironde is formed by the confluence of the Garonne and Dordogne rivers and is the largest estuary in western Europe with a total surface area of approximately 635 km$^2$. The Gironde can be categorised as a macrotidal estuary with a mean spring tidal range of 4.5 m at the mouth of the estuary. Fig. 1 shows the extent of the computational domain which spans a distance of approximately 170 km. The open seaward boundary is located between 10 and 20 km into the Bay of Biscay whilst the two landward boundaries are located at La Réole on the Garonne and Pessac-sur-Dordogne on the Dordogne.

B. The Malpasset dam-break

The second test case involves the simulation of the Malpasset dam-break [6,7] which occurred in the Reyran valley in the south of France on 2nd December 1959, following a period of heavy rain. The sudden and unexpected collapse of almost the entire wall of the 66 m high, 223 m long Malpasset dam, caused a flood wave of 50 million cubic metres of water to flow into the Reyran valley. Fig. 2 shows the computational domain used in the present study.

IV. MESH SPLITTING

The SELAFIN format grids used by TELEMAC-2D require knowledge of the 2-D coordinates, the bathymetry, the connectivity between the nodes, the location of each node (whether it is on a physical boundary or not) and the imposed boundary conditions. The boundary conditions are stored in an ASCII file whereas the remaining information is contained in a single binary file. Running in parallel follows the same strategy; each MPI task reads two files, one for the geometry, and the other for the parallel communications and the boundary conditions. Extra files might also be required, for example to set up the mass flow rate or the water level variations at the inlet boundaries, or the specification of meteorological data such as wind speed and direction.

Figure 1. Computational domain for the Gironde study.

Figure 2. Computational domain for the Malpasset study.
handle larger meshes due to memory constraints and the fact that the algorithm loops have not been optimised.

A new serial mesh-splitting tool has therefore been designed to overcome these problems and is described in the following section.

A. Description of the mesh-splitting algorithm

Globally refining grids by splitting each triangular element of the original mesh (OM) into four sub-triangles of the new mesh (NM) requires the determination of the mid-point of each edge of OM. These mid-points occur twice for internal faces and have different global indices whereas boundary face mid-points only occur once. A temporary new mesh connectivity list is built from all the new nodes, including the ones counted twice. This list is four times larger than OM’s connectivity list. The next step consists of merging all OM’s mid-points/NM’s new nodes and updating the NM connectivity list. This is performed in the following stages:

- From the over-estimated list of NM’s nodes, a one-dimensional array is assembled from the two-dimensional nodal coordinates using the following equation: \( XY = \alpha X + \beta Y \), where \((X,Y)\) are the nodal coordinates and \((\alpha,\beta)\) are suitable weighting coefficients. The \(\beta Y\) term is used to differentiate nodes that have the same abscissa but different ordinates. In the present study, the weighting coefficients have been set to \((\alpha,\beta) = (10^{10},10^{-10})\). It should be noted that the one-dimensional array, \(XY\), is stored in quadruple precision. Having to handle quadruple precision is a clear limitation since the GNU Fortran compiler is not able to support quadruple precision arithmetic. One way of overcoming this problem is to define a new data type \((X,Y)\) which contains \(X\) and \(Y\) in double precision, and define a new ‘comparison’ operator such as \((X,Y) \leq (X',Y')\). The piece of code for sorting would have to be modified in order to handle this new data type.

- The one-dimensional array, \(XY\), is then sorted into ascending order so that all the nodes counted twice appear consecutively.

- The nodes are then recounted using a new global index which only counts identical nodes once. The connectivity list of NM is then updated with the new node list of global indices.

The physical boundary condition list also has to be updated accordingly. After each refinement, the bathymetry has to be linearly interpolated from the coarse grid onto the finer grid.

B. Mesh refinement study

The new mesh-splitting tool has been tested on both the Gironde and the Malpasset computational domains. Refining the Gironde flow domain is particularly challenging due to the presence of seven islands within the estuary. In contrast, the Malpasset refinement is somewhat simpler since the computational domain does not contain islands.

The mesh refinement process for the Gironde is illustrated in Fig. 3 which shows a section of four separate meshes of increasing spatial resolution within the estuary. The grid refinement computations were performed on a fat node of an IBM POWER7 cluster. The original mesh (0\(^{th}\) level), composed of 96,773 nodes and 188,219 (~0.2 million) elements, was successively refined to obtain meshes containing approximately 0.8, 3 and 12 million elements, respectively. The finest mesh illustrated in Fig. 3 is composed of 6,044,358 nodes and 12,046,016 elements.

Table I details the computational time required for the mesh refinement process on an IBM POWER7 cluster and shows the computational efficiency of the numerical algorithm. The mesh refinement process was continued for five levels of refinement, yielding a final mesh composed of 96,453,546 nodes and 192,736,256 elements. The smallest elements are approximately 5 m in size on the coarsest grid and about 0.16 m on the finest grid.

<table>
<thead>
<tr>
<th>Level</th>
<th>Nodes</th>
<th>Elements</th>
<th>Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0(^{th})</td>
<td>96,733</td>
<td>188,219</td>
<td>—</td>
</tr>
<tr>
<td>1(^{st})</td>
<td>381,771</td>
<td>752,876</td>
<td>3.26</td>
</tr>
<tr>
<td>2(^{nd})</td>
<td>1,156,424</td>
<td>3,011,504</td>
<td>16.10</td>
</tr>
<tr>
<td>3(^{rd})</td>
<td>24,134,738</td>
<td>96,453,546</td>
<td>3090.97</td>
</tr>
</tbody>
</table>

Figure 3. Detail of the most refined part of the Gironde grid for the original (0\(^{th}\) level) mesh and the first three levels of refinement.

<table>
<thead>
<tr>
<th>Level</th>
<th>Nodes</th>
<th>Elements</th>
<th>Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3(^{rd})</td>
<td>6,044,358</td>
<td>192,736,256</td>
<td>3090.97</td>
</tr>
<tr>
<td>4(^{th})</td>
<td>24,134,738</td>
<td>96,453,546</td>
<td>3090.97</td>
</tr>
<tr>
<td>5(^{th})</td>
<td>96,453,546</td>
<td>3,011,504</td>
<td>16.10</td>
</tr>
</tbody>
</table>
An additional test was performed for the Malpasset computational domain. Table II summarises the times required to reach seven levels of grid refinement and a final mesh of 213,061,121 nodes and 425,984,000 elements. The tests demonstrate that meshes of more than 400 million elements can readily be generated using the proposed serial technique.

**Table II. CPU Time (s) (IBM POWER7) to Compute Seven Levels of Grid Refinement from an Original (0th Level) Mesh of the Malpasset Flow Domain Composed of 0.02 Million Triangular Elements.**

<table>
<thead>
<tr>
<th>Level</th>
<th>Nodes</th>
<th>Elements</th>
<th>Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0th Level</td>
<td>13,541</td>
<td>26,000</td>
<td>—</td>
</tr>
<tr>
<td>1st Level</td>
<td>53,081</td>
<td>104,000</td>
<td>0.27</td>
</tr>
<tr>
<td>2nd Level</td>
<td>210,161</td>
<td>416,000</td>
<td>1.17</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Level</th>
<th>Nodes</th>
<th>Elements</th>
<th>Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3rd Level</td>
<td>836,321</td>
<td>1,664,000</td>
<td>5.48</td>
</tr>
<tr>
<td>4th Level</td>
<td>3,336,641</td>
<td>6,656,000</td>
<td>27.37</td>
</tr>
<tr>
<td>5th Level</td>
<td>13,329,281</td>
<td>26,624,000</td>
<td>127.76</td>
</tr>
</tbody>
</table>

**V. TESTS ON VARIOUS PARTITIONERS**

The current serial version of PARTEL (version 6.0) uses METIS 4.0 as the partitioner. However, METIS 4.0 is known to be very memory consuming. Instead, four other partitioners have been tested including, METIS 5.0, ParMETIS 4.0, SCOTCH 5.1.12 and PT-SCOTCH 5.1.12. The Gironde Estuary and the Malpasset dam-break test cases have been run on a PC and an IBM POWER7 cluster. Unfortunately, PT-SCOTCH returns errors on the IBM cluster and these are currently being investigated by the main developer of the software.

**A. METIS/ParMETIS [8]**

METIS is a set of serial programs for partitioning graphs, partitioning finite-element meshes, and producing fill reducing orderings for sparse matrices. The algorithms implemented in METIS are based on the multilevel recursive-bisection, multilevel k-way, and multi-constraint partitioning schemes.

ParMETIS is an MPI-based parallel library that implements a variety of algorithms for partitioning unstructured graphs, meshes, and for computing fill-reducing orderings of sparse matrices. ParMETIS extends the functionality provided by METIS and includes routines that are especially suited for parallel AMR computations and large scale numerical simulations. The algorithms implemented in ParMETIS are based on the parallel multilevel k-way graph-partitioning, adaptive repartitioning, and parallel multi-constrained partitioning schemes developed at the Karypis Laboratory [8].

Version 5.0 of METIS and version 4.0 of ParMETIS are used in the present study. A huge effort has been made to optimise memory consumption in the latest releases of these codes. As both software codes are graph-based, a good strategy is to build a dual mesh of the existing grid and to transform it into a graph before partitioning since partitioning by elements has been shown to provide better quality results than partitioning by nodes. This option is provided in both METIS and ParMETIS.

**B. SCOTCH/PT-SCOTCH [9]**

SCOTCH is a project carried out by the Satan team of the Laboratoire Bordelais de Recherche en Informatique (LaBRI) in France. It is part of the ScALApplix project of INRIA Bordeaux-Sud-Ouest. Its purpose is to apply graph theory, with a divide and conquer approach, to scientific computing problems such as graph and mesh partitioning, static mapping, and sparse matrix ordering, in application areas ranging from structural mechanics to operating systems or bio-chemistry. The SCOTCH distribution is a set of programs and libraries which implement the static mapping and sparse matrix reordering algorithms developed within the SCOTCH project. PT-SCOTCH is the parallel version of SCOTCH.

Version 5.1.12 of SCOTCH has been used in this study. Unlike METIS and ParMETIS, SCOTCH/PT-SCOTCH do not provide tools for building dual meshes since they only deal with graphs. In contrast, a function has been built into METIS/ParMETIS to handle meshes directly. The function to perform the grid-to-dual-graph operation in METIS/ParMETIS avoids using a less optimised Fortran function. SCOTCH and PT-SCOTCH require a strategy to define how to perform the partitioning. The default strategy is used here.

**C. The Gironde Estuary**

Each of the partitioners was used to process the 0th level Gironde mesh (~0.2 million elements) into 8 sub-domains; this initial study was performed using a local PC. Fig. 4 shows that each partitioner creates substantially different arrangements of sub-domains, as can be seen in the Garonne and Dordogne rivers.

A more elaborate series of tests were then performed on the IBM POWER7 cluster using METIS, SCOTCH and ParMETIS. These tests considered the time required to complete the partitioning process. Fig. 5 shows the time spent by the partitioners for various levels of grid refinement; all three partitioners exhibit a linear behaviour with the level of grid refinement, with ParMETIS being the fastest and SCOTCH the slowest. The number of partitions for each grid was selected so as to obtain an average of 3250 elements per sub-domain since previous tests [5] have demonstrated that TELEMAC-2D offers very good speed-up on a variety of computer platforms when reducing the number of elements per sub-domain from 6500 to 3250.
Fig. 6 compares the time spent by ParMETIS to partition the Gironde grids as a function of the refinement level, for different numbers of cores. In general, the 16-core simulations are the fastest.

**Figure 4.** Sub-domains generated by each partitioner for the Gironde study; METIS (upper left), SCOTCH (upper right), ParMETIS (lower left), PT-SCOTCH (lower right).

**Figure 5.** Partitioner times for each of the Gironde grid refinement levels.

**Figure 6.** Time spent by ParMETIS as a function of the grid refinement level for the Gironde study.

D. The Malpasset dam-break

The first test for the Malpasset dam-break study consisted of running each of the partitioners on a PC to create 8 sub-domains for the 1st level of refinement mesh (~0.1 million elements). Fig. 7 again shows that the various partitioners
produce slightly different results (see for example, the boundary between sub-domains, A and B, in Fig. 7). A second series of tests were then performed on the IBM POWER7 cluster using METIS, SCOTCH and ParMETIS to assess the time required to complete the partitioning process.

Figure 8. Partitioner times for each of the Malpasset dam-break refinement levels.

Fig. 8 shows the execution time of METIS, SCOTCH and ParMETIS for all seven levels of grid refinement for the Malpasset study. The same criterion of 3250 elements per sub-domain was used when selecting the number of partitions. ParMETIS was run on 16 cores except for the original mesh, where only 8 cores were used because the number of partitions to satisfy the 3250 element criterion is only 8, and the number of cores should always be lower than or equal to the number of sub-domains, otherwise the communication is too costly. The results show that ParMETIS is the fastest of the partitioners whereas SCOTCH is the slowest.

Fig. 9 compares the time spent by ParMETIS to partition the Malpasset grids as a function of the refinement level for different numbers of cores. A speed-up was still observed for 96 cores but additional tests should be conducted to see if the partitioning time decreases with larger core counts. The 7th level of grid refinement requires at least 32 GB of RAM per ParMETIS task and therefore the tests on 32, 64 and 96 cores could not be performed because only 3 fat nodes are available on the POWER7 cluster, the fourth being a standalone node.

An additional test has shown that SCOTCH can partition the 7th level of refinement for the Malpasset study into 294,912 subdomains which represents the total number of cores on the IBM BlueGene/P located at Juelich [10].

VI. PARALLEL PRE-PROCESSING (PARTEL_P)

To overcome the 2-10 million element grid limit of the serial pre-processor, PARTEL, a parallel version has been developed within the PRACE-IIP project [11], called PARTEL_P, which runs on NPROCS cores and partitions grids into NSUBS sub-domains. It should be noted that the current version of PARTEL_P does not support parallel IOs nor the method of characteristics for the pre-processing stage.

A. Format of the files output by PARTEL

Two files per sub-domain are output by PARTEL; a geometry file in SELAFIN format and a boundary file in ASCII format following the TELEMAC-2D standard. The geometry file contains a header, and the number of elements, nodes, physical boundaries and interfaces for a given sub-domain. It also contains the local connectivities of the nodes, the local-to-global node table and finally the coordinates and/or other quantities known at each node. The boundary file contains the information for the physical boundaries, the number of interfaces with other sub-domains, and the information required to handle the interfaces. Each physical boundary requires knowledge of the neighbouring nodes located in a different sub-domain. The treatment of the interfaces is more complex as the number of contiguous sub-domains has to be known, as well as their partition index. The interfaces also have to be sorted into ascending order to comply with the TELEMAC-2D standard.

B. Description of PARTEL_P

PARTEL_P is actually split into two parts; PARTEL_P_1 is used to generate NPROCS files to be read by PARTEL_P_2 so as to reduce memory consumption. These two programs will be merged in the future since both PARTEL_P_1 and PARTEL_P_2 are run on the same number of cores.

PARTEL_P_1 is used to distribute the information of NSUBS/NPROCS sub-domains over the NPROCS cores. The initial stage is mainly serial as no attempt has yet been made to improve the IO operations. The input parameters are read, i.e. the name of the geometry file, the name of the boundary file, NSUBS and the library used to partition the grid. Each processor reads the geometry and the boundary...
domains also has to be computed. The boundary information for the neighbouring nodes located in a different sub-domain but on a different core to PARTEL_P_2. Interfaces are not dealt with at this stage.

PARTEL_P_2 is run on NPROCS cores. It first reads the input parameters, i.e. the name of the original geometry file, the name of the original boundary file, NSUBS and NPROCS. Each core then reads the files output by PARTEL_P_1 which contains information for NSUBS/NPROCS sub-domains. The number of elements, nodes, physical boundaries, and interfaces per sub-domain are easily computed. This information, together with the knowledge of the sub-domain local connectivity and coordinates, helps build the NSUBS geometry files that are read by TELEMAC-2D. The local-to-global node table is also easily accessible.

The main issues arise when building the physical boundary information for the neighbouring nodes located in a different sub-domain and on a different core to PARTEL_P_2. Interfaces are not dealt with at this stage.

The neighbouring nodes located on the same core but in a different sub-domain have to be identified. Working on a given core, all the physical boundaries are gathered in an array containing the global index. This array is sorted in ascending order and global indices that occur twice, or more, indicate that the corresponding nodes belong to several sub-domains. Their neighbours are easily identified and the array is sorted back to its original structure to comply with the TELEMAC-2D standard.

The interfaces are treated globally. A loop over all the NSUBS/NPROCS sub-domains allows the code to gather the interfaces of all the NPROCS cores before using MPI_Allgatherv to get their global index, as well as the index of the sub-domain they belong to. This array is sorted by global indices in ascending order. The number of consecutive occurrences, NINTERF, of a given global index indicates that the same interface belongs to NINTERF sub-domains and these partition indices have to be saved. To comply with the TELEMAC-2D standard, the information per interface has also to be sorted. All this information is then distributed in two stages, first onto the NPROCS, using an MPI_Scatterv command, and then to the NSUBS sub-domains.

The information relating to the physical boundaries and the interfaces is finally copied into the boundary files which are read by TELEMAC-2D.

C. Timings for PARTEL_P_1 and PARTEL_P_2

PARTEL_P_1 and PARTEL_P_2 have been run to pre-process the 5th level of refinement for the Gironde Estuary test case, and the output will be used in the next section to test TELEMAC-2D.

<table>
<thead>
<tr>
<th>PARTEL_P_1</th>
<th>PARTEL_P_2</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>4096</td>
<td>2073</td>
<td>189</td>
</tr>
<tr>
<td>8192</td>
<td>2421</td>
<td>284</td>
</tr>
<tr>
<td>16384</td>
<td>2876</td>
<td>413</td>
</tr>
<tr>
<td>32768</td>
<td>3941</td>
<td>748</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PARTEL_P_1</th>
<th>PARTEL_P_2</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>4096</td>
<td>2859</td>
<td>201</td>
</tr>
<tr>
<td>8192</td>
<td>3021</td>
<td>262</td>
</tr>
<tr>
<td>16384</td>
<td>3671</td>
<td>364</td>
</tr>
<tr>
<td>32768</td>
<td>4878</td>
<td>587</td>
</tr>
</tbody>
</table>

Tables III and IV indicate the total time spent by PARTEL_P to pre-process the grid into 4096, 8192, 16384 and 32768 sub-domains respectively, using METIS and SCOTCH as the partitioner. Overall, partitioning by METIS allows a faster pre-processing. All PARTEL_P_1 simulations are faster when METIS rather than SCOTCH is used as the partitioner. However, PARTEL_P_2 is normally faster when SCOTCH is used. A more thorough study should be able to confirm whether this is due to the fact that the edge-cut should be smaller with SCOTCH, which has a direct impact on the global communications used in PARTEL_P_2.

VII. SCALING PERFORMANCE OF TELEMAC-2D

The 5th level of refinement for the Gironde Estuary test case (~200 million elements) has been used to evaluate the performance of TELEMAC-2D on 32,768 cores of Argone’s IBM Blue Gene/P [12]. PARTEL_P was used to perform the pre-processing with both METIS and SCOTCH being used as the partitioner.

The positive stream-wise implicit (PSI) advection scheme was selected since PARTEL_P does not yet support the method of characteristics. The scaling performance of TELEMAC-2D was evaluated using simulations of 60 seconds (1200 time steps). The CPU time is reported as the time for the executable to complete ($T_{TOTAL}$), as well as the time difference between the end and the beginning of the main program, $homere_telemac2d.f$ ($T_{SOLVER}$).
Fig. 10 shows that $T_{\text{SOLVER}}$ decreases linearly as a function of the number of cores, whether METIS or SCOTCH is used as the partitioner. Good performances are observed with about 6100 elements per core. A 65,536 sub-domain simulation would help assess the performance of TELEMAC-2D with about 3000 elements per core. However, $T_{\text{TOTAL}}$ shows a different behaviour, with no real speed-up for the 32,768-core simulations. This might be explained by the time spent opening files, and the way the system manages the simulations.

![Gironde Simulation](image)

**Figure 10.** Scaling performance of TELEMAC-2D for the 5th level of grid refinement for the Gironde Estuary simulation.

VIII. CONCLUSIONS AND FUTURE WORK

This paper has described the latest developments for running TELEMAC-2D on massively parallel computer architectures. An efficient serial global mesh refinement technique has been developed that can readily create meshes containing 400 to 500 million elements.

To overcome the mesh-size limitations of the existing serial pre-processor, PARTEL, a parallel version of the system tool has been developed called PARTEL_P. This new tool has been shown to be capable of pre-processing a 200-million element grid on up to 32,768 sub-domains and its output has successfully been used to demonstrate good scaling performance of TELEMAC-2D on Argonne National Laboratory’s IBM Blue Gene/P.

The next stage is to be able to run TELEMAC-2D on more than 32,768 cores. This should be possible by implementing some form of memory optimisation, for example using compressed sparse row (CSR) format for some of the arrays. There is also a need to test the new pre-processing tool, PARTEL_P, with ParMETIS and PT-SCOTCH in order to complete the performance comparisons for different partitioners.

ACKNOWLEDGEMENTS

The authors acknowledge that the developments outlined in this paper have been achieved with the assistance of the high-performance computing resources (Tier-0) provided by PRACE on Jugene, based in Germany. This research also used resources of the Argonne Leadership Computing Facility at Argonne National Laboratory which is supported by the Office of Science of the U.S. Department of Energy under contract DE-AC02-06CH11357.

The authors would also like to thank the UK Engineering and Physical Sciences Research Council (EPSRC) for their support of Collaborative Computational Project 12 (CCP12) and the Distributed Computing Group at STFC Daresbury Laboratory. In addition, the authors would like to thank Nathalie Durand from EDF R&D for providing the data for the Gironde study.

REFERENCES

HPC for sensitivity studies: simulations with TOMAWAC and TELEMAC-3D

Noémie DURAND, Sébastien BOURBAN
HR Wallingford Limited, UK
n.durand@hrwallingford.co.uk

Charles MOULINEC, Robert BARBER,
Yoann AUDOUIN, Xiaojun GU, David EMERSON
STFC Daresbury Laboratory, UK

Abstract—High Performance Computing (HPC) is useful in a range of scientific applications, from running computationally-intensive high-resolution models to running a large number of smaller simulations either to calibrate the model (for example, using genetic algorithms) or to assess a range of design options or model scenarios. One particularly promising area for HPC is when carrying out sensitivity studies. In this case, a large number of simulations are performed using a systematic variation in the values of certain input parameters in order to determine the effects of each of the parameters on the model results. This type of application is non-intrusive in the sense that no modifications are required to the model source code. Instead, the only changes that need to be made are to user-generated subroutines and to the files that control the values of the input parameters.

In this paper, we describe a script which has been developed for use on an iDataPlex cluster (STFC Daresbury Laboratory) and a Cray XE6 supercomputer (UK National Supercomputing Service). The script enables concurrent simulations to be submitted to the system via the queuing facility available on parallel clusters. This is not only relevant for serial instances of the code but is also valid for parallel simulations. Using the script for parallel instances allows the elapsed time for sensitivity studies to be reduced by a factor of approximately NCSIMS × ICORES OPT, where NCSIMS is the number of concurrent instances and ICORES OPT is the optimum number of cores used for running each parallel instance.

I. INTRODUCTION

TOMAWAC [1,2] is a third generation spectral wave model which accounts for wave generation, refraction, shoaling, non-linear wave interactions and energy dissipation due to white-capping, bottom friction and depth-induced breaking. It was developed by EDF R&D as part of the TELEMAC finite element hydrodynamic modelling system [3-5]. TELEMAC-3D [6] is the 3D hydrodynamic module within TELEMAC which solves the complete Navier-Stokes equations.

In many instances, it is necessary to run TOMAWAC or TELEMAC-3D as part of a sensitivity study. This requires the code to be run many times using a systematic variation in the input parameters. This paper describes the development of a script to enable concurrent simulations to be submitted to the job queuing facility on parallel clusters. Most of the script is used to control the simulations, i.e. creation of temporary system directories from which the model can be run locally, creation of the local executables, and retrieval of the simulation results. The script is primarily independent of the queuing system. Only the last step of running the model is dependent on local environment variables and other platform-specific commands. The main entries for the script are the number of simulations, the names of the steering files, the name of the TELEMAC executable and the number of MPI tasks per parallel simulation.

II. HARDWARE/SOFTWARE

A. IBM iDataPlex cluster [7]

STFC Daresbury Laboratory hosts an IBM iDataPlex system (see Fig. 1) composed of a total of 76 nodes (912 cores), distributed as follows; 38 nodes for the exclusive use by STFC Daresbury Laboratory, 20 nodes for external consultancy and 18 nodes for the University of Huddersfield. Each node has 12 Westmere cores, i.e. two Intel X5650 six Core 2.66 GHz processors. A total of 24 GB RAM is available per node. A Qlogic QDR InfiniBand system is used as the switch.

Figure 1. The iDataPlex cluster at STFC Daresbury Laboratory.
The iDataPlex cluster uses the Portable Batch System (PBS) to manage the queues. The Fortran compiler employed on the iDataPlex system is version 11 of the Intel compiler and the TELEMAC system is compiled with the -O3 optimisation option. Parallel communications are handled in TELEMAC using MPI; version 1.4.3 of OpenMPI is currently used on the iDataPlex system.

B. Cray XE6 [8]

HECToR Phase 2b (Cray XE6) (see Fig. 2) is the UK National Supercomputing Service hosted at the University of Edinburgh. The system is contained in 20 cabinets and comprises a total of 464 compute blades. Each blade contains four compute nodes composed of two 12-core AMD Opteron 2.1 GHz Magny Cours processors (24 cores per node); this amounts to 44,544 cores in total. Each 12-core socket is coupled using a Cray Gemini routing and communications chip, and each 12-core processor shares 16 GB of memory, giving a system total of 59.4 TB. The theoretical peak performance of the Phase 2b system is over 360 Tflop/s.

III. DESCRIPTION OF THE SCRIPT

A bash script has been developed that builds a second script which is referred to, hereafter, as the job submission script. The latter is submitted to the queuing system in order to run the concurrent simulations. These may be run in serial, or in parallel, depending upon the user’s specific requirements and the size of problem being considered. A typical scenario for running concurrently is when performing a large number of similar simulations in the context of a sensitivity analysis.

The bash script contains seven entries as follows:

- the name of the job submission script,
- the name of the module of the TELEMAC system to be run (in this case, tomawac or telemac3d). The name of the module is referred to as telmodule,
- the number of concurrent simulations (or telmodule instances), NCSIMS. These are referenced by the variable, ICSIM, which runs from 1 to NCSIMS,
- the number of processes per node on the targeted cluster. In theory, this number should be 12 for the iDataPlex cluster and 24 for the Cray XE6, but might be reduced because of memory constraints,
- the total number of cores used by each telmodule simulation. This is equal to 1 if each telmodule simulation is run in serial; otherwise, this is set to the number of sub-domains that the grid is to be partitioned into,
- the estimated number of elapsed hours that the job will take, so as to inform and help with the booking process on the cluster, and finally,
- the estimated number of minutes (again to inform and help with the booking process on the cluster).

A pre-processing stage precedes the job submission and is performed as follows. The names of the parameter files for all concurrent simulations are listed in a file called casfile.dat (NCSIMS lines). A loop over these filenames is then performed that creates NCSIMS temporary directories and files by running telmodule casname(ICSIM). If different user-subroutines are used from one ICSIM instance to another, a new executable is created.

The names of the temporary telmodule directories are then written to a file named dirfile.dat in an attempt to simplify the rest of the submission process.

A loop over the directory names in dirfile.dat creates symbolic links to the temporary files in DIR(ICSIM) with ICSIM varying between 1 and NCSIMS. This completes the pre-processing stage.

The job submission script is then built from the bash script. The operating principle of the job submission script is described in the broadest of terms, without going into the details of queuing-system-dependent settings.

A loop over the NCSIMS simulations is performed:

- While ICSIM is less than NCSIMS+1
- Change to directory DIR(ICSIM)
- Run the ICSIMth executable
- End of the While loop

The job submission script is then submitted to the queuing system. The specified number of hours and minutes are used by the queuing system to determine which queue should be used to run the job, as the queues are different for short and long runs.
IV. COSTING THE SIMULATIONS

The tendering process for consultancy projects and the preparation of research proposals for computing time on High-End facilities often require an accurate estimate to be made of the total elapsed time necessary to complete the simulations and also the cost of the total CPU time (the cost being based upon the number of core-hours). In the present study, a series of formulae have been developed that help determine the cost of the numerical simulations. Before detailing the cost formulae, it is necessary to list the notation and the assumptions that have been made in the analysis. It is assumed that:

- ICSIM denotes the ICSIM\textsuperscript{th} of the NCSIMS telmodule instances.
- each telmodule instance is assumed to run on ICORES, where ICORES ranges from ICORES\textsubscript{REF} to NCORES. Here, ICORES\textsubscript{REF} is the number of cores used by the reference simulation, with ICORES\textsubscript{REF} = 1 for serial instances. The reference simulation is defined as the one using the smallest number of cores that will allow the telmodule instance to run.
- all NCSIMS instances are run on the same number of cores, ICORES.
- T(ICSIM,ICORES) is the time for a given ICSIM\textsuperscript{th} instance to complete.
- T\textsubscript{ICORES} is the time for the slowest of all the NCSIMS instances to complete and is defined as
  \[ T_{ICORES} = \max \left( T(ICSIM,ICORES) \right) \text{ for } 1 \leq ICSIM \leq NCSIMS. \] (1)

The total time for the job to complete can then be estimated as

\[ TOT_{ICORES} = T_{ICORES} + T_{SYSTEM}, \] (2)

where T\textsubscript{SYSTEM} is the time required by the system to manage the job involving the concurrent simulations.

The overall speed-up of the telmodule instances run on ICORES for the NCSIMS simulations can therefore be defined as

\[ \text{SPEED-UP(ICORES)} = \frac{TOT_{ICORES\textsubscript{REF}}}{TOT_{ICORES}}. \] (3)

The best performance, SPEED-UP\textsubscript{OPT}, is achieved when the instances are run on ICORES\textsubscript{OPT} cores and can be defined as

\[ \text{SPEED-UP}_{OPT} = \text{SPEED-UP}(\text{ICORES}_{OPT}) = \max \left( \text{SPEED-UP}(\text{ICORES}) \right) \text{ for } \text{ICORES}_{REF} \leq \text{ICORES} \leq \text{NCORES}. \] (4)

Now if the reference simulations were run consecutively on ICORES\textsubscript{REF} cores, and assuming that T\textsubscript{SYSTEM} is negligible, then the total elapsed time, T\textsubscript{EL,CONSEC\textsubscript{REF}}, to complete all the consecutive simulations would be given by

\[ TOT_{EL,CONSEC\textsubscript{REF}} = \sum_{ICSIM=1}^{NCSIMS} \left( T(ICSIM,ICORES\textsubscript{REF}) \right). \] (5)

Assuming that all the values of T(ICSIM,ICORES\textsubscript{REF}) are of the same order of magnitude, then the summation can be approximated by

\[ TOT_{EL,CONSEC\textsubscript{REF}} \approx NCSIMS \times T_{ICORES\textsubscript{REF}}. \] (6)

If all NCSIMS instances were run concurrently on ICORES\textsubscript{OPT} cores, then it would take T\textsubscript{EL,CONCURR\textsubscript{OPT}} seconds to complete the simulations. The duration of the project would then be reduced by approximately

\[ TOT_{EL,CONSEC\textsubscript{OPT}} = TOT_{EL,CONSEC\textsubscript{REF}} - TOT_{ICORES\textsubscript{OPT}}. \]

Moreover, it is also possible to estimate the total CPU requirement when NCSIMS concurrent runs are each performed on ICORES:

\[ TOT_{CPU,CONCURR} = NCSIMS \times ICORES \times T_{ICORES}. \] (7)

It is again assumed that all the timings for T(ICSIM,ICORES) are of the same order of magnitude. The CPU cost of the concurrent simulations, relative to the cost of the reference concurrent simulations can then be expressed as

\[ \text{RELATIVE\_COST} = \frac{TOT_{CPU,CONCURR\textsubscript{OPT}}}{TOT_{CPU,CONCURR\textsubscript{REF}}} = \frac{ICORES_{OPT} \times T_{ICORES_{OPT}}}{ICORES_{REF} \times T_{ICORES_{REF}}}. \] (8)

Equation (8) can be used to estimate the cost of the optimised concurrent simulations compared to the reference concurrent simulations.

V. TESTS USING TOMAWAC

A. Description of the problem

The first test case considers the simulation of the morphological processes that affect dredging spoil that has been used to create an artificial island. Although the island is not intended for infrastructural development, there is concern that the spoil material could suffer erosion, with sediment being eventually transported back to the original dredged channel. HR Wallingford has developed a series of numerical models to study the coastal morphological processes affecting the site and to determine the potential for sediment to affect the dredged channel. Waves contribute to the re-suspension and transport of sediments in the shallows and breaking zone. The test case described here only considers the wave modelling aspect of the overall study.

A TOMAWAC model was set up to account for the wave transformation processes (e.g. refraction, shoaling and depth-induced breaking) as the waves propagate from deep to shallow water. Currents can also affect the waves as they travel nearshore, depending on whether the currents follow or...
counter the predominant wave direction, and therefore wave-
current interactions have been accounted for in the present
model. The TOMAWAC mesh used in the study was
composed of 57,177 elements and is shown in Fig. 3.

Offshore model data were obtained for a point on the
TOMAWAC boundary. These data cover a period from
January 1983 to December 2002. The offshore wave climate
is presented as a wave rose in Fig. 4, indicating the frequency
of occurrence of certain wave conditions, discretised into
directional sectors and wave height bins. In this case, each
ring on the wave rose represents a frequency of occurrence of
10%. Referring to Fig. 4, it is clear that north-westerly waves
are the predominant offshore waves in the region of interest.
The significant wave heights are below 1.5 m for about 90%
of the 20-year record but extreme wave heights can be
greater than 4 m.

B. Analysis of the results

It is usually not practical to run the wave model directly
for the duration of the offshore wave record. Instead, a look-
up table approach is often preferred, whereby the wave
model is run for a range of discrete wave and wind
conditions, representative of the offshore wind and wave
climates. Multi-dimensional look-up tables are then created
relating the modelled offshore wave and wind conditions to
the nearshore wave conditions at the site. These look-up
tables are applied to the offshore time series to provide
congruent long-term nearshore time series. Ideally, the
model would be run for thousands of conditions in order to
cover all realistic combinations. Added complexity is
introduced when the state of the tide (water level and current
variations) plays a significant role in the wave conditions at
the site.

Due to constraints with HR Wallingford’s computational
resources, the original study using TOMAWAC was run for
41 conditions representative of the offshore wave climate,
considering wave directions between 270°N and 60°N and a
water level of Mean High Water. Local wave generation for
directions between 60°N and 270°N was included, based on
fetch length and wind speeds, and follows the method
proposed by Hasselmann et al. [9]. The nearshore wave
climate resulting from this analysis is presented in Fig. 5 and
was then used in conjunction with flow model results and
sediment and borehole data to assess the long term stability
of the sea bed around the island, and the potential for infilling
of the dredged channel as a result of erosion on the island.

Fig. 6 displays a typical result from one of the simulations,
and shows the spatial variations of predicted significant wave
height at the site. In this figure, the grey arrows are a visual
representation of the mean wave direction. For clarity, the
vectors are not shown for all grid points.
C. Use of the script on the iDataPlex cluster and the Cray XE6

For the purposes of this investigation into the wider use of HPC in consultancy and research, the same wave model was run on the iDataPlex cluster and the Cray XE6. All 41 instances of the TOMAWAC study (NCSIMS = 41) were set to run for 2000 time steps. In this particular case, ICORESREF = 1. Running all the instances concurrently, albeit in serial, takes approximately 25,700 s and 33,358 s on the iDataPlex cluster and the Cray XE6, respectively. The simulations on the iDataPlex cluster are faster for two reasons; firstly its processor is faster than the Cray XE6 processor, and secondly, the internal vectorising options in the Intel compiler generate faster executables, in general, than the GNU compiler.

Figs. 7 and 8 present the time to solution for each simulation (full squares) as well as the time required for the jobs to complete (open circles). When using a small number of cores per simulation, the total time required for the job to complete is very close to the time for the slowest simulation. However, when ICORES becomes large (see for example, ICORES = 192 or ICORES = 384 in Fig. 8), TSYSTEM is no longer negligible. On the Cray XE6, comparison between ICORES = 192 and ICORES = 384 shows that the speed-up based on the simulation time is acceptable (1.42) whereas the speed-up based on the job time is poor (1.09).

The value of SPEED-UPOPT is equal to 5.47 for the iDataPlex cluster compared to a theoretical value of 6 assuming a linear speed-up. The corresponding value for the Cray XE6 is 110.09 compared to the theoretical value of 384. This lower than theoretical performance can mainly be attributed to the small size of the grid.

For the iDataPlex cluster, the total elapsed time, TOT_ELCONSECREF, to complete all the consecutive simulations is approximately equal to 1,053,700 s (about 12 days); the corresponding figure for the Cray XE6 is 1,367,678 s (about 16 days). In contrast, running concurrent simulations on the optimum number of cores (ICORESOPT = 6 for the iDataPlex cluster and 384 for the Cray XE6) leads to TOTICORESOPT = 4700 s and 303 s, respectively.

Since TOTICORESOPT is extremely small compared to the total elapsed time, TOT_ELCONSECREF, the reduction in time when running concurrent parallel simulations is about 12 days on the iDataPlex cluster and about 16 days on the Cray XE6. The fact that ICORESOPT is, for both machines, the highest number of cores that could be used per simulation, demonstrates excellent performance of TOMAWAC, even for such a relatively small problem size (57,177 elements).

Figs. 9 and 10 show the relative cost of running each simulation on more than ICORESREF cores, and show that there is a benefit of running TOMAWAC with concurrent parallel simulations on the iDataPlex cluster but there is a penalty when running on the Cray XE6. However, the vastly reduced time to solution is thought to more than compensate for the increased computational costs.

Figure 7. Time to solution for the 41 concurrent TOMAWAC simulations on the iDataPlex cluster.

Figure 8. Time to solution for the 41 concurrent TOMAWAC simulations on the Cray XE6.
VI. TESTS USING TELEMAC-3D

A. Description of the problem

In this case study, HR Wallingford had an advisory role in support of the design and the acquisition of planning permission for a proposed coastal development. Since the project remains confidential, references to names, locations and model values have been modified or removed; thus any resemblance with any particular site is purely coincidental. The bathymetry of the site is shown in Fig. 11.

For this project, HR Wallingford carried out physical model tests aimed at measuring wave overtopping along a new seawall under a range of waves with different return periods and different states of the tide. It was noticed that the combination of strongly oblique waves and a (very nearly) vertical seawall give rise to a local wave travelling along the seawall, termed for this study a ‘stem-wave’ but sometimes also referred to as a ‘Mach-stem wave’. These stem-waves result from the non-linear interaction between the incident and reflected wave along vertical breakwaters for a specific range of oblique incident angles.

A range of numerical models have been developed for the site in order to study more closely the phenomenon of ‘stem wave’ generation along the protected coastline. The numerical study considered the severity of any ‘stem wave’ enhancement, whether that effect might continue to increase beyond the area represented in the physical model, as well as related effects which might cause problems for the adjoining site (in the direction of wave travel). The numerical modelling work was carried out using the TELEMAC-3D module of the TELEMAC system.

The TELEMAC-3D model was set up in non-hydrostatic and wave resolving modes in order to predict non-linear wave propagation, refraction and shoaling in front of the seawall. In this model, the size of the elements varies from 1 m near the seawall to approximately 5 m at the offshore boundary.

Extreme offshore wave conditions were derived based on 20 years of model wind and wave data. The first step was to propagate these conditions closer to the site (not detailed here). Sensitivity tests to the state of the tide demonstrated that extreme nearshore wave conditions were well represented by modelling the conditions at high water. Selected outputs from the wave transformation model were in turn used as input to TELEMAC-3D. Were it not for the time constraints of this short-term project, it would have been desirable to simulate a long record of randomly generated waves. Instead, only a small number of regular, yet non-linear waves, were generated by imposing the water level and the vertical pressure and velocity profiles at the offshore boundary.
B. Analysis of the results

The preliminary results presented here are for waves of amplitude 5 m and of period 10 s, approximating the conditions with a 1-in-200 year return period (see Fig. 12). It is shown that the waves undergo refraction and shoaling as they propagate towards the coast, reaching the seawall with an angle that is less oblique than the incident angle. Local stem-wave like effects were also observed in the model although the effect reduces as the depth of water at the seawall toe decreases. TELEMAC-3D was able to confirm physical model observations. Significantly more computer time would be required to analyse the output from the numerical model statistically.

C. Use of the script on the Cray XE6

A computationally more demanding test has been performed running 12 parallel concurrent TELEMAC-3D simulations, each composed of a horizontal grid of 376,158 elements and 10 layers (3,761,580 prisms in total). This test could only be carried out on the Cray XE6 since the iDataPlex cluster does not have enough cores for a relevant demonstration. All 12 instances of TELEMAC-3D (NCSIMS = 12) were set to run for 6000 time steps. Each simulation was run on at least 24 cores (i.e. ICORESREF = 24). This also corresponds to the number of cores on each node of the Cray XE6.

Fig. 13 presents the time to solution for each simulation (full squares) together with the time required for the jobs to complete (open circles). Comparison between ICORES = 384 and ICORES = 768 shows that the speed-up based on the simulation time (1.71) and the speed-up based on the job time (1.63) are very similar. This is mainly because only 12 concurrent simulations have to by dealt with by the system and therefore TSYSTEM is negligible for such a small number of simulations. The value of SPEED-UPOPT for the TELEMAC-3D jobs on the Cray XE6 is equal to 33.49 compared to a theoretical value of 32.

Finally, Fig. 14 shows the relative cost of running each simulation on more than ICORESREF cores. The analysis indicates that it is actually still cheaper, in terms of the total CPU time, to run on 768 cores rather than on 24 for this particular test case, highlighting the benefits of running concurrent TELEMAC-3D instances in parallel.

VII. CONCLUSIONS AND RECOMMENDATIONS

This paper has presented a script which can be used to run concurrent parallel simulations on High-End computing resources. The script is specifically aimed at sensitivity analyses where it is necessary to run a large number of identical simulations using different input parameters.

The script enables concurrent simulations to be run either in serial or in parallel, depending upon the user’s specific requirements and the size of problem being considered. The results show that the time to solution for both TOMAWAC and TELEMAC-3D can be dramatically reduced with the help of HPC facilities. This will not only improve future calibration exercises but will also allow far
more simulations to be run in order to assess a wider range of design options or model scenarios. In turn, this should lead to increased fidelity of wave-climate predictions for both research and consultancy projects.

ACKNOWLEDGEMENTS

This work was initiated by the South-East England Development Agency (SEEDA) which aims to promote strong relationships between academia and businesses in South-East England. In this context, HR Wallingford was granted a SEEDA Voucher covering STFC Daresbury Laboratory staff and computing time.

The authors would also like to thank the UK Engineering and Physical Sciences Research Council (EPSRC) for their support of Collaborative Computational Project 12 (CCP12) and the Distributed Computing Group at STFC Daresbury Laboratory.

REFERENCES


Abstract—The present study investigates the need to consider the modifications of the waves fields by the ambient water-depths and currents for sediment transport computations with the TELEMAC modelling system. The application is dedicated to the outer Seine estuary (France, English Channel). The wave propagation module TOMAWAC is coupled with the bidimensional horizontal circulation module TELEMAC 2D. Times histories of depth-averaged mean currents and waves heights are compared with field data collected in the access channel to the harbour of Le Havre. Predictions exhibit a local increase by 30 % of the wave height induced by the current. The sensitivity study compares numerical results issued from SISYPHE integrating hydrodynamics of uncoupled and coupled TOMAWAC – TELEMAC 2D simulations. Computations are performed considering successively suspended transport of cohesive mud and bedload transport of sand. The effect of the hydrodynamic coupling is analysed through the variability of the total maximum wave and current bottom shear stress, the maximum suspended sediment concentration and bedload transport rates and the resulting seabed evolutions.

I. INTRODUCTION

The importance of combined surface gravity waves and currents on nearshore sediment transport has long been recognized ([1], [2]). The acquisition of the ability to model accurately the interactions between waves and currents is fundamental in many aspects of coastal, estuarine and offshore engineering. Ways in which waves interact with the currents include at the scale of the continental shelf [3] (1) the interactions between the wave and current bottom boundary layers and (2) the modifications of the wave field by the ambient current. The first type of interaction has been the subject of numerous modelling dedicated to sediment transport ([4], [5]) as it determines the increase of the total wave and current bottom shear stress [2]. Nevertheless, investigations dedicated to the second type of interaction were restricted to the quantification of the wave height modulation and the associated storm surge ([6], [7]).

The purpose of the present study is to analyse the significance of this interaction for sediment transport. The application is dedicated to the outer Seine estuary (France, English Channel) (Fig. 1) characterized by strong interactions between wave and tide with (1) spring tidal range of 7 m and current amplitude of 1.5 m/s [8] and (2) annually wave height over 5 m offshore [9].

At the scale of the outer Seine estuary, the modelling is based on the hydro-informatic finite elements system TELEMAC (section II). The hydrodynamic modelling (section II-A) is based on the coupling of the wave propagation module TOMAWAC [10] with the bidimensional horizontal (2DH) circulation module TELEMAC 2D [11]. The coupling is restricted to (1) the modifications of the wave fields by the time-varying water depths and currents and (2) the interactions between the wave and current bottom boundary layers ignoring the generation of currents by the waves. Sediment transport and morphodynamic evolution of the seabed are computed with the module SISYPHE [12] (section II-B) considering successively two grain-size classes of bottom sediments: mud and sand.

Hydrodynamic numerical results are compared with field data collected with two current meters and two wave buoys in the access channel to Port 2000 (harbour of Le Havre) (section III-B). This comparison exhibits for the month of November 2008 the importance of the modifications of the wave field by the currents for predictions of the significant wave height near the harbour of Le Havre (section III-B). The significance of the modification of the wave field by the ambient water depths and currents is further investigated by identifying the variability of “key” parameters for sediment transport in November 2008 (sections III-B and III-C): (1) the total maximum wave and current bottom shear stress, (2) the maximum suspended sediment concentration (SSC) and bedload transport rates and (3) the resulting seabed evolutions.

II. MODEL DESCRIPTION

The modelling procedure is conducted in three steps successively dedicated to (1) the hydrodynamics of the circulation and wave, (2) the sediment transport and (3) the seabed morphological evolution.

A. Hydrodynamic Modules

Simulations of the circulation and wave propagation are performed with the two nesting modelling systems (i) MISTRAL (Modélisation Intégrée pour la Simulation des TRAnports Littoraux) [13] at the regional scale of the English Channel and the North Sea and (ii) TELEMAC ([10], [11]) at the local scale of the Bay of Seine and the infrastructures of the harbour of Le Havre (Fig. 1). Each modelling system integrates
wave propagation and circulation modules. MISTRAL is based on the wave propagation module SWAN (Simulating Waves Nearshore) [14] and the three-dimensional (3D) circulation module COHERENS (Coupled Hydrodynamical Ecological model for RegioNals and Shelf seas) [15]. TELEMAC considered the wave propagation module TOMAWAC [10] and the 2DH circulation module TELEMAC 2D [11]. The coupling procedure between the circulation and wave propagation modules integrates (i) the interaction between the wave and current bottom boundary layers and (ii) the modifications of the wave fields by the time-varying water depths and currents.

The effects of the interaction between the wave and current bottom boundary layers are implemented in the circulation modules through the increase of the apparent bottom roughness parameter $z_0$ felt by the current above the wave boundary layer. The computational method is based on the formulation proposed by [16]. The module of the total maximum wave and current bottom shear stress is given by

$$\tau_{b,w} = \rho u_{*,w}$$

(1)

where $\rho = 1025$ kg/m$^3$ is the density of clear water and

$$u_{*,w} = u_{*,c} \left[ 1 + 2 \left( \frac{u_{*,w}}{u_{*,c}} \right)^2 \cos \phi_{bw} + \left( \frac{u_{*,w}}{u_{*,c}} \right)^4 \right]^{1/2}$$

(2)

is the total maximum wave and current bottom shear velocity, $u_{*,c}$ is the shear velocity arising from the current alone and computed from the depth-averaged current $\mathbf{U}$ following a Chezy’s law. $u_{*,w}$ is the shear velocity associated with the wave and given by

$$u_{*,w} = \sqrt{\frac{1}{2 f_w} \frac{U_w}{u_{*,c}}}$$

(3)

where $U_w$ is the wave bottom orbital velocity issued from the wave propagation module, $f_w$ is the wave friction factor evaluated with the empirical relations

$$f_w = 0.13 (k_s a_b)^{0.4} \quad \text{if} \quad k_s a_b < 0.08$$

$$f_w = 0.23 (k_s a_b)^{0.62} \quad \text{if} \quad 0.08 \leq k_s a_b < 1$$

$$f_w = 0.23 \quad \text{if} \quad k_s a_b \geq 1$$

(4)

where $a_b$ is the near-bottom excursion amplitude and $k_s = 30 z_0$ is the Nikuradse parameter with $z_0$ the bottom roughness parameter.

Finally, $\phi_{bw}$ is the angle between the wave and current directions and the expression of $z_0$ is given by

$$z_0 = \frac{1}{\delta_w} \left( \frac{u_{*,w}}{u_{*,c}} \right)^{1/2}$$

(5)

where $\delta_w$ is the thickness of the wave bottom boundary layer parametrized according to [2]. Equations (1) to (5) must in principle be solved with an iterative procedure as the current shear velocity depends on $z_0$, which in turn depends on $u_{*,w}$.

Following [17], a simpler numerical approach is considered whereby the value of $z_0$ calculated at each time step is used in the computational method at the next time step.

The wave propagation modules integrates the effects of the time-varying water depths and currents computed by the circulation modules. The major associated modifications have recently been reviewed by [18]. The variation of the water depths modulates the dissipation of the wave energy by bottom friction and the wave breaking. Opposing tidal flow induces steepening of the incident wave field thus increasing the wave height. Finally, the combination of the time-varying water depths and currents leads to wave refraction and a semi-diurnal variability of the incident wave energy at specific coastal locations. A noticeable resulting effect of these processes is the tidal modulation of the wave height in the coastal region.

Further details about the coupling between the different circulation and wave propagation modules are available in [19].

B. Sediment Transport Modules

SISYPHE [12] computes the bedload and suspended load of bottom sediments as a function of the hydrodynamic parameters issued from modules TELEMAC 2D and TOMAWAC.

Suspension is computed using a concentration-based approach where the instantaneous SSC $C_s$ of mud satisfies the depth-averaged advection-diffusion transport equation

$$\frac{\partial}{\partial t} (h C_s) + U_{conv} \frac{\partial C_s}{\partial x} + V_{conv} \frac{\partial C_s}{\partial y} =$$

$$\frac{1}{h} \left( \frac{\partial}{\partial x} \left( h \epsilon C_s \right) + \frac{\partial}{\partial y} \left( h \epsilon \frac{\partial C_s}{\partial y} \right) \right) + (E - D)_{conv}$$

(6)

where $U_{conv}$ and $V_{conv}$ are the depth-averaged convective flow velocities in the $x$ and $y$ directions, respectively. $t$ denotes times,
where the empirical coefficient $M$ is set to $M=0.0005$ kg/m$^3$/s and $u_s$ is the critical shear stress velocity for erosion. The mathematical expression of the deposition rate is

$$D = w_c C_b \left[ 1 - \left( \frac{u_s}{u_w} \right)^2 \right] \text{ for } u_s < u_w$$

where $w_c$ is the settling velocity of suspended sediment and $u_w$ is the critical shear stress velocity for deposition.

Bedload is computed with the formulation extended by Bijker [20] in combined waves and currents conditions. The instantaneous bedload transport rate expressed as

$$q_b = \phi_b \left[ g(s-1)d^3_b \right]^{1/2}$$

where

$$\phi_b = b \theta_u \exp \left( -\frac{0.27}{\theta_u} \right)$$

is the dimensionless bedload transport rate and $\theta_u$ and $\theta_e$ are the non-dimensional shear stresses associated with the current alone and the combined wave and current, respectively. The Bijker formula is recommended by [21] in conditions of combined wave and current for its reliability, simplicity and flexibility. Finally, $b$ is set to $b=2$ according to [22].

The resulting evolution of the seabed is computed by solving the Exner equation extended to total load (including the suspended load).

C. Model Setup

Further details about the setup of the regional circulation and wave propagation modules COHERENS and SWAN are available in [19]. The present section focuses on the implementation of the TELEMAC modelling system in the outer Seine estuary.

TELEMAC 2D is set-up on an outer domain #1 covering the Bay of Seine between the longitudes 1°380 W and 0°433 E and the latitudes 49°253 N and 50°005 N (Fig. 1). The computational domain comprises 8,708 nodes and 16,414 finite elements with a size of 4 km offshore to a few tens of meters close to Le Havre. The time step is set to 10 s. The bottom friction coefficient is computed following a Chezy’s law and the heterogeneous roughness parameter derived from the observed grain size distribution dataset [23]. The circulation module incorporates the wind fields from the database of the National Centers for Environmental Predictions (NCEP) [24]. TELEMAC 2D is driven by the free surface elevations and the depth-averaged currents extracted at one hour intervals from COHERENS regional simulations. The wave fields are provided every 90 min from SWAN regional simulations. Finally, an average flow of 450 m/s is prescribed at the entrance of the Seine river [25].

TOMAWAC is implemented on a inner domain #2 close to the Seine estuary extending from 0°245 W to 0°354 E and 49°253 N to 49°743 N (Fig. 1). The computational mesh comprises 2,462 nodes and 4,507 finite elements with a size of 2 km offshore to a few tens of meters close to the harbour of Le Havre. The module runs with 30 exponentially spaced frequencies ranging from 0.05 Hz to 1 Hz, 15 evenly spaced directions (resolution of 24°) and a time step of 20 s. The sink term of dissipation by bottom friction is parametrized with an uniform bottom friction coefficient $C_b=0.038$ m/s$^3$ according to [26]. TOMAWAC is driven by the wave components extracted every 90 min from SWAN regional simulations. The time-varying water depths and currents are given every 30 min by TELEMAC 2D.

SISYPHE is set-up on the inner domain #2 with the same computational mesh as for the wave propagation module. Total water depths and depth-averaged currents are interpolated from TELEMAC 2D simulations in the inner domain. The bottom friction coefficient is parametrized as in the circulation module. The bed is considered of an uniform single grain size with a rigid bed set 100 m below the superficial sediment layer. In order to investigate the effects of the coupling between the circulation and wave propagation module on sediment transport, simulations are performed considering successively (1) suspended transport of cohesive mud and (2) bedload transport of sand. Consolidation is not taken into account. Sensitivity analyses performed by [27] reveal that consolidation slightly affects the suspended sediment patterns. Furthermore, the localisation of computed deposition patterns appears globally in agreement with field data as most of the mud deposited on neap tide is resuspended during spring tides. The mud shear strength is set to the mean value of 0.5 N/m$^2$ which corresponds to a critical shear stress velocity of $u_s=0.022$ m/s [27]. This value is a little high for a fresh deposit (fluid mud) but low for a consolidated mud. The critical stress for deposition is taken equal to 10 N/m$^2$ ($u_s=0.098$ m/s) following [27]. Flocculation processes are not integrated but implicitly accounted for through a high settling velocity of 1 mm/s according to [28]. Bedload considers sand of a diameter of 200 $\mu$m. The critical shear stress is computed according to [29] with a value of 0.17 N/m$^2$. Finally, the sediment transport module runs with a time step of 10 min during the month of November 2008.

In order to investigate the effects of the modification of the wave field by the time-varying water depths and currents, three numerical experiments are conducted (Tab. 1). Experiments E1 considered the effects of the currents alone without taking into account the superimposed effects of the waves for sediment transport. Experiments E2 and E3 integrate the effects of the waves for sediment transport without and with the modification of the wave field by the ambient water depths and currents. In each experiments, the circulation module integrates the increase of the apparent bottom roughness parameter felt by the current above the wave boundary layer. The fields of the water depths and currents used in each experiments remain consequently the same. This makes it easier to compare the numerical results between each experiment.
III. MODEL APPLICATION AND RESULTS

A. Comparison With Point Measurements

TELEMAC 2D predictions of the depth-averaged currents are compared with measurements realized in the access channel to Port 2000 (harbour of Le Havre) in mean water depths of 19 m at points C1 (λ=0°091 E, ϕ=49°476 N) and C2 (λ=0°077 E, ϕ=49°484 N) (Figs. 1, 2 and 3). The current measurements were realized over the two spring tide periods of 15-16 November 2005 at point C2 and 2-3 January 2006 at point C1. The instrumentation is an upward looking Acoustic Doppler Current Profiler (ADCP) placed on the bottom. The numerical results reproduce the flood/ebb asymmetry at the two measurements points characterized by (1) a magnitude of the flood three times greater than during ebb and (2) a duration of the flood limited to four hours whereas extended to seven hours for ebb. The magnitude of the flood peak is approached with a difference lower than 15 %. A slight phase lag with an average value less than 15 min is however noticed in the comparison of the measured and predicted depth-averaged current directions. Nevertheless, these numerical predictions are considered reliable for an integration in the wave propagation module.

TOMAWAC predictions of the significant wave height are compared with measurements realized at points H1 (λ=0°104 E, ϕ=49°465 N) and H2 (λ=0°087 E, ϕ=49°475 N) as part of an observational system to regulate the ship routing close to the harbour of Le Havre (Fig. 1). The period of comparison corresponds to the month of November 2008 characterized by continuous records with a tidal range of 7 m in November 2008 and a significant wave height of 2.8 m in 23 November 2008 at point H1. When the effects of the time-varying water depths and currents are integrated (case E3), the numerical results reproduce the semi-diurnal variation of the wave height (Fig. 4). An overall good agreement is thus obtained between predictions and measurements. A slight overestimation of the wave height is however noticed at point H2 during the storms of 21 and 24 November 2008. This tendency is reduced at location H1 with a difference lower than 12 % in 21 November 2008. Considering the difficulties to compute the tide-induced modulation of the wave height [30], the present wave modelling seems acceptable to proceed further analysis.

B. Effects of the Coupling on the Hydrodynamics

The comparison of the predictions at points H1 and H2 between cases E2 and E3 exhibits the importance of the tide in the semi-diurnal wave height modulation near the harbour of Le Havre (Fig. 4). The wave height (1) increases by about 30 % at high tide and (ii) diminishes by about 10 % at low tide in the access channel to Port 2000. Further analyses [19] reveal that current-induced refraction appears to be one of the main mechanism responsible for the tidal waves height variation in

<table>
<thead>
<tr>
<th>Experiments</th>
<th>TELEMAC 2D with time-varying waves</th>
<th>TOMAWAC without time-varying water depths and currents</th>
<th>TOMAWAC with time-varying water depths and currents</th>
</tr>
</thead>
<tbody>
<tr>
<td>E1</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E2</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>E3</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>

Figure 2. Measured (black) and computed (blue) time series of (a) the module and (b) direction (clockwise from the north) of the depth-averaged currents at location C1 in January 2006 (from [19]).

Figure 3. Measured (black) and computed (blue) time series of (a) the module and (b) direction (clockwise from the north) of the depth-averaged currents at location C2 in November 2005 (from [19]).
the access channel to Port 2000. At the regional scale of the outer Seine estuary, the effects of the tidal currents overcome globally the effects of the water depths for the variation of the wave height with local exceptions in shallow waters off the cape of La Hève, the shoals of the Seine estuary entrance and along the southern coastline.

The effect of this coupling on the hydrodynamics is further investigated focusing on the total maximum wave and current bottom shear stress (Eq. 1) as it is a key parameter for sediment transport. Figure 5-a and 5-b displays the maximum values of the bottom current shear stress and the total maximum wave and current bottom shear stress for case E3 in November 2008. When waves are neglected (Fig. 5-a), the maximum shear stress appear in the three access channels to the Seine river with an average value of 5-6 N/m². Highest values over 9-10 N/m² are obtained at the entrance of the river and along the southern breakwater of the harbour of Le Havre. This patterns appear also with the superimposed effect of the waves (Fig. 5-b) as the inner Seine estuary is dominated by the action of the tidal current. Nevertheless, waves exhibit the total bottom shear stress over the shoals of the outer Seine estuary and the nearshore areas with average values of 11-12 N/m². Figure 5-c shows the relative difference of the maximum value of the total wave and current bottom shear stress predicted in November 2008 between cases E3 and E2:

\[
\text{RelDiff} = \frac{\sigma_{b,c,w}^{\text{max}}(E3) - \sigma_{b,c,w}^{\text{max}}(E2)}{\sigma_{b,c,w}^{\text{max}}(E2)} \times 100
\]

The modification of the waves components by the time-varying water depths and currents has a major impact in the shallow waters of the outer Seine estuary. The maximum value of the total wave and current bottom shear stress is thus increasing by 50 % over the shoals of the Seine river entrance and decreasing by 8 % in the three access channels. These differences may be attributed to the effects of the time-varying water depths. Over these regions, the reduction of the water depths favours the action of the wave on the seabed in spite of an increase dissipation of the wave energy with bottom friction and wave breaking.

C. Effects of the Coupling on the Sediment Transport

The effect of the modification of the wave fields by the ambient water depths and currents is further analysed by mapping the maximum SSC of mud (Fig. 6) and bedload transport rates of sand (Fig. 7) in November 2008 at the scale of the outer Seine estuary. Waves exhibit the suspension of bottom mud sediments over the shoals of the Seine river entrance with depth-averaged SSC reaching 4 g/l (Fig. 6-a). The hydrodynamic coupling (case E3) globally increases the maximum SSC with a relative difference over 30 % in the shallow waters of the southern Seine estuary (Fig. 6-b). Nevertheless, the depth-averaged SSC appears to decrease by 30 % close to the shoals of the Seine river entrance in relation to the reduction of the total wave and current bottom shear stress (Fig. 5-c). Advection processes are seemingly playing a role in the variability of the SSC restricting the correlation between the suspension and the bottom shear stress (Figs. 5-c and 6-b). The effect of the hydrodynamic coupling is less significant for the maximum bedload transport rates of sand (Fig. 7-a). The tidal currents have a major influence on bedload with highest transport rates along the three access channels to the Seine river. A close relationship is thus obtained between the maximum current bottom shear stress (Fig. 5-a) and the maximum bedload transport rates when waves are integrated (Fig. 7-a). The increase of the total wave and current bottom shear stress with

![Figure 4](image)
the hydrodynamic coupling (Fig. 5-b) results in a slight increase by 0-10 % of the total bedload transport rate (Fig. 7-b).

The resulting seabed evolutions at the end of November 2008 (Figs. 8 and 9) exhibit also the influence of the hydrodynamic coupling on sediment transport. Concerning the suspension of mud, waves increase the erosion of the shoals at the entrance of the Seine estuary and the deposition in the central channel located close upstream (Fig. 8-a). The hydrodynamic coupling tends to favour this evolution increasing noticeably the deposition patterns (Fig. 8-b). Numerical results present a slight final evolution by bedload of sand (Fig. 9-a) with tendencies to fulfil the channel entrance to the Seine river as well as the access channel to Port 2000 (harbour of Le Havre). Weak erosion appears over the northern shoal. The hydrodynamic coupling tends to increase the erosion at the entrance of the Seine estuary (Fig. 9-b).

IV. CONCLUSIONS

A numerical modelling based on circulation, wave propagation and sediment transport modules has been implemented in the outer Seine estuary to investigate the effect of the modification of the wave field by the time-varying water depths and currents on sediment transport. The main outcomes of the present study are the following:

1. The time-varying currents have a major influence in the modulation of the significant wave height near the harbour of Le Havre.
2. The integration of the time-varying water depths in the wave propagation module is fundamental to estimate the increase of the wave bottom shear stress and the total wave and current bottom shear stress in shallow waters.
3. The integration of the time-varying water depths and currents in the wave module is leading to differences in sediment transport modelling, particularly noticeable for the SSC and the associated seabed evolution.

The present study is restricted to the combined effect of water depths and currents on sediment transport predictions. A prospective will consist in drawing the line between the impact of the water depths and currents. A second prospective is the implementation in the modelling application of sand-mud mixtures formulation for erosion/deposition near the bottom and consolidation effects.

ACKNOWLEDGEMENTS

The authors are particularly grateful to Dr. José Ozer (Management Unit of the North Sea Mathematical Models, Belgium) for supplying us with tidal harmonic constituents issued from a long term run of COHERENS at the scale of the North-West European continental shelf and Dr. Fabrice Ardhuin (Institut Français de Recherche pour l’Exploitation de la MER, France) for giving us access to wave data issued from regional runs of WaveWatch III. Simulations were performed on computer facilities CAPARMOR (Calcul PARallèle Mutualisé pour l’Océanographie et la Recherche). The present paper is a contribution to the CETMEF-IUEM joint research program MEMPHYS (Mesure Et Modélisation des Processus Hydrodynamiques et Hydro-Sédimentaires dans les écosystèmes côtiers, http://www.memphy-lgce.fr)
REFERENCES


Figure 8. (a) Seabed evolution by suspension of mud at the end of the month of November 2008 for case E3. (b) Differences between the final evolution computed for cases E3 and E2.

Figure 9. (a) Seabed evolution by bedload of sand at the end of the month of November 2008 for case E3. (b) Differences between the final evolution computed for cases E3 and E2.
Numerical modelling of the Vaugris reservoir on the Rhône with Telemac 3D

Olivier BERTRAND1, Antoine LAVEDRINE1, Adlane REBAI1, Eric DAVID1, Gilles PIERREFEU2, Pierre ROUMIEU2

1 SOGREAH ARTELIA GROUP, 6 rue de Lorraine, 38130 Echirolles, France
2 Compagnie Nationale du Rhône, 4 rue Chalon-sur-Saône, 69007 Lyon, France

olivier.bertrand@arteliagroup.com

Abstract—Production of hydroelectricity, a clean, renewable form of energy, is growing constantly and could be optimised both by carrying out works to improve existing power plants and by taking certain technical aspects into account when building new facilities. This approach is already underway, in particular through the launching of studies to rehabilitate existing facilities, although they do not systematically incorporate technological advances that would enable energy production to be optimised.

The PENELOP2 R&D project aims to improve and promote this approach applied to low-head hydropower schemes, which represent the largest proportion of hydroelectricity generated in France and around the world, with the goal of substantially improving performance. The pilot site of the project is the Vaugris dam on the Rhône river. This construction is composed of a spillway on the right bank, the power plant (with 4 generating sets) and a lock on the left bank.

To meet the objectives of the project, and study hydraulic losses, a numerical model of the currents through the Vaugris reservoir was constructed with Telemac 3D. The model was calibrated and validated for various modes of operation of the plants using current and water level measurements. The capacity of the model to simulate the current field in the complex geometry of the water chamber, with the inclusion of its main geometric features, intones, bulb turbine, is also analysed.

I. INTRODUCTION

Climate change and deteriorating air quality have led the international community to take into account the impact of human activities on the environment. In France, following the introduction of a “Climate Plan” in 2004, the law of 13 July 2005 stipulated that the country would have to reduce its greenhouse gas emissions by 3% per year. It also called for France to diversify its sources of energy production by developing renewable energy (RE).

Hydropower, unlike other types of RE such as wind and solar, provides a continuous and predictable supply. Production can be adapted more quickly to respond to network requirements. Hydropower is therefore the most widely used and competitive type of RE found on the market and accounted for 85.9% of RE production in France in 2007.

Low-head hydropower schemes are those that operate on a run-of-the-river basis on rivers with a high discharge and head of less than 20m. They consist of a dam to create a reservoir, a water intake to channel the flow, a power plant to house the turbine (often a bulb unit) and finally a tail race to return water to the river downstream. The total length of these facilities is often less than 100-200m.

Low-head turbines are extremely sensitive to the quality of flow. Any loss of uniformity caused by disturbance at the intake is immediately felt at the turbine and results in production losses that can quickly become significant.

Certain low-head facilities thus experience difficulties when the flow of water is disturbed, and consequently a loss of efficiency and head (of the order of a few per cent). These problems affect the entire facility, including the upstream and downstream sections and turbines, and therefore require thorough investigation and analysis. Reducing the problems that cause disturbances in the hydraulic passage could significantly improve energy production and would enable the capacity of new facilities to be optimised right from the design stage.

PENELOP2 (Performance ENergetiques, Economiques, et environnementaLes des Ouvrages de Production hydroélectrique de basse-chute – Energy, economic and environmental performance of low-head hydro production structures) is a collaborative research project being conducted by a consortium of companies and university laboratories including the Compagnie Nationale du Rhône (CNR), Alstom Hydro France, Sogreah Consultants, In Vivo Environnement, Actoll, JKL Consultants and Grenoble INP. PENELOP2 is approved by the Tenerrdis competitivity cluster, with funding granted in the framework of the 9th Fonds Unique Interministériel (FUI) programme. The aims of the project may be grouped under four general headings:

• Understanding, qualifying and quantifying inadequate performance at hydropower plants on site.
• Devising new systems for representing in detail what is observed on site.
• Systematically studying the origins and consequences of disturbances in flows.
• Studying ways of monitoring these disturbances and proposing innovative processes and technologies for controlling and correcting them.
Sogreah’s Hydraulic Modelling and Software division is responsible for the numerical model design and construction aspects, and is to produce all the models of the reservoir. One of the keys to success will be to link up the various 3D models of flows upstream, downstream and in the water chamber correctly with those of flows in the turbine and draft tube designed by Alstom.

Vaugris dam on the Rhône was chosen as pilot site as its power losses resulting from head losses in the head race leading to the turbine have been clearly identified by the operator, the Compagnie Nationale du Rhône (CNR). The power plant adjoining the dam, which has a capacity of 18 MW, comprises four bulb units and has a maximum head of about 7m.

This site is to be used to validate the numerical models by comparing their results with measurements taken on site, and to quantify the impact of current patterns on the performance of the power plant. Subsequently, the models will be used to improve the efficiency of the generating sets by testing different geometrical configurations (shape of the invert, contraction, etc.).

The Telemac modelling system was designed initially to study free-surface flow only [1]. The model of the reservoir presented here is to be used to validate the techniques for taking into account confined flows and submerged structures in a first attempt to produce a comprehensive representation of the flows involved.

II. MODEL OF THE RESERVOIR

A. Footprint and bathymetry

All the data required for constructing the model were supplied by the CNR, including bathymetric surveys upstream and downstream of the dam and drawings of the power plant with its hydraulic equipment. The model footprint covers the entire low-water bed of the Rhône over a distance of about 2km upstream the dam [2]. The model also includes the lock approach channel on the left bank (Fig. 1).

B. Grid and boundary conditions

The horizontal grid comprises more than 6500 nodes and 12600 elements. The mole area between the dam and the plant was represented in particularly fine detail as it is here that recirculation is liable to occur, causing loss of capacity in bulb unit 4, which is next to it. The size of the mesh segments varies from 1m to in the immediate vicinity of the water intakes for the four bulb units to 50m upstream of the model, with 3m for the area adjacent to the mole and storage dam (Fig. 2).

The 3D grid is built on the basis of the horizontal grid, which is reproduced 20 times along the water column. The model’s boundary conditions are not of the usual kind. The velocity distribution upstream of the power plant depends directly on the discharges passing through each bulb unit. It is therefore necessary to prescribe discharges downstream of the model, at each bulb unit. The same type of condition is prescribed upstream and the only place where the free surface is controlled is the lock on the left bank of the model.

In order to represent flows immediately upstream of the power plant as accurately as possible, the boundary condition downstream of the model is modified to take into account the submerged inlet of the bulb units. To do so, the real flow sections are calculated and the normal velocities at these outlets are prescribed as a function of the required discharges.

III. MODEL OPERATION

A. Available measurements

A campaign of velocity measurements (ADCP readings) was carried out on 25 November 2010 by the CNR [3]. These measurements were taken on 8 profiles downstream of the plant and 11 profiles upstream in two different plant configurations. In the morning G2 and G4 (right bank next to the dam) were operating at full capacity and G1 (left bank) was providing additional capacity to reach close to 800m³/s (Fig. 3). In the afternoon, G2 was replaced by G3 (config-
uration not shown here). Throughout the measurement period, discharge at the dam was nil.

According to the in-house tests performed by the CNR laboratory, the ADCP devices used gave results with a level of uncertainty of around 5% concerning discharges in steady conditions, based on the average of a series of 4 successive transects. The uncertainty with regard to the instantaneous velocity values is of the order of 10%.

The upstream velocity profiles P1 (profile 1) to P4 demonstrate the influence of the bulb units on the velocity distribution in the section. From P1 to P8, the velocities are higher on the LB (power plant side). The velocities on the following (P9 to P11) are uniform over the entire section. On profile P1, the closeness of the reinforced concrete affects the ADCP compass, as shown by the differences between the ADCP and GPS paths.

Figure 3. Flow rate of the Rhône and through the bulb units on 25/11/2010.

The measurements and model are exploited in the rest of this article in the power plant’s morning operating configuration.

B. Results

In order to avoid the effects of boundary conditions on the model (wave reflection during changes in power plant configuration), it is only run here under stationary discharge conditions. The ADCP profiles exploited are shown in Fig. 4. These profiles are the most representative of the lack of flow uniformity upstream of the bulb units. Fig. 5 to Fig. 8 compare the current intensities indicated by the model results and the measurements.

The lack of spatial uniformity in the velocities immediately upstream of the bulb units appears to be well reproduced by the model. While the discharges flowing through bulb units 2 and 4 are practically the same, the current distribution is not symmetrical and tends to show that the effect of the feeder canal is significant.

The contraction of the current on the right bank and along the mole produces a local increase in velocity opposite bulb unit 4. Fig. 7 and Fig. 8 show that the current tends to become uniform upstream even if the impacts of the dam on the right bank and of the power plant are still perceptible.
In order to quantify the results obtained, Table I gives the discharge, cross-section and mean velocity calculated from the ADCP measurements and obtained from the model. All these results show that circulation inside the reservoir is accurately represented by the model.

To analyse flows immediately upstream of the power plant in greater detail, a new model of bulb unit 4 at Vaugris is now presented.

IV. MODEL OF BULB UNIT 4 AT THE POWER PLANT

The purpose of this model is to consider the civil works part of bulb unit 4 of the power plant upstream of the turbine.

A. Footprint and construction

In order to represent flows on the right bank of the power plant as accurately as possible, a local model of flow inside bulb unit 4 (connected to the dam) was built. In addition to this initial approach, the feasibility of using Telemac for this type of modelling must be validated by additional work using a confined-flow model (OpenFOAM). The long-term aim is to combined the model of the reservoir and that of bulb unit 4 in a single model.

<table>
<thead>
<tr>
<th></th>
<th>Profile 1</th>
<th>Profile 2</th>
<th>Profile 3</th>
<th>Profile 4</th>
<th>Profile 5</th>
<th>Profile 6</th>
<th>Profile 7</th>
<th>Profile 8</th>
<th>Profile 9</th>
<th>Profile 10</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q$</td>
<td>944</td>
<td>796</td>
<td>789</td>
<td>761</td>
<td>789</td>
<td>797</td>
<td>789</td>
<td>797</td>
<td>789</td>
<td>789</td>
</tr>
<tr>
<td>$S$</td>
<td>1527</td>
<td>1256</td>
<td>1264</td>
<td>1349</td>
<td>2955</td>
<td>3451</td>
<td>2874</td>
<td>3201</td>
<td>2927</td>
<td>3226</td>
</tr>
<tr>
<td>$U$</td>
<td>0.62</td>
<td>0.63</td>
<td>0.62</td>
<td>0.59</td>
<td>0.26</td>
<td>0.23</td>
<td>0.30</td>
<td>0.25</td>
<td>0.27</td>
<td>0.24</td>
</tr>
</tbody>
</table>

The model footprint is shown on Fig. 9. The model extends from the screens upstream of the power plant (i.e. the downstream boundary of the previous 3D upstream model) to the turbine bulb.

This type of model requires several adaptations of the code to take into account the confined flow inside the power plant and the turbine bulb. The contraction at the bulb unit inlet is forced by prescribing a spatially varying pressure field. The hypothesis chosen is that of hydrostatic pressure in order to set the free surface elevation at the desired level.

In parallel with this free surface processing, friction is taken into account in the boundary condition of the equations of motion by calculating the shear velocity $u_*$ in rough friction conditions. Finally, for this initial approach, the turbulent viscosity is modified with this proximity of the upper “wall”, still applying the same type of processing that is normally used on the bottom of the domain.
Further downstream the water chamber is taken into account in the usual manner with a free surface that changes both spatially and in time. Finally, the outflow condition is modified by subtracting the planned area of the turbine bulb from the flow section. The 3D grid is shown in Fig. 10.

Figure 10. Three-dimensional grid of the model of bulb unit 4 incorporating free surface forcing to represent the contraction.

B. Preliminary results

The first results are promising (Fig. 11). The flow characteristics are accurately represented. A new campaign of measurements is to be carried out, incorporating a frame supporting numerous ADCP and ASFM sensors in the stop log groove. This campaign will enable the pertinence of this type of modelling to be assessed.

Figure 11. Intermediate result concerning flows inside bulb unit 4.

V. CONCLUSION

The market for renovating and optimising low-head power plants is set to develop continuously as a result of the increased importance that the international community is giving to renewable energy, especially hydroelectricity. It is necessary to carry out in-depth analysis and identify disturbance factors in order to ensure gains in performance. Indeed, low-head turbines are extremely sensitive to the quality of flow. Any loss of uniformity caused by disturbance at the intake is immediately felt at the turbine and results in production losses that can quickly become significant.

On the basis of this work, it was possible to build various complementary operational models. The first measurements performed throughout the reservoir were used to qualify the model upstream of Vaugris power plant. Velocity mapping should now help to identify the initial factors that are disturbing flow.

Rough modelling of one of the plant’s bulb units should help to fine-tune the representation of flows around the power plant when the various models are linked up. However, this is the limit of validity of the Telemac software as a bulb unit consists of numerous complex features (Fig. 12).

Figure 12. Representation of bulb unit 4 at Vaugris power plant using the OpenFOAM software.

It is by multiplying these approaches, which combine on-site measurements and numerical models, that pertinent solutions will be found for fine-tuning hydraulic assessments of low-head schemes. Numerical modelling is now recognised as a reliable way of representing physical phenomena and a technical reference in assessing projects. However, all these approaches are a challenge for the scientific modelling community, which only joint program-mmes like PENELOP2 can handle.

ACKNOWLEDGEMENT

This work was carried as part of the PENELOP2 project approved by the Tenerrdis competitiveness cluster, with funding granted in the framework of the 9th Fonds Unique Interministériel (FUI) programme.

REFERENCES

Modeling of a managed coastal Mediterranean wetland with TELEMAC-2D: the Vaccarès lagoons system (Rhone delta, Camargue, France)

Olivier BOUTRON, Aurélien LOUBET, Philippe CHAUVELON
Fondation Tour du Valat, Centre de recherche pour la conservation des zones humides méditerranéennes, Le Sambuc, 13200 Arles, France
boutron@tourduvalat.org
Olivier BERTRAND
ARTELIA, 6 rue de Lorraine, BP 218, 38432 Echirolles Cedex, France
Patrick HÖHENER
Laboratoire Chimie Provence, Université de Provence, 3 place Victor Hugo 13331 Marseille cedex 3, France
Eric COULET, Yves CHERAIN
SNPN, Centre d’Information de la Réserve Nationale de Camargue, La Capelière, 13200 Arles, France

Abstract— The “Ile de Camargue”, the central part of the Rhône delta (Fig. 1) delimited by the two embanked branches of the Rhône river, is a complex hydroosystem submitted to high natural hydrological variability due to the combination of a Mediterranean climate and the artificial hydrological regime imposed by human water management. It includes agricultural drainage with low elevation gradient, marshes and the brackish shallow Vaccarès lagoons system whose connection with the sea is managed. During rice cultivation, large amounts of water are pumped from the Rhône river for irrigation, generating significant water fluxes. This complex hydrosystem is particularly representative of human influence on water and salinity regimes in a lagoon system. It has been listed as an area for nature conservation since 1927, and provides critical habitats for a wide range of terrestrial and aquatic wildlife. This study presents a two-dimensional hydrodynamic modeling of the “Vaccarès lagoons system” using TELEMAC-2D, with special attention to water fluxes and salinity. Rain and evaporation, which have a great influence on water levels and salinity in this area, were implemented in the code. Water circulations in the lagoons are mainly induced by the wind, thus a spatial interpolation of the wind was tested. Water depths and salinities were simulated on four months. Results were compared with experimental data acquired on several stations for water levels and salinities. TELEMAC-2D appears to be a promising tool to simulate the Vaccarès lagoon system. However this paper presents preliminary results and further studies are needed to calibrate and validate the model.

I. INTRODUCTION

The Ile de Camargue basin is the central part of the Rhône Delta in the South of France, included between the two branches of the Rhône river (see Fig. 1). The higher parts of this area of about 800 km² are devoted to agricultural land (420 km²), mainly rice fields, whereas wetlands, brackish lagoons and marshes occupy lands of lower elevation [1]. This system includes three interconnected lagoons, which form the “Vaccarès lagoons system”. These interconnected lagoons are the Vaccarès lagoon, the “Impériaux” lagoon (IL), and the complex “Lion/Dame” lagoon (LDL) (Fig. 1). The Vaccarès lagoon is the only one receiving runoff from rice farming. “Impériaux” is the only lagoon exchanging water with the sea, through the sea connection at “La Fourcade” (see Fig. 1).

The aquatic ecosystem in the center of the “Ile de Camargue” is internationally recognized as a biosphere reserve within the framework of the UNESCO’s Man and Biosphere Programme. The “Ile de Camargue” is an important area of reproduction of a wide variety of aquatic organisms and birds, and in addition an important resting area for migrating birds.

Tourism activities prevail in the western part of the delta. In the low lands surrounding the lagoons, large freshwater marshes (25 km²) are managed for waterfowl hunting, nature conservation or exploited for their reed beds. Agricultural land borders the park to the North and South-East, and most of it is devoted to intensive flooded rice cultivation. The rice parcels are irrigated during the crop period (from mid-April or early May, to September) by irrigation pumping stations taking water from both arms of the river Rhône. On an unpolderized agricultural area of 112 km² in the north-eastern part of the Ile de Camargue and around the Vaccarès lagoon, the runoff of rice paddies is directly discharged to the lagoon system through two drainage channels (“Fumemorte” (FUM) and “Roquemaure” (ROQ), see Fig. 1), whereas in the northern and south-eastern parts, the runoff of 310 km² of polderized rice paddies is pumped back to the Rhône River or to the Mediterranean sea [1]. The discharge of irrigation induced drainage water to the lagoons is about 480,000 m³ per day during the crop period. This counterbalances partially the salinity increases in the lagoons due to high evaporation rates and due to seawater intrusion, but carries a load of anthropogenic substances into the protected area [2-3]. Therefore, there is an ongoing debate on the management of water in this area.
Figure 1. Map of the “Ile de Camargue” hydrosystem delimited by the two branches of the Rhône river (“Petit Rhône” and “Grand Rhône”). “SMM” and “SDG” are the cities of “Saintes-Maries de la Mer” and “Salins de Giraud”.

The Camargue regional nature park is separated from the Mediterranean Sea by a sea dike. Sea-lagoon exchanges through the sea connection at “La Fourcade” (see Fig. 1) are managed by 13 manual sluice gates. In autumn and winter, the gates are opened during northerly wind to decrease water level in the lagoons. In spring, one to three gates are opened to allow fish recruitment into the lagoons [4]. Water exchanges through the gates are mainly seawards. During sea storms, sluice gates are closed. The exchange of water from the lagoons with the Mediterranean Sea is then limited [1], and the major pathway for water loss from the Ile de Camargue is evaporation during the summer months. Frequent strong winds tend to homogenize salt concentrations within each lagoon and force water exchanges between the lagoons.

This study aims to model water and salt fluxes on the Vaccarès lagoons system at short time period with great spatial resolution. Results of hydrodynamic simulations are also used in conceptual models used for long term runs and prospective analysis, to help stakeholders for water and salt management in this area.

II. METHODOLOGY

A. Monitoring data

1) Hydro-meteorological data

Wind (speed and direction), temperature, precipitation data were measured continuously at stations A and B (see Fig. 1), and averaged on a 60 minutes basis. Hourly potential evaporation was calculated at the same sites as described in detail by P.W. Brown (http://ag.arizona.edu/azmet/et2.htm).

2) Water Levels and salinity

Water levels have been monitored continuously at 9 sites (see Fig. 1), on a 15 minutes basis, since 2002. Water conductivity is measured monthly (converted to salinity) at five locations (see Fig. 1) in the Vaccarès lagoon and the southern lagoons since 1970. In addition, conductivity data are acquired at a time step of 30 minutes at stations C1 since 2003 and C2 and C3 since March 2011 (see Fig. 1).

3) Flow data

At the outlet of the Fumemorte canal, discharge was monitored continuously on a 30-min basis from 1993 to August 2008 using an automatic ultrasonic flow meter, which became out of order afterwards. Since August 2008, we then used a rainfall runoff model to estimate the discharge at the outlets of the Fumemorte and Roquemaure canals [5].

At the Fourcade (lagoons-sea connection, see Fig. 1), the discharge Q (m3/s) through the gates is calculated using the following hydraulic equation:

\[
Q = N \left( \frac{2H}{3} \right)^{3/2} KL \sqrt{g \left[ 1 - \left( \frac{H'}{H} \right)^{4/3} \right]}^{0.85} \tag{1}
\]

where \( L \) is the width of one sluice gate (m), \( N \) is the number of opened sluices, \( H \) the experimental upstream water height above the sill (m), \( H' \) the experimental downstream water height above the sill (m) and \( K \) the discharge coefficient depending of the characteristics of the sluices (–).

4) Bathymetry and vegetation

The bathymetric data used came from echo-sounding campaigns for the Vaccarès lagoon and from manual depth measurements for the lower lagoons and the marshes. Maps of the sea grass bed from 2004 were used to describe the hydraulic roughness in the hydrodynamic model (Réserve Nationale de Camargue, unpubl. data).

B. The numerical model

The two-dimensional mathematical model computing transient flows in the Ile de Camargue was constructed with TELEMAC-2D [6]. The TELEMAC-2D code solves the following four hydrodynamic equations simultaneously:

\[
\frac{\partial h}{\partial t} + \mathbf{u} \cdot \nabla h + h \nabla \cdot \mathbf{u} = S_h \tag{2}
\]

\[
\frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla (u) = -g \frac{\partial Z}{\partial x} + S_u + \frac{1}{h} \left( h v \nabla u \right) \tag{3}
\]

\[
\frac{\partial v}{\partial t} + \mathbf{u} \cdot \nabla (v) = -g \frac{\partial Z}{\partial y} + S_v + \frac{1}{h} \left( h u \nabla v \right) \tag{4}
\]

\[
\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla (T) = S_T + \frac{1}{h} \left( h v \nabla T \right) \tag{5}
\]

where \( h \) is the depth of water (m), \( t \) the time (s), \( u \) and \( v \) the velocity components (m/s), \( S_h \) the source or sink of fluid.
Rain and evaporation, which have a great influence on water levels and salinity in the Ile de Camargue, were calculated in equation (2) with:

\[ S_h = R - E \]  

where \( R \) is rain intensity (m/s) and \( E \) the evaporation rate (m/s), derived from experimental data.

### D. Salinity modelling

\( S_p \) is determined considering a constant mass of salt, with a variation in the volume of water due to the rain and/or the evaporation.

### E. Spatial wind interpolation

To take into account spatial variability of wind, we used the spatial interpolation as described in equations (7–12):

\[ S_x = \frac{1}{h} \rho_{air} a_{wind} U_{wind} \sqrt{U_{wind}^2 + V_{wind}^2} \]  

\[ S_y = \frac{1}{h} \rho_{water} a_{wind} V_{wind} \sqrt{U_{wind}^2 + V_{wind}^2} \]  

\[ U_{wind} = U_A \left( \frac{D_B}{D_A + D_B} \right) + U_B \left( \frac{D_A}{D_A + D_B} \right) \]  

\[ V_{wind} = V_A \left( \frac{D_B}{D_A + D_B} \right) + V_B \left( \frac{D_A}{D_A + D_B} \right) \]  

\[ D_A = \sqrt{(X_A - X)^2 + (Y_A - Y)^2} \]  

\[ D_B = \sqrt{(X_B - X)^2 + (Y_B - Y)^2} \]

where \( S_x \) and \( S_y \) are the source or sink terms in the dynamic equations (m/s), \( a_{wind} \) the wind forcing coefficient (\(-\)), \( U_{wind} \) and \( V_{wind} \) the components of wind velocity (m/s), \( \rho_{air} \) and \( \rho_{water} \) the densities of air and water (kg/m\(^3\)), \( X_A \), \( Y_A \), \( X_B \) and \( Y_B \) the horizontal space coordinates of the two meteorological stations A and B (Fig. 1), \( D_A \) (m) the distance from the meteo station A \((X_A, Y_A)\) to a point of coordinates \((X, Y)\), and \( D_B \) (m) the distance from the meteo station B \((X_B, Y_B)\) to a point of coordinates \((X, Y)\).

### F. Mesh and altitude

The non-structured grid consisted of some 17767 triangular elements of varying sizes and 9745 nodes (Fig. 2). The triangular elements ranged in size from 250 meters in the different lagoons to 2 meters in the different channels between IL and LDL lagoon sub units, and between the Vaccarès lagoon and LDL.

The resulting altitude is shown in Fig. 3. The different lagoons of the Vaccarès lagoons system are characterized by a minimum altitude of -2mNGF with a maximum altitude of 1.9mNGF. For a mean water level in the Vaccarès lagoons system of 0 mNGF, the area covered is about 110 km\(^2\), and the volume of water is about 108 \(10^6\) m\(^3\). For a mean water level of 0.5 mNGF, the lagoons store a total volume of about 163 \(10^6\) m\(^3\). The French topographic datum (zero NGF) was fixed in Marseille at the end of the 19th century by averaging local mareographic records between 1885 and 1897 [7].
55 m\(^{1/3}\)/s for a bed with vegetation and equal to 120 m\(^{1/3}\)/s when no vegetation was observed.

### H. Boundary and initial conditions

Three liquid boundaries were considered for i) the lagoons-sea connection (“Fourcade”, see Fig. 1), ii) the FUM canal and iii) the ROQ canal. At the “Fourcade” boundary, semi-experimental water flow (see (1)) and experimental salinity (see gauge C1 in Fig. 1) were imposed. At the FUM and ROQ boundaries, water flows estimated with the rainfall runoff model [5] were imposed. Other boundaries are rigid boundaries.

Initial salinities and water levels were derived from experimental data routinely acquired each month at six locations (see “salinity measurement” in Fig. 1) during five field campaigns (16/03/2011; 14/04/2011; 18/05/2011; 10/06/2011 and 30/06/2011). The punctual measurements were extrapolated to the different lagoons to obtain the initial conditions of water level and salinity.

### I. Events simulated

According to available data, simulations were conducted from March 2011 to June 2011. In accordance with the field campaigns, four runs were performed:

- Run 1: from 16/03/2011 to 14/04/2011
- Run 2: from 14/04/2011 to 18/05/2011
- Run 3: from 18/05/2011 to 10/06/2011
- Run 4: from 10/06/2011 to 30/06/2011

### III. Preliminary results

This study is a preliminary work, more experimental data are needed to calibrate and validate the model. However, preliminary results are presented here.

#### A. Model performance criteria

To measure the model performance for each stations, two criteria were used, the mean absolute error (MAE) and the root-mean-square error (RMSE) of difference between simulated and measured water levels and salinities.

#### B. Water levels

Experimental and simulated water levels where compared for stations C2, C3 and WL1, as shown in Fig 1. An example of comparison is given Fig. 4 for the station C2 for Run 2.

MAE and RMSE of difference between simulated and measured water levels are given in Table 1 for Runs 1, 2, 3 and 4. MAE values range from 0.01m for station C2 and Run 1, to 0.05m for station C3 and Run 2 and 3. RMSE values range from 0.02m for stations C2 and WL1 for Runs 1 and 4 respectively, to 0.06m for station C3 and Run 2. Higher values are generally observed for stations C3 and WL1. This can be explained by problems with experimental water levels monitored at station WL2 (South-East of the Vaccarès lagoon, see Fig. 1). We could not use these data to determine the initial water levels conditions in the model for the Vaccarès lagoon. In a first approach initial water level in the Vaccarès lagoon was chosen as equal to water level monitored in station WL3. However at the beginning of Runs 1, 2, 3 and 4, wind conditions induce higher water levels in the western part of the Vaccarès lagoon than in the eastern part. This implies lower simulated flow from the Vaccarès lagoon to the LDL lagoon than the observed one, and lower simulated water level in the LDL than the observed one.

#### C. Salinities

Experimental and simulated salinities where compared for stations C2 and C3, as shown in Fig 1. Examples of comparisons are shown for Run 2 for stations C2 (Fig. 5) and C3 (Fig. 6).

MAE and RMSE of difference between simulated and measured salinities are given in Table 2 for Runs 1, 2, 3 and 4. MAE values range from 1.5g/L for station C2 and Run 4, to 5g/L for station C3 and Run 4. RMSE values range from 2g/L for station C2 and C3 and Run 4 to 5g/L for station C3 and Run 4. These rather high values can be explained by a lack of experimental data to describe new initial conditions for water levels have to be used, using data of stations WL3 and WL4, and using a new station in WL3.

#### TABLE I. MEAN ABSOLUTE ERROR (MAE) AND ROOT-MEAN-SQUARE ERROR (RMSE) OF DIFFERENCE BETWEEN SIMULATED AND MEASURED WATER LEVELS FOR RUNS 1, 2, 3 AND 4.

<table>
<thead>
<tr>
<th>Station</th>
<th>MAE (m)</th>
<th>Run 1</th>
<th>Run 2</th>
<th>Run 3</th>
<th>Run 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>C2</td>
<td>0.01</td>
<td>0.03</td>
<td>0.02</td>
<td>0.04</td>
<td></td>
</tr>
<tr>
<td>C3</td>
<td>0.04</td>
<td>0.05</td>
<td>0.05</td>
<td>0.03</td>
<td></td>
</tr>
<tr>
<td>WL1</td>
<td>0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.02</td>
<td></td>
</tr>
</tbody>
</table>

![Figure 4. Experimental and simulated water levels in the station C2 (see Fig. 1) for Run 2 (from 14/04/2011 to 18/05/2011).](image-url)
the initial conditions for salinities in the model. Additional salinity data are then needed to calibrate and validate the model.

Figure 5. Experimental and simulated salinities at the station C2 for Run 2 (from 14/04/2011 to 18/05/2011).

Figure 6. Experimental and simulated salinities at the station C3 for Run 2 (from 14/04/2011 to 18/05/2011).

IV. CONCLUSION

TELEMAC-2D appears to be a promising tool to simulate the Vaccarès lagoons system. However this paper presents preliminary results. Further studies are needed to calibrate and validate the model.

First, it appears necessary to acquire more experimental data. Continuous salinity data are indeed only available for three stations (C1, C2 and C3, see Fig. 1). New stations are planned in 2011 to have more spatial continuous data for salinities and water levels. In addition, water discharges at the outlet of the Fumemorte and Roquemaure canals are estimated with a rainfall runoff model. Installation of flow meters are planned to have physical measurement of these discharges.

Another important point would be to measure flows in the channels between the Impériaux lagoon (IL) and the Lion/Dame lagoon (LDL), and between the Vaccarès lagoon and the Lion/Dame lagoon, to calibrate and validate the model considering flows.

It would also be interesting to have data of another meteostation, to test other spatial interpolation for the wind effect.

In the presented simulations, maps of the sea grass bed from 2004 were used to describe the hydraulic roughness in the hydrodynamic model. New maps are planned for 2011 and 2012 to have a better description of the hydraulic roughness.

It would also be interesting to test TELEMAC-3D in this area as we expect a vertical variation of the salinity in the different lagoons.

ACKNOWLEDGEMENT

The authors would like to thank: The MAVA and PRO VALAT Foundations, the French Ministry of Ecology, Energy, Sustainable Development and Sea for research grants within the GICC2 – IMPLIT project; LITEAU2 – GIZCAM Project, ERA NET CIRCLE – MEDCODYN project; and the “Provence-Alpes-Cote d’Azur Region” for a PhD grant.

REFERENCES

Three-dimensional flow patterns at a river difluence on the alluvial system of the Paraná River, Argentina

Mariana I. MORELL, Carlos A. VIONNET
Centre for Water & Environmental Studies at FICH
UNL & CONICET
Santa Fe, Argentina
cvionnet@fich1.unl.edu.ar

Pablo A. TASSI
Laboratoire National d’Hydraulique et Environnement
EDF R&D
Chatou, France
pablo.tassi@edf.fr

Abstract— Whilst identification of secondary flows may be straightforward in open-channels with regular geometry and slowly varying plane curvature, it is not so in the case of natural meandering streams, whose boundaries are loose and irregular. Indeed, due to the continuously changing plan form and variable bed topography, the hydrodynamics of a natural meandering stream is rather complex. The flow field is strongly three-dimensional (3D), and in each cross-section of the meandering stream a cross-flow develops. Thus, on the basic flow there is superimposed a flow in the transverse direction which occupies the large part of the cross-section, whose formation is understood in terms of the mechanical imbalance between the local elevation of the free surface and the centrifugal force induced by channel curvature. The behavior of the cross-flow measured at the outlet of the Colastiné River, Argentina, where the flow diverts in two branches forming an almost T-shaped difluence, is briefly analyzed here with the open source code Telemac-3d. This communication reports the comparison of cross-flows captured with two acoustic Doppler current profiler (aDcp) in the study site against numerical solutions obtained with models of increasing complexity: i) hydrostatic 3D model, ii) full non-hydrostatic 3D model, in conjunction with the zero-equation and two-equations turbulence models.

I. INTRODUCTION

In the last decades, theoretical [12,1] as well as experimental research [2,3] on circular open-channel flows have emerged with the hope of uncovering part of the mechanisms responsible for river meandering. Despite the abundant research available on the subject, insight into the relevant meander processes is still incomplete [6,3]. Just to describe various stages of channel development, numerical models require the ability to compute the interaction among bend flow, transport of bed sediments, and failure of erodible banks over a wide range of time-space scales [8,3].

It has long been known that in meandering open-channels, the flow curvature gives rise to secondary circulation resulting in the classical 3D helical motion, partially responsible for the bank erosion processes observed in natural channels [1,2,3,13]. Whilst identification of secondary flows may be straightforward in open-channels with regular geometry and slowly varying plane curvature, it is not so in case of natural streams, whose boundaries are loose and irregular. Due to the changing planform and bed topography, the hydrodynamics of a natural meandering stream is rather complex. A cross-flow is superimposed to the basic flow, whose formation is understood in terms of the mechanical imbalance between the local elevation of the free surface and the centrifugal force induced by channel curvature. Secondary currents indeed represents a local process that scales with channel width $b$ and water depth $h$, and behaves different depending on the aspect ratio $\beta = b/h$ [9]. Most known field data related to cross-flow formation lies within $10 \leq \beta \leq 15$ [10].

The present paper focuses on the hydrodynamics of curved open-channel flows, whose accurate description is required for further understanding of river meandering. To that aim, the behaviour of the cross-flow measured at the outlet of the Colastiné River, where the flow diverts in two branches forming an almost T-shaped difluence (Fig. 1), is analyzed by comparing detailed field data with numerical results obtained from mathematical models of increasing complexity, as more mechanisms are brought into play.

Therefore, the present paper’s objective is to report some preliminary comparison of cross-flows captured with two acoustic Doppler current profiler (aDcp) in the study site, where the incoming flow experiences an acute turn

Figure 1. Study site on the alluvial system of the Paraná river, nearby Santa Fe city, Argentina.
which makes it prone to centrifugal effects, against numerical solutions obtained with 3D formulations tied to the zero-equation turbulence model (constant eddy viscosity value) and the two-equations standard $k-\epsilon$ model [5, 18, 22].

Next, a description of the mathematical models used are given first, starting with the classical explanation of the formation of a single cell of cross-stream circulation as reviewed by Engelund [1]. Then, few details of the 3D models are given since they are fully described elsewhere (see e.g. [18]), followed by the method used to capture field data. Preliminary results show that all models yield sounded solutions. Nevertheless, it is shown that in order to compute the figure of merit to characterize their performance, an unbiased treatment of the field data should be obtained first.

II. MATERIALS & METHODS

A. Mathematical Models

Many engineering problems involving water motion can be treated as shallow turbulent flows, where the shallowness condition $h/l \ll 1$, valid whenever the depth $h$ of the water layer is small compared to the wavelike extent $l$ of the fluid motion, is achieved. Flow in compound channels and coastal waters are just few examples of turbulent flows that can be analyzed with the shallow water assumption, also known as the long-wave approximation.

Two mathematical models describing the 3D velocity field $(u, v, w)$ and the water depth $h$ (bounded from below by a fixed bed and from above by a free surface), based upon the shallowness condition $h/l \ll 1$, are tested here numerically with two different formulations of TELEMAC-3d [18, 21]. The first one is a layered-average formulation obtained from the Navier-Stokes equations previously averaged in the sense of Reynolds (RANS), whereas the second refers to the full 3D RANS formulation [18]. Nevertheless, it is appropriated to highlight first the salient aspects of known analytical results on curved open-channel flows (see e.g. [1, 7]).

1) Conceptual Model of Helical 3D Flow: The formation of a single cell of cross-stream circulation is well understood [12, 1]. The long wave approximation reduces the fluid motion in vertical direction $z$ to a mechanical balance between gravity and pressure yielding the hydrostatic pressure distribution $g + \rho \partial p / \partial z = 0$, where $g$ is the acceleration of gravity, $\rho$ the fluid density, and $p$ the fluid pressure. Now, and with reference to Fig. 2, if $(r, \theta, z)$ are the cylindrical co-ordinates in radial, azimuthal, and upward directions, respectively, the radial velocity component, $u_r$, occurs in planes perpendicular to the primary-flow component $u_\theta$ and is originated by the centrifugal acceleration $u_\theta^2 / r$ due to channel curvature. Then, a simple order of magnitude analysis [12] reduces the set of equations governing the flow on curved open-channels to

\[
\frac{\partial (ru_r)}{\partial r} + \frac{\partial u_r}{\partial \theta} = 0 \quad (1)
\]

\[
g S + \epsilon \frac{\partial^2 u_\theta}{\partial z^2} = 0, \quad (3)
\]

where $z_b$ is the free surface elevation above datum, $\epsilon$ is the eddy viscosity coefficient --assumed constant-- and $S$ is the longitudinal channel bed slope which satisfies $Sr = S_0 R$, where $S = -dz_b/d\theta$ and $S_0$ is the slope along the channel centre at $r = R$. Here, $z_b$ is the bed elevation above datum.

Direct integration of (3) yields a parabolic distribution for $u_\theta$, resolved by Engelund [1] after assuming a free-slip velocity at the bed level. Then, since the stream-wise velocity component, $u_\theta$, varies from nearly zero at the bed to a maximum value at or near the surface, centrifugal effects are greater near the surface and less intense toward the bed. The centrifugal force is mostly counterbalanced by the radial pressure gradient, which has been assumed to be dominated by a hydrostatic balance manifested as a local hydraulic gradient in radial direction, $\partial z_b / \partial r$ (known as the transverse elevation phenomenon of the free surface). The balance of both forces can hold only for a certain single element, situated somewhere close to the central portion of the water column and moving with a velocity equal to $u_\theta^p$. For particles moving near the upper portion of the water column with velocity $u_\theta > u_\theta^p$, the centrifugal force will be greater than the hydrostatic pressure gradient. These particles will be conveyed away from the centre of curvature. On the contrary, particles situated in the lower portion of the water column, for which $u_\theta < u_\theta^p$, will be moving toward the centre of curvature (Fig. 2). Integration of (2) yields a polynomial of 6th degree that fits a distribution similar to the profile depicted in Fig. 2. From continuity considerations, a non trivial vertical velocity component $u_z$ will develop near the lateral banks, and the velocity field will acquire a 3D helical flow pattern round the river bend.

A difficulty arises whenever the cross-stream is to be captured in the field, where the turbulent flow is far from being uniform and the bed geometry is irregular.

2) Hydrostatic 3D Model: Giving the simple mechanical unbalance of forces that triggers the formation of a single cell of secondary circulation, it is natural to invoke the shallowness of the flowing water layer to approximate the vertical momentum equation with an hydrostatic balance of forces. Additional hypotheses support the use of constant eddy viscosity for the prediction of turbulence mixing, as
well as the so-called Boussinesq approximation where the variation of density is neglected everywhere except in the
buoyancy term, if any, allowing thus for a linear change
between temperature differences at bottom and top of each
fluid layer. The remaining momentum conservation laws at
each layer in the horizontal directions are to be solved in
combination with the layer-averaged continuity equation
\[ \frac{\partial \rho u}{\partial x} - \frac{\partial \rho v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]
(4)

where \((x, y)\) represents the Cartesian coordinates in
the horizontal plane, and \(w_{upper}\) and \(w_{lower}\) are the vertical
component of velocity at the upper and lower limits of each
layer of size \(\Delta h\), respectively. Finally, the solution of the
overall mass-conservation law yields the water depth, and
with it, the position of the free surface. A detailed
description of this type of model can be found in [4, 18].

3) Full 3D RANS Model: The governing equations can be
found elsewhere [see e.g. 18, 21], which are essentially
the full 3D Navier-Stokes equations previously averaged in
the sense of Reynolds, albeit with the pressure force term
split here into a hydrostatic component and a dynamic
component, \(p_d\):
\[ p = p_{atm} + p_0 \delta (z_0 - z) + \int_L \frac{\Delta \rho}{\rho_0} dz_p + p_d \]
(5)

This splitting is required not only to inherit part of the
shallowness condition invoked before, but also for its
intrinsic numerical stability advantages when there is a
deal-off between the static pressure force component and
the local gradient of the free surface [11]. Above, \(p_{atm}\) is
the atmospheric pressure, and \(p_0\) a fluid density reference
value.

B. Field Data Acquisition

1) Study site: Field measurements at the study site along
predetermined transects are being conducted periodically
since 2004, where the Colastiné River diverts in two
branches (Fig. 3). One of this branches leads to the Santa
Fe's city harbour through a channel excavated artificially at
the beginning of the XXth century. As a consequence of
this “artificially” induced river curvature, the incoming
flow from the Colastiné River experiences an acute turn at
the channel inlet which makes it prone to inertial effects by
centripetal forces. The field site is within the alluvial
system of the Paraná River (Fig. 1), which is a large low
gradient sandy river with a water surface elevation drop of
the order of 3 to 5 cm per km, i.e., \((3–5) \times 10^{-5}\). The river
bed is characterized by fine and medium size sands, with
banks composed of approximately 4-6 m layer of clay and
silt overlying coarse sands.

2) Field Equipment: Two different aDcp have been
used systematically mostly in low-medium flow conditions
(Table 1), a SonTek RiverSurveyor and TRDI Rio Grande
operating at 1000 kHz and 1200 kHz, respectively. Water
velocity and bathymetry data were collected using one of
the aDcp in tandem with a digital 210 Hz Raytheon single
beam echo-sounder, in turn coupled to a differential Global
Positioning System (dGPS) receiver Leika with Real-Time
Kinematic (RTK) technology, which provided centimeter-
level accuracies in \((x, y)\) and \(z\) \(\pm 0.02\) m in planar and
\(\pm 0.04\) m in vertical dimensions, respectively. For sake
of simplicity, description of adopted field procedures are
mostly restricted here to the SonTek aDcp, whose proprietary
software package RiverSurveyor was used for data
acquisition and integration with position information from
the dGPS. Similar procedures were later followed for the
TRDI Rio Grande. The aDcp were mounted on the side of a
fiberglass-hull vessel of 6.4 m in length, when a combined
bathymetry and flow velocity field survey were carried out
using the moving vessel methodology [16, 17]. A second
serial port to collect dGPS input signals was attached to the
on-board computer during the surveys, whose geographical
data were later converted to TM (Transverse Mercator)
coordinates, whereas the aDcp internal compass and tilt
sensor (roll/pitch) referred water velocities components in
terms of East-North-Up (ENU) coordinates.

3) Flow Measurements: Exploratory measurements
were taken on November 4, 2004, when the vessel surveyed
few transects roughly orientated perpendicular to the
expected primary flow direction. Then, once secondary
cells were detected with the first version of an in-house
computational code devised to process the field data, more
careful campaigns for data collection were organized. On
April 27, 2006, each cross section to be measured was
drawn in advance following rays that departed from a
virtual centre of curvature, and orientated approximately
perpendicular to the true channel inner bank (Fig. 3). The
virtual centre of curvature was defined by fitting a circle to

| Table I. Field Data Trips, \(z_a\): water stage measured at SPé's Harbour, \(Q\): mean discharge at channel inlet, \(V\): mean-vessel velocity, \(\Delta z\): cell (bin) size, and \(\Delta t\): sampling interval |
|-----------------|----------|--------|---------|------|
| Date | \(z_a\) [m] | \(Q\) [m³/s] | \(V\) [m/s] | \(\Delta z\) [m] | \(\Delta t\) [s] |
| 2004\(^a\) | 11.35 | 856 ± 98 | 0.7 | 0.90 | 10 |
| 2004\(^b\) | 11.35 | 917 ± 101 | 0.7 | 0.50 | 5, 10 |
| 2007\(a\) | 12.33 | 663 ± 116 | 1.5 | 1.10 | 10 |
| 2008\(b\) | 10.83 | 622 ± 119 | 1.2 | 0.75 | 10 |
| 2009\(a\) | 13.32 | 1083 ± 126 | 0.6 | 0.90 | 10 |
| 2010\(b\) | 12.83 | 1096 ± 73 | 1.4 | 0.25 | 0.59 |

\(a\) SonTek, \(b\) TRDI
Intrinsic operational limitations of aDcp architecture renders them unable to measure near all cross-section boundaries. The Sontek aDcp used has a profiling capability ranging from 1.2 m to 40.0 m, with cell-sizes going from 0.25 m to 5.00 m, and minimum blanking distance of 0.7 m. The equipment was set with a minimum blind distance from the water surface to the centre of the first bin of 1 m approximately \([= 0.7 \text{ m (blanking distance)} + 0.2 \text{ m (probe submergence)} + \Delta z/2]\). With this setting, the blind distance at the top layer (near the free surface) and the less resolved bottom layer (near the riverbed) rendered an unmeasured depth ranging from 55% to 15% for water columns located in shallow and deep zones, respectively.

Other issue was related to the lateral coarseness of the data given the narrow width of the channel inlet. On the time spanned by the sampling interval, the aDcp profiles the water column hundreds of times, whose backscatter data is then properly averaged and assigned to each bin centre in vertical direction, and to the midpoint of the travelled distance \(\Delta s\) covered between time \(t\) and \(t+\Delta t\). The lateral vessel displacement \(\Delta x\) can be estimated with the expression \(\Delta s \approx V \Delta t\), where \(V\) is vessel speed (Table 1). For a fixed \(\Delta t\), the higher \(V\) the coarser the water column is. However, as it is shown later, the window of bulk moving water scanned was large enough to capture cells of cross-flow.

C. Digital Terrain Model Generation

A semiautomatic approach to generate high quality DTM from scatter elevation points, surveyed from a moving boat with an echosounder and an aDcp both connected with a dGPS, and different procedures were bundle [27]. A data manager was developed based on separated steps to merge point data and line and polygon data from complementary sources and/or interfaces. This topographic data manager tool comprises the use of SMS (Surface Modelling System) developed by the EMRL at Brigham Young University [24], the visualization tool Tecplot [25] and a set of in-house routines written in Fortran 95 [27]. It allows the user to alter the outcome of the interpolation from the scatter data using different interaction tools and/or criteria, which may guide the user with the continuous assessment of the DTM generation process. The source data must be in plain ASCII format specifying \((x,y,z)\), as defined above, with the inclusion of: i) the scatter set of bed elevation data, ii) the boundary data along riverbanks and limiting cross-sections in form of lines and polygons produced by the SMS, whose elevation is chosen by the user. Here, the scatter set of elevation data comprises isolate points collected along vessel paths with an echosounder, and depth data estimated through the four rays readings of the aDcp TRDL, corrected by roll and pitch with the aid of the VTM [26].

Then, an adaptive tessellation of the domain is constructed with a Delaunay triangulation for the scatter point set. A Delaunay triangulation for a set \(P\) of points in the plane is a triangulation \(DT(P)\) such that no point in \(P\) is inside the circumcircle of any triangle in \(DT(P)\). Connecting the centers of the circumcircles produces the Voronoi diagram or Thiessen polygons of the surrounding scatter points, such that there is only one Thiessen polygon in the triangular irregular network (TIN) for each scatter point. The TIN so obtained, exported with Tecplot in ASCII format, enclosed a bounded domain that contains elements located outside some concave portion of the riverbanks. Those elements are then deleted by hand from the TIN data, and the scatter set of elevation data is interpolated using linear base functions onto a regular grid, whose size is defined by the user. The algorithm defines which node of the regular grid is wet (inside the domain) or dry (outside the domain), information that is later used to pass a 2D Laplacian kernel to smooth out the resulting interpolation. Then, a triangular finite element mesh (FEm) with the expected fine detail on critical areas, as well as fitted along internal boundaries where the flow has been measured in the field, is produced with SMS and later exported into Telemac format with an user specific interface [32]. The elevation data exported to Telemac is that of the regular grid containing both constant elevation points (outside the domain boundary), and highly accurate albeit properly smoothed bed data (inside the domain). The Fudaa interface of the open source Telemac System finally bounds the FEm with the exported bed bathymetry, mesh that is later used by Telemac-3d in the computations (Fig. 4).

D. Finite Element Computations

One of the reasons the Finite Element Method (FEM) is being increasingly used to study environmental problems...
involving river and tidal flows is because its ability to ease the treatment of boundary conditions, bottom topographies and geometrically complex domains with high accuracy [23]. The 3D numerical codes used in this work belong to the open source Telemac System [18, 20], which solves the 3D Navier-Stokes equations with a FEM discretization under a hydrostatic and non-hydrostatic approximations [18, 21]. Telemac-3d is currently developed by the research and development department of Electricité de France (EDF) and the Telemac Consortium [20]. The Telemac-3d code has been fully parallelized using the Message Passing Interface (MPI).

The hydrostatic approximation consist on neglecting the vertical acceleration, diffusion and source term in the momentum equations. The non-hydrostatic approximation is based on the pressure decomposition into hydrostatic and hydrodynamic parts, allowing an accurate computation of the vertical velocity, which is now coupled with the whole system of equations. The overall algorithm for the solution of the hydrostatic 3D model is given hereafter: (i) computation of the advected velocity components by solving the advection terms in the momentum equations; (ii) determination of the new velocity components by taking into account the diffusion and source terms in the momentum equations (intermediate velocity field); (iii) computation of the water depth from vertical integration of the continuity equation and momentum equations by excluding the pressure terms; and (iv) determination of the vertical velocity \( w \) from the continuity equation and computation of the pressure step by the Chorin method [18]. The overall algorithm for the solution of the 3D non-hydrostatic model can be summarized as: (i) a hydrostatic part, which is almost exactly to the hydrostatic model described before, with the exception that the vertical velocity is also advected and diffused (in this step the free surface function is also determined); and (ii) a non-hydrostatic part, in which the velocity field is corrected by the dynamic pressure gradients in order to fulfill the divergence-free constraint [18, 21].

The 3D finite element mesh is obtained by first dividing the two-dimensional domain with non-overlapping linear triangles and then by extruding each triangle along the vertical direction into linear prismatic columns that exactly fit the bottom and the free-surface. In doing so, each column can be partitioned into non-overlapping layers, requiring that two adjacent layers comprise the same number of prisms. Turbulent stresses and turbulent fluxes are modelled using turbulent viscosity and turbulent gradient diffusion hypothesis, which introduce eddy viscosity and eddy diffusivity, respectively. Several turbulence-closure models are available in Telemac-3d [18]. Two turbulence closure models are used here: a constant eddy viscosity model and the standard \( k-\varepsilon \) turbulence model [5, 22].

In the present study, boundary conditions can be specified as follows: at the inflow boundary, all flow components are prescribed by imposing a velocity profile to provide a certain inflow discharge; at outlet boundaries, the normal gradients of all variables are set equal to zero. This homogeneous Neumann boundary condition implies that the surface integrals resulting from integration by parts in the variation formulation vanish. On the solid boundaries, the velocity tangential and normal to the boundary are set to zero. Finally, the position of the water surface is determined as described in [18, 21]. For the \( k-\varepsilon \) model, the boundary conditions are specified according to Burchard [22].

In all computations the domain is discretized with an unstructured triangular mesh consisting of 6623 triangular elements and 15 layers in the vertical direction, corresponding to 99,345 prisms. The time step is set equal to 0.1 s. For a given initial condition consisting on a water surface elevation of 13 m and velocity components equal to zero, the steady state is reached after about 100,000 time steps, corresponding to a physical time of about 2 hours 45’. For all simulations, the inflow discharge was 2416 m\(^3\)s\(^{-1}\) and a fixed surface elevation of 13 m was imposed at the outflow boundaries. When ran on eight processors of a Z600 HP workstation, the typical convergence time of the simulations presented in this paper was close to 35’.

### E. Treatment of Field Data

Occasionally, reported results seem to be vague and open to different interpretations depending on the method used to extract secondary currents from the field data. The vast majority of researchers resort to the so-called Rozovskii method, which isolated the excess (or deficit) of the transverse velocity component relative to the respective depth-averaged value on any vertical profile. In brief, the method accounts for a rotation of the planar velocity vector with respect to the direction of the depth averaged velocity vector [12]. However, the procedure depends upon having zero net secondary discharge at the vertical, a condition normally used to close the mathematical problem posed by (1)–(3) albeit unrealistic in practical situations (where no fluid particle remains in the cross-wise plane as it is advected downstream by the primary flow).
Dinehart and Burau [14] proposed a method in two steps: firstly, a bend-crossing plane of velocity vectors from aDcp data is derived, secondly, elements of the backscatter intensity planes are used to guide an interactive alignment of the averaged velocity grids previously obtained. They found the bend-crossing plane through a section straightening procedure, where the velocity ensembles are spatially translated to a straight line defined by a mean crossing line fitted along multiple transects.

Two types of outputs are reported here, depending on the instrument used to capture the field data (Sontek or TRDI), and the algorithm used to extract the secondary currents from it. Usually, the procedure involves two steps, firstly using proprietary software, and then using some sort of an ad hoc software. For data captured with the TRDI aDcp, the 3D flow velocity data was first filtered and exported with the proprietary software WinRiver II [30], and later visualized with the VMT code [26]. In case the Sontek aDcp was used instead, the 3D flow velocity data was filtered and exported into spreadsheet files written in ASCII format with the aid of the proprietary ViewADV program [28]. These files were later processed with an in-house code written in Fortran 95 to get the transverse velocity field. The first version of the in-house code included: (1) conversion of coordinates from geographical to TM coordinates, (2) identification of data outliers, (3) generation of local coordinates along the cross-wise plane with reconstruction of the bed bathymetry.

The local ENU coordinates are formed from a plane tangent to the Earth's surface at the study site. Contrary to the usual convention of naming the east with \( x \), and the north with \( y \), the RiverSurveyor program measures water motion in 3D with \( x/north \), \( y/east \), and \( z/up \) [28]. Then, and as long as the collected data is corrected by the magnetic declination bias, both TM (Gauss-Krüger) and ENU coordinates are fully compatible, with the up component pointing out in the direction opposite to gravity.

The in-house coded developed for the Sontek aDcp data decomposes both the 2D planar (depth-averaged) and the full 3D vectors into tangential (along the cross-wise plane) and normal (along the stream-wise plane) components of the absolute velocity relative to ground, with the addition of the up component for the later case. Consequently, the field data collected along transects was always referred in geographical and ENU coordinates as well. The Gauss-Krüger coordinate system used by the Argentine Geographic Military Institute to make topographic maps of the national territory is based upon a TM projection, with origin of coordinates in the intersection of the South Pole with the central meridian of each band. Strict applications of the TM formulas, with south latitudes negative, results in the derivation of the correct easting and northing. Therefore, a module with the new World Geodetic System (WGS84) as the reference system was implemented, adopting the TM formulas given by Snyder [29] in combination with a procedure similar to that proposed by Dinehart and Burau [14] to project the 3D velocity field data. The location of the projection plane was dictated by the mean vessel trajectory during the surveys. The tangential and normal components were computed for each profile over the mean plane.

Finally, both tangential and up components defined the projected cross-stream motion along the mean plane.

### III. RESULTS

Curvilinear open-channel flows induce centrifugal forces which generate secondary currents and super-elevation of the water surface, which significantly may influence the 3D flow patterns since the cross-flow can be up to 40-50% of the bulk streamwise velocity. The selected surveyed cross-sections (XS) are showed in Fig. 5, numbering them from 0 (XS-0) at the upstream inlet boundary to 5 (XS-5) towards the downstream boundary in the navigation channel. It is seen in Fig. 6 that the secondary circulation is indeed strong, with transverse velocity on the order of 0.40 ms\(^{-1}\), which is about 50% of the primary velocity component. The 2D velocity field depicted in Fig. 6 was captured during the 2009 campaign (Table 1) with the Sontek aDcp, and isolated according to the procedure aforementioned. The cross-stream flow pattern shown in Fig. 6 corresponds to cross-section 3 (XS-3), whose relative location within the computational domain is indicated in Fig. 5.

Fig. 7 shows cells of secondary motion captured with the TRDI aDcp, plot produced with the VMT [26] toolbox. Here, it is worth to mention that VMT computes secondary currents according to the Rozovskii procedure, whose suitability to isolate cells of cross-flow from a skewed primary flow is now under close scrutiny among different authors [26, 31].

This reach of the river diffuence, characterized by the strong asymmetry of the bed topography, and consequently, of the flow dynamics, should exhibit some characteristics behavior such as acceleration, stagnation and flow deflection, whose features are seen to be well captured by the numerical solutions depicted in Fig. 8. As mentioned previously, the velocity distribution at the free surface has been computed with the hydrostatic and non-hydrostatic 3D models, in conjunction with the zero-equation and two-equations standard k-\( \varepsilon \) models. Further insight into the
complex flow pattern can be observed in Fig. 9, where the formation of a separation zone deviates the flow towards the left bank at the inlet of the navigation channel.

Computed secondary flow patterns in XS-2, a bit downstream of the diffuence (see Fig. 5 for surveyed cross-sections) are showed in Fig. 10. The results obtained with the hydrostatic and non-hydrostatic 3D models, in conjunction with the zero-equation and two-equations standard $k-\varepsilon$ models show some variations along the vertical distribution of the streamwise velocity. Nevertheless, besides an optical effect due to the 3D projection algorithm used to plot the solution, the numerical results along XS-2 show a significant net unidirectional flow component in transverse direction from outer to inner regions [10]. This is induced by mass conservation to compensate the flow acceleration in the inner regions of the bend, clearly shown in the contour values of the flow module (see Fig. 10).

IV. CLOSURE

Since no unbiased algorithm is yet available to project the 3D computed velocity field onto the selected cross-sections [31], no judgment on the capability of the different mathematical models to fit observed data can be made at this point. Despite the increasing popularity of aDcp to study 3D flows, several issues must be overcome when matching observed cross-flow along bends with numerical results. One of this issues is related to the way the field data is captured and later treated, and the other is related with the use of different forms of the conservation laws and the effect the eddy viscosity model may have onto the solution. An unified algorithm to isolate cross-flow from a skewed discharge, either from field or numerical data, somehow is still lacking. Captured field data at some verticals (Fig. 6) resembles the cross-flow of Fig. 2. The figure of merit to judge the performance of different formulations of the governing equations of curved open-channel flows shouldn’t be far from the simple model (1)–(3) solution.

![Figure 8. Velocity at the water surface (in ms$^{-1}$). Snapshots (a) and (b): 3D hydrostatic + zero and 2 $k-\varepsilon$ turbulence models resp. Snapshots (c) and (d): 3D non-hydrostatic + zero and $k-\varepsilon$ turbulence models, respectively.](image)

![Figure 9. Velocity field and module (in ms$^{-1}$) at the separation zone.](image)

ACKNOWLEDGEMENT

Support from CONICET and UNL, and EDF R&D as well, is gratefully acknowledged.
REFERENCES


Shear-driven flushing and circulation in a marina in the United Arab Emirates

Enda MURPHY, James WALKER
Sogreah Gulf – Artelia Group
Dubai, United Arab Emirates
enda.murphy@ae.arteliagroup.com

Abstract— Flushing or residence times are typically used to assess water quality potential in marinas and other semi-enclosed water bodies. Recent publications have focussed on developing and revisiting general guidelines to improve water exchange in tidal marinas by optimizing basin and entrance geometry. However, these guidelines are based on specific cases where water exchange is strongly tide-driven and frequently do not apply to micro-tidal sites. In this work, we focus on a real-world example of a marina on the edge of a micro-tidal channel, where water exchange is strongly influenced by transverse velocity shear at the interface between the channel and the marina basin. A TELEMAC-2D hydrodynamic model of the channel and marina basin was developed, calibrated and validated using field measurements of current speeds and water levels. The numerical model was used to assess flushing times for different marina and entrance configurations. The results demonstrate a particular example where dead-zone models of water exchange, traditionally applied to evaluate mass transport in rivers and groyne fields, provide a better means to guide optimization of basin and entrance geometry.

I. INTRODUCTION

Flushing or residence times have historically been used as a first step in assessing water quality in marinas, harbours and coastal basins [2, 5, 9, 20, 22, 28, 29]. Recent publications have offered guidance in relation to optimal basin geometries (e.g. plan form factor, aspect ratio, tidal prism ratio, curvature, relative entrance area) to help achieve rapid renewal [2, 5, 28, 29]. However, these guidelines have been developed for the particular case where water exchange is strongly tide-driven and are not widely applicable, particularly in micro-tidal regions such as the Persian Gulf, where mean spring tidal ranges are typically less than 2m.

For marinas in micro-tidal areas, the tidal prism is seldom sufficient to ensure water renewal by purely tide-driven exchange. Where water exchange is dominated by shear-driven circulation and lateral transfer of momentum at the interface between the marina and the adjacent water body (i.e. a mixing layer), there is a strong analogy to groyne fields and other cases involving flows containing quasi-stagnant peripheral areas (dead zones). In these cases, dead-zone models for mass transport may present a better alternative to guide optimization of basin and entrance geometry.

In this paper, a case study is presented whereby a numerical hydrodynamic model was used to evaluate flushing times in a marina on the edge of a micro-tidal channel in the United Arab Emirates. A number of basin and entrance geometries were tested to investigate the impacts on residence times. We investigate the applicability of dead-zone models of water exchange, traditionally used to evaluate mass transport in rivers and groyne fields, to the case of a marina on the edge of a micro-tidal channel.

II. DEAD-ZONE MIXING PROCESSES AND MODEL FOR WATER EXCHANGE

The effect of quasi-stagnant, or “dead” zones on water exchange and mass transport in rivers has been the subject of considerable research over the past 35 years [1, 4, 6, 7, 13, 14, 24, 26, 27, 30, 31, 35]. Dead zones, which are defined by [24] as local areas of the flow cross section with relatively still water, or no net downstream velocity, are created in rivers by the presence of meanders [27], natural or manmade peripheral embayments (including marinas) [32], groyne fields [32, 33, 34], vegetated zones [10, 11, 12, 13, 14, 19, 21], hyporheic zones [13, 21, 35], pools and riffles [13].

Dead zones on the sides of channels result in shear flow at the interface with the main channel and an associated inflection point in the streamwise velocity profile. The latter is a characteristic of mixing layers, commonly observed in terrestrial [25] and aquatic vegetated flows [10], and is a prerequisite criterion for instability [16, p. 499]. Mixing layers are susceptible to Kelvin-Helmholtz instability, such that transport across the layer is typically dominated by large scale vortex structures [10, 32]. The formation of vortices and the efficiency of exchange between the dead zone and the main channel, which are separated by the mixing layer, are subject to local flow conditions and the geometry of the dead zones [12, 30, 32]. The rate of renewal of water in the dead zone is also strongly influenced by secondary gyres [32].

Most dead zone models consider exchange between stagnant zones and the main stream to be well represented by a first order process [4, 7, 30, 32]:

$$\frac{\partial C}{\partial t} = b(C - C_s)$$  

(1)

where $C$ and $C_s$ are the concentration of a conservative solute in the dead zone and main stream, respectively, $t$ is time, and $b$ is an exchange coefficient with dimensions $T^{-1}$. For a dead
zone of uniform concentration, the inverse of the exchange coefficient provides a measure of the time scale for water renewal within the dead zone, commonly referred to as the mean residence time, \(T_R\). For an ideal (plane) mixing layer \([25]\), the exchange coefficient is known to be a function of the difference between the net streamwise velocity in the main stream and the dead zone, \(\Delta U = U - U_M\) \([12, 19]\), and the width of the layer. Assuming zero net flow in the dead zone and considering that the thickness of the plane mixing layer is constrained by the width of the dead zone perpendicular to the flow, \(W\) (Fig. 1), it follows that

\[
b = b_{\text{ideal}} = \frac{kU}{W} \tag{2}
\]

where \(k\) is a dimensionless coefficient, referred to as an “entrainment coefficient” by \([30, 32]\). In the case of transverse mixing between a main stream and a peripheral dead zone of finite depth (i.e. a real mixing layer), the formation of large scale vortices (which controls the rate of exchange) is also dependent on the limiting water depth \([32]\). Using mass conservation principles, Weitbrecht et al. \([32]\) showed that in such cases, the exchange coefficient is given by the dead-zone model as

\[
b = \frac{kU}{W} \frac{h_E}{h_M} \tag{3}
\]

where \(h_E\) is the depth at the entrance to the dead zone and \(h_M\) is the average depth in the dead zone, as illustrated in Fig. 1. Thus, by determining the coefficient \(k\), shear-driven exchange between the main stream and a side-embayment and the associated mean residence times can be evaluated.

For the specific case of a tidal channel, \(U = U(t)\). Here, we make the assumption that (3) holds approximately for \(U = U_{\text{rms}}\), implying quasi-steady state conditions.

Valentine and Wood \([30]\) considered a value of \(k = 0.02\) to be generally appropriate for a rectangular cavity in a channel bed, consistent with previous experiments for side cavities (cited in \([30]\)). For a range of groyne field geometries and configurations, Weitbrecht et al. \([32]\) determined via experiments entrainment coefficients ranging from 0.012 to 0.051. They developed an empirical relationship between \(k\) and the dead zone geometry, through a nondimensional shape parameter that is fundamentally analogous to a hydraulic radius:

\[
R_B = \frac{WL}{h_S(W + L)} \tag{4}
\]

where \(L\) is the length of the dead zone and \(h_S\) is the flow depth in the main stream (Fig. 1). Weitbrecht et al. \([32]\) contend that the formation of large scale vortices (which controls the rate of exchange) is dependent on the stability of the mixing layer governed by the ratio of the width (~ \(L\), \(W\)) to the water depth in the channel (~ \(h_S\)). In the specific case of a shallow marina on the edge of a deeper channel, with one or more breakwaters partially protruding across the entrance (Fig. 1), we argue that the length scales controlling the development of vortices are the entrance width, \(L_E\), the average depth in the marina, \(h_M\), and \(W\). Thus, we propose a modified shape parameter:

\[
R_{BM} = \frac{WL_E}{h_M(W + L_E)} \tag{5}
\]
Figure 2. Numerical model domain, mesh and boundary conditions.

The model bathymetry for the pre-development scenario, which was used to calibrate and validate the model, is shown in Fig. 3, referenced vertically to the project datum (approximately equivalent to mean sea level). Open water boundary conditions were applied as follows (as indicated in Fig. 2):

- A temporally varying water level along the southern boundary; and
- Temporally varying water levels and currents (velocity component normal to the boundary) along the northern boundary.

B. Field Data

To support the numerical modelling, a programme of hydrographic field surveys was implemented for the project, which included:

- Measurement of directional current speeds using bed-mounted acoustic Doppler current profilers (ADCPs) at three locations in the channel; and
- Measurement of water levels using calibrated tide gauges at three locations in the channel.

Details of the ADCPs (ADCP01, ADCP02 and ADCP03) and tide gauges (SG01, SG02 and SG03) are listed in Table I. The locations of the field instruments in the tidal channel are shown in Fig. 3.

Ideally, hydrographic field measurements to support calibration and validation of a tidal hydrodynamic model should be carried out for a minimum period of 15 days to capture variations over a full spring-neap tidal cycle. However, due to project time constraints, the recording durations were shortened to approximately 5 days and 11 days for the ADCPs and tide gauges, respectively.

Since field data was available for only a limited duration, harmonic constituents were estimated from the field data and then used to approximate the time series of water levels and currents over an extended period spanning a full spring-neap tidal cycle.

C. Model Calibration and Validation

The model was calibrated using the field measurements of water levels and currents. Model input variables, such as bed roughness coefficients and turbulence parameters, were adjusted incrementally within appropriate ranges and simulations were implemented for the period corresponding to the field surveys. The final calibrated model parameters (Table II) were then selected based on goodness-of-fit and visual inspection of the time series measurements and predictions at the stations closest to the development site (SG02 and ADCP02).

Observed and predicted free surface elevations, depth-averaged current speeds and depth-averaged current directions are shown for the calibration period (19/8/2010 – 24/8/2010 for currents and 17/8/2010 – 28/8/2010 for water levels) in Fig. 4 for the locations nearest the site (SG02 and ADCP02). Visual inspection indicates good agreement between the field measurements and model predictions, with
TABLE I. FIELD INSTRUMENT DETAILS

<table>
<thead>
<tr>
<th>Station ID</th>
<th>Instrument Type</th>
<th>Location</th>
<th>Approx. Depth (m)</th>
<th>Recording Interval (min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADCP01</td>
<td>Acoustic Doppler current profiler</td>
<td>North channel entrance</td>
<td>6.4</td>
<td>10</td>
</tr>
<tr>
<td>ADCP02</td>
<td>Acoustic Doppler current profiler</td>
<td>Adjacent to site</td>
<td>6.9</td>
<td>10</td>
</tr>
<tr>
<td>ADCP03</td>
<td>Acoustic Doppler current profiler</td>
<td>South channel entrance</td>
<td>6.0</td>
<td>10</td>
</tr>
<tr>
<td>SG01</td>
<td>Tide gauge</td>
<td>North channel entrance</td>
<td>n/a</td>
<td>10</td>
</tr>
<tr>
<td>SG02</td>
<td>Tide gauge</td>
<td>Adjacent to site</td>
<td>n/a</td>
<td>10</td>
</tr>
<tr>
<td>SG03</td>
<td>Tide gauge</td>
<td>South channel entrance</td>
<td>n/a</td>
<td>10</td>
</tr>
</tbody>
</table>

Figure 3. Model bathymetry and locations of field instruments.

the phasing and magnitude of water levels, peak current speeds and directions all captured accurately by the hydrodynamic model. This assessment is supported by the statistics of the fit, with correlation coefficients for measured versus predicted values close to 1 ($R^2 = 0.997$ and 0.848, for water levels and depth average current speeds, respectively). Predicted and observed rms depth average current speeds differ by less than 2.5% (0.43m/s observed versus 0.44m/s predicted).

Once the final model parameters were established through calibration, a full spring-neap tidal period was simulated using the extended time series, and goodness-of-fit between the reconstructed time series and model-predicted values was reassessed (validation). Results showed that the calibrated hydrodynamic model accurately captured the spring-neap variation in water levels and depth-averaged currents. The phasing and magnitude of water levels, peak current speeds and directions were all found to be in good agreement for the validation period, with correlation coefficients $R^2 = 0.996$ and 0.890 for water levels and depth average current speeds, respectively. Differences in rms depth average current speeds were less than 1.5%.

D. Flushing Assessment Methodology

Flushing was assessed by introducing a conservative (i.e. non-decaying in time) tracer to the hydrodynamic model with an arbitrary initial value of 100 everywhere within the marina basin. The hydrodynamic model was then used to simulate the advection and dispersion of the tracer over a 15-day period beginning on a neap cycle, to quantify the exchange with outside waters. The simulation was chosen to begin during neap conditions as this typically provides the most conservative time estimate of tide-driven water exchange.

E. Marina Layouts

To evaluate the flushing efficiency of different marina layouts and configurations, the following four scenarios were investigated using the numerical model (Fig. 5):

- Layout 1 – base case marina layout;
- Layout 2 – raised bed elevations in southern parts of the marina basin;

TABLE II. SUMMARY OF MODEL PARAMETERS

<table>
<thead>
<tr>
<th>Model Process</th>
<th>Parameter and Units</th>
<th>Range of Values Tested During Calibration</th>
<th>Final Calibrated Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bed friction</td>
<td>Chezy coefficient ($m^{1/2}$/s)</td>
<td>25 to 60</td>
<td>60</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Constant eddy viscosity, $\nu_T$ ($m^2$/s)</td>
<td>0.01 to 0.1</td>
<td>0.01</td>
</tr>
<tr>
<td>Physical properties</td>
<td>Seawater density (kg/m$^3$)</td>
<td>1030</td>
<td>1030</td>
</tr>
<tr>
<td>Wetting / drying</td>
<td>Tidal flats</td>
<td>Wetting / drying included</td>
<td>Wetting / drying included</td>
</tr>
</tbody>
</table>
Figure 4. Time series and regression analysis of field measurements and model predictions for the calibration period.
• Layout 3 – no breakwater on the southern side of the marina entrance;
• Layout 4 – no breakwater on the northern side of the marina entrance.

IV. RESULTS AND DISCUSSION

Results for each of the numerical simulations are listed in Table III. Fig. 6 shows the tide- and basin-averaged relative tracer concentration, \( \frac{C}{C_0} \) versus time (normalized by the semi-diurnal tidal period, \( T_p \approx 12h \)) for each of the simulations. Tide averaged values were computed by twice applying a running average filter to the results, using a time window \( T_{\text{mean}} = T_p \). For each layout, the exchange coefficient, \( b \), was evaluated from a least squares exponential fit to the basin- and tide-averaged tracer concentration (e.g. as described in [27]).

The formation of large scale vortices at the interface between the marina and the main channel was observed for all layouts, and was most evident in the results for Layout 3 (Fig. 7), which incidentally, was the layout with the widest entrance and the shortest mean residence time. This suggests shear-generated circulation is the dominant exchange mechanism.

Fig. 8 shows the entrainment coefficient, \( k \), determined from (3) and the best fit \( b \) for each simulation, plotted against the modified shape parameter in (5). There is reasonable (although not statistically significant [23]) correlation between \( k \) and \( R_{DM} (R^2 = 0.7221, n = 4, \alpha = 0.05) \), suggesting the dead zone model and the length scales adopted in the modified shape parameter are generally valid. By contrast, there is no apparent correlation between residence times and \( R_p \) (Table III) or other traditional indicators for tide-driven exchange (e.g. tidal prism ratio, relative entrance area [5]).

Figure 5. Marina layouts, entrance configurations and bathymetries investigated using the numerical model.
TABLE III. LAYOUT DEFINITIONS AND NUMERICAL MODEL RESULTS

<table>
<thead>
<tr>
<th>Layout</th>
<th>$U_{rms}$ (m/s)</th>
<th>W (m)</th>
<th>$L_E$ (m)</th>
<th>L (m)</th>
<th>$a$ (m$^2$)</th>
<th>$A$ (m$^2$)</th>
<th>$h_E$ (m)</th>
<th>$h_M$ (m)</th>
<th>$h_S$ (m)</th>
<th>$b \times 10^6$ (s$^{-1}$)</th>
<th>$T_R$ (days)</th>
<th>$k$ (−)</th>
<th>$R_D$ (−)</th>
<th>$R_{tot}$ (−)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.2</td>
<td>215</td>
<td>35</td>
<td>160</td>
<td>112</td>
<td>34320</td>
<td>3.2</td>
<td>7.0</td>
<td>2.5</td>
<td>4.8</td>
<td>0.005</td>
<td>13.1</td>
<td>9.4</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.2</td>
<td>215</td>
<td>35</td>
<td>160</td>
<td>112</td>
<td>34320</td>
<td>3.2</td>
<td>7.0</td>
<td>3.4</td>
<td>3.4</td>
<td>0.007</td>
<td>13.1</td>
<td>10.3</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.2</td>
<td>215</td>
<td>77</td>
<td>160</td>
<td>246</td>
<td>34430</td>
<td>3.2</td>
<td>6.0</td>
<td>7.0</td>
<td>1.7</td>
<td>0.015</td>
<td>13.1</td>
<td>17.7</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.2</td>
<td>215</td>
<td>56</td>
<td>160</td>
<td>179</td>
<td>34570</td>
<td>3.2</td>
<td>7.0</td>
<td>2.6</td>
<td>4.3</td>
<td>0.006</td>
<td>13.1</td>
<td>13.9</td>
<td></td>
</tr>
</tbody>
</table>

$^a$ Values in parentheses are adjusted for a central entrance, as suggested (for tide-driven exchange) in [5].

The computed values of $k$ are generally lower than the ranges determined experimentally by [30] and [32]. This may be explained by the fact that, for the layouts investigated, the large scale vortices did not penetrate fully to the inner side of the marina. This inhibits mixing in areas of the marina not encompassed by the mixing layer, resulting in lower values of $k$. It is also important to note that the experiments by [30, 32] were confined to unidirectional flows. For the numerical simulations presented in this paper, part of the mass of tracer removed from the marina during flood tide was observed to return during ebb flow conditions, increasing the mean residence time and leading to reduced values of $k$. This suggests that further work is needed to investigate how the quasi-steady approximation, $U = U_{rms}$, affects the dead zone model predictions, since observations have shown that mass transfer rates for oscillatory flows may be significantly different to those for comparable unidirectional currents [17].

The lower values of $k$ are consistent with the value of turbulent diffusivity for the tracer implemented in the numerical model, $D_T = 0.01$m$^2$/s, which gives a turbulent Schmidt number (the ratio of the eddy viscosity to the turbulent diffusivity of the tracer) $S_C = 1$. This falsely implies that mass is transported as efficiently as momentum, when typically $S_C < 1$ for shear layers [12, and references therein].

As $k$ is a measure of the efficiency of transport across the mixing layer, it is useful to consider factors affecting the development of the layer, one of which is that its growth may be constrained by the time scale for reversal of flows. However, the length of the marina entrance ($L_E$) also plays a role in determining $k$. For groyne fields and dead zones in rivers, the entrainment coefficient in upstream areas is known to differ significantly from values downstream, which are generally quite constant [1, 31, 32]. This is because the mixing layer is not fully developed over short distances [11] and because particles do not have sufficient time to sample the full range of velocities in the main stream and dead zones (i.e. Fickian conditions have not been reached). For marinas with short entrances, fully developed mixing layers may never be practically realised, due to impingement on the downstream edge. In this case, complex feedback mechanisms not represented by the dead zone model may also persist [18].

For Layout 4, the computed value of $k$ was not wholly consistent with the trend observed for the other layouts (Fig. 8). This may be explained by the entrance being more centred on the marina, resulting in two counter-rotating circulation cells within the basin (Fig. 9). Reduced rates of exchange for marinas with central entrances compared to
offset entrances have been identified for marinas subject to tide-driven exchange [5, 20, 28, 29]. For such cases, it has been proposed by others to treat the marina as two mirror-image basins with offset entrances [5]. If we apply this approach for Layout 4 (Fig. 10), significant correlation between $k$ and $R_{DM}$ is obtained ($R^2 = 0.9602$, $n = 4$, $\alpha = 0.05$).

Figure 8. Dimensionless exchange coefficient $k$ as a function of the modified shape parameter $R_{DM}$.

Interestingly, very few of the published guidelines and relationships for tidal exchange in marinas hold true for this study. For example, water exchange generally tended to decrease with increasing $A/a$, the ratio of the basin area to the entrance cross-sectional area, which conflicts with guidelines in [5] for marinas dominated by tide-driven exchange. There are significant differences in the computed mean residence times for the various layouts, despite only small changes in the tidal prism ratio.

V. CONCLUSION

A TELEMAC-2D numerical hydrodynamic model was used to evaluate flushing and mean residence times in a marina on the edge of a micro-tidal channel. The results show that, for conditions where shear-driven exchange dominates, dead zone models for water exchange provide a better means to guide optimization of marina and entrance geometry than traditional guidelines developed for tide-driven exchange.

As the time scales for flushing and water renewal at micro-tidal sites typically exceed the tidal period, the quasi-steady assumption made here is not strictly valid, and further research is needed to extend the dead zone model to oscillatory flow in a rigorous way. Other opportunities to improve the model are to consider the effects of partially developed mixing layers and flow interactions with the downstream edges of marina entrances on dead zone residence times.

Further work could be undertaken to generalise the dead zone exchange model to different entrance configurations (e.g., offset entrances, centred entrances and multiple entrances) and to irregularly shaped marinas (i.e., where $W$ and $L$ cannot be clearly defined), and to investigate the transition from tide- to shear-driven flushing regimes.

Figure 9. Snapshot of relative tracer concentrations and current vectors for Layout 4, showing counter-rotating circulation cells.

Figure 10. Dimensionless exchange coefficient $k$ as a function of the modified shape parameter $R_{DM}$, with adjustment to the value for Layout 4 for a centred marina entrance.

ACKNOWLEDGEMENTS

The authors wish to thank Jack Bokaris for obtaining permission to use the case study materials presented herein.

REFERENCES


Abstract — Complex, high resolution, finite-element TELEMAC-2D grids have been developed to simulate ocean/fluvial interaction in three shallow, intensively managed Welsh estuaries. In each case, important ecosystems and developed areas are exposed to flooding from both tidal and fluvial events. Of particular concern is how the estuaries will be affected by future sea-level rise due to climate change, and how best to manage the estuaries in the future.

Each grid has been generated using BlueKenne© and comprises several sub-grids to resolve important aspects of the estuary, such as river channels and flood embankments. Accurate representation of these features is paramount to flood risk modelling. A selection of present-day mean and extreme hydrodynamic/fluvial scenarios has been simulated, as well as future climate change scenarios. These simulations have highlighted the need for local management so that coastal flooding and morphological change are minimised, whilst preserving important ecosystems such as protected salt marshes and peat bogs. Alterations were made to the grids in order to address possible management options such as coastal realignment. In this way, key implications for flood risk, sediment transport and morphological change can be predicted which will aid management decisions.

I. INTRODUCTION

Some of the most challenging environments faced by ocean modellers are estuaries. These regions are where fresh river water interacts with saline water to produce strong baroclinic currents; where shallow water depths generate highly nonlinear tidal perturbations; and where complex topography and anthropogenic coastal interventions ultimately control circulation. Estuaries provide important modelling case studies because they are often surrounded by protected ecosystems, and harbour significant human populations which exploit the area for leisure and tourism.

Around the coast of Wales, UK, there are several estuaries which urgently require coastal management to reduce flood risk. These estuaries have been heavily managed over the past century and coastal realignment has reduced their natural tidal prisms. De-facto coastal defences are becoming overtopped more regularly, due to sea-level rise and increased storm surge and high river flow events. Flooding episodes drain slowly and hence impact on important ecosystems and infrastructure. In order to understand the influence of climate change on coastal flooding, and the impact of different management options, a three-year project in collaboration with the Countryside Council for Wales (CCW) was undertaken where TELEMAC-2D was applied to three representative Welsh estuaries (Fig. 1), namely the Dyfi Estuary [1-3], the Burry Inlet [4], and the Mawddach Estuary [5]. A fundamental aim of the study was to develop hydrodynamic solutions based upon accurate, two-dimensional model grids of each estuary. The grids incorporate anthropogenic features, such as coastal breakwaters, railway embankments and sea walls, whilst resolving small-scale bed features in the near shore zone and extending off-shore to allow the simulation of tidal propagation. Each application has investigated flooding, circulation and sediment transport for a selection of mean and extreme climatic scenarios.

The three case studies are described in Section II. Application of TELEMAC-2D, the flow module, and SISYPHE, the sediment transport module, to each estuary is explained in Section III, including model validation and a summary of the scenarios that have been modelled. Some of the key results are presented in Section IV, followed by discussions and conclusions in Section V.

Figure 1. Map showing the three Welsh estuary case studies (boxed areas), modelled using TELEMAC-2D (1-The Burry Inlet, 2-the Dyfi Estuary, and 3-the Mawddach Estuary). The domains overlie model output of bathymetry in the Irish Sea from a (finite-difference) POM simulation, which was used to provide elevation/velocity boundary harmonics for each of the nested TELEMAC-2D models.
II. CASE STUDIES OF WELSH ESTUARIES AND RATIONALE

This paper summarises three model studies of estuaries on the Welsh coast. The three regions are dynamically similar in that they are typically shallow and comprise medium grained sands, representing the later stages after the Holocene transgression (rapid infill of deep wide estuaries). The estuaries have a main ebb-dominated channel, flanked by extensive (flood-dominated) tidal flats and salt marshes. In all cases, the estuary mouth is constricted in width due to the presence of a spit. As a consequence, the strongest tidal flow occurs through the estuary mouth. Another common feature of all the case studies is that the natural size and tidal prism of the estuaries has been reduced over the past century by coastal realignment and land reclamation. This process occurred during war times when land for agriculture was at a premium. Embankments and coastal defences that were previously sufficient for withstanding extreme high water levels are now being overtopped, and these flooding episodes are becoming more regular.

The Burry Inlet, south Wales, is the largest of the three estuaries (16 km in length with a spring tidal prism of 1×10^{12} m^3), and contains a sizable population on its north coast which is at high risk from flooding. The south coast contains extensive protected salt marshes which are also at risk from increased tidal inundations, together with increased sediment erosion. There is also a risk that the spit could be breached, due to wave-induced erosion on its seaward side, which would expose the salt marsh directly to the open sea.

The natural tidal prism of the Dyfi Estuary, mid-Wales, is severely restricted by an (active) railway embankment, and also embankments on the river flood plains to protect agricultural land. Because of these embankments, high river flow that coincides with high spring tide leads to significant flooding at Machynlleth, a town near the tidal limit. Issues that now face coastal management here are whether to reinforce the embankments, or realign some of them to reduce flood risk, at the cost of land depletion, some of which comprises protected marsh ecosystems.

The third case study, the Mawddach Estuary in mid-Wales, faces many similar management issues as the neighbouring Dyfi Estuary because it is a similar inter-tidal environment that has also been affected by coastal realignments. Fig. 2 shows sub-sections of the model grid of the Mawddach Estuary, generated using BlueKenue®. The top panel illustrates the bathymetry of the estuary and evaluation of the surrounding terrestrial areas. The bottom panel shows a section of the mesh in the lower-estuary which is composed of several sub-meshes so that important features such as the main ebb channel (green mesh) and embankments or flood defences (red lines) are correctly resolved. For all case studies, bathymetric data comprised high-resolution LIDAR (Light Detection And Ranging) data within the estuary, and either Admiralty chart data or boat survey measurements off-shore. Unfortunately, bathymetry of the river channels was not available and, so, LIDAR-measured river heights were reduced by 2 m uniformly.

III. MODEL APPLICATION

TELEMAC-2D is an ideal modelling framework for the estuarine environment due to its finite element grid which allows graded mesh resolution. Areas that require high bathymetric accuracy (e.g. complex coastlines, meandering river channels, or anthropogenic features such as embankments and sea defences) can be well resolved, whereas the off-shore grid-spacing can be increased to maximise computational efficiency.

The model, used here in 2D, vertically averaged mode (V5P9), is based on the shallow water Saint-Venant equations of momentum and continuity, derived from the Navier-Stokes equations by taking the vertical average, see [6]. A hydrostatic assumption is valid in this application where bed slopes are small and, hence, vertical accelerations caused by the pressure are balanced by gravity. The classical \( k-e \) turbulence model has been adapted into vertically averaged form to include additional dispersion terms [7]; this parameterisation of the internal friction has been used throughout this study with a constant friction coefficient of \( 3\times10^{-2} \) m, implemented in Nikuradse’s law of bottom friction. Turbulent viscosity has been set constant with a viscosity (molecular + turbulent) coefficient equal to \( 10^{-3} \) m^2 s^{-1}. Coriolis effects have also been included. The simulations were forced with tidal elevations.
and velocities at the off-shore boundaries (calculated from an outer nested model [8]), and with known river flowrates at the inland river boundaries. Atmospheric forcing has been neglected in the simulations. A tracer was simulated as a proxy for salinity with constant seawater temperatures of 10°C. Therefore, densities are updated according to the salinity gradients. However, in the presence of tidal flats, erroneous salinity values were encountered. Unfortunately, there was insufficient time available to investigate this issue further, but the hydrodynamics were not affected.

Many processes in the coastal zone, such as sediment dynamics, often need to be modelled in a three-dimensional framework, especially when strong vertical stratification is present, e.g. [9]. However, 3D modelling requires extensive field validation which was not available for the present study. In practice, the present estuaries are essentially vertically well mixed in terms of density stratification because the freshwater input is small in comparison with the tidal prism [10]. It is therefore sufficient for this study to use TELEMAC-2D since the vertical approximations associated with averaging are likely to be small.

SISYPHE is the sediment transport module coupled with TELEMAC-2D to produce simulations of bed evolution. The Soulsby-Van Rijn transport formula [11] was used here where total (bed load plus suspended load) sediment transport is calculated. An equilibrium model was then used for bed level changes resulting from divergences in the transport (i.e. an advection-diffusion scheme was not implemented). The transport rate formula is expressed:

\[ q_{tw} = A_s \bar{U} \left( \frac{U^2}{C_D} \right)^{0.5} \left( \frac{U_{rms}}{U_{cr}} \right)^{2.4} \]  

(1)

The formula (1) applies to total sediment transport rate per width of the flow in combined waves and currents, where \( A_s \) represents the bed load plus suspended load transport, \( \bar{U} \) = depth-averaged velocity, \( U_{rms} \) = root-mean-square wave orbital velocity (here set to zero), and \( C_D \) = drag coefficient due to the current alone. The threshold current velocity [12] is expressed:

\[ U_{cr} = 0.19 \left( \frac{d}{h} \right)^{0.1} \log_{10} \left( \frac{4h}{d} \right) \]  

(2)

for \( 1 \times 10^{-4} < d < 5 \times 10^{-4} \) m, where \( d \) = mean grain diameter and \( h \) = water depth.

The Soulsby-Van Rijn formula is intended for conditions in which the bed is rippled with a bed roughness length scale implicitly equal to 6 mm. The ‘sloping bed’ term that would appear in a complete statement of the Soulsby-Van Rijn formula is handled separately by an equivalent ‘switch’ in SISYPHE. The module also includes a parameterisation of secondary currents due to vertical eddy circulation. The surface wave module, TOMAWAC, was not utilised; therefore, all simulations hereafter do not account for surface wave effects. While wave effects are important on the open coast [10], the present focus is on the interior of the respective estuaries only.

The flow model was validated in the Dyfi Estuary using data collected during 9th – 21st July, 2007 [2]. Tidal elevations were measured at 3 locations within the estuary (Aberdyfi, Dyfi Junction, and Pennal Bend; points (a) to (c), respectively, in Fig. 5). TELEMAC-2D was run for this period, forced with tidal elevations and with river inputs based on the observed river flowrates. The data and model simulations were mainly in good agreement at Aberdyfi, as shown in Fig. 3a. A high run-off event can clearly be seen in (b) and (c) after 100 hours, followed by a slowly diminishing water level after the rain event onto which the tidal signal was superimposed. The elevations in the river are poorly simulated at times, most notably during the first half of the field survey. River elevations are difficult to simulate in relatively small estuaries because the detailed shape of the channel and position of the tide gauge have a great influence on the local water level. Unfortunately, there was insufficient sediment transport data available for validation of the SISYPHE module. However, TELEMAC has been tested extensively elsewhere against similar case studies [e.g. 13-15].

A set of tidally-forced, idealised model scenarios, together with fluvial inflow from the river channels, has been designed as a sensitivity exercise whereby the simulations represent mean and ‘extreme’ climatic conditions, both in the present-day and in the future. In this way, the specific model input variables that affect the circulation (tidal amplitude, storm surge, river flowrates and sea-level rise) can be examined. The scenarios are summarised in Table 1. For all the scenarios, the model was forced with the primary tidal constituents (M2 and S2) and run for a period of 8 days in order to capture both neap and spring tidal conditions.
### Table 1. Summary of the modelled scenarios.

<table>
<thead>
<tr>
<th>Model Scenario</th>
<th>Duration and Tidal forcing</th>
<th>Storm surge</th>
<th>River flowrates</th>
<th>Sea-level rise</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Baseline mean case</td>
<td>--</td>
<td>---</td>
<td>Annual mean</td>
<td>---</td>
</tr>
<tr>
<td>2 Extreme storm surge</td>
<td>Neap-Spring cycle</td>
<td>1 in 100 yrs (= 2 m)</td>
<td>Annual mean</td>
<td>---</td>
</tr>
<tr>
<td>3 Extreme high river flowrates</td>
<td>M2+S2 tidal amplitudes</td>
<td>---</td>
<td>1 in 100 yrs</td>
<td>---</td>
</tr>
<tr>
<td>4 Extreme low river flowrates (drought)</td>
<td>---</td>
<td>1 in 100 yrs (= 2 m)</td>
<td>0.0</td>
<td>---</td>
</tr>
<tr>
<td>5 Combined extreme conditions</td>
<td></td>
<td>1 in 100 yrs (= 2 m)</td>
<td>1 in 100 yrs</td>
<td>---</td>
</tr>
<tr>
<td>6 Future extreme conditions</td>
<td></td>
<td>100 yrs (= 1.0 m)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### IV. RESULTS

In each of the three estuarine case studies, increased sea levels of 1 m and above had the most dramatic influence on flooding episodes, rather than high river events. Such a circumstance is caused by an extreme storm surge or due to predicted sea-level rise in 100 years. Presently, most embankments and coastal defences are not overtopped during high spring tide, yet with increased sea levels of 1 m or more, overtopping occurs and protected marshes and urban areas are flooded. For example, Fig. 4 shows high spring tide water levels in the lower Mawddach Estuary, with and without a 2 m storm surge. The embankments shield the village of Fairbourne and the protected Arthog Bog from flooding in the former case, but not with an extreme storm surge. Another important result, that is generic to all three case studies, is that by removing some or all of the embankments, tidal inundations and excess fluvial waters are able to spread out laterally across the flood plains, which reduces water levels up-stream in the river channels. Therefore, informed management decisions can reduce flood risk in low-lying areas in the upper estuaries.

The Dyfi Estuary simulations show that with no coastal management in the future, areas of salt marsh will diminish due to coastal squeeze, and the scarce ecosystem of Borth Bog will change due to increased flooding. Borth Village will also be at risk to flooding. However, the model predicts that coastal realignment can protect some areas of Borth Bog and Borth Village, while allowing the salt marsh to migrate landwards. For example, Fig. 5 shows flooding for three management scenarios. For option 1, the embankments (which act as de-facto flood defences) have been reinforced by raising their height by 2 m. The present estuary shape is maintained but increased sea level will cause areas of salt marsh (located on the estuary-side of the embankments) to diminish. For option 2, a section of the rail line has been realigned further south. This allows the northern bog (which is less important as most of it comprises agricultural land) to flood so that salt marsh migration can take place, yet protects the majority of the peat bog and residential properties in the south. Option 3 shows that the entire Borth Bog will flood in extreme conditions (Scenario 5) if this section of the embankment is removed.

The morphodynamics of shallow, vertically well mixed estuaries, characterised by tidal flats and deeper channels, have been investigated to find out what contributes to flood/ebb-dominant sediment transport in localised regions (see [2] for more details). Applied to the Dyfi, the results show that shallow water depths lead to flood-dominance in the

---

**Figure 4** - A sub-section of the Mawddach Estuary, showing flooding at high spring tide (top) and with an additional 2 m storm surge (bottom). Bathymetry in dry areas is shown, where several embankments (coastal defences) can be identified (light brown lines). The bathymetry is overlain with water depths (light blue) and velocity vectors.

**Figure 5** - The Dyfi Estuary management options. Option 1: blue areas show flooding when existing rail embankments (solid black line) are raised by 2 m. Option 2: green areas show additional flooding when a section of the rail embankment is realigned further south (dashed black line). Option 3: orange areas show further flooding when the same section of the rail embankment is removed all together. The validation points (a - c) are marked (see Fig. 3).
inner estuary while tidal flats and deep channels cause ebb-dominance in the outer estuary. With an artificially ‘flattened’ bathymetry (i.e. no tidal flats), the net sediment transport switches from ebb-dominant to flood-dominant where the parameter $a/h$ (local tidal amplitude ÷ local tidally-averaged water depth) exceeds 1.2. Sea-level rise will reduce this critical value of $a/h$ and also reduce the ebb-directed sediment transport significantly, leading to a flood-dominated estuarine system. A similar pattern, albeit with greater transport, was simulated with tidal flats included and also with a reduced grain size.

Simulations in the Burry Inlet show that there is an increasing risk of flooding on the urbanised north coast, which could affect the towns of Llanelli and Burry Port, and also in the southern estuary salt marsh which is of great environmental importance (see Fig. 6). An extreme tidal/fluvial simulation (Scenario 5), for example, increased the tidal prism by 100%, compared with Scenario 1, and shifted the tidal limit 3 km further up-stream. Velocities increased by 50% throughout the estuary. Consequently, Whiteford Spit experienced significant erosion with sediment transported northward and deposited off-shore in the ebb delta.

Fig. 6 shows the effects on sediment transport (compared with the present-day Grid 1) of two management options: (i) removing (Grid 2) and (ii) restoring (Grid 3) a breached training wall in the main estuary channel. Velocities and sediment transport rates were reduced locally, where the wall had been removed, but increased further up-estuary. The re-built wall constricts the flow which increased erosion either side of the wall, but reduced sediment transport further up-estuary.

The Mawddach Estuary simulations have shown that removing some embankments, hence increasing the size of the estuary and its tidal prism, will reduce flood risk in the upper estuary and in the river channels. Consequently, flood risk in the town of Dolgellau (see Fig. 2) will be reduced. This is because intruding tidal and fluvial waters are able to spread out laterally across the flood plains, reducing water levels upstream. Fig. 6 shows water heights over a spring tidal cycle, at four locations within the Mawddach Estuary (see also Fig. 2), for Scenario 1 simulations on three different grid configurations (Grid-1: present estuary shape, Grid-2: natural estuary shape - all embankments removed, and Grid-3: managed estuary shape - some embankments removed). In the upper estuary and river channel, water heights are reduced by up to 1 m for Grids-2 and -3, where the flood plains have been opened up allowing water in the channel to spread out laterally. This is a very important result for coastal management as it suggests that coastal realignment, which is also relatively cheap, can be highly effective in reducing flood risk up-estuary.
V. DISCUSSION

Estuarine environments have always been populated, originally due to trade and industry, but more so today because people are attracted by their outstanding natural beauty. In order to preserve these environments for future circumstances involving climate change, coastal management strategies are essential. Through this study, and other modelling projects, interested authorities in Wales are now realising the potential that modelling has in aiding coastal management.

This study has shown that TELEMAC-2D is a suitable modelling framework for estuarine case studies, for the most part because of the finite-element grid (orthogonal or curvilinear grids will not generate comparable resolution as efficiently). Simulations of very shallow water depths and tidal flats (where cells can ‘dry out’) are also essential features which are included in TELEMAC-2D. However, secondary baroclinic flows, which are often generated in estuaries (although shallow depths ensure a vertically mixed water column for the present case studies), will not be simulated in a depth-averaged model and, therefore, three-dimensional modelling will be required.

Nevertheless, there are some major shortcomings to this study, caused primarily by coarse grid resolutions and limited model validation. Whilst grid resolutions as fine as 10 m seem sufficient for most coastal applications, increasing the resolution further would significantly improve the accuracy of the simulations. For example, small-scale features such as creeks can have a significant impact on drainage times on the flood plains. Impermeability of the bed together with unresolved drainage channels lead to unrealistic ‘pooling’ of water behind overtopped embankments. Therefore, the severity of flooding may be over-estimated in some of the simulations. Such errors could be reduced with additional coding to either mimic an impermeable bed or realistic drainage, both of which would require validation.

Future, similar, estuarine modelling will require accurate high-resolution (<5 m) bathymetric data, including off-shore measurements and bathymetry of the river channels. Unfortunately, time constraints of the present study did not allow for extensive validation of the hydrodynamics and sediment transport. Given more time, these issues could be easily addressed which would greatly improve the results presented here. All that being said, the models have given considerable insight into local circulation patterns that were previously not understood. The results presented here have provided a foretaste of the vast potential that can be gained from forecast simulations, from the perspective of coastal planners.

ACKNOWLEDGEMENTS

This was a collaborative study with the Countryside Council for Wales (CCW), spearheaded by Dr. Rod Jones. His guidance and expertise were much appreciated. Dr. Simon Neill (Bangor University) provided modelled tidal boundary data. LIDAR bathymetry data was provided by the Environment Agency Wales.

REFERENCES

Modelling and analyzing dredging and disposal activities using Telemac, Sisyphe and DredgeSim

Christian MAERKER, Andreas MALCHEREK
Institute of Hydro Sciences
University of the German Armed Forces Munich
Neubiberg, Germany
christian.maerker@unibw.de

Judith RIEMANN, Thomas BRUDY-ZIPPELIUS
River Systems
Federal Waterways Engineering and Research Institute
Karlsruhe, Germany
thomas.brudy-zippelius@baw.de

Abstract—High amounts of sediments are dredged regularly within channels and harbour basins to provide the required water depths for safe shipping. Hence different dredging and disposal strategies as a part of sediment management concepts are investigated to reduce the economic and ecological impacts of the maintenance of waterways. In order to model dredging and disposal strategies the software package DredgeSim was developed. This package is coupled to Telemac and Sisyphe, which provide the initial data. The three simulation components are exchanging information, as their results are affecting each other. This paper presents first practical experiences and typical applications for modelling dredging and disposal activities. Furthermore, one application for artificial bed load campaigns along a stretch of the River Rhine is discussed in depth.

I. INTRODUCTION

Different sediment management concepts are used within rivers, estuaries and harbour basins in order to react on negative effects of sediment transport. To avoid instabilities of structures next to erosion hot spots or an abatement of the water level, grain feeding measures are applied as a response on a sediment deficit. The sediment material which is disposed on such locations stems from dredging operations performed at forehand within the water or as artificial bed load supply from the outside. In the latter case additional sediment material is introduced to the system. An example of such an activity is the grain feeding campaign along the barrage of Iffezheim at the River Rhine in Germany, where 200 000 m$^3$ of sediments are disposed annually as artificial bed load supply in order to stabilize the bottom and the water level [1].

Furthermore, sediment management concepts contribute to the maintenance of waterways to provide the required water depths for safe shipping. Construction measures like groynes or current deflection walls are used to prevent the accumulation of sediments along critical stages of channels and harbour entrances. Quite often sediment deposition has to be removed by dredging. E.g. 15 to 25 Million m$^3$ are dredged annually within the Elbe Estuary [2]. The main part of those dredged sediments is disposed at other locations. Hence, dredging and disposal strategies are developing to keep the economic and ecologic impact of such maintenance actions small. They play a major role within sediment management.

Due to the extent of such anthropogenic induced sediment movement it can be necessary to take the influences of dredging and disposal actions into account when analyzing the sediment transport regime of waterways. In order to model such influences in hydrodynamic-morphodynamic-numerical simulations the software package DredgeSim was developed. DredgeSim coupled to Telemac and Sisyphe allows the modelling of dredging and disposal activities for maintenance or expansion works as well as artificial bed load supply [3]. The model can be used to analyze and optimize sediment management systems.

II. MODELLING OF DREDGING AND DISPOSAL ACTIONS

There are different possibilities to compute dredging and disposal operations in numerical models. Modelling such activities will result in a modification of the node depths of the used computational mesh around the dredging and disposal areas. Furthermore, if a multi-fraction sediment transport model is used, the sediment distribution on the bottom can change at disposal sites. If dredged material is transferred to a disposal site it is necessary to recalculate the sediment fractions.

A. Computing Dredging Actions in DredgeSim

In DredgeSim dredging operations can be computed in two ways:

- For time steered dredging a dredging time and an amount of dredged material is defined. Dredging is initiated if the simulation time reaches the defined time frame. The node depths resulting from this dredging activity are then calculated from the amount of dredged material and the size of the dredging area.

- Or a dredge criterion is defined. Then dredging is initiated, if too much sediment material is accumulating at certain stages. The depths within a predefined area are checked at defined time intervals on whether a critical value is reached due to deposition of sediments. In this case the affected nodes are dredged to a defined dredging depth (Fig. 1). Either node depths or node water depths can be checked in a dredge criterion. Dredged sediment volumes are then calculated from the difference between
the actual node depths and the dredging depth and the
area of the dredging site.

In order to compute the composition of the soil mixture, the
dredged sediment fractions that are supposed to be dumped on
a disposal site are saved.

![Figure 1. Dredge criterion to initiate dredging operations in DredgeSim.](image)

### B. Computing Disposal Activities in DredgeSim

The computation of disposal actions in DredgeSim is tied
to the modelling of dredging operations. If a dredge criterion
is used, a subsequent modelling of disposal of the dredged
material can follow by specifying a disposal site. From the
saved volume of dredged material and the extent of the
disposal site a new bottom depth is calculated. As these
bottom depths are achieved by dividing the disposal volume
by the area of the disposal site, this value applies for all nodes
inside the disposal site (Fig. 2, left). Such an approach is
acceptable for disposal sites with a rather flat bathymetry. The
new sediment distribution on the disposal site is calculated
from the dredged and the original sediment distribution on the
disposal site.

A disposal action following a time steered dredging
operation is modelled in a similar way. From a specified
disposal volume and the size of the disposal area a new
bottom depth resulting from the dumping of the sediment
material is calculated like explained above. The sediment
distribution on the disposal site is again calculated from the
dredged sediment fractions and the soil composition on the
disposal site.

Another feature of DredgeSim is the computation of a
disposal activity to fill scours. Contrary to the modelling of
disposal operations described above, the new disposed surface
is plain. So, different node depths need to be calculated for all
nodes inside a scour hole. Therefore a filling line needs be
calculated respecting the main requirement that the volume
can be stored underneath it (Fig. 2, right).

![Figure 2. Disposal of dredged material with a constant heightening and with variable depths inside scour holes.](image)

This is done by an iterative procedure, as the position of
the filling line is affected by the effective disposal area which
in return is specified by the position of the filling line.

Furthermore, DredgeSim offers the possibility to model
artificial bed load supply. Those operations are time steered
operations for the disposal of sediments as well. Thus the
required input data are nearly the same. Only the composition
of the disposed material must be defined explicitly. This way,
external sediments can be introduced into the system. All key
functionalities of DredgeSim and their main use cases are
illustrated in Fig. 3.

![Figure 3. Options to define dredging and disposal operations in DredgeSim.](image)

### III. USED MODELLING PACKAGE

The DredgeSim package was developed as an additional
component to existing sediment transport models. The
sediment transport model calculates the natural sediment
transport induced by the flow regime. The dredging module
then takes into account dredging and disposal works within the
actual bed topography.

#### A. Software Structure

The presented modelling structure consists of three
components: Telemac2D, Sisyphe and DredgeSim. Telemac2D
computes the flow regime, which is needed to determine the
forcing data for the sediment transport, which is calculated by
Sisyphe. Therefore, Telemac2D and Sisyphe are exchanging
data of velocities, water levels and bed levels. Thus the
interaction between flow regime, sediment distribution and bed
level changes can be computed.

DredgeSim communicates with Sisyphe in order to get the
actual information about the topography (node depths $z_i$)
and the composition of the bottom (sediment fractions $j$ per node $i$
$\text{d}_{ij}$). After dredging and disposal operations are computed in
DredgeSim, changed bottom depths and in case of disposal
areas the new sediment composition of the bed has to be
updated in Sisyphe. The used model components and data flow
are shown in Fig. 4.
B. Input Files of DredgeSim

DredgeSim requires information about the dredging and disposal activities which shall be modelled and the size and position of the operational areas (Fig. 5). The first is defined in a steering file, which allows the definition of different sediment management activities according to the functionalities being available. The operational areas are set in an extra input file – the IPDS-File (Initial Physical Data Set). Within this module package different shape formats can be defined to distinguish regions where initial parameters vary from their global values, e.g. a local sediment distribution or the salinity. In DredgeSim it is used to define the position and size of dredging and disposal sites.

Furthermore, a reference surface water level can be defined. This surface level is used to define a critical water depth. At coastal regions the reference surface level is normally defined as sea level (0 m). Along inland rivers of Germany it is calculated from statistical values of free surface levels measured in the past, therefore varying in its spatial distribution.

![Figure 5. Required input files for DredgeSim.](image)

IV. TYPICAL USE CASES

All key functionalities were developed in order to model typical aspects and use cases of maintenance actions along waterways. The most important are illustrated below.

A. Predicting Dredging Needs to Analyze Maintenance Strategies

A simple model of a meandering channel with two bends was built to simulate a prediction of dredging needs for two different dredging strategies. The model has a width of 200 m and a length of 3500 m. The initial water depth is 4 m along the whole channel. The sediment distribution consists of very fine sand, according to the Udden-Wentworth-Scale. After a constant discharge period of one year typical bed level change patterns are developed. Fig. 6 (left) shows sedimentation at the inner banks (white) and erosion at the outer banks (black).

In a first study it was assumed that the water depths need to be maintained. A minimum value of around 2.5 m was supposed to be guaranteed. For that, a dredge criterion was applied, where dredging is initiated on nodes which reach the critical water depth of 2.5 m. Then the affected nodes were dredged down to a water depth of 3 m. The results of the bottom evolution following this dredge criterion are shown in Fig. 6 (middle). The dredging needs resulting from this operation are 4567.14 m$^3$ of sediments.

In a next step this model was extended by including the disposal of the dredged sediments. The deep stages of the outer banks were chosen as disposal sites. The disposal strategy applied here consisted of two features: The dredged volume was deposited on the nearest disposal site. The scour disposal option was chosen. This means, that the filled material in the scour had a levelled surface. The results of the bottom evolution following this dredge criterion are shown in Fig. 6 (right). The dredge volume for this scenario increases up to 5198.01 m$^3$. This is due to the remobilization of the disposed matter from the scours.

The comparison of both dredged volumes can help to evaluate the maintenance strategy. In this example, the amount of dredged material is increased only slightly, if the dredged material is disposed along the outer river bends. Therefore this seems to be a good disposal place.

![Figure 6. Bottom evolution due to natural morphodynamics (left), natural morphodynamics with dredging (middle) and natural morphodynamics with dredging and disposal of dredged material (right) [5].](image)

B. Time Steered Operations

Especially along waterways and harbours, where dredging and disposal activities occur frequently, the effect of such anthropogenic measures can not be neglected. Their impact on the evolution of the riverbed can be massive. The functionality of time steered dredging and disposal operations can be used in hindcast models to evaluate their effect on the total change in the bathymetry of the riverbed over a defined period. The usually well documented data about dredging and disposal measures (such as date, time and amount of dredged and
dumped sediments) can be used as input information for DredgeSim.

Fig. 7 shows the computed bed evolution in a simulation, where only the maintenance efforts were taken into account. All dredging and disposal events of the year 2000 along the Mühlhamer Bend in the River Danube were used as input data to calculate the resulting bed level change. Influences of the natural morphodynamics were neglected herein. Such a result already shows the high impact of anthropogenic influences on the bathymetry of this region. In further simulations the total bed level change induced by natural and anthropogenic effects on the morphodynamics can be computed. From these results, both their effects can be analyzed in detail.

Figure 7. Maintenance measures over one year along the Mühlhamer Bend at River Danube and their effect on the topography.

C. Determining the Filling Capacity in Scour Holes

The time steered disposal functionality in combination with the scour disposal option can be used to determine the capacity to store sediments in scour holes. For applying the scour disposal option an upper value for the filling line needs to be defined. This marks a horizon, up to which material can be disposed (Fig. 8). In a use case to determine the volume of sediments which can now be dumped underneath this maximum filling line, the disposal volume on defined disposal area can be varied until this height is reached.

In Fig. 8 the maximum filling line was defined at a total height of 302.5 m. A disposal area was roughly defined around the deepest parts of a scour hole (see the red line in Fig. 8, left). The disposal volume was determined to be 70 000 m$^3$. The resulting topography of the scour filling is shown in Fig. 8 (right). From this values and the initial bathymetry (Fig. 8, middle) the bottom evolution is computed (Fig. 8, left). It can be seen that the nodes inside the defined disposal area are all heightened by a different value due to the initial topography and the maximum filling line. Fig. 8 (below) further shows the filling process of the scour over the simulation time.

V. PRACTICAL APPLICATION

This section presents an example of modelling artificial bed load supply measures to stabilize the riverbed at a River Rhine reach between Iffezheim and Mainz.

In order to stabilize the riverbed and through this hold up the water level, artificial bed load supply campaigns are conducted along some erosive stretches of the River Rhine between Iffezheim and Mainz (Fig. 9). Downstream of the barrage of Iffezheim around 200 000 m$^3$ sediments are disposed annually on different disposal fields (Fig. 10). The disposed material comes from gravel-pits located in the area of Iffezheim.

It is essential to consider these anthropogenic measures in order to successfully simulate the sediment transport regime of this river stretch. Only if all main influences on the hydraulic and morphologic system are taken into account reliable results for possible optimization scenarios can be expected.

Therefore, the simulation package Telemac2D, Sisyphe and DredgeSim was applied between Iffezheim and Speyer (Rhine-km 334.7 – 394.3) in order to model common artificial bed load supply measures [6]. The computed drifting and transport velocities of the disposed sediment fractions are compared with measurements carried out during a tracer experiment by the Federal Institute of Hydrology (BfG) [7]. The functionality of modelling artificial bed load supply in DredgeSim was used. Over a simulation period of five years all five disposal fields were fed with 100 000 to 300 000 m$^3$/year, varying by discharge. The distribution of the disposed sediment was represented by four classes of sediment with the following mean grain diameters:
Figure 9. Artificial bed load supply at the River Rhine (picture taken by WSA Freiburg).

- $d_{i,m} = 2.0 \text{ mm}$
- $d_{i,m} = 11.0 \text{ mm}$
- $d_{i,m} = 21.0 \text{ mm}$
- $d_{i,m} = 42.0 \text{ mm}$

Those grain sizes resemble typical grain sizes dumped at such campaigns. For the dumping process the amount of sediment volume and the disposing time periods for the five disposal sites must be defined. Furthermore, the grain size distribution has to be specified.

The distribution and the names of the sediment classes are given in DredgeSim (Fig. 12). The grain size of the sediment is defined in Sisyphe. Therefore, all classes on the riverbed and the ones being disposed have to be specified there.

The computation mesh used in this study consists of approximately 117 000 elements. The bed load transport in Sisyphe was calculated with the formula of Meyer-Peter and Müller. Additionally, exposure effects were regarded following the approach of Karim, Holly and Yang. The Nikuradse law was used to determine the roughness coefficient for the hydrodynamic computation.

The results of the simulation of each sediment class of the artificial bed load campaign after one year due to drifting from the disposal site are shown in Figure 11.

The front velocities (velocities of the fastest particles) of the four sediment classes were computed:

- $10.5 \text{ km/a}$ for $d_{i,m} = 2.0 \text{ mm}$ (class 0–8 mm)
- $18.5 \text{ km/a}$ for $d_{i,m} = 11 \text{ mm}$ (class 8–16 mm)
- $6.5 \text{ km/a}$ for $d_{i,m} = 21 \text{ mm}$ (class 16–32 mm)
- $2.5 \text{ km/a}$ for $d_{i,m} = 42.0 \text{ mm}$ (class 32–64 mm).

The observed front velocities of similar sediment classes of the tracer experiment are:

- $11.5 \text{ km/a}$ (class 4–8 mm)
- $11.5 \text{ km/a}$ (class 8–16 mm)
- $8.9 \text{ km/a}$ (class 16–31.5 mm)
- $7.6 \text{ km/a}$ (class 31.5–45 mm)
- $4.4 \text{ km/a}$ (class 45–62 mm).

The computed transport velocities of the first, the third and the fourth sediment class are comparable to the measured ones. The modelled transport velocity of class $d_{i,m} = 11 \text{ mm}$ is significantly faster than the observed one. As the measurements of the tracer experiment were carried out at different discharge conditions, the results of the simulations and the measurements are not directly comparable. A further explana-
The model shows a good validity concerning the transport behaviour of grain feeding sediment classes. Therefore, it can be applied for computations of different sediment management scenarios and the reliable evaluation of such optimization studies. The results of a calibrated model can then be used for analyzing and evaluating maintenance measures.

VI. SUMMARY

In order to model dredging and disposal strategies due to the maintenance of waterways the software package DredgeSim was developed. This package is coupled to Telemac and Sisyphe, which provide the initial data. First practical experiences and typical applications for modelling dredging and disposal activities have been presented. An application for artificial bed load campaigns along a stretch of the River Rhine has been further discussed.

It has been shown how a modelling package of Telemac2D, Sisyphe and DredgeSim can be used to analyze maintenance actions with numerical models. As the whole package is MPI-parallelized an application on high performance computers is possible. A potential application is the use in hindcast models to estimate the effects of anthropogenic influences on the total bed evolution over a certain time period. Further applications are the use in forecast models to predict maintenance efforts, the simulation of different
dredging strategies or the possibility to analyze the transport paths of sediments disposed by artificial bed load supply.

VII. OUTLOOK

With the presented modelling package the effects of new maintenance concepts with regard to anthropogenic changes from dredging and disposal activities can be computed and studied. Optimization of dredging strategies is necessary for a sustainable sediment management along navigation channels [8]. Numerical models which are able to take into account dredging and disposal actions can support this optimization process in the future. Further research is needed, especially regarding applied optimization and interpretation techniques. A challenge will be the direct use of computational results to solve an optimization task, as a lot of regional parameters have to be considered.

REFERENCES

URL: http://www.hamburg-port-authority.de.
[4] Bundesanstalt für Wasserbau, Information on the IPDS-concept for defining initial physical datasets (German).
URL: http://www.baw.de.
Bed roughness feedback in TELEMAC-2D and SISYPHE

David L. McCANN, Alan G. DAVIES, James D. BENNELL
School of Ocean Sciences, Bangor University
Menai Bridge, Anglesey, UK, LL595AB
d.l.mccann@bangor.ac.uk

Abstract— An adaptation to TELEMAC-2D and SISYPHE which allows the feedback of bed roughness to the flow calculation has been implemented in a validation case in the Dyfi Estuary, Wales, UK. The dimensions of dune-scale bed forms measured from a swathe sonar survey in July 2007 are compared with bed roughness lengths calculated by a coupled simulation showing very promising agreement given the simplifying assumptions made in the process. The effect of bed roughness feedback to TELEMAC-2D on the sediment transport rate subsequently modelled by SISYPHE is also explored for a simple M2 harmonic tide. The sediment transport magnitude is shown to increase significantly in one area while decreasing in another, suggesting that roughness feedback can significantly affect the pattern of sediment transport and therefore morphological change.

I. INTRODUCTION

Beds of non-cohesive sediment, such as in many coastal environments, are rarely, if ever, flat. Sedimentary bed forms are generated by an interaction between the flow and the erodible bed. The frictional force generated between the flow and the bed is heavily affected by the presence and form of these sedimentary structures. Form drag, caused by the pattern of pressure around the bed form due to flow separation over the crest, generates a frictional force which is often the dominant resistive force between bed and flow in shallow, coastal tidal environments.

The hydraulic roughness of the bed has a direct effect on the flow velocity through the roughness length $z_0$ occurring in the logarithmic velocity profile. As such, for a given driving pressure gradient, an increase in the hydraulic roughness of the bed results in a decrease in the depth-averaged flow velocity $U$. However, bed roughness also has a direct impact on the kinematic bed shear stress $\tau_0$, both through the drag coefficient $C_D$ and its previous effect on $U$, which is subsequently amplified (either positively or negatively) as the bed shear stress is proportional to the drag coefficient multiplied by the square of the velocity. The effect of this change in bed stress is transmitted to the magnitude of sediment transport, being further enhanced as $\tau_0$ enters most sediment transport formulae as the Shields parameter $\theta$ raised to a power greater than one.

The effect of bed roughness on the computation of flow in complex, process-based numerical models such as TELEMAC-2D has been explored in the past. However research has been concentrated on the impact of ripple-scale features (bed forms with wavelength much smaller than the flow depth). In contrast, the impact of sub grid scale dunes (bed forms with wavelength that scale on the flow depth) is poorly understood. Although dune roughness is sometimes accounted for by increased (yet still static) values there is no direct, concerted link between the roughness generated and used by SISYPHE and that used by TELEMAC-2D.

For the most part, the lack of research into the influence of dunes on coupled morphological models is due to a lack of field evidence which requires a greater spatial range (due to the size of dunes) than that for ripples. A recent field campaign in the Dyfi Estuary, Wales, U.K., however, provided an extensive, fine-scale map of the seabed using swathe sonar. A significant amount of data was collected around the Dyfi ‘scour pit’ – a permanent feature of the estuary maintained by converging tidal currents and stable substrates [1]. In this paper a coupled morphological model of the Dyfi Estuary is presented which contains a mechanism to feed back bed roughness into TELEMAC-2D. The model has been run to simulate a period of 21 days in July 2007, which coincided with the swathe sonar bathymetric survey with the aim of reproducing the bed roughness derived from these measurements. The simulated free surface is validated by tide gauge data obtained during the same field campaign. The effect of coupling the roughness between TELEMAC-2D and SISYPHE on the subsequent sediment transport rate for a simplified tidal case in the Dyfi is also explored.

II. METHOD

The bed roughness predictive procedure of Van Rijn [2] has been implemented within SISYPHE in the subroutine RIDE. Here the total hydraulic roughness $k_r$ is calculated from the roughness contributions of ripples $k_{r,m}$ megaripples $k_{r,m}$ and dunes $k_{r,d}$ separately and then combined via quadratic summation. The roughness contributions are calculated from three continuous curves:

$$k_{r,m} = f_s D_{50} [85 - 65 \tanh(0.015(\psi - 150))]$$  \hspace{1cm} (1a)

$$k_{r,m} = 0.00002 f_s h [1 - \exp(-0.05\psi)] (550 - \psi)$$ \hspace{1cm} (1b)

$$k_{r,d} = 0.00008 f_s h [1 - \exp(-0.02\psi)] (600 - \psi)$$ \hspace{1cm} (1c)
where $f_s$ is a granular scaling coefficient (equal to 1 in this case), $D_{50}$ is the median grain diameter and $\psi$ is the mobility parameter:

$$\psi = \frac{U^2}{g(s-1)D_{50}}$$

(2)

where $g$ is acceleration due to gravity, $s = \rho_s/\rho$ ($\rho_s$ is the sediment density and $\rho$ the fluid density). The three roughness contributions are then combined to form the total hydraulic roughness by

$$k_s = \sqrt{k_{s,m}^2 + k_{s,r}^2 + k_{s,d}^2}$$

(3)

with the values of $k_s$ at each grid node. This file is subsequently read at the next computation time-step by TELEMAC-2D (reading $k_s$ into the subroutine CHESTR), with SISYPHE reading in $k_{s,r}$ internally. In this way the depth-averaged flow calculated by TELEMAC-2D is based on the total roughness of the flow, whereas close to the bed where small-scale roughness is most important SISYPHE only uses the ripple roughness to calculate sediment transport. The Dyfi model was run on a grid (created using BlueKenue) of 91,000 nodes, with grid resolution fining from 2km at the offshore liquid boundary to a constant 15m within the interior of the estuary basin. Both rivers (the Dyfi and the Leri, Figure 1) were included using channel meshes with a constant along-channel resolution and a fixed number of points across-channel. Bathymetry (Figure 1) was provided by a combination of a LiDAR survey from 2004 (provided by the Environment Agency Wales) and the Dyfi model grid of P.Robins (Bangor University), which contained a mixture of LiDAR and Admiralty Chart data. Tidal boundary conditions were assigned using the TELEMAC-2D subroutine BORD, with eight tidal constituents provided by S. Neill (Bangor University), tuned to a start-time corresponding to the 9th July, 2007. Phases and amplitudes of each constituent were mapped to each boundary node creating a boundary-varying tidal condition. The Dyfi and Leri river inputs were taken as the 2007 annual mean flow-rates (20m$^3$s$^{-1}$ and 1.7m$^3$s$^{-1}$ respectively) from river flow-rate data provided by the Centre for Agriculture, Fisheries and Aquaculture Science (CEFAS).

The model was run for the simulation period of 12 days with a time-step of 5s. TELEMAC-2D and SISYPHE were run together in coupled mode, with the latter using the sediment transport formula of Bijker. Grain size was set to 220$\mu$m (an average grain size for the Dyfi [3]). Tidal elevation data from a gauge at Aberdyfi Pier (North bank opposite the spit, Figure 1) over the survey duration was utilised to validate the free surface.

On the 9-11th July, 2007 a swathe sonar bathymetric survey was carried out in the Dyfi as part of a monitoring campaign funded by the Centre for Catchment and Coastal Research (CCCR). The instrument (on loan from R. Bates at the Univer-
sity of St. Andrews) was a SEA Ltd `SWATHplus-H' system with a 468kHz transducer pole mounted over the side of a small vessel. Bed elevations were calculated from the depth beneath the instrument, the elevation of the vessel on the free surface was obtained using a Leica 1200 RTK-GPS, and tidal data from two gauges was used to make corrections. The raw data was processed by R. Bates to 1m resolution in the horizontal and 10cm in the vertical, producing a map of bed elevations which could be used to extract the dimensions of dune-like bed forms with wavelengths of the order of tens of meters (Figure 2). Due to the need for a minimum depth of water for the side scan instrument to operate correctly, measurements were taken at high-water and could only be taken a short distance up the main tidal channel. Extensive measurements were taken around the scour pit and estuary mouth however, where relatively deep water made wide-area bed scans possible.

To extract individual bed forms from the sonar data set 103 profiles were taken, each selected to lie as close as possible to perpendicular across bed form crests. Profiles were also chosen to only intersect areas containing bed forms (much of the channel section of the sonar data was devoid of bed forms). As such the profiles varied considerably in length, from a single bed form to hundreds of meters. The profiles were then run through a turning points algorithm to identify the coordinates of bed form crests and troughs. To calculate the bed form length \( \lambda \) and height \( \eta \) individual bed forms were identified as two troughs with an intermediate crest. With the assumption that the bed forms were long-crested and two-dimensional in plan form, \( \lambda \) can be defined as the distance between two successive trough points. However many of the bed forms were found to lie on a mean bed slope and so the true on-ground distance between two trough points would be greater than the distance between their respective \( x \) coordinates. Bed form length was therefore taken in all cases as the length of line connecting both trough points in \( x,z \) space. The bed form height \( \eta \) was similarly taken from a line perpendicular to that between two troughs and intersecting the intermediate crest point. Each measurement of length and height was co-located with the 'real-world' \( x,y \) coordinate of the corresponding bed form crest, calculated by interpolating from the \( x,y \) coordinates of the end-points of each profile.

2730 individual bed forms were measured, covering much of the area around the scour pit as well as an up-stream portion of the main ebb channel (Figure 3).

To compare \( k_s \) values from the model with bed form dimensions extracted from the sonar survey the simplifying assumption was made that, as dune roughness is believed to dominate the total roughness and \( k_s \approx 0.5\eta \) [1,4], \( 2k_s \) is approximately equal to \( \eta \). Extracted bed form heights were then interpolated onto a regular grid and two profiles (A-B and C-D) were taken (Figure 4), one through the major-axis of the scour pit and one further into the estuary interior. Similar profiles of modelled \( k_s \) were taken for comparison.

A second model condition was run to assess the impact of the \( k_s \) feedback approach in TELEMAC-2D on calculated velocities and the resulting transport of sediment in SISYPHE running in coupled mode. A simplified tidal case with one harmonic constituent (M\(_2\) tide, amplitude 1.8m) was run on the same grid. In case 1 roughness feedback to TELEMAC-2D was not activated, with only the ripple component of roughness being passed to SISYPHE for use in the sediment transport calculation. In this case TELEMAC-2D was run with a separate, constant roughness of 1cm. Case 2 was the same, only with \( k_s \) feedback to TELEMAC-2D activated.

III. RESULTS

Figure 5 shows the free surface elevation from the Aberdyfi tide gauge (black line) and the model (red line) over the validation simulation period, with the sonar survey period shown as the grey area between the 9\(^{th}\) and 11\(^{th}\) July. The model can be seen to reproduce the observed tide relatively well, although there is a phase shift between model and observations which is most likely due to a lack of tidal
constituents or error in their implementation around the liquid open boundary (for example start time or nodal corrections). However the model reproduces the tidal envelope relatively well and captures the amplitude of the spring tide.

Figure 6 shows time series (centred around spring tide) of the modelled velocity components, free surface elevation, skin friction Shields parameter and total and ripple roughnesses ($k_s$, and $k_{s,r}$ from equations 3 and 1a respectively) from a point in the centre of the scour pit (Figure 4, point E).

![Figure 6](image)

**Figure 6.** Time series of model parameters during peak spring tides from a point at the centre of the scour pit (Figure 4, point E). From top to bottom: Free surface elevation, $u$ and $v$ velocity components (black and red lines respectively), $\theta_s$ and $\theta_{s,r}$ (black and red lines respectively) and in the bottom panel the total bed roughness $k_s$ (black line, left axis) and ripple component of roughness $k_{s,r}$ (red line, right axis).

The effect of tidal velocities (and therefore bed shear stress and Shields parameter) on the calculated values of $k_s$ and $k_{s,r}$ is seen to vary significantly through the tidal cycle. Total and ripple roughness lengths can be seen to be out of phase with each other as equation 1a gives larger values of $k_{s,r}$ for low flow conditions whereas equations 1b and 1c provide the opposite ($k_{s,d}$ dominates $k_s$ when dunes are present). Importantly, the bottom panel in Figure 6 highlights the ephemeral nature of roughnesses predicted with the Van Rijn method. Although a critical Shields criterion is included (preventing bed forms to adapt in flows below the threshold of motion) the equilibrium nature of the predictor forces bed forms to adapt at the same rate as the change in tidal currents. In reality there is a development time to equilibrium dimensions (and therefore equilibrium $k_s$) associated with bed forms which is dependent on flow conditions which, in the case of ripples alone, can be of the order of hours to hundreds of hours [5]. Due to the continually changing tidal flow conditions the equilibrium roughness for any given flow might not be achieved. Further work needs to be conducted to introduce history effects into the roughness computation.

![Figure 7](image)

**Figure 7.** Top panel: profile A-B of measured bed form height (black line shows data smoothed to 15m model resolution, dashed black line shows raw data), average $k_s$ over the simulation period (solid red line), maximum $k_s$ over the simulation period (solid blue line) and maximum $k_s$ over neap tides (dashed blue line). Bottom panel: profile A-B of bathymetry (in m below ODN) from the sonar survey (black line) and from the model (red line).

![Figure 8](image)

**Figure 8.** Top panel: profile C-D of measured bed form height (black line shows data smoothed to 15m model resolution, dashed black line shows raw data), average $k_s$ over the simulation period (solid red line), maximum $k_s$ over the simulation period (solid blue line) and maximum $k_s$ over neap tides (dashed blue line). Bottom panel: profile C-D of bathymetry (in m below ODN) from the sonar survey (black line) and from the model (red line).

Figures 7 and 8 show profiles of modelled $k_s$ and measured bed form heights $\eta$ along lines A-B and C-D respectively, with the roughness scaled to $2h$, for comparison. This scaling is based on the assumption that dune (and megaripple) roughness dominates the total roughness as ripple roughness is proportional to $\eta$ rather than $0.5\eta$ [6]. The maximum value of $2k_s$ corresponding to springs and the maximum $2k_s$ over neaps are both over-estimated towards the mouth of the Dyfi (left of Figure 7) –being over two times the measured bed form height in the scour pit. Here the model predicts a maximum roughness corresponding to large dunes– over 0.8m in height in the centre of the scour pit. However in shallower water the maximum $2k_s$ begins to show a better comparison with the data, reproducing an increase in measured roughness as the profile intersects the main tidal channel (~1.9km, Figure 7). The mean $2k_s$ over the simulation period shows much better comparison with measured bed form heights, for the most part capturing the trend of bed form height. The difference in bathymetry between the survey and model must be taken into account, especially in shallow water, as $h$ has a significant effect on megaripple and dune roughness (and therefore $k_s$). The difference in bathymetry is due to the time difference between bathymetry in the model (Lidar survey from 2004) and the sonar survey (July 2007). Figure 8 shows similar agreement between the average $2k_s$ and $\eta$, capturing the general trend of the measured data despite differences in bathymetry one again.
In practice, the maximum $2k_s$ may be unrealistic due to the absence of history effects on dune development. With an ’equilibrium approach’ such as this, peaking current speeds during flood and ebb are capable of producing very large roughnesses (albeit briefly) whereas in reality the bed may not be able to adapt to high flow conditions in time. The average $2k_s$ may therefore be a better approximation of roughness due to bed forms with a history effect in operation. The period at which the sonar survey was conducted must be taken into account as well –being over neap tides and at high water slack only. The mean conditions over the simulation period may also have a better correspondence with neap conditions as evidenced by the better agreement with the data. It must also be noted that comparison is only possible due to the scaling of $k_s$ to approximate $\eta$. Van Rijn’s method calculates the physical hydraulic roughness of the bed directly, bypassing the need to model bed form dimensions explicitly. It is, in practice, a simple step to scale $k_s$ back to bed form height and, although it may not be philosophically correct, it appears to work very well in the case of this Dyfi data.

Figures 9 and 10 show time series from the M2 tidal cases 1 and 2 (without $k_s$ feedback and with $k_s$ feedback, respectively). Figure 9 shows a time series from a point in the centre of the scour pit (Figure 4 point E, average depth 12m) and Figure 10 shows a time series from a point mid-way up the estuary in the centre of the main channel (Figure 4 point F, average depth 3.5m). In the deeper case (Figure 9) a decrease in flow magnitude in case 2 is evident, with an average decrease in velocity of 8%. Although this is relatively minor it leads to a much larger change in total bed shear stress – in the coupled case an increase of over 122% is seen at peak flow. This is due to the significant increase in $k_s$ between case 2 and 1 (over 40 times greater at peak flow) which results in a vastly increased drag coefficient and therefore bed shear stress. The subsequent average increase in total sediment transport magnitude over two tides is 74%. This shows how the non-linear nature of the sediment transport equations can amplify a small change in hydrodynamic input into a large change in sediment transport rate. Although the magnitude of the sediment transport rate is increased in the scour pit, the result from a point further up-estuary (where water is significantly shallower) suggests that the change in sediment transport rate is inverted (Figure 10). Here although velocity and stress are decreased and increased as before, the resulting sediment transport magnitude is decreased by an average of 37%. Here the differences in ripple roughness are much smaller thanks to the lower flow speeds. Yet the flow velocity is slowed to a greater extent in the feedback case as the flow is much shallower and so the added dune roughness (~0.1m in case 2 rather than 0.01m in case 1) has a larger effect on the flow. Bijker’s sand transport formula is a total-load formulation with a suspended transport fraction which is based on the flow speed. It is therefore possible that, at this shallower location, the suspended fraction becomes relatively more important to the total transport rate. As this is tied to the flow speed (which is slowed to a lesser extent in case 2) this might account for the decrease in total transport. What is important, however, is that the addition of roughness feedback to TELEMAC-2D, although not directly affecting sediment transport via processes at the small-scale, is able to significantly alter the magnitude and pattern of sediment transport in a shallow, tidal setting. This will have a significant effect on morphological development which should be further explored.

IV. DISCUSSION

The addition of roughness feedback to TELEMAC-2D is principally very important as sub-grid scale dunes, although not having a direct effect on local, small-scale processes important to sediment transport, can have a significant effect on total sediment transport rates once their impact on flow velocities has been taken into account. It is planned for future versions of TELEMAC-2D and SISYPHE to feature this method [7]. However the neglect of history effects on bed form development in a tidal environment needs to be addressed – for this it is proposed that a half-life decay curve could be implemented, with the maximum allowed rate of decay of bed form height scaling on the time scale required for equilibrium dimensions to be achieved. This maximum rate could be implemented relative to the model time step. We may theorise that the addition of history effects will
dampen the general time-pattern of calculated bed roughness so that it approaches the average over a tidal cycle. However its effect on deeper channels where larger dunes may become permanent features and on the subsequent propagation of the tide should be explored fully.

The difficulty in validating bed roughness on the scale of dunes over wide areas of bed is due to the large spatial and temporal time-scales involved. The use of the swathe sonar survey conducted in the complex, tidal environment of the Dyfi has allowed some insight into the implications of roughness feedback based on the use of Van Rijn’s [2] method for the prediction of \( k_s \). The agreement between model and data is encouraging, especially considering the numerous simplifying assumptions made in the comparisons. Bed forms in the natural environment are a product of a number of hydrodynamic and sedimentary factors, only some of which have been addressed using TELEMAC-2D and SISYPHE in this paper. For example only a single grain size has been used whereas the Dyfi does display a degree of variability [3]. Secondly, the presence of waves and their subsequent modification of the bottom boundary layer has been ignored from this study and, while they may have negligible effect in the estuary interior, the study location at the mouth of the Dyfi does experience a degree of wave ingress from Cardigan Bay [8].

ACKNOWLEDGEMENTS

This work was undertaken as part of a PhD funded by the EPSRC at Bangor University, U.K. The sonar bathymetric survey of the Dyfi was undertaken as part of a field campaign funded by the Centre for Catchment and Coastal Research (CCCR). Lidar data for the Dyfi model grid was provided by the Environment Agency Wales. Many thanks to R. Bates at the University of St. Andrews for post-processing of the sonar data and to S. Neill at Bangor University for supplying (and help with implementing) tidal constituent data.

REFERENCES


A continuous sediment layer concept for Sisyphe

Uwe MERKEL
Consulting Hydraulic Engineer
Karlsruhe, Germany
uwe.merkel@uwe-merkel.com

Rebekka KOPMANN
Project Team Modelling of Sediment Transport
Federal Waterways Engineering and Research Institute (BAW)
Karlsruhe, Germany
rebekka.kopmann@baw.de

Abstract— Sediment layer thickness and grain size distributions influence the bed erosion stability and the flow field due to grain roughness and bed form roughness. Vice versa flow sorts sediments and develops bed forms. Therefore many hydrodynamic numerical simulations cannot be completed successfully without considering flow-sediment interaction. In Sisyphe sediment transport, sediment sorting and development of bed forms are all highly influenced by one parameter, the active layer thickness. The concept of active layer has been developed in 1971 by Hirano and expanded by Ribberink among others. With new high performance computers, it is possible to overcome several limitations of this meanwhile 40 year old concept. The limitation to 9 layers, the a priori chosen layer thicknesses and the continuous remixing of the top layer strongly influences the sediment transport.

The authors were inspired by continuous vertical sorting models, which were examined in Delft during the last decade by Astrid Blom among others. The new approach still uses an active bed zone, similar to the active layer, but all sedimentation, erosion and change of grain fractions is stored in a high resolution depth profile for each node of the hydraulic mesh, instead of 9 discreet layers. For evolution calculations, at each time step the active layer is updated with averaged data from the vertical sorting profile. With this new concept it is possible to avoid smearing effects in grain fraction calculations. This leads to a better reproduction of the natural sediment profile and thus to a better prediction of the transport processes.

To validate this concept, comparisons were made with the Hirano / Ribberink approach with flume data from Astrid Blom. Ongoing validations with more flume experiments will open the way to further developments. A modular addition of algorithms for compacting or moving of fines within a coarse matrix is possible, as the implementation of this storage concept is kept similar to the classic layer & fraction approach in Sisyphe.

I. STATE OF THE ART & LIMITATIONS OF SEDIMENT LAYER MODEL OF SISYPHE V6P0

Considering sediment distribution and a vertical sorting of the sediments is essential for a successful modelling and prediction of river morphology. Sediment parameters and flow interact together and influence each other in a complex way. E.g. grain roughness is dependent on sediment distribution, which is influenced by bed forms. Both change the flow and vice versa the flow changes the sediment distribution and develops bed forms. Therefore many hydrodynamic numerical simulations cannot be completed successfully without considering flow-sediment interaction.

In Sisyphe sediment transport, sediment sorting and development of bed forms are all highly influenced by one parameter: the active layer thickness. The concept of an active layer (HR-VSM) has been developed in 1971 by Hirano [5] and expanded by Ribberink in 1987 [6] among others. The idea was that flow interacts with a fully mixed top-most layer. The active layer describes the common depth of morphological processes in the riverbed per time step. The active layer thickness is usually chosen between 3d90 and the mean height of bed forms. For numerical reasons it is the maximum depth

Figure 1. Scheme of the classic Hirano/Ribberink vertical sorting model (HR-VSM) and the later explained continuous vertical sorting model (C-VSM).
that can be eroded in one time step. Below the active layer follows another theoretical layer, the active stratum. It is used to refill or reduce the active layer to the predefined thickness after evolution calculations changed the active layer thickness. Below these 2 layers up to 7 more storage layers can hold different sediment mixtures until they are activated by erosive processes. Within 1 time step evolution only affects the active layer and the active stratum, see Fig. 1.

This meanwhile 40 year old concept was developed during a time where the average computational performance was $10^{10}$ times less than in 2011. It incorporates several limitations:

- The number of layers is limited to 9 layers.
- The a priori chosen layer thicknesses depend on dune heights, grain roughness, depth of the rigid bed, mesh density and other parameters.
- The continuous remixing of the top layer strongly influences the sediment transport e.g. development of bed forms.

While the first two limitations could be removed, the last one requires a new concept.

The problem of a fully mixed active layer becomes obvious in the flume experiment explained later. Dunes occurred but could only be simulated, if the active layer thickness is set to be equal the mean dune height. Due to this the modeller needs to know a priori the expected dune regime and he can’t change it in case of time dependent flow conditions. Otherwise an active layer thickness calculated according to the grain size (e.g. $3d_{50}$) results in an armoured bed without dune development (see Fig. 2). Another negative effect is the inability to preserve thin but prominent layers, e.g. the armoured bottom of dunes in Fig. 1.

Due to strong averaging and clipping effects of the HR-VSM, good results can be achieved only in a wider spatial context. Reason is the modification of the active layer, which might influence results stronger than any other parameter, including $d_{50}$. Unfortunately the active layer thickness (ALT) is a deterministic, theoretical mean value, originally meant to describe the thickness of the morphological active top layer of the bed. It is difficult to measure and its natural complement strongly varies with shear stress and many other variables. Choosing the ALT beforehand the calculation, e.g. without knowing dune heights, leads to mixing and smearing effects of the grain fractions within the upper 2 layers. This happens because the ALT is deterministic and all changes in volume have to be passed proportionately through to the active stratum. The active stratum might grow to unlimited size without any internal vertical discretization.

Fig. 2 shows the ALT problematic exemplarily along the middle axis of a flume experiment calculated by the authors with Telemac 2D coupled with Sisyphe v6p0 using different ALTs. Case II “ALT = mean dune height” fits the average measured fractions the best. Here mean dune height doesn’t represent the mean value of all dune crowns, but the dune amplitude, see Ribberink [6].

II. THE VERTICAL SORTING EXPERIMENTS OF BLOM ET AL.

Astrid Blom [1,2,3,4] conducted flume experiments at Delft Hydraulic Laboratories in 1998 to investigate vertical sorting processes. She used the obtained data to develop her own vertical sorting model. The authors decided to use these experiments as validation cases as well.

The appendent laboratory flume was 50m long, 1m wide and filled with an artificial three modal grain mixture (0.00068, 0.0021 and 0.0057 m, 33% each). For a discharge of 0.267...
m³/s a slope of 0.0018 produced a normal flow depth of 0.386 m for case “B2”. The sediments were recirculated (see Fig. 4). In both physical and numerical experiments the field of flow requires the first half of the flume to develop constant conditions, thus only the second half is used for comparison.

Blom numerically simulated the morphology of the flume experiments with different vertical sorting models, using a constant backwater curve and not a multidimensional hydrodynamic numerical model like Telemac2D or 3D. Among other methods she experimented with an own continuous vertical sorting model, which is in fact a storage model with a high, but limited number of very fine layers. Evolution is calculated with classic approaches like van Rijn or Meyer-Peter & Müller.

Erosive impact forces lose their power over penetration depth according to a probability density function, which equals the ALT concept in case of a constant distribution function. Compared to her own Hirano & Ribberink implementations the

Case I: Sedimentation

Old C-VSM for t₁

active layer: averaged from sorting profile

“deposition of fraction I” added to sorting profile

Z_B
Z_F

New C-VSM for t_{i+1}

New HR-VSM for t_{i+1}

(Sisyph v6p0)

coarsening!!!

Case II: Erosion

Old C-VSM for t₁

active layer: averaged from sorting profile

Active Layer after “erosion of fraction I” eroded material subtracted from sorting profile

Z_B
Z_F

New C-VSM for t_{i+1}

New HR-VSM for t_{i+1}

(Sisyph v6p0)

still fines available !!!

**Figure 3.** The changed and finer book keeping of Continuous Vertical Sorting Profiles (C-VSM) avoids smearing problems due to averaging in the classic Hirano/Ribberink layer method (HR-VSM). This sketch shows the behavior of both book keeping algorithm for one time step.
C-VSM succeeds when using averaged vertical sorting profiles. This academic numerical model didn’t adapt the hydrodynamic and the surface after every time step, it was custom made for this single project.

III. IMPLEMENTATION OF A CONTINUOUS VERTICAL SORTING MODEL (C-VSM)

Similar to the vertical sorting model of Astrid Blom [3] we decided to add a depth dependent storage model with unlimited resolution for the grain sizes. The transport model remains unchanged. Both are kept in separate modules to enable an independent development. The addition of a consolidation model as well as an implementation of statistical methods for the erosive impact depth are possible for future developments without major changes.

The new storage model can be described as book keeping model as shown in Fig. 4. It is a data set of virtual layers, theoretically unlimited in their numbers, thicknesses and grain size fractions. A drilling profile is the physical equivalent. For better visualization the grain size fractions of each layer are sorted from fine (left) to coarse (right) (legend: see Fig. 1). In contrast to the classic Hirano-Ribberink layer model there are no theoretical limitations to the discretization of thicknesses, but the capabilities of the hardware.

Transport model calculations of Sisyphe are not touched by the implementation of the C-VSM. The Hirano / Ribberink concept (HR-VSM) is still used to calculate evolution based on the active layer. The grain size fractions are now taken from the VSP by calling ADD_VSP_Layer & ADD_Fraction. The transport model remains unchanged. Both are kept in separate modules to enable an independent development. The addition of a consolidation model as well as an implementation of statistical methods for the erosive impact depth are possible for future developments without major changes.

The new storage model can be described as book keeping model as shown in Fig. 4. It is a data set of virtual layers, theoretically unlimited in their numbers, thicknesses and grain size fractions. A drilling profile is the physical equivalent. For better visualization the grain size fractions of each layer are sorted from fine (left) to coarse (right) (legend: see Fig. 1). In contrast to the classic Hirano-Ribberink layer model there are no theoretical limitations to the discretization of thicknesses, but the capabilities of the hardware.

Transport model calculations of Sisyphe are not touched by the implementation of the C-VSM. The Hirano / Ribberink concept (HR-VSM) is still used to calculate evolution based on the active layer, but the content of the active layer changes. The grain size fractions are now taken from the VSP and averaged over the ALT for each time step. Therefore it is called “Projected Layer HR-VSM” (PL-VSM). The main benefit is the conservation of any layering that is finer than the ALT without changing the classic transport models.

Fig. 3 shows the difference between the storage models in HR-VSM and the C-VSM in case of sedimentation and erosion.

Extraction of fines and burying of coarses can be found in the BLOM flume model case B2 as well as in many rivers, contrary to the armouring example in Fig. 2. The HR-VSM might not be able to reproduce this. A theoretical example would be a channel with erosion and deposition, dependent on turbulence and multidimensional effects. We use an ALT of 0.2 m, which is chosen to 50% of the expected average dune height. Fig. 5 shows a typical vertical sorting profile developing in 2 time steps.

The HR-VSM will develop as follows:

- A first deposition phase of 15 cm of fine material on coarse material will be saved in an active layer mixture of 75% fine + 25% coarse material. \( T = 1 \).
- A second deposition phase will mix 10 cm fine with 10 cm of (75% fine + 25% coarse) resulting in 12.5% coarse saved in the active layer. \( T = 2 \).
- If flow conditions change to erosion, coarse material is not moving.
- Depositing additional fine material (less than the ALT in 1 time step) will always result in numerical lifting of coarse to the top layer due to the averaging.

The C-VSM model places the newly deposited material on top and does not mix it with the underlying material. It will develop as follows:

- A first deposition phase of 15 cm of fine material on fine material will be saved on top of the VSP without mixing. An active layer mixture of 75% fine + 25% coarse material is averaged from the VSP. No changes to HR-VSP so far. \( T = 1 \).
- A second deposition phase will save 10 cm fine material on top of the underlying material in the VSP. Averaging the top 20 cm for the new active layer mixture results in 100% fine material, which might result in a full erodible layer when it comes to erosion. \( T = 2 \).
- Additionally the bed roughness changes, as a function of the mean diameter \( d_{50} \).

The practical implementation in Sisyphe is a set of bief objects, defined in DECLARATIONS_SISYPHE.F. They describe fractions over the depth for each mesh point. In the LAYER.F module the active layer mixture is averaged each time on demand from the C-VSP by calling MAKE_ActiveLayer. If new material is deposited according to BEDLOAD.F it is set on top of the VSP by calling ADD_VSP_Layer & ADD_Fraction.
If a certain volume is eroded from the active layer, it will be removed in the vertical sorting profile starting form top by calling RM_Fraction. (Other options are possible, e.g. erode equally over the ALT.)

IV. FIRST RESULTS OF THE NEW C-VSM FOR SISYPHE

Figs. 6 and Fig. 7 show first results of simulations of case “Blom B2” calculated with Sisyphe v6p0 and Sisyphe v6p0 + C-VSM. A 2D mesh with 2193 nodes and an average edge length of 0.1 to 0.25 had sufficient density for this case. Fig. 6 shows initial and developed drilling profiles every 5 meters along the middle axis of the flume for both sorting models. Fig. 7 shows the mean grain size $d_{50}$ of the surface after 4h for the classic HR-VSM, the C-VSM and the PL-VSM (which is the averaged from the top 3 cm of the C-VSM).

The C-VSM case shows exactly the burying effects observed in the flume experiments and described in the last chapter with example values.
- Only the fine and parts of the medium grain fraction are moving.
- Fine grains are soaked out from sub surface. Remaining coarse grains fill the gap and create a coarse layer with lower elevation.
- Deposition of fine grains is on top of the existing sediments, eventually burying the coarse material.
- Continuing erosion / deposition cycles pull out fine and bury the coarse grains, which are resistant to erosion.

In the experiment the fraction of the fine material reached around 70% after 4h. In the numerical model more than 42% were reached. This correct tendency can be improved by calibration.

V. CONCLUSION & OUTLOOK

The continuous vertical sorting model (C-VSM) already shows promising results that overcome many limitations of the layer concept (HR-VSM) in Sisyphe v6p0. This is despite the fact that it is still implemented with a strong focus on code compatibility for validation purposes, which restricts some possibilities, like dynamic active layer thicknesses.

The smearing effects and the lack of vertical discretization of the bottom are improved and now only limited by computational power. With the new model it is possible to keep minor but prominent grain mixture zones even after a high number of time steps.

Figure 6. Drilling profiles along flume middle axis at initial time 0 h (top) and after 4 h shown as HR-VSP (middle) and C-VSP (bottom) (orange: fine material, grey: median, blue: coarse). Simulation with recirculating sediment, old layer concept with 9 layers.
Further validation cases will be calculated to prove the superiority of C-VSM. The new storage model has the following advantages:

- A dynamic active layer thickness that now is more independent of smearing effects of grain size fractions due to less averaging processes.
- Depth functions (probability density functions) for the impact of the shear stress instead of a fixed average active layer thickness. For an overview over these functions see e.g. Malcherek [8]
- Consolidation models for time-dependent porosity changes of sedimentation zones.

The sub models provide an interface for important future developments. They are inspired by Hiranos original idea, where the active layer thickness is the depth at which morphological activity normally stops. Until now, this depth is an empirical mean value, hard to measure and

- has growing uncertainties the coarser the spatial steps gets (mesh width),
- is sensitive to the length of the observed morphological activity (time step),
- and dependent on the shear stress magnitude.

ACKNOWLEDGEMENT

The authors thank Astrid Blom for her advice and the fruitful discussions.

REFERENCES

New developments and validations for the 3D sediment transport modelling within the Telemac Modelling System

Pablo TASSI
EDF R&D – Laboratoire National d’Hydraulique et Environnement (LNHE)
Chatou, France
pablo.tassi@edf.fr

Catherine VILLARET & Jean-Michel HERVOUET
EDF R&D – Laboratoire National d’Hydraulique et Environnement (LNHE)
Saint Venant Laboratory for Hydraulics
Chatou, France

Abstract—In this work, some recent developments for the 3D sediment transport modelling are presented in the framework of the Telemac Modelling System. The mathematical formulation of the implemented modules is discussed and some special issues arising from the treatment of the bottom boundary are addressed. In particular, the choice of the near-bed concentration and reference level is analyzed and a new methodology is introduced to avoid infinite concentrations at the bed level, while conservation properties are upholding. The model is verified by comparison with the well-known analytical solution of Rouse and validated with 2D simulations and experimental data from the trench evolution setup of van Rijn. The sensitivity to the sediment parameterizations and turbulence closure relationships is also analyzed. In all cases, the 3D model performs well when compared against analytical results and measurements of velocity and suspended sediment profiles, and shows good agreement in reproducing changes of the bed.

I. INTRODUCTION

Over the years, two-dimensional (2D), depth-averaged numerical models have been developed to predict sediment transport rates and changes of bed level [15]. The 2D approach, strictly valid under well established hypothesis, has been widely used by the engineering community to compute medium to long term bed evolution in large-scale applications. However, situations where strong secondary flows, stratification effects or complicated spiral motions are present can only be represented realistically by three-dimensional (3D) models. Furthermore, detailed 3D modelling of flow and suspended sediment transport can provide useful information on the complex flow structures characterized by strong vertical gradients of both velocity and suspended sediment concentration in the near-bed boundary layer [4].

Despite some recent progress in the development of full 3D models, see for example [1, 8, 14, 16] and references therein, some relevant issues such as the inherent difficulty to capture the vertical flow structure, the dependence of model results to the bottom boundary conditions and the choice of turbulence closures has not been yet emphasized enough. In this work, we address some of these issues by comparing 2D and 3D numerical simulations of coupled flow and suspended sediment transport on the basis of the open-source Telemac Modelling System [11].

In the 2D model, the depth-averaged suspended sediment concentration is calculated by solving an advection-diffusion equation. In this 2D transport equation, a correction factor can be applied to the advection term in order to account for the non-uniform vertical distribution of flow velocity and concentration over the depth [7]. In the 3D model, the flow field is computed by solving the continuity equation and the Reynolds-averaged Navier-Stokes equations. The Reynolds stress tensor is modelled by suitable turbulence closure relationships. The suspended sediment load is then calculated by solving the full 3D convection-diffusion equation for the suspended sediment concentration distribution. As pointed by Begnudelli et al. [2], sediment transport rate predictions can be highly sensitive to the choice of the near-bed concentration and reference level. Therefore, a methodology is proposed to avoid infinite concentrations at the bed level, while conservation properties are upholding.

The capabilities of both 2D and 3D models are demonstrated by comparing numerical results against analytical solutions and experimental data. First, we perform numerical simulations in steady, uniform flows for a prescribed flow rate and variable bed roughness. The 3D model results are validated against classical turbulent boundary layer concepts, like the logarithmic velocity profile and the Rouse distribution of concentration. The very large gradients of velocity and sediment concentration within the near-bed boundary layer are accurately captured by using a suitable refined vertical grid. Then, both 2D and 3D models are applied to simulate the laboratory experiment of a migrating trench [12]. The experience of a migrating trench
in a flume is used to validate and compare the 2D and 3D numerical results of velocity profiles, concentration distribution and bed deformation. This reference test has been selected as a validation test by various authors using 2D and 3D numerical models, see e.g., [1, 8, 14].

This paper is organized as follows. In Section 2 the 2D and 3D model equations are presented for flow and suspended sediment transport. In Section 3, the computational framework for the solution of the 2D and 3D models is briefly introduced and some important issues arising from the implementation of the 3D sediment transport model are discussed. In Section 4, the 3D model is first verified with the analytical solution of the uniform flow and sediment transport in a steady, uniform channel and the results are compared with the classical and corrected 2D model. Then the model is validated with the experimental setup of van Rijn [12]. Finally, the conclusions and outlook of the work are presented in Section 5.

II. MATHEMATICAL FORMULATION

In this section, we briefly describe the 2D and 3D models for flow and sediment transport.

A. Three-dimensional hydrodynamic model

In this work, the 3D flow field is determined by solving the continuity and Reynolds-averaged Navier-Stokes equations in the Cartesian coordinate system:

\[
\begin{align*}
\frac{\partial u_i}{\partial x_i} &= 0 \\
\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} &= F_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j}
\end{align*}
\]  

where the summation convention for repeated indices is used. Above, let \( x = (x_1, x_2, x_3) = (x, y, z) \) denote the spatial coordinates; \( t \geq 0 \) the time; \( u_i = (u, u, u_j) = (u, v, w) \) the local time-averaged components of the flow velocity, \( F_i \) the components of external forces, such as gravity, Coriolis force, etc.; \( p \) the mean pressure; \( \rho \) the fluid density and \( \tau_{ij} \) the components of the stress tensor calculated with the Boussinesq hypothesis and related to the gradients of the velocity and the turbulence eddy viscosity \( \nu_i \). In this work, two turbulence closure models are used to compute \( \nu_i \), namely the mixing length and the standard \( k-\varepsilon \) turbulence models, see e.g., [9, 10].

The continuity and Reynolds-averaged Navier-Stokes equations (1) are completed with initial and boundary conditions. At the inlet boundary, the uniform or the classical rough wall logarithmic profiles can be applied:

\[
u_i(z) = \frac{u_*}{\kappa} \log \frac{z}{z_0}
\]  

where \( z \) is the vertical distance from the theoretical bed level, \( u_* \) is the shear velocity, related to the bed shear stress by \( u_* = \sqrt{\tau_{ij}/\rho} \), \( \kappa \) is the von Karman constant (\( \approx 0.40 \)), \( z_0 \) is \( k_i / 30 \) a length scale related to the bottom roughness and \( k_i \) the Nikuradse's equivalent bed roughness. A sketch of the theoretical distribution of \( u(z) \) is given in Figure 1.

At the outlet boundary, the normal derivatives of the flow variables are set to zero. At the sidewalls, the velocity tangential and normal to the boundary are set to zero (no-slip condition). Finally, the position of the water surface is determined from the solution of the depth-averaged shallow-water continuity equation, see [6]. The input boundary conditions for the \( k-\varepsilon \) model are taken from [3]:

\[
k_{*i}(z) = \frac{u_*^2}{\kappa} \left( 1 - \frac{z}{h} \right)
\]  

\[
\epsilon_{*i}(z) = \frac{u_*^2 (1 - z/h)}{\kappa z_s}
\]  

with \( h \) the water depth and \( \epsilon_{*i} = 0.09 \) a coefficient. The eddy viscosity \( \nu_i \) is related to \( k \) and \( \varepsilon \) by

\[
\nu_i = c_k \frac{k_i^2}{\varepsilon}
\]

B. Two-dimensional hydrodynamic model

To obtain the 2D, depth-averaged, horizontal shallow water equations, the continuity and Reynolds-averaged Navier-Stokes equations (1) are integrated over the depth \( h \) using the Leibniz’ integral rule, replacing the mean pressure \( p \) by the hydrostatic pressure and adopting a movable free surface level \( k_* = h + z_{so} \) with \( z_{so} \) a smooth function representing the bottom level, eventually resulting in:

\[
\begin{align*}
\frac{\partial h}{\partial t} + \frac{\partial h U_j}{\partial x_j} &= 0 \\
\frac{\partial U_i}{\partial t} + \frac{\partial h U_i U_j}{\partial x_j} &= -gh \frac{\partial z_{so}}{\partial x_i} + \frac{1}{\rho} \frac{\partial h T_{ij}}{\partial x_j} - \frac{1}{\rho} \tau_{ij}
\end{align*}
\]  

where \( U_i = (U_1, U_2) \) are the components of the depth-averaged flow velocity in two space dimensions; \( g \) denotes the gravitational acceleration; the bottom shear stress \( \tau_{ij} \) can be modelled by the well-known quadratic friction law \( \tau_{ij} = C_f \rho |U_i| U_j \), with \( |U| = \sqrt{U_i U_j} \) the Euclidean norm of the horizontal velocity vector and \( C_f \) the friction coefficient determined as function of equivalent bed roughness \( k_* \). In this work, the stresses at the free surface are not considered.

The components of the tensor \( T_{ij} (i, j \neq 1, 2 = x, y) \), accounting for the depth-averaged normal and horizontal shear stresses, are related to the gradients of the depth-averaged velocities assuming a constant horizontal diffusion coefficient. Details of the full derivation of the shallow water equations, as well as different boundary conditions for different flow regimes can be found in [13].

C. Three-dimensional suspended sediment transport model

The suspended sediment load is calculated by solving the full 3D advection-diffusion equation for the suspended sediment concentration distribution, expressed as

\[
\begin{align*}
\frac{\partial c}{\partial t} + \frac{\partial u_i c}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \frac{\partial c}{\sigma_j \partial x_j} \right) &= \frac{\partial}{\partial x_j} \left( \nu_j \frac{\partial c}{\partial x_j} \right)
\end{align*}
\]
with \( c = c(x,t) \) the suspended sediment concentration, \( w_i > 0 \) the vertical-settling sediment velocity and \( \sigma_u \) the turbulent Schmidt number, assumed here to be one. The advection-diffusion equation (6) is completed with initial conditions specifying \( c(t = 0) \) and boundary conditions as follows.

At the inlet boundary, a local equilibrium concentration profile can be specified. This profile can be derived from equation (6) by assuming uniform and steady flow conditions. If a parabolic distribution of turbulent eddy diffusivity coefficient is adopted, then the vertical distribution of suspended sediment concentration is the classical Rouse profile:

\[
\frac{c}{c_{ref}} = \left( \frac{h/z - 1}{h/z_{ref} - 1} \right)^{w_i/\kappa u_t},
\]

where \( c_{ref} \) and \( z_{ref} \) are the equilibrium near-bed concentration and reference level, respectively. In this work, numerical results are performed by using either the formula of van Rijn [12] or the Zyserman and Fredsoe [17] for the reference concentration and associated reference level, where \( c_{ref} \) is a function of the local skin friction. The Rouse number \( w_i/\kappa u_t \) evidences the effect of gravity against the turbulent diffusion. Equation (7) clearly shows that the concentration is equal to zero at the free surface and can be infinitely large as the distance \( z \) tends to zero as the turbulence vanishes. This particular issue is discussed in more detail in Section 3.

At the outlet, the normal gradients of the concentration are set equal to zero. A similar boundary condition can be specified at the sidewalls of the model. At the free surface, the net vertical sediment flux is set to zero, thus:

\[
\left( v_i \frac{\partial c}{\partial z} + w_i c \right)_{z = z_{ref}} = 0
\]

At the bottom, a Neumann type boundary condition is specified, in which the total vertical flux equals the net sediment transport:

\[
\left( v_i \frac{\partial c}{\partial z} + w_i c \right)_{z = z_{ref}} = D - E
\]

The deposition \( D \) and entrainment \( E \) fluxes can be expressed as \( E = w_i c_{ref} \) and \( E = w_i c_{ref} \) respectively [5]. To compute the deposition rate \( D \), the concentration at the bed level \( z = z_0 \) must be determined, as described in Section 3.

D. Two-dimensional suspended sediment transport model

Integrating Eq. (6) over the depth, the 2D depth-averaged horizontal suspended sediment transport model is obtained:

\[
\frac{\partial C}{\partial t} + \frac{\partial U_{conv,j} C}{\partial x_j} = \frac{\partial}{\partial x_j} \left( v_i \frac{\partial C}{\partial x_j} \right) + \frac{E - D}{h}
\]

where \( C \) is the depth-averaged suspended-load concentration and \( U_{conv,j} \) are the corrected convective velocity-cities accounting for the effects of the heterogeneous vertical distribution of suspended sediment, as defined in [7]. In (8) it is assumed that \( z_{ref} \ll h \).

E. Bed evolution

By considering only suspended-load sediment transport, the bed evolution is function of the net sediment flux near the bed given by:

\[
(1 - p') \frac{\partial z_p}{\partial t} = D - E
\]

where \( z_p \) is the position of the bed above datum and \( p' = 0.40 \) is the bed porosity. Once the flow variables are determined by solving the hydrodynamics and suspended sediment transport equations (for 3D or 2D models), changes of bed level are computed from (9) for the cell coincident to the bed, calculating at each time step the net sediment flux \( D - E \). Further details on the derivation of the mass balance equation (9) and coupling strategies can be found in [15].

III. NUMERICAL IMPLEMENTATIONS

In this section, some important issues arising from the implementation of the 3D sediment transport model are addressed. For the 2D sediment computations, the reader is referred to the Sisyphe documentation [11].

A. Telemac finite element system

The computational framework for the implementation of the different models is the Telemac Modelling System [11]. This is an open-source, sequential and parallel free-surface solver based on the finite volume and finite element methods. Details of the numerical formulations for the models are given in [6] and not repeated here.

For the 3D model, the domain composed of prismatic elements is built from an unstructured triangulation of the 2D domain, then repeated along the vertical in superimposed layers from the bottom to the free surface. The largest concentration gradients in the model are expected to occur close to the bottom surface. This is where the highest vertical resolution is required and the performance of the numerical model is significantly diminished. Therefore, simulations are performed here with a number of layers vertically distributed with a geometric progression.

Figure 1. Schematic view of the model variables.
B. Bottom boundary layer model

The treatment of the near bed boundary layer in the 3D model is particularly important for sediment transport, which strongly depends on an accurate determination of the bed shear stress and the friction velocity.

According to classical boundary layer concepts, the turbulent eddy viscosity coefficient is proportional to the theoretical (mean) bed level, located at some intermediate point. The vertical balance between the gravity term due to the theoretical (mean) bed level, located at some intermediate point, and the settling and turbulent diffusion no longer holds at the bed level. The vertical balance between the gravity term due to the theoretical (mean) bed level, located at some intermediate point, and the vertical balance between the gravity term due to the theoretical (mean) bed level, located at some intermediate point, and the vertical balance between the gravity term due to the theoretical (mean) bed level, located at some intermediate point.

In the hydrodynamic model, the mesh is shifted in the vertical direction by a fixed value $z = z_0$ (see Figure 1) relative to the theoretical bed position. The bed origin in the model therefore corresponds to the origin of the velocity profile. This numerical artefact avoids the problem of infinite concentration at the bed. The vertical grid is then refined such that the first grid elevation scales with the bed roughness (of the order of few millimetres) and the consequent layers follow a geometric progression in the vertical direction, increasing the vertical resolution from the bottom to the top. A detail of the mesh with a geometric progression in the vertical direction is shown in Figure 3.

C. Sediment boundary conditions

The erosion flux $E$ is calculated as an explicit function of the reference concentration at $z = z_{ref}$. The deposition flux $D$ is calculated as an implicit function of the concentration at the (model) bed level ($z = z_0$). In most practical applications, the reference level $z_{ref}$ is generally greater than $z_0$. The value of the concentration at $z = z_0$ is then obtained by extrapolating the reference concentration from the reference level $z = z_{ref}$ to the model bed level $z = z_0$ (Figure 1), assuming an exponential concentration profile which is consistent with the linear diffusion coefficient:

$$ c(z = z_0) \approx c_{ref} \left( \frac{z_{ref}}{z_0} \right)^{w_u/\kappa u} \quad (10) $$

IV. NUMERICAL EXAMPLES

In this section, the 3D model results are first compared with the well-known logarithmic velocity and sediment concentration Rouse profiles and validated with the trench migration experiment of van Rijn [12].

A. Fully developed uniform steady flow

Both 2D and 3D models are first applied to the simple test case of a fully developed uniform steady flow over a flat bed. Model results can be validated against the classical analytical solution of the Rouse concentration profile and logarithmic velocity profile (which are strictly valid up to the free surface assuming a parabolic distribution for the turbulent eddy viscosity). The main objective of this test is to validate the 3D model resolution, such as number of planes, etc., as well as to test the consistency between the 2D and 3D approach: both models are expected to give the same integrated results in this simple configuration as stated in the introduction, regarding the value of friction velocity, suspended load transport rate and mean depth-averaged concentration. For this test, a flow rate $Q = hu = 0.22 \text{ m}^3 \text{s}^{-1}$ and a water depth $h = 0.40 \text{ m}$ are specified at the channel inlet and outlet, respectively. The simulations are performed with the reference concentration formula of Zyserman and Fredsoe [20], with $d_{50} = 0.16 \text{ mm}$, $k_0 = 2.5 \text{ cm}$ and $w_s = 1.50 \text{ cm} \text{s}^{-1}$.

In both 2D and 3D models, the reference concentration is calculated as a function of skin friction and applied at the reference level ($z_{ref} = 3d_{50}$). Results of shear velocity $u_s$, depth-averaged concentration $C$ and solid discharge $Q_s$ at the channel outlet are summarized in Table 1. For the 3D model, the values of the depth-averaged concentration are obtained by integrating the concentration $c(z)$ over the water depth.

The effect of the vertical mesh resolution for 6, 12 and 18 layers is presented in Figure 2 for the velocity and concentration profiles. As expected, increasing the number of layers shows convergence to steady profiles. It can also be shown that the gradients of velocity and concentration can be well captured with a reasonable number of layers in the vertical direction.

<table>
<thead>
<tr>
<th>Model</th>
<th>Type</th>
<th>$u_s$ ( (\text{cm.s}^{-1}) )</th>
<th>$c$ or $C$ ( (\text{g.l}^{-1}) )</th>
<th>$Q_s$ ( (\text{m}^2 \text{s}^{-1}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>30 layers</td>
<td>3.87</td>
<td>0.268</td>
<td>1.16 x10^{-5}</td>
</tr>
<tr>
<td>3D</td>
<td>“classical”</td>
<td>3.87</td>
<td>0.231</td>
<td>1.74 x10^{-5}</td>
</tr>
<tr>
<td>3D</td>
<td>“corrected”</td>
<td>3.87</td>
<td>0.231</td>
<td>1.31 x10^{-5}</td>
</tr>
</tbody>
</table>

B. Flow in a migrating trench

The laboratory experiment, conducted by van Rijn [12], was performed in a straight channel at Delft Hydraulics, and the geometry of the experimental facility was as follows: 30 m long and 0.50 m wide with vertical side walls. The channel was filled with a 0.20 m thick layer of sand with median grain size $d_{50} = 160 \text{ m}$. The average velocity was 0.51 m/s and the water depth was approximately equal to 0.39 m at the channel inlet. The experiment considered in this work involved a trench with side slope 1:3.

To mimic the laboratory conditions with the model, a constant water depth of 0.3775 m above the bottom of the flume was imposed at the downstream outlet, and a constant discharge of 0.09945 m$^3$s$^{-1}$, was specified at the upstream inlet. Flow and suspended sediment transport were computed with a fixed bed until steady flow conditions were reached in order to initialize the sediment and flow velocity, and with a movable bed afterwards in which the trench propagates in the direction of the flow. For this configuration, the flow decelerates by keeping the cross-trench water flux constant, resulting in the deposit of the upstream portion, where the
bottom profile tends to be linear and the erosion of the downstream portion, where the disturbance is travelling downstream.

The modelled channel was represented using a mesh of 68,210 elements. For this model, 20 vertical layers were used, a thickness of the top layer of about 0.05 m and the thickness of the bottom layer equals of about 0.005 m.

The steady-state measured and computed velocity and sediment concentration profiles at five sections corresponding to 7 m, 9 m, 10.5 m, 12.75 m and 14 m from the channel inlet at the centreline of the flume are shown in Figures 3 and 4. In all figures, the circles correspond to experimental data.

Figures 3 and 4 show the velocity and suspended sediment concentration profiles at steady-state, obtained with the near-bed reference concentration approaches of van Rijn and Zyserman and Fredsoe using the mixing-length turbulence model. As showed, the velocity profiles are well captured by both approaches while van Rijn approach captures better the concentration profiles than the Zyserman and Fredsoe formula.

Results of the migration of the trench after 15 hours are showed in Figure 5. The computations are performed with the reference concentration of van Rijn and the mixing-length turbulence model. For the adopted configuration the results obtained with both models match the measurements of velocity, although some discrepancies are observed for the suspended sediment profiles. A detail of the bed level profile at $t = 0$ and $t = 15$ hours is showed in Figure 5. For the “classical” and “corrected” 2D model, the migrated distance of the trench is captured well but the trench filling is much too large in comparison with observations. Nevertheless, some little improvement is observed when the velocity correction terms are included [7]. The discrepancies between 2D simulations and observed data could be associated to the fact that the shallow water hypothesis are no longer valid for this channel configuration. The 3D model captures very well the migrated distance of the trench and the bed level profile is in good agreement with the observed data.
V. CONCLUSIONS

In this paper, a 3D model for the flow, suspended sediment transport and bed evolution has been presented and compared with 2D simulations and experimental data. The model, implemented on the basis of the open-source, finite element/finite volume Telemac Modelling System, showed good agreement when compared against analytical and laboratory measurements of velocity and suspended sediment profiles. Furthermore, the simulations yielded successful predictions of the bed evolution.

The 2D simulations were performed with the “classical” shallow water equations and a “corrected” formulation that takes into account the fact that the largest part of the sediment is transported near the bed [7]. As expected, the classical 2D approach fails to reproduce accurately the observed data. The corrected approach introduces some improvements in the solution but the trench filling is still much too large in comparison with observations. The 3D model captures very well the migrated distance of the trench and the bed level profile is in good agreement with the observed data after 15 hours. The results are also in agreement with simulations of Lesser et al. [8] and Warner et al. [14].

Future work will examine on one hand the ability of the model to reproduce complex 3D flow structures, such as helicoidal flows in curved channels, on the other hand the coupling of the suspended transport with bed-load transport and the modelling of variable-size sediment.

REFERENCES

Large scale morphodynamic modeling of the Gironde estuary

Catherine VILLARET1,3, Nicolas HUYBRECHTS2,3, Lan Anh VAN2

1Laboratoire National d’Hydraulique et Environnement
Electricité de France R&D
6 Quai Watier, 78400 Chatou, France
catherine.villaret@edf.fr

2Saint Venant Laboratory for Hydraulics (Université Paris Est, joint research unit EDF R&D – CETMEF – Ecole des Ponts ParisTech), 6 quai Watier, 78400 Chatou, France

3Roberval Laboratory, LHN (joint research unit UTC-CETMEF), UMR CNRS 6253, Centre de recherches de Royallieu, B.P. 20529, 60206 Compiègne Cedex, France

Abstract— Previous attempts to model sediment transport and morphodynamics in complex estuarine conditions have been limited by the use of simplifying methods in order to reduce computational cost. Thanks to tremendous progress in numerical methods and extensive use of parallel processors, the open source finite element Telemac system (release v6p1) is applied to represent the medium term bed evolution in the largest estuary in France, the Gironde macro-tidal estuary. After calibration, the 2D hydrodynamic model (Telemac-2d) is validated by comparison with some recent data set, combining tidal flow and velocity measurements at different locations along the estuary. For morphodynamic modeling, the effect of sand grading is incorporated into Sisyphe, in order to represent schematically the high variability in the sediment bed composition. The effect of cohesive sediments is also examined using a recently developed model of consolidation.

I. INTRODUCTION

The objective of this work is to develop a realistic morphodynamic model which can be applied to predict accurately the sediment dynamics and medium term bed evolution in the central part of the estuary, where drastic bed evolutions have been reported as a result of sand banks formation and secondary mid-channel deposit. Bed evolutions can be either due to human activities or to natural origins, and may also be attributed to dredging activities. This model can be used as an operational tool by end-users and engineers to test solutions to prevent unwanted erosion or deposition in strategic areas.

Our framework is the finite element Telemac system (release 6.1), where the 2D approach is selected as a good compromise between model accuracy and computational cost. A local morphodynamic model was previously developed for the central part of the estuary [13]. An embedded model strategy was chosen to impose the hydrodynamic boundary conditions under schematic forcing conditions, for a single neap-spring tidal cycle and constant mean flow rate. This method allows saving computational time, but induces additional uncertainties related to the treatment of boundary conditions.

Thanks to tremendous progress in the numerical method and use of parallel processors, the computational domain is here extended and represents the whole 150 km long estuary, including the Dordogne and Garonne main tributaries, while the maritime boundary conditions are now imposed at a distance of approximately 30 km from the coast line. We also include a more realistic representation of the hydrodynamic forcing (including seasonal variations in the river flow rates) and sediment transport processes (including sand grading effects, bed roughness prediction, consolidation algorithm).

We start in Part 2 with a general description of the Gironde estuary, its hydrodynamics and sediment characteristics. In Part 3, we present the coupled 2D hydrodynamic and morphodynamic models: the method of Van Rijn [11] implemented in release 6.1 is applied in order to predict the bed roughness. The advantage of this method which can be applied as an alternative to a calibration procedure for the bed friction coefficient, is to reduce the possible inconsistency between morphodynamics and hydrodynamics [15]. In Part 4, the model is applied to reproduce the effect of grain size distribution on bed evolutions and variability in the flow rate. Finally in Part 5, the effect of cohesive sediments is examined by using a recently implemented multilayer algorithm for consolidation [12]. We present here a preliminary model comparison with 5-years of measured bed evolution (1995–2000) and also with some recent data sets including velocity and turbidity measurements at different points along the estuary (September, 2009).

II. DESCRIPTION OF THE GIROND Estuary

A. Study area

The Gironde estuary, located southwest of France, extends from the confluence of the Garonne and Dordogne rivers to the mouth on the Atlantic coastline (Figs 1&2). Its width ranges from 3.2 to 11.3 km downstream. The Gironde
can be subdivided into three parts: the upper river part, the central part and the downstream maritime part. The central part is characterized by a complex geomorphology, with different channels separated by elongated sand banks. The estuary can be classified as macro-tidal, hyper-synchronous and with an asymmetric tide (4 h for the flood versus 8 h 25 for the ebb). The ocean coast line induces a strong forcing with tidal amplitude at the mouth of the estuary ranging from 2.2 m to 5.4 m during a fortnightly spring-neap cycle. The cumulated river discharge from both rivers (Dordogne and Garonne) ranges from 50 to 2000 m$^3$/s. During flood events, the river flow rate becomes occasionally greater than 5000 m$^3$/s. The centurial average value of the fluvial flow rate reaches 1000 m$^3$/s, 65% of it coming from the Garonne River.

B. Hydrodynamic data

The tide propagation can be analyzed through water levels, which are measured every 5 min at nine hydrometric stations along the estuary from the Verdon station at the mouth to the harbour of Bordeaux, located 10 km upstream of the confluence between the Dordogne and Garonne rivers (Fig. 2). Measurements of flow rates are available every hour at the upstream boundary.

Velocity measurements are sparser in comparison to the water level data. For instance, ADCP velocity profiles were measured by EDF R&D in August 2006 at 3 points located along the same cross section, approximately 5 km downstream Pauillac station (Fig. 2) and at 5 points along the estuary from September to October 2009 (7 points were measured, as shown on Fig. 1, but only 5 of them were successful). Both events are used to calibrate and validate the hydrodynamic model [5].

C. Bathymetry surveys

The bed evolutions are measured through bathymetry surveys made every 5 years, since it takes about 4 years to cover the whole estuary from Bordeaux to Verdon station. As mentioned in the introduction, a better accuracy is expected in the central part of the estuary and model validation will be focused on this part. A rather coarse grid is applied in both maritime and fluvial parts, where the bathymetry will not be updated. In the central part of the estuary, morphodynamic features evolved drastically from 1994 to 2005 and more detailed bathymetric data sets are available for years 2000 and 2005. The 1995 bathymetry is used as an initial condition of the morphodynamic model whereas the 2000 data is used to compare the measured bed evolutions with model predictions. The 2005 bathymetry is prescribed.

D. Characteristics of bed material

The bed composition is highly variable in space: gravel and sand can be found at the mouth of the estuary whereas, in the tributaries, the bed channel is dominated by the presence of mud. Information concerning the bed material is generally provided qualitatively: areas of sand, mud or gravel are reported on maps (see Fig.3). At the mouth of the estuary, the median diameter of the bed material ranges within 0.25 and 0.38 mm (Port Autonome de Bordeaux, 2002).

More quantitative information on the bed composition is available in the central part of the estuary. Two measurement campaigns were performed in 2006 and in 2009 by EDF R&D. In 2006, bed samples were collected downstream of the Patiras island (see Fig. 3). Analysis of these samples reveals that 55% of the bed material is cohesive (finer than 0.063 mm) and 45% non-cohesive, with median diameter $d_{50} = 0.21$ mm. The second campaign provides more detailed quantitative information on the spatial variation of the bed composition. According to Boucher (2009), three types of sediment bed composition can be identified with mud only, sand only (63µm < $d_{50}$ < 2mm) and sand mud mixtures, as shown on Fig. 2. Sand is dominant in the deeper channels, whereas the tidal banks are dominated by the presence of mud.

D. Turbidity measurements

The suspended load and related water quality parameters have been measured at various stations along the estuary (www.magest.u-bordeaux1.fr) since 2005. This yearly monitoring gives some qualitative information on the turbidity variation along the estuary as a response of seasonal variation in the river flow rates.

During the September-October 2009 survey, the attenuation of the ADCP velocity signal is interpreted in terms of turbidity level and converted in g/l using the linear relation proposed by [2] Measurements are summarized in Table 1. The turbidity level is very high upstream in the Dordogne tributary with maximum values up to 8 g/l (Point 7), and progressively reduces to less than 1.05 g/l in the central part and to less than 0.05 g/l at the mouth of the estuary (Point 1). Those observations qualitatively match the observations reported in [2]: for instance, the turbidity level at Pauillac station fluctuates with maximum values of the order of 3 g/l (July 2005).

To the authors’ knowledge, no information is available on the bed load transport rates, although the presence of mega ripples and dune in this zone is an indicator of active sand transport.

Figure 1. Location map (indicating ADCP velocity measurement).
well adapted to cover large scale domain and allows to refine zones of particular interests (e.g. the central part of the estuary), while the upstream maritime and downstream river part is more coarsely represented (300 m between nodes in the streamwise direction).

The numerical domain covers the whole estuary: from the Bay of Biscay (mouth near Verdon, Fig. 2) to La Reole and Pessac, considered as the limit of the tide influence in the tributaries. The unstructured triangular mesh comprises 22650 nodes [5]. The cell lengths extend from 50 m in the refined central part and up to 2 km in the maritime boundary. The current release 6.1 of Telemac-2d is used to solve the shallow water equations. This version benefits from optimized finite element schemes and a full parallelization of the code. Moreover, a recently developed algorithm for tidal flats allows to ensure both mass conservation and positive water depth [4]. The numerical domain is extended into the coastal zone (30–40 km from Verdon station) in order to impose the tide height in deep water. The tidal components are issued from a global oceanic model [7].

B. Sediment transport model

The morphodynamic model (Sisyphe release 6.1) solves the Exner equation and splits the total load into bed- and suspended-load. The bed load is estimated by a semi-empirical formula (e.g. Meyer-Peter Muller [8]) whereas the suspension load is calculated by solving an additional transport equation for the depth-averaged sediment concentration. The erosion and deposition fluxes, which enter both the Exner equation and the transport equation, are expressed as a function of an equilibrium concentration, which is also calculated using a semi-empirical formula [16].

In order to solve the advection term of the suspended-load transport equation, a new algorithm has been implemented, to ensure a fully mass conservative scheme. This new scheme is based on finite volumes methods and allows calculating fluxes of sediment along segments forming the individual triangular elements. The treatment of tidal flats is based on positive water depth algorithm and is fully mass conservative [4].

The depth-averaged velocity field calculated by Telemac-2D needs to be corrected to account for the fact that most suspended sediment is transported by the near-bed velocity field. This correction leads to a reduction of transport rates, as detailed in [6]). The correction of the velocity, which is a multiplication by a space-dependent factor in the range [0;1], must be applied to the fluxes themselves, at the edge level, in order to avoid unphysical results. The average correction of the two points forming the edge is chosen so far. This method is available in the current release 6.1 which is used in this application.

The corrected 2D-velocity field does not obey the shallow water continuity equation and this requires a specific treatment in finite volumes advection schemes. Mass conservation is still ensured but monotonicity is spoiled, and this could threaten the numerical stability, especially in dry zones. Stability is eventually obtained by adding the settling velocity term in an implicit way in the advection. By this

TABLE I. TURBIDITY MEASUREMENTS AT DIFFERENT STATIONS ALONG THE ESTUARY. A CONVERSION FACTOR OF 0.0023 IS APPLIED TO CONVERT THE TURBIDITY IN CONCENTRATIONS [2].

<table>
<thead>
<tr>
<th>Measurement point</th>
<th>C(g/l)</th>
<th>Point 1 (depth 32m)</th>
<th>Point 4 (depth 6m)</th>
<th>Point 7 (depth 7m)</th>
<th>Maged July 2005 Paulliac</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.023–0.046</td>
<td>0.12–1.03</td>
<td>6.9–8.05</td>
<td>0.5–3.25</td>
<td></td>
</tr>
</tbody>
</table>

III. MODEL SETUP

A. Hydrodynamic model

Numerical computations are performed with the open source TELEMAC finite element system (see the site www.telemacsystem.com) developed at EDF R&D [3]. The use of unstructured meshes and finite elements methods is
way, even places where the water depth is close to zero (dry zone) will come up with a finite value of concentration.

The effect of sand grading is based on Hirano’s active layer concept, whose thickness is set to 10 cm, which is of the order of magnitude of bed roughness as predicted in the central part of the estuary [14]. In Part III, the grain size distribution in the model remains here in the non-cohesive range ($d_{50} > 60 \ \mu m$). In Part V, we present some preliminary results on the effect of cohesive sediment which is based on a multilayer consolidation model [12].

IV. LARGE SCALE HYDRODYNAMIC MODEL

A. Boundary conditions

Flow rates are imposed at the upstream boundary and the tide height at the maritime downstream boundary for the hydrodynamics. The tide height is composed of 46 harmonic waves [5]. Special attention must be paid to the boundary conditions for the suspended load. When the flow exits from the domain, the concentration of suspended sediment is a degree of freedom and is naturally derived from the knowledge of the concentration inside the domain. When the flow enters the domain, the concentration coming from outside is unknown and it is therefore chosen to apply an equilibrium concentration.

B. Friction coefficients

Friction coefficients are calibrated using water levels and velocities measurements of the 2006 survey. The method is explained in details by [5]. The hydrodynamic friction coefficient is first estimated by using a bed roughness predictor [11] and needs to be further adjusted to account for various sources of uncertainty in the model.

The bed roughness predictor takes into account the effect of both spatial and time variation of the friction coefficient. Model results compare reasonably well with the observations of tidal amplitude and velocity, although the velocity were slightly underestimated by the 2d model in comparison to measurements in the central part of the estuary, as discussed in [5]. For this reason, the bed roughness coefficients, converted into Strickler coefficients, were time-averaged and slightly adjusted to get a set of calibrated Strickler values which were applied in the morphodynamic model application.

The estuary is split into four zones of constant friction coefficients (Fig. 2): 37.5 m$^{1/3}$/s in the mouth, 67.5 m$^{1/3}$/s in the central part, 70 m$^{1/3}$/s for the Garonne River and 60 m$^{1/3}$/s for the Dordogne River.

C. Model validation

Two sets of data have been used for model calibration and validation. Figure 4 shows the comparison between velocity measurements and tidal measurements at the mouth of the estuary (Verdon station) and at the centre (Pauillac) for the 2009 survey (October spring tide).

An accuracy of less than 10 cm in the water level is obtained.

V. LARGE SCALE MORPHODYNAMIC MODEL

A. Multi-grains sand transport model

Sediment transport predictions are highly sensitive to the sediment granulometry and bed composition as well as to the choice of transport formula. In the present application, transport rates are dominated by the presence of very fine particles in suspension. The suspended load is highly sensitive to the choice of settling velocity, which can be deduced from the grain diameter using a semi-empirical formula [10]. The reference length delineating the bed-load and suspended load is taken at 0.5 $k_s$ as suggested by van Rijn, where $k_s$ is the equivalent bed roughness. Influences of the ripples on the skin friction and thus on the transport rates are incorporated for all the performed computations.

The variability of the sediment distribution along the Gironde estuary is schematized by assuming an initial uniform sediment distribution for each geo-morphological unit. In the upper river part, the bed sediment is composed of silt ($d_{50} = 60 \ \mu m$), whereas the maritime part is made of medium sand ($d_{50} = 310 \ \mu m$). In the central part, the bed is made of a mixture of 50% of fine sand ($d_{50} = 210 \ \mu m$) and
50% of silt \( (d_{50} = 60\ \mu m) \). The grain size distribution calculated by the model after one year is shown on Fig.5.b. In the maritime part, the fine sediment is flushed out and deposited offshore, which is in qualitative agreement with observations (see Fig. 5.a). In the central part, very fine (cohesive) sediment is dominant downstream of the Patiras island and deposits area, whereas coarser sediment is predominant in the deeper channels, as observed by [1].

Model results are in qualitative agreement with turbidity measurements from the September 2009 campaign. Time-varying concentrations calculated at point P1 (mouth of the estuary), P4 (central part) and P6 (Dordogne) are shown in Fig. 6. In comparison to the data (see also Table 1), the model tends globally to overestimate the peaks in concentrations by approximately a factor 2 to 5. Best agreement in the central part, both at P4 and at Pauillac station, is obtained by adjusting the settling velocity to 1.8 mm/s for the finer grain size and by lowering the empirical coefficient in the van Rijn formula (0.05 instead of 0.15). These model parameters are retained for the morphodynamic simulation.

**B. Medium term bed evolution**

In the large scale morphodynamic model, the sequence of dry or flood seasons can be imposed at the upstream boundaries based on measured flow rates. Variation of river discharges from January 1\textsuperscript{st} 1995 to December 31\textsuperscript{st} 2000 is shown on Fig. 7. On this figure, the sequence of dry and flood seasons is clearly seen. For instance in the Garonne River, the flow rate decreases down to 60 m\(^3\)/s during the dry season and reaches its maximum, up to 4000 m\(^3\)/s, during winter floods.
The predicted bed evolution (Fig. 8.b) is overall, in both qualitative and quantitative agreement with the 5-year differential bathymetry, shown in Fig 10a. The growth rate of the Patiras island and associated deposition rates of the fine particles downstream of the island are over-estimated by roughly a factor 2, which is consistent with sediment transport rates estimations.

VI. COHESIVE SEDIMENT PROCESSES

Non-cohesive sediments, consisting of sand, are characterised by their diameter and exhibit stable properties in time, while cohesive sediments, consisting of mud, silt and clay, are subject to consolidation and obey different laws of transport, erosion and deposition.

A. Erosion and deposition laws

Cohesive sediments are transported in suspension (no bedload) and the erosion and deposition fluxes are calculated according to Partheniades’ erosion law. The erosion rate $E$ is zero except when the bed shear stress $\tau_0$ exceeds the critical erosion rate $\tau_e$:

$$\frac{\partial \sigma'}{\partial t} + \frac{\partial}{\partial z} \left( \frac{\rho_f - 1}{\rho_s} \frac{d}{dx} k \frac{\partial \sigma'}{\partial x} \right) + \frac{\partial}{\partial z} \left( \frac{1}{\gamma'_f 1 + e \frac{\partial e}{\partial z}} \frac{\partial}{\partial z} \right) = 0$$

where $k$ is the bed permeability, $e$ the void ratio, $\sigma'$ the effective stress, $\rho_f$ and $\rho_s$ the fluid and sediment densities, $g$ the gravity. $K$ and $\sigma'$ are determined form constitutive equations, in order to reproduce observations. The accuracy of the concentration profiles depends on the number of layers. Finally, we set the concentration as $C = \rho_s / (1 + e)$.

We use 10 sediment layers (with fixed concentrations) and time-varying thickness. The model results are in good agreement with measured profiles, as shown on Fig. 9.

C. Model set-up

In order to initialize the bed structure, the model is run for one month of pre-simulation. In zones of deposit (North of Patiras island, tidal flats) the top layer increases ($C_s = 100 \text{ g/l}$) and the bed is covered of soft mud (see Fig. 10.a). The deeper navigation channel (Fig. 10.b), where the currents are stronger, the top layer is eroded and the sediment bed is made of consolidated mud (3rd layer becomes the top layer: $C_s = 200 \text{ g/l}$).

Figure 9. Comparison between measurements (top) and model results (bottom).
The concentration formula, associated with a bed roughness predictor. Despite the fact that the suspended load transport rates of the finer sediment class are overestimated (by roughly a factor 2), results for the medium term bed evolution are in qualitative agreement with observations.

Cohesive sediment transport processes have also been introduced and change quite drastically the model predictions. Qualitatively, the multi-layer consolidation algorithm is able to reproduce the observed spatial variation in the bed structure (soft mud in the deposit area and tidal flats). However, bed evolutions are overestimated and further validation is required.

**D. Preliminary morphodynamic model results**

The cohesive sediment transport model is now applied to simulate the bed evolution in the period 1995−1996. The preliminary results show quite drastic bed evolutions and reproduce the formation of the Patiras bank in the lee of the central island (Fig. 11). This feature is clearly observed in the data (Fig. 8.b). However it is overestimated in the model predictions. Model parameters still need further validation.

**VII. CONCLUSIONS**

The Telemac finite element system, which has been applied to predict the medium term bed evolution (5 years) in the Gironde estuary (150 km long). The model includes a more detailed representation of physical processes: bed friction factor, sand grading, realistic hydrodynamic forcing. For 3 grain sizes (non-cohesive sediment) the CPU time is approximately 10 hrs for 1 year (using 4 processors).

The high variability in the sediment distribution (mixed sediment in the center part of the estuary) has been schematized by assuming non-cohesive sediments with variable grain size (Part IV) and cohesive sediments (pure mud) with variable properties (Part V). This is a rather schematic representation and the effect of mixed sediment still needs to be accounted for.

In Part IV, the grain size distribution is schematized by setting fine sediment in the river tributaries, coarser grains in the maritime part and mixture of fine and very fine sediments in the center part. The morphodynamic model has been validated against observations (turbidity measurements and differential bathymetry from 1995 to 2000). The best agreement is obtained by use of the van Rijn reference concentration formula, associated with a bed roughness predictor. Despite the fact that the suspended load transport rates of the finer sediment class are overestimated (by roughly a factor 2), results for the medium term bed evolution are in qualitative agreement with observations.

**REFERENCES**


Numerical modelling of oil spill in inland waters

Cédric GOEURY
Saint-Venant Laboratory for Hydraulics
Joint research unit EDF R&D, CETMEF, Ecole des Ponts
Chatou, France
Cedric-externe.goeury@edf.fr

François THOUVENEL
VEOLIA, Environment Research and Innovation (VERI)
Rueil Malmaison, France

Jean-Michel HERVOUET1,2
1Saint-Venant Laboratory for Hydraulics
Joint research unit EDF R&D, CETMEF, Ecole des Ponts
2EDF R&D, National Laboratory of Hydraulics and
Environment (LNHE)
Chatou, France

Isabelle BAUDIN-BIZIEN
VEOLIA, Environment Research and Innovation (VERI)
Rueil Malmaison, France

Abstract— The European Water Framework Directive together with the requirement to monitor water resources for drinking as well as leisure and industrial purposes, have substantially increased the demand for water-quality evaluation and monitoring systems. The Migr'Hycar research project was initiated to provide decisional tools, and to fulfil operational needs, for risks connected to oil spill drifts in continental waters (rivers, lakes, estuaries). Within the framework of the Migr'Hycar project, a new numerical oil spill model has been developed by combining Lagrangian and Eulerian methods. This model enables to simulate the main processes that act on the spilled oil: advection, diffusion, evaporation, dissolution, spreading and volatilization. Though generally considered as a minor process, dissolution is important from the point of view of toxicity. The Lagrangian model describes the transport of an oil spill near the free surface.

To model dissolved oil in water, an Eulerian advection-diffusion model is used. The fraction of dissolved oil is represented by a passive Eulerian scalar. This model is able to follow dissolved hydrocarbons in the water column (PAH: Polycyclic Aromatic Hydrocarbons). The Eulerian model is coupled with the Lagrangian model. In parallel with model development, two types of experiments on the behavior of hydrocarbons have been carried out:

- Static chemical laboratory experiments in order to study the kinetic of dissolved petroleum in a beaker.
- Dynamic experiments in artificial river facility. After releasing refined commercial products (fuel and heavy oil) into an artificial channel, the aim of these experiments was to study the drift of the oil spill and the dissolution in the water column.

Static experiments allow a calibration of evaporation, dissolution and volatilization mass transfer coefficients used in the model. Then, the model is validated with the artificial river experiments. Comparisons of numerical results with measured data are presented in this paper.

I. INTRODUCTION

Although in almost half of all instances of contamination the exact cause is never determined, oil spills can be due to human error, accidental or voluntary discharge of cargo residues, domestic or industrial tank overflow, leakage from fuel stations, traffic accidents or fire, amongst other causes.

When faced with hydrocarbon contamination of inland waterways, authorities and other organizations can seldom rely on dedicated decision-making tools to intervene in an effective way.

Whereas considerable management and monitoring resources are rapidly deployed for off- or inshore oil incidents, the more frequent occurrence of continental water pollution is dealt with using relatively modest means. A limited grasp of the nature and magnitude of such events often renders both industry and government powerless in controlling their impact.

The Migr'Hycar research project (www.migrhycar.com) was initiated to provide decisional tools, and fulfil operational needs, for risks connected to oil spill drifts in continental waters (rivers, lakes, estuaries). These tools are meant to be used in the decision-making process after an oil spill pollution and/or as reference tools to study scenarios of potential impacts of pollution on a given site. The Migr’hycar consortium has been organized to closely match project objectives and comprises modelling technology developers (EDF, Saint-Venant Laboratory for Hydraulics, VEOLIA), researchers with long-standing experience of hydrocarbon physicochemical behaviour (Agribusiness laboratory LCA, CEDRE), engineering consultants liaising closely with local and regional authorities (SOGREAH), two water intake operators directly concerned with project-related issues and well experienced in applying protective warning systems (EDF, VEOLIA), and a major player in the oil industry (TOTAL). The consortium has therefore the expertise required to develop a surface-water risk monitoring and prevention system against oil spillage contamination.

In this study, a hybrid two-dimensional trajectory and fate model has been developed to simulate the process of advection, turbulent diffusion, evaporation and dissolution in the water column. The developed model has then been applied to simulate an oil spill in the Gironde estuary and in an artificial river.

In section 2 we briefly describe the conceptual model. In section 3 we deal with the main physical phenomena and show...
how they are modelled. Finally our first results and validation cases are described in section 4.

II. CONCEPTUAL MODEL


The hybrid oil spill model we introduce here combines an Eulerian and a Lagrangian approach. It is new due to the fact that the Lagrangian model describes the transport of an oil spill near the surface. The oil slick is represented by a large set of small hydrocarbon particles. Each particle is a mixture of hydrocarbons. That is why in this model particles are represented by component categories (PAH, pseudo-components characterized by distillation curves), and the fate of each component is tracked separately.

Each particle has an area, a mass, its element number, its barycentric coordinates within this element, physical-chemical properties of each component, amongst other properties, associated to it. This model allows the main processes that act on the spilled oil: advection, effect of wind, diffusion, evaporation, dissolution, to be simulated. Though generally considered as a minor process, dissolution is important from the point of view of toxicity. To model PAH dissolution in water, a Eulerian advection-diffusion model is used. The fraction of each dissolved PAH is represented by a passive Eulerian scalar and its quantity directly depends on the dissolved mass of particle PAH component.

III. PHYSICAL PROCESSES

When an oil spill occurs, the slick moves due to advection and diffusion phenomena. At the same time, the mass of the oil slick changes because of evaporation and dissolution (Figure 1). Therefore, the fate and transport oil spill processes described in the following paragraph need to be included in the oil spill model.

A. Transport processes

1) Advection

a) In 2D: Free surface velocity evaluated

Telemac-2D solves the depth-averaged free surface flow equations. Therefore, in order to calculate the surface slick displacement, we need to evaluate the surface velocity using the depth-averaged velocity. The hypothesis of logarithmic profile for the vertical velocity has been made in order to estimate the surface velocity. In fact, we calculate the averaged velocity by integrating the logarithmic profile. A function of the mean velocity, the surface velocity and the friction velocity is deduced. We specify the friction velocity at the bottom thanks to the friction coefficient. In this way the following relationship for the surface velocity is obtained:

\[ u(h) = \left( \frac{1}{\kappa} \sqrt{\frac{C_f}{2}} \right) u \]

where \( u(h) \) is the surface velocity, \( \langle u \rangle \) is the depth-averaged velocity, \( \kappa \) is the Karman constant, \( C_f \) is the friction coefficient.

b) Wind effect on the oil spill drift.

We consider a float which moves due to wind and current. A solid body submerged in a fluid which moves with constant velocity is subjected to the following force:

\[ F = \frac{1}{2} \rho S C_d \left| u - v \right| \]

where \( u \) is the fluid velocity, \( S \) is the surface of the solid, \( \rho \) is the density, \( C_d \) is the drag coefficient. If the solid moves with velocity \( v \), it is necessary to replace the vector \( u \) by the vector \( v - u \).

The float is subjected to wind and current (Figure 2). At steady state, Newton’s second law allows us to write the following relationship:

\[ \rho S \left[ C_{d,w} \left| v - u \right| (v - u) \right] + \rho S \left[ C_{d,w} \left| v - u \right| (v - u) \right] = 0 \]

where \( v \) is the body velocity, \( \rho_a \) is the air density , \( \rho_w \) is the water density, \( C_{d,w} \) is the drag coefficient in the water, \( C_{d,wind} \) is the drag coefficient in the wind, \( S_a \) is the area of the solid in the wind, \( S_w \) is the surface of the solid in the water, \( u \) is the current velocity, \( u_w \) is the wind velocity.

If the float velocity \( v \) is expressed in a basis formed by \( u \) and \( u_w \) which are known, a solvable system of equations is obtained. This system has the following solution:

\[ v = \left( \frac{1}{\rho} \right) \left[ C_{d,w} \left| u - u_w \right| (u - u_w) \right] + \left| \left( C_{d,w} \left| v - u \right| (v - u) \right) \right| \]
\[ \mathbf{v} = \frac{\mathbf{u}_c + \beta \mathbf{u}_w}{1 + \beta} \]
\[ \beta = \frac{\rho_o S_w c_{d,w}}{\rho_n S_n c_{d,n}} \] (4 & 5)

The drag coefficient for a petroleum slick is a function of the area of the slick, so the drag coefficients \( C_{d,w} \) and \( C_{d,n} \) are equal. We thus obtain:
\[ \beta = \frac{\rho_o}{\rho_n} = 0.036 \] (6)

We can deduce that the oil spill transport induced by wind is 3.6 % of the wind velocity. A similar result is suggested in [10]. Thus, in light wind without breaking wave conditions, oil spill drift induced by wind is 3.5 % of the wind velocity.

2) Diffusion

A drifting substance, for instance petroleum parcels submerged in a current, will diffuse. This diffusion is mostly induced by the turbulent flow. In order to take this phenomenon into account, a stochastic approach is adopted. The hypothesis of “white noise” is made in order to consider the random displacement of a petroleum parcel in water. This hypothesis allows us to define the particle displacement like a Markov process which means that each particle displacement at each time step is independent of its displacements at previous time steps.

Contaminant dispersion is modelled using one governing equation, namely the Advection-Diffusion equation [3]:
\[ \frac{\partial hC}{\partial t} + \nabla \cdot \left( h \mathbf{u} C \right) = \nabla \cdot \left( \frac{h}{\sigma_v} \nabla C \right) \] (7)
where \( h \) is the water depth, \( C \) the depth-averaged pollutant concentration, \( \sigma_v \) is the neutral turbulent Schmidt number, \( \mathbf{u}_c \) is the turbulent viscosity. The turbulent Schmidt number can be set to \( \sigma_v = 0.72 \) [5].

A transformation will be applied to the Advection-Diffusion equation (7) to obtain a Lagrangian equation, according to the following process. The first step in this transformation is to interpret the concentration \( C(X,t) \) as a probability \( P(X,t) \) of finding a particle at a location \( X \) at a time \( t \). Then, using mass conservation, we develop and simplify the previous equation, which leads to:
\[ \frac{dP}{dt} = -\left[ \mathbf{u} \right] - \frac{1}{h} \nabla \left( \frac{h V}{\sigma_v} \right) \nabla P + \frac{V}{\sigma_v} \nabla \cdot \nabla P \] (8)

This equation is called the Fokker Planck equation. The main benefit of having rewritten the equation in form is that it is equivalent to the Ito stochastic differential equation [1]:
\[ \mathbf{X}(t + \delta t) = \mathbf{X}(t) + \left[ \mathbf{u} \right] - \frac{1}{h} \nabla \left( \frac{h V}{\sigma_v} \right) \delta t + \sqrt{\frac{2V}{\sigma_v}} \delta \xi(t) \] (9)
where \( \delta t \) is the time step, \( \xi(t) \) is a vector with independent, standardized random components.

B. Weathering processes

1) Spreading

The spreading process is the most important weathering process. In fact, all mass transfer phenomena which occur during an oil spill are influenced by the area of the surface slick. Oil discharged onto a water surface will immediately start to increase its surface area. This slick expansion is controlled by mechanical forces such as gravity, inertia, interfacial tension and viscosity. Fay (1971) [16] developed a three-phase spreading theory. The three phases are:
- 1st phase: gravity (spreading) and inertia (retardation) dominate;
- 2nd phase: gravity and viscous (retardation) forces dominate;
- 3rd phase: surface tension (spreading) and viscous forces dominate.

Some authors neglect the inertial forces which are important only in the first phase of spreading [7]. The spreading phenomenon is described as follows:
\[ S - \operatorname{arctan}(\lambda S) = 4\pi \mu \] (10)
with:
\[ \mu = \frac{\sigma_{sw} - \sigma_{ow} - \sigma_{aw}}{K} \quad a = \frac{2}{3V} \sqrt{\frac{2\mu}{K} \Delta \rho_{w}} \] (11)
where \( \sigma_{sw} \) is the water-air surface tension, \( \sigma_{ow} \) is the oil-air surface tension, \( \sigma_{aw} \) is the oil-water surface tension, \( K \) is the friction coefficient at the oil water interface, \( V \) is the volume of oil spilled, \( g \) is the gravity, \( \Delta \) is a parameter which relates the oil and water densities: \( \Delta = (\rho_o - \rho_w)/\rho_o \) with \( \rho_o \) and \( \rho_w \) the oil and water density respectively.

Moreover, according to [7], some experiments show that more than 90% of the surface slick is controlled by gravity. This area is surrounded by a thinner oil slick controlled by surface tension. In this paper, the third phase is neglected according to these observations. With this hypothesis and by considering the friction coefficient \( K = (\rho_o \nu \gamma e)/\rho_w \) (where \( e \) is the slick thickness and \( \nu_o \) is the oil kinematic viscosity), the slick surface evolves according to the following equation:
\[ S = \left( \frac{27\pi V^3 \Delta g}{2 \nu_o} \right)^{1/4} \] (12)

2) Mass transfer processes

The mass transfer between two phases is quantified by theories. These theories are based on the hypothesis that the mass transfer resistance is located close to the interface between the two phases. Whitman [15] has suggested one of these theories. In the next sections, all processes are based on the Whitman theory which formulates the mass transfer flux for every mass transfer phenomenon.

a) Evaporation

Evaporation is the most important mass transfer process that oil undergoes after spillage. In a few days, light crude or
refined products can lose up to 75% of their volume. An understanding of evaporation is important both from the practical viewpoint of cleaning up spills and for developing predictive models.

The evaporation model uses a basis on Stiver and Mackay’s work [12]. An expression of the evaporated fraction is determined thanks to the molar flux expression of Mackay and Matsugu [6] and the thermodynamic equation:

\[
dF_{evap} = \exp \left[ \ln \left( \frac{P_a \nu}{RT} \right) + \frac{\Delta H}{RT_a} \left( 1 - \frac{T_a}{T} \right) \right] \frac{k_{evap} S}{V_0} dt \quad (13)
\]

where \( F_{evap} \) is the evaporated fraction, \( P_a \) is the atmospheric pressure, \( \nu \) is the molar volume, \( R \) is the universal gas constant, \( T \) is the temperature, \( K_{evap} \) is the mass transfer coefficient, \( \Delta H \) is the molar enthalpy, \( T_a \) is the boiling temperature, \( S \) is the area of the oil slick and \( dt \) is the time step.

With the previous expression, a pseudo-component approach is adopted. In the latter, crude oils and refined products are modelled as mixtures of discrete non-interacting components. Each pseudo component (PC) is treated as a single substance with an associated boiling temperature. Then distillation data is used to determine the properties of the PC’s. The data are stored as pairs of values: the temperature of the distillate (\( T_{\text{dist}} \)), and the molar fraction of component \( i \) in oil (\( X_i \)). The mass evolution of each particle component \( i \) at every time step is deduced:

\[
mass_{\text{comp}(i)}(t) = mass_{\text{comp}(i)}(t-1) - \rho_c V_a X_i dF_{evap} \quad (14)
\]

where, in any expression \( A_n \), the subscript \( n \) denotes the variable \( A \) of the component \( i \).

b) Dissolution

Dissolution is an important phenomenon from a toxicological and environmental point of view, although it only accounts for a negligible fraction of the oil mass. In fact, the oil quantity concerned by this process is about 1% of the initial mass of oil. Due to their physico-chemical properties, only PAHs need to be considered in the water.

The concentration of dissolved PAH \( i \) in the water column at time \( t \) as a function of the concentration (\( C \)) at the previous time step (\( t-1 \)) is given by the following relation:

\[
C_i(t) = S_i X_i + [C_i(t-1) - S_i X_i] \exp(-\alpha \delta t) \quad (15)
\]

where \( \delta t \) is the time step, \( S_i \) is the component \( i \) solubility in water, \( X_i \) the molar fraction of component \( i \) and \( \alpha = (K_{\text{dist}} A_p) / V \) where \( K_{\text{dist}} \) is the mass transfer coefficient, \( A_p \) is the particle area, \( V \) is the node volume. The order of magnitude of the dissolved mass transfer coefficient \( K_{\text{dist}} \) is of several cm/h [11], [4].

Thus, thanks to the relationship which links mass with concentration, the mass loss at time \( t \) for each component \( i \) can be deduced:

\[
mass_{\text{comp}(i)}(t) = mass_{\text{comp}(i)}(t-1)
\]

\[
= [1 - \exp(-\alpha \delta t)] S_i X_i - C_i(t-1) V \quad (16)
\]

In Telemac-2D each variable is defined on every node of the mesh. If we consider a particle \( P \) inside an element (Figure 3), it is important to define the dissolved mass of the particle at each element node.

The coefficient \( \alpha \) must be defined at each node \( j \). For this, we define the reduced particle area as

\[
A_p \text{SHP}(j) = A_p(j) \quad (13)
\]

where \( \text{SHP}(j) \) is the barycentric coordinate at the node \( j \) and \( A_p(j) \) is the reduced area at node \( j \).

An area is defined around each mesh node, according to a method described in [3]. The volume \( V \) is obtained by multiplying the node area by the depth of the node (Figure 4).

The previous steps allow the coefficient \( \phi_i \) to be calculated at each node \( j \) of each element that contains a particle. The dissolved mass of PAH \( i \) in the water column is defined at each node by the following relation:

\[
mass_{\text{dissolved}}(i)(t) = [1 - \exp(-\alpha \delta t)] S_i X_i - C_i(t-1) V \quad (18)
\]

The total amount of dissolved mass for each particle is:

\[
mass_{\text{dissolved}}(i)(t) = \sum_{j=0}^{n_{\text{tot}}} mass_{\text{dissolved}(j)}(i)(t) \quad (19)
\]

A problem can occur if \( mass_{\text{dissolved}} \) is bigger than the mass of the PAH component. In this case, the dissolved mass needs to be multiplied by a coefficient:

\[
\text{coefficient} = \frac{mass_{\text{comp}(i)}}{\sum_{j=0}^{n_{\text{tot}}} mass_{\text{dissolved}(j)}(i)(t)} \quad (20)
\]

Thus, the quantity of tracer at the time step \( t \), at node \( j \), added by dissolution is defined by
c) Volatilization

Oil components dissolved in the water can be volatilized to the atmosphere if they are not covered by the oil surface slick. The volatilization flux is expressed as follows:

\[ F_i = -K_{vol} C_i \]

where \( F_i \) is the mass flux of component \( i \) (Kg/m\(^2\)·s), \( C_i \) is the concentration of component \( i \) in the water (Kg/m\(^3\)) and \( K_{vol} \) is the overall volatilization rate coefficient (m/s).

IV. EXPERIMENTAL DEVICES.

A. Kinetic experiments

Within the Migr’HyCar project, a static test campaign was conducted by the Agribusiness laboratory LCA located in Toulouse (France).

1) Experimental protocol

During the kinetic experimentations, eight hydrocarbons were tested. The aim of these experiments is to study the hydrocarbons dissolution phenomenon. In a beaker, the tested hydrocarbons are in contact with water during two days. Then, some water samples are taken at different times. An analysis of each sample allows to define the PAHs concentration and the hydrocarbons kinetic is obtained.

2) Results

These hydrocarbon kinetics are used to calibrate the numerical model. In fact, the mass transfer coefficients \((K_{vol}, K_{dif} \text{ and } K_{evap})\) are obtained by fitting the numerical results of the kinetic experiments with the experimental data.

![Figure 5. Heavy fuel kinetic.](image)

Figure 5 shows the result obtained for the heavy fuel with calibrated coefficients such as \((K_{vol} = 1.05\times10^{-6} \text{ m/s}, K_{dif} = 6.81\times10^{-6} \text{ m/s} \text{ and } K_{evap} = 5.27\times10^{-4} \text{ m/s})\).

B. Artificial river experiments

In addition to previous laboratory tests, an artificial river test campaign was conducted by Veolia Environment Research and Innovation (UBA, Berlin). During the tests, four hydrocarbons were tested: heavy fuel oil, home heating oil, kerosene and SP95E10. The main objective of these experiments is to observe the capacity of the pollutant to dissolve PAHs according to four variable parameters, such as flow velocity, the injection position, the oil volume and the presence of obstacles.

1) Experimental device

The UBA (Umweltbundesamt German Federal Agency for the Environment) has on its site 16 identical systems of artificial rivers with each 100 m in circumference. Among these rivers called FSA (acronym for Fluss und StillgewässersimulationsAnlage: simulator rivers and lakes), eight are located outdoors. A water flow is generated in these rivers with a screw pump. A system for continuous measurement of physical parameters river is installed for each river, and there is one weather station.

For the purpose of the project, two rivers were linked together to increase the installation length and sinuosity (Figure 6).

![Figure 6. Artificial river sketch.](image)

2) Experimental protocol

The release of the hydrocarbon is achieved through a ring on water surface, the pollutant is injected inside it (Figure 7). Then, the ring is removed to allow the pollutant transport.

![Figure 7. Release device (left); heavy fuel spilled around obstacle (right).](image)

To observe the evolution of the concentration of dissolved PAHs, a fluorescence probe is used. Every morning a sample called "white" is made to know the initial concentration of PAHs already present in the channel. When the signal (°) is approximately on the peak, a water sample is taken during 30 seconds using an automatic device located with the probe. The samples are then sent to the CEDRE for the analysis of dissolved concentrations of PAHs in samples. For each sample, there is therefore a concentration of total PAHs (ng/L) and a probe signal (%).

3) Results
With the various tests carried out the same day and with the same hydrocarbon, it is possible to draw a calibration curve. This curve will allow to convert the signal probe in % into a total PAH concentration in ng/l (Figure 8). Then, the real concentration as function of time is obtained by subtracting the value of the corresponding blank.

The profiles obtained are used to calculate the mass balance (dissolved mass/injected mass) and validate the numerical model.

V. NUMERICAL RESULTS

A. Spreading process

Some spreading experiment has been presented in [9]. During these experiments, a volume of hydrocarbons was spilled in a rectangular Plexiglass tank. The evolution of the slick surface is followed thanks to a camera which allows to quantify the area increasing of the surface slick. In this paper, the result for the heavy fuel called “#6 fuel” is used. The results is summarized in the following table:

<table>
<thead>
<tr>
<th>Time (min)</th>
<th>Area</th>
<th>M^2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.04</td>
<td>0.0037</td>
</tr>
<tr>
<td>3</td>
<td>0.10</td>
<td>0.0093</td>
</tr>
<tr>
<td>4</td>
<td>0.13</td>
<td>0.0121</td>
</tr>
<tr>
<td>5</td>
<td>0.15</td>
<td>0.0139</td>
</tr>
<tr>
<td>6</td>
<td>0.16</td>
<td>0.0147</td>
</tr>
<tr>
<td>8</td>
<td>0.17</td>
<td>0.0158</td>
</tr>
<tr>
<td>10</td>
<td>0.18</td>
<td>0.0167</td>
</tr>
<tr>
<td>15</td>
<td>0.19</td>
<td>0.0177</td>
</tr>
<tr>
<td>20</td>
<td>0.20</td>
<td>0.0186</td>
</tr>
<tr>
<td>25</td>
<td>0.22</td>
<td>0.0204</td>
</tr>
<tr>
<td>30</td>
<td>0.23</td>
<td>0.0214</td>
</tr>
<tr>
<td>35</td>
<td>0.26</td>
<td>0.0242</td>
</tr>
</tbody>
</table>

A comparison between numerical results and these experimental data can be done, the result is presented in Figure 9.

The numerical result are in good agreement with the experimental data. At the beginning of the spreading process, the model overestimates the spreading of the surface slick. Then, the model superimposes with the experimental results.

B. The artificial river simulation

To evaluate the hybrid oil spill model performance, a simulation is carried out with the artificial river experimental conditions.

1) Mesh characteristics

The mesh to model the artificial river includes only triangular elements. It is composed of about 15000 elements and 7500 nodes. The maximum distance between two nodes is 0.2 m. A detail of the mesh is shown in Figure 10.

2) Simulation results

In this paper, only heavy fuel spill in the artificial river is considered. The following heavy fuel parameters are used:

\[ \rho (at\ 20°C) = 950\ Kg/m^3 \]
\[ \nu_o (at\ 20°C) = 4465\ cSt \]

The turbulent viscosity \( \nu_t \) used in the transport model of the surface slick is calculated thanks to a depth-averaged \( k-\varepsilon \) turbulence model.

Figure 8. Process to obtain results.

Figure 9. Spreading evolution of heavy fuel (“#6 fuel”).

Figure 10. Artificial river curve mesh.

Figure 11. Artificial river simulation.
In Figures 11 and 12, the black dots represent the oil surface slick whereas the scalar tracer represents the dissolution phenomenon in the water column.

Figure 12. Oil spill simulation in the artificial river curve, without obstacles (top) and with obstacle (bottom).

The oil surface slick movement is well modelled, as shown in Figure 12. In fact, on the right part of the figure, the movement of oil particles in the surface slick tail is induced by the turbulence phenomenon.

For what concerns the dissolved petroleum in the water column, the first case is an heavy fuel spill which occurs in the first artificial river straight line close to the screw pump. The flow velocity is imposed to 0.1 m/s and there is no obstacle in the channel. The numerical and experimental concentration peak appears at the same time as show in Figure 13.

Figure 13. Concentration evolution in the water column (case 1).

So, we can deduce that the transport is well modelled. Moreover, the dissolved hydrocarbons concentration in the water column has the same order of magnitude and compares well with experiment. More precisely, the numerical model seems to overestimate the maximum of dissolved concentration and to underestimate the dispersion phenomenon.

Two other cases have been carried out to confirm the previous observations. These two cases concern an oil spill which occurs in the first artificial river curve, with obstacles in the channel.

The river flow is imposed to 0.1 m/s in the second case (Figure 14, left), and 0.2 m/s in the third case (Figure 14, right). The previous observations are confirmed with these cases. However, the numerical concentration peaks for each case arrive earlier in comparison to the experimental concentration peak. This phenomenon can be explained by the outdoor conditions which cannot be modelled, such as gusts of wind.

Figure 14. Concentration evolution in the water column: case 2 (top) and case 3 (bottom).

To conclude, these results show the model capacity to follow the dissolved PAHs evolution in the water column according to the release position, the oil spill volume and the presence of obstacles. Nevertheless, a complementary study is necessary in order to exemplify the capability of the numerical model to deal with different petroleum products.

C. Gironde estuary simulation

Some water intake operators and a nuclear power plant are located on the coast of Gironde estuary. An oil spill can have a strong impact on the management of these industries. So, it is important to be able to model accurately an oil spill which would occur in the estuary.
In the simulation presented on Figure 15, a hypothetical heavy fuel spill is considered to occur with an initial oil volume of 1 m³. After one day, the shape of the slick is shown on Figure 15. Even if the surface slick is not stranded, some heavy fuel reached the estuary coast. This phenomenon can have a strong impact for intake operators. Thus, it is important to follow up the dissolved oil in the water column and the surface slick for operational management of risks.

VI. CONCLUSION

The hybrid model has been developed to simulate oil spill in continental waters. This model predicts the movement of an oil slick, the fate processes and the dissolution concentration of oil in the water column. This oil slick model has been coupled with the Telemac hydrodynamic model. Verifications of the model were carried out by comparing numerical and experimental results. In fact, static experiments have allowed to calibrate the numerical model use for the validation. Then, the calibrated model has been tested by reproducing dynamic artificial river experiments. The obtained results for heavy fuel spill are close to the experimental data. However, even if the numerical results are promising, a complementary study is necessary in order to validate the fate oil spill model for other petroleum products.

Application of the model to a case of oil spill in Gironde estuary exemplifies the capability of the model to deal with different phenomena.

ACKNOWLEDGEMENT

The Migr’Hycar research project is supported by the French Research Agency ANR as a result of the PRECODD 2008 Call for proposals. The PRECODD Eco-technology and Sustainable Development Programme is a research scheme aimed at supporting emerging techniques, procedures and concepts that can help control the environmental impact of industrial and urban activity.

REFERENCES

Simulation for climate change and indicator of vulnerability on four French sandy beaches

Philippe LARROUDE
Laboratoire des Ecoulements Géophysiques et Industriels
UJF-Grenoble Université, Grenoble, France
larroude@hmg.inpg.fr

Abstract— First, we established a procedure for binding three codes to simulate realistic or idealized climates. This procedure is validated in terms of hydrodynamics and morpho-dynamic evolution. These models have been used as part of a cycle of meteorological simulations describing the evolution of monthly events or hydrodynamic factors. Then, the vulnerability can be studied: the vulnerability of the coast will be defined and studied on the basis of in situ observations and model results will come from a set of simulations based on different scenarios (current and 2030). We will evaluate, for all four French sites, the parameter of vulnerability against this set of scenarios.

I. INTRODUCTION

The nearshore region frequently exhibits complicated motions. This complexity is perhaps particularly prominent in the changes that can take place in the morphology of many beaches.

On this nearshore region the climate change induced vulnerability is defined by the Intergovernmental Panel on Climate Change (IPCC) as the combination of sensitivity to climatic variations, probability of adverse climate change, and adaptive capacity. As stated by the IPCC (Watson et al., 1997 [15]), the “coastal systems should be considered vulnerable to changes in climate”. In these areas, amongst the most serious impacts of sea-level rise are erosion and marine inundation.

Within this context, the present paper will give the methodology for the modelling approach to analyse the vulnerability of several beaches on the French coast and, more particularly, on four beaches complementary in terms of hydrodynamic forcing. The coast of metropolitan France is composed of 30% sandy coasts and is potentially vulnerable. All these studies are involved in the VULSACO project.

All these assumptions should, of course, be systematically checked, the purpose of the exercise being to assess, through mid-term bathymetric evolution simulation. Then, vulnerability can be studied: the vulnerability of coast/beach will be defined and studied based on in-situ observations and model results will be taken into account as a modulator of the physical vulnerability.

The understanding of these processes needs, at this time the in situ data but also the development of models, mathematics and numerical codes. Hence, following the work of De Vriend (1987) [3] and De Vriend & Stive (1987) [4], we try to improve the classic quasi-steady procedure. This methodology for morphodynamic evolution is also used more recently in Smit et al. 2008 [12]. The objectives of this work will be therefore to model and to simulate processes of sedimentary transport on sandy beaches with varied weather conditions in the medium term time scale (from a few days to a few months). The coastal morphology evolution cannot be represented with average climatic conditions but needs to simulate such extreme events as storms and therefore, in a long term approach, the morphological evolution is the result of the combination of storm events and calm periods.

II. DESCRIPTION OF THE BEACH

In this national programme we are looking at the climate change influence on four different beaches in France. These beaches are representative of linear sandy beaches of the coastal region. 31% of the French coastline is composed of sandy beaches. They are also representative of forcing and of various important factors.

Figure 1. a) Lido de Sète, b) Truc Vert, c) Noirmoutier (in yellow the barrier beaches and in blue the flooding area) and d) Dunkirk (from Idier et al. 2007, [9]).

The four studied sites (see figure 1) were chosen to have complementarily hydrodynamic contexts (covering some of
the possible hydrodynamic and wave conditions on the French metropolitan coast, Table 1):

1) Sète: The site studied is located in the northern part of the Lido de Sète. The beach is a linear sandy beach with one or two offshore bars. The tourism of this area and the local fisheries are important factors to be taken into consideration. Several measurement surveys have already been carried out on this site (PNEC programme), as well as numerous development studies.

2) Truc Vert: The site is characterized by its high exposure to waves from the Atlantic. There are rhythmic surf zone bars and their related morphodynamic self-organization. The system of bars/baïnes in the intertidal zone has already been studied in Castelle et al. (2006) [1].

3) Noirmoutier: On the coast of the Noirmoutier (Vendée) peninsula, a succession of three barrier beaches demarcating zones liable to flooding in the west can be observed: two in the north supported on reef flats and one (the longest) able to move in the south.

4) Dunkirk: The site is located between Dunkirk and the Belgian border. Dunkirk is on the boundary of a sedimentary layer stretching up eastwards to the Belgian border. Data is available concerning the following fields of study: morphodynamic, hydrodynamic and aerodynamic conditions. The system is complex with the presence of banks in the open sea which attenuate the energy of waves and protect the coast, but which also reduce sedimentary provisions towards beaches (by diminishing the intensity of oscillatory currents onshore). Table 1: Synthesis of the main hydrodynamic environments of the 4 sites studied

### Table 1

<table>
<thead>
<tr>
<th>Number</th>
<th>Zone</th>
<th>Wave exposure</th>
<th>Wave exposure</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Mediterranean</td>
<td>Micro</td>
<td>High</td>
</tr>
<tr>
<td>2</td>
<td>Truc Vert</td>
<td>Micro</td>
<td>High</td>
</tr>
<tr>
<td>3</td>
<td>Noirmoutier</td>
<td>Macro</td>
<td>High</td>
</tr>
<tr>
<td>4</td>
<td>Dunkirk</td>
<td>Macro</td>
<td>Moderate</td>
</tr>
</tbody>
</table>

**III. MODEL AND METHODOLOGY**

Using the computer code 2DH Telemac, we set up a quasi-permanent binding calculations for the wave (wave modeling is done through the Artemis code that solves the equation of Berkhoff with process integration dissipation by wave breaking and bottom friction), for the calculation of the hydrodynamics and for the simulation of the sea bed evolution with a choice of sediment transport formulae (Camenen & Larroudé, 2003 [2]) (Fig. 2). The calculation chain Telemac is a complete model using the finite element method and allows the realization of various sedimentary hydrodynamic calculations.

The equations of the three modules are detailed in Hervouet (2007) [6]. This modeling methodology morphodynamics of sandy beaches is already validated in terms of mesh, time step and convergence in Falquès et al. (2008) [5] and Larroudé (2008) [10].

Figure 2. Technical drawing (Artemis-Telemac-Sisyph: ATS) loop on a time step weather event (between t1 and t2) used in our simulations. For the Noirmoutier and Dunkirk site we used (Tomawac-Telemac-Sisyph: TTS).

Figure 3. Comparison Hs, Ux (cross shore) and Uy (long shore) between the numerical values and in situ measurements on the device VEC 3 (all data in situ: EPOC Univ. Bordeaux).

Figure 3 shows comparisons with in-situ data (here the Truc Vert, see Idier et al. (2011) [9]) for other sites see Maspataud et al. (2010) [11] and ANR-Vulsaco reports. The physical presentation of the sites is described in Vinchon et al., 2008 [14].
IV. RESULTS

We will present two ways of looking at the vulnerability of beaches by analyzing the results of simulations of different scenarios.

The first method is based on the method described in Idier et al. (2006) [7]. We will initially look at the maximum grain size mobilized with a simpler approach. Indeed the calculation of the stress at the bottom will be estimated only from the velocity of the simulations after coupled waves-tides and/or waves only after the site. It also determines the critical Shields using the equation proposed by Soulsby and Whitehouse in Soulsby (1997, p105 [13]). Then there is the maximum size of grain mobilized by inverse method (see results in Tables II to V).

<table>
<thead>
<tr>
<th>TABLE II. RESULTS OF THE MAXIMUM DIAMETER MOBILIZED (M) WITH THE INVERSE METHOD OF SÈTE SITE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Site 1: Sète</td>
</tr>
<tr>
<td>---------------</td>
</tr>
<tr>
<td>Point 1.1</td>
</tr>
<tr>
<td>Point 1.2</td>
</tr>
<tr>
<td>Point 1.3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TABLE III. RESULTS OF THE MAXIMUM DIAMETER MOBILIZED (M) WITH THE INVERSE METHOD OF TRUC VERT SITE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Site 2: Truc Vert</td>
</tr>
<tr>
<td>-------------------</td>
</tr>
<tr>
<td>Point 2.1.1</td>
</tr>
<tr>
<td>Point 2.1.2</td>
</tr>
<tr>
<td>Point 2.1.3</td>
</tr>
<tr>
<td>Point 2.2.1</td>
</tr>
<tr>
<td>Point 2.2.2</td>
</tr>
<tr>
<td>Point 2.2.3</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TABLE IV. RESULTS OF THE MAXIMUM DIAMETER MOBILIZED (M) WITH THE INVERSE METHOD OF NOIRMOUTIER SITE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Site 3: Noirmoutier</td>
</tr>
<tr>
<td>---------------------</td>
</tr>
<tr>
<td>Point 3.1</td>
</tr>
<tr>
<td>Point 3.2</td>
</tr>
</tbody>
</table>

In these tables the columns represent the four scenarios: baseline scenario for each site is the scenario constructed from current data and a simulation case where premium is increased by 120%, an event that we change the height of significant waves at 10% off and if you change the direction of the swells offshore. We make calculations of maximum grain size mobilized within a few points along cross-shore profile at each site (see Figure 4).

This criterion is to be confirmed by the analysis of all the simulations, but it does not seem completely relevant to the vulnerability or not the beach.

It can be completed by the second method in this paper: the analysis of the temporal evolution of cross-shore profiles for each site for the same scenarios presented above. This criterion is not convincing at sites where tidal currents are predominant (Noirmoutier and Dunkirk). For these sites where we are less convinced by the relevance of our results with our morphodynamic simulations TTS. Figure 5 shows that the study on the sea bed evolution of cross-shore profile can be complementary to a vulnerability study by conventional indicators. Indeed we see the influence of the increase of the storm surge, the change in direction for the incident wave or the increase of the significant height of waves on the evolution of multiple profiles on the site of the Truc Vert. This influence could be important and we also analyze that these sea bed evolution are very dependent of the choice we made on the sediment transport formula. These results have to be completed with further study.

V. CONCLUSIONS

This work allowed us to validate and improve our procedures for calculating the evolution of morpho-dynamic calculation chain Telemac. We show in this paper a simplified approach of first calculating the data as an indicator of vulnerability of sandy beaches. This approach shows the limits and must be complemented by a full analysis of simulations of different scenarios and a more classical approach of the evaluation indicators.

ACKNOWLEDGEMENT

This work was partly funded by the National Research Agency (ANR) through the VMC program (project VULSACO, No. VMC06 ANR-009).
Figure 4. Cross-shore profiles of the four sites used for this study with the position of points used for calculations of maximum grain sizes.

Figure 5. Profile cross shore of the Truc vert site for the four scenarios of the Table III.

REFERENCES


Adaptive vertical layering in TELEMAC-3D

Chris J. CAWTHORN, Sébastien E. BOURBAN, Chris T. MEAD, Matthew J. WOOD, Michael .S. TURNBULL

Hydrodynamics and Metocean Group, HR Wallingford
Howbery Park, Wallingford
Oxfordshire, OX10 8BA, UK
Correspondence to: s.bourban@hrwallingford.com

Abstract—Many processes in environmental hydraulics exhibit sharp spatial gradients of some physical variable(s) in a small localised part of the overall water column. Examples of this include spreading of dense or buoyant plumes and thermal or saline stratification in reservoirs. In this paper, we demonstrate a robust adaptive mesh redistribution (AMR) method coded for TELEMAC-3D. The AMR method aims to capture these sharp gradients without requiring an excessive number of mesh layers or any prior knowledge of the flow structure. Rather than increasing the number of mesh planes in regions of sharp spatial gradients, the idea of mesh redistribution is to maintain a fixed number of planes that move in response to the local solution structure. The movement of the planes is governed by a diffusion equation; an approach that is discussed in Ref. [1]. This approach is similar to that used in the popular GETM software (Ref. [2]). Mesh plane elevations linked to gradients in tracer concentration only are discussed in this paper, although the extension to include velocity shear and/or bathymetry in the equations governing plane placement is expected to be straightforward.

We present preliminary results demonstrating that the AMR method can adapt to relatively thin tracer plumes without the increased mesh resolution that would be required with some form of sigma mesh. Comparisons are drawn with an alternative approach in which plane elevations are specified by the user based on some a priori knowledge of the flow structure. The AMR method, which requires neither prior information about the flow nor user input, can be seen to give very similar results for the spreading of dense and buoyant plumes.

1. I N T R O D U C T I O N

Whether in the form of velocity shear layers or rapid changes in saline or thermal stratifications, thin layers with large gradients form an important part of many hydrodynamic processes. Such sharp spatial gradients can pose a troublesome problem for the numerical modelling of fluid flows. In order to accurately capture such layers, it is necessary to have a mesh that is sufficiently well resolved. In the context of TELEMAC-3D, which uses a set of identical two-dimensional meshes (referred to as ‘planes’) stacked vertically (see Ref. [3] for details), one must aim to decrease the spacing between mesh planes in regions of sharp vertical variations.

The spacing between mesh planes in TELEMAC-3D can be reduced in two ways. The simplest method is to use many horizontal planes in the TELEMAC-3D mesh. Even with a standard sigma-mesh (for which the planes are equispaced), this can provide adequate resolution of important thin layers if a large number of planes are used. However, this dramatically increases the cost of the computation. It is also wasteful in the sense that some areas of the mesh will inevitably have fine vertical resolution where capturing the fine detail of the dynamics is unnecessary.

A slightly more sophisticated approach would be to modify the CALCOT subroutine to place the layers at specific positions in the water column, carefully chosen based on the expected solution behaviour. Although this can be an effective technique, it is limited by the fact that the user must have some prior knowledge of the flow structure before modifying the layer positions. This means that such an approach must usually be made in an iterative manner: gradually adjusting the plane positions and re-running the simulation with the goal of converging on some ‘optimal’ configuration.

The aim of this paper is to introduce an automatic mesh layering approach to TELEMAC-3D, which we refer to as adaptive mesh redistribution (AMR). The idea is to devise an algorithm that moves the plane positions based on certain aspects of the local solution structure. For example, one might wish to concentrate layers in regions of sharp velocity gradient, tracer gradient or arbitrary linear combinations of these two variables.

Adaptive mesh refinement itself is not a new idea. In fact, the general formulation presented here was introduced by Winslow in 1966 [1]. Since that time, AMR methods have been substantially refined, but the general principles remain the same. We use a version of Winslow’s “variable diffusion” approach described in more detail in [4], with a few refinements of our own designed specifically to work on the type of layered mesh used by TELEMAC-3D.

An outline of the AMR scheme implemented in TELEMAC-3D, as well as the underlying mathematics, is given in section II. Some simulation results are presented in section III, where comparisons are drawn with a simple sigma mesh. Finally, a discussion of the results and some possible extensions follows in section IV.
II. ADAPTIVE MESH REDISTRIBUTION SCHEME

The type of adaptive mesh refinement that we have introduced to TELEMAC-3D is based on the variational formulation used originally by Winslow [1]. In this section, we give a brief overview of the variational approach, and describe its specific implementation in TELEMAC-3D.

A. Variational principle for mesh redistribution in 1D

Suppose that the computational domain is represented in one dimension by \( n \) nodes \( \{ \xi_i : i = 1, n \} \), with

\[
0 = \xi_1 < \xi_2 < \ldots < \xi_n = 1
\]

and that the real domain (for example, the interval \( a < x < b \)) is then represented at the points \( \{ x_i : i = 1, n \} \). We then define a one-to-one mapping, \( X(\xi) \), such that

\[
X(\xi_i) = x_i, \quad \text{with} \quad X(0) = a \quad \text{and} \quad X(1) = b.
\]

The variational approach is to find the mesh map \( X(\xi) \) that minimises a functional of the form

\[
E[X(\xi)] = \int \omega(X(\xi)) \left| X'(\xi) \right|^2 \, d\xi
\]

(1)

where a dash (') indicates differentiation of a function with respect to its argument. The *monitor function*, defined here by \( \omega(x) \), is a positive definite function that in general depends on the structure of the solution that is to be calculated using the AMR approach. Typically, one might want to focus the mesh resolution in regions where the gradient of the function \( f(x) \) is large. In such a case, an appropriate monitor function would be

\[
\omega(x) = \sqrt{1 + a \left( \frac{df}{dx} \right)^2}
\]

(2)

where \( a \) is a tuning parameter that will be discussed later.

The Euler-Lagrange equation for \( X(\xi) \) associated with minimising the functional (1) is

\[
\frac{d}{d\xi} \left[ \omega(X(\xi)) \frac{dX}{d\xi} (\xi) \right] = 0
\]

(3)

subject to the boundary conditions \( X(0) = a, X(1) = b \).

The adaptive mesh refinement method therefore consists of solving a diffusion equation for the mesh node positions at each timestep. The diffusion coefficient is spatially-varying, and depends upon the current solution. This makes it possible to attract nodes to regions of interest, such as where the solution has large gradients.

B. Implementation in TELEMAC-3D

The ‘variable diffusion’ method described above has been implemented in TELEMAC-3D, and has been released as part of version 6.1. It is accessible by using the keyword *MESH TRANSFORMATION = 5*, and functions as an additional option in the CALCOT subroutine. It works by using an iterative (Gauss-Seidel) approach to solve the diffusion equation (3) on each vertical line of nodes in the three-dimensional mesh.

The method is currently implemented to follow only the gradient of tracer 1, using the monitor function shown in (2), with \( f(x) \) representing the concentration of tracer 1. The extension to consider other physical variables and higher derivatives ought to be, in principle, a straightforward task.

In order to make the adaptive layering scale-free, the tuning parameter in (2) is chosen independently on each vertical line of nodes to be

\[
a = a(x, y) = \frac{C}{\max \left| f'(x) \right|^2}
\]

(4)

where the maximum runs over each of the nodes on the vertical line. This choice means that a very large solution gradient in one part of the mesh will not affect the mesh layering in another region with smaller (yet still significant) gradient. The constant parameter, \( C \), can be tuned to increase or decrease the sensitivity of mesh layer positions to tracer concentration gradient. Large values of \( C \) can produce too large a deformation of the mesh planes, resulting in numerical instability. If \( C \) is too small, however, then the planes movement will not result in sufficient resolution of sharp solution gradients. In practice, we have found values of \( C \) ranging from 10 to 100 to be a good compromise between these two extremes for all of the examples studied.

Strong horizontal variation in the plane positions of a 3D mesh in TELEMAC-3D can have a destabilising effect on the simulation. In order to reduce the horizontal variation of layer positions on a local scale, the monitor function is smoothed using a simple low-pass filter in two dimensions before solving the diffusion equation (3) for layer positions.

The presence of maxima or minima in tracer concentration in the interior of the water column (as opposed to extrema at the free surface or bottom boundaries) raises a small problem for the AMR method. Because such extrema have low gradients, the AMR scheme will attempt to move solution points away from any local maxima or minima. When the solution is interpolated onto the new layer positions, this can change the position and magnitude of the extremum, resulting in a form of numerical diffusion. In the TELEMAC-3D implementation, we have attempted to eliminate this problem by first locating any local extrema in each water column, and ensuring that such points must feature in the new mesh configuration. This ensures that no interpolation takes places at extreme points, so the solution magnitude there cannot be diminished.

III. SOME EXAMPLES

We now illustrate the automatic mesh redistribution method in TELEMAC-3D by considering some simple examples. In each case, the layer positions are modified according to the tracer concentration gradient alone.
A. Tracer advection over a compound slope

The first example of adaptive layering concerns a buoyant tracer released at the upstream boundary of a straight channel. The channel bed is formed from two planar slopes with different inclinations and a constant flow rate from left to right is imposed at the upstream boundary.

Fig. 1 shows the resulting plume using a sigma mesh, whilst Fig. 2 shows the same plume simulated using the adaptive layering approach described in Section II. The AMR algorithm has moved three of the four internal planes into the spreading front of the plume, whereas the plume front only occupies a single mesh layer in the sigma-mesh case. The plume is also more concentrated towards the free surface when using the AMR approach, indicating a reduction in numerical diffusion caused by the divergence of the planes in the sigma mesh.

Finally, note that the adaptive layering reduces to equispaced sigma-layering ahead of the spreading plume, where there is no tracer concentration gradient. This demonstrates that the adaptive layering will only take effect in regions of sharp tracer gradient, leaving the mesh unchanged elsewhere.

B. Dense tracer source in a straight horizontal channel

For our second example we consider a point source of dense tracer located at the bottom of a straight channel with rectangular cross-section. The channel is 1km long, 100m wide and 10m deep. A depth-averaged velocity of 1m/s is applied along the channel. At the source, dense fluid with an excess salinity of 215 parts-per-thousand is added at a rate of $0.5m^3/s$. These source and flow parameters are typical of those found in studies of hypersaline discharge dispersion conducted by HR Wallingford.

TELEMAC-3D was first used to establish a steady velocity profile without the tracer source, and then this steady profile was used as an initial condition for a second computation including the dense tracer source. Fig. 3 shows a comparison of the results of simulations of this dense plume taken 240 minutes after release began.

By comparing panels (a) and (b) of Fig. 3, it is clear that the plume modelled using the adaptive mesh contains more mesh planes than that modelled with the sigma mesh, despite the sigma mesh having almost twice as many planes overall. The focussed resolution gives the resulting plume a more realistic shape. This is particularly apparent at the upstream end of the plume, where the AMR result shows a more rounded front than that predicted using a sigma mesh.

It is interesting to note that additional simulations carried out using a sigma mesh with more layers (not shown) seem to suggest a convergence towards a tracer concentration distribution very similar to that obtained using the AMR approach.

Panels (c) and (d) of Fig. 3 clearly show that the horizontal spreading of tracer is strongly affected by the choice of layering strategy. The dense plume spreads further across the flow using the sigma mesh when compared to the AMR approach. We believe that this cross-flow spreading follows from a ‘blocking’ phenomenon, caused by the spurious sharp front upstream of the source. This obstructs the ambient flow, forcing fresh water around the dense plume. For the adaptive mesh, the plume occupies a smaller vertical extent, thus it has a reduced effect on the ambient flow. Dense fluid is therefore mostly swept directly downstream, with minimal cross-stream spreading. This hypothesis is supported by investigation of the velocity fields in both cases (not shown).

This example highlights a very important fact about vertical mesh spacing in TELEMAC-3D. It shows that a crude sigma mesh may predict spurious flow patterns if insufficient mesh planes are used. The fact that the spreading of the dense plume tends to resemble that obtained using AMR as the number of sigma mesh planes are increased suggests that the predictions of the AMR method are more accurate than those obtained using a sigma mesh, even when using far fewer planes.

The implications of these observations for studies of dense discharges, though significant, are not discussed here. Interested readers are directed to Ref. [5] for a discussion of the importance of mesh plane positions in discharge studies.
C. Saline lock exchange

For our final example, we turn to a simple lock-exchange problem, in which relatively dense and relatively buoyant dense fluids are initially contained side-by-side in a cuboidal container. As the denser fluid sinks, it spreads beneath the less dense fluid, driven by gravity. This example is distinct from the previous examples in that the tracer concentration (which is directly linked to the fluid density) is the only factor driving the fluid flow. In the earlier examples, the applied ambient flow was large enough to dominate the dynamics.

Fig. 4 and Fig. 5 show the solution and mesh layers for the lock exchange flow using sigma and adaptively-layered meshes, respectively. The most important feature to note is that the shape of the lock exchange current (as visualised by the colours in Figs. 4 and 5) is essentially the same in each case. This demonstrates that the AMR algorithm is not degrading the quality of the solution. In fact, the clustering of mesh planes near the spreading and receding fronts of the current ought to make the solution more accurate at these key locations.

In the centre of the domain, the transition between dense and light fluid (visually, the transition from red to blue) is sharper in Fig. 5 than in Fig. 4, indicating once more that the AMR method reduces vertical numerical diffusion by increasing the mesh resolution in the transition region.

At the ends of the domain, the tracer concentration is essentially constant within each water column, so the equispaced sigma-mesh is retained.

IV. DISCUSSION

We have introduced to TELEMAC-3D a powerful method of automatically increasing mesh resolution in key portions of the water column. It is important to note that this adaptive mesh redistribution does not change the number of nodes or elements in the mesh. Furthermore, the mesh redistribution algorithm adds only a modest cost in terms of computation time.
Figure 5. Lock exchange flow using the AMR algorithm. Colours represent the salinity distribution.

The AMR algorithm that is currently implemented in TELEMAC-3D release 6.1 is a first example of what will eventually become a general adaptive layering toolbox for use in TELEMAC-3D. At present, the AMR implementation allows layers to cluster where the tracer concentration gradient is largest, and can be easily generalised to focus on velocity gradient or linear combinations of velocity and tracer gradients in much the same way as other established software, notably the GETM package [2].

One can imagine cases for which it is more appropriate to consider higher spatial derivatives of physical variables. An example of this might feature relatively small deviations from a strong saline background gradient. Interesting behaviour may occur where the salinity gradient changes sharply, but an AMR approach based on salinity gradient alone will not cluster layers at such locations. Instead, basing the mesh redistribution on the curvature of the salinity function (for example) ought to provide increased resolution in regions of interest. This can be achieved by considering a different monitor function in place of (2), with the curvature operator replacing gradient. Higher derivatives still can be tracked with equally straightforward modifications.

We look forward to further development of the AMR options in TELEMAC-3D, which we hope will be carried out both at HR Wallingford and in cooperation with other open-source developers. As part of the development process, we would greatly welcome any feedback from users regarding generalisations or suggestions for improvement.

REFERENCES


Revisiting the Thompson boundary conditions

Jean-Michel HERVOUET, Christophe DENIS
1 Laboratoire National d’Hydraulique et Environnement
2 SINETICS
Electricité de France R&D
6 Quai Watier, 78400 Chatou, France
j-m.hervouet@edf.fr

Eric DAVID
Sogreah (Artelia Group)
6 rue de Lorraine
38130 Echirolles, France
eric.david@sogreah.fr

Abstract — the Thompson boundary conditions are used for situations where data are lacking, e.g. in tidal computations where only the free surface is known. They can also allow a free exit of waves through an open boundary. The technique, originally published by Thompson in [1] and described in [2], is based on the theory of characteristics, which was applied so far in a direction normal to the boundary. Adapting Thompson boundary conditions to domain decomposition parallelism revealed a weakness of this approach which requires specific advection fields for every boundary point. These advection fields should have been transmitted to every processor, and this was considered too cumbersome. A modified theory is presented, which consists of applying the theory of characteristics in a direction following the flow. The resulting advection fields do not depend on the original boundary point, thus the standard method for characteristics in parallel may be used.

I. INTRODUCTION

The Thompson boundary conditions are used in Telemac-2D, in cases where data are unknown on open boundaries. This is the case with tidal computations when only the free surface is known, or when a wave exits an open boundary. The original method uses the theory of characteristics, linearized in a direction normal to the boundary. In Thompson publication finite differences are employed to solve the 3 advection problems of the method, this was done mainly because at that time regular grids were common practice. Eric David, at Sogreah, then resorted to the method of characteristics itself to solve these problems on unstructured grids. At that time (1999) it precluded parallelism. Then Jacek Jankowski (BAW Karlsruhe) wrote an amazing parallel version of the method of characteristics (module "streamline" in library BIEF). More recently, module streamline was adapted by Christophe Denis (Sinetics, EDF R&D) for dealing with a list of points that are not necessarily linked to mesh nodes, to enable the treatment of particles on one hand, and Thompson boundary points on the other hand. This was not the end of the story. As a matter of fact, the advection fields requested by Thompson boundary points depend on the starting point, and these specific fields must be defined for the whole domain. In parallel this implies that every Thompson boundary point has to send its advection fields to all processors, in case its characteristic path-lines would go to another sub-domain. This was considered too cumbersome, a dead end. Moreover, the Thompson theory leads to the fact that two nearby boundary points may have their characteristics path-lines crossing, because linearization was done in two different directions. This is somewhat against the nature of characteristics that do not cross unless they carry the same invariant. For all these reasons it was considered that the theory had to be modified. It seems natural that the linearization direction should be the direction of the flow. It is what is attempted here. We shall first fully explain what was done in previous versions, and then we shall move to the new idea.

II. A DETAILED EXPLANATION OF THE ORIGINAL TECHNIQUE

We explain hereafter in more detail what is said in Reference [1] page 105 to 108. We neglect diffusion and start from the conservative form of Saint-Venant equations, put in the following form taken from [1] at page 31, using the fact that the free surface $Z_s$ is the bottom topography plus the depth $h$.

\[
\frac{\partial h}{\partial t} + \text{div}(h\vec{u}) = Sce
\]  
(1)

\[
\frac{\partial (hu)}{\partial t} + \frac{\partial}{\partial x}(huv + g h^2/2) + \frac{\partial}{\partial y}(huv) = -gh\frac{\partial Z_f}{\partial x} + h F_x
\]  
(2)

\[
\frac{\partial (hv)}{\partial t} + \frac{\partial}{\partial x}(huv) + \frac{\partial}{\partial y}(huv + g h^2/2) = -gh\frac{\partial Z_f}{\partial y} + h F_y
\]  
(3)

We write:

\[
F = \begin{pmatrix} h \\ hu \\ hv \end{pmatrix}
\]  
(4)

\[
G_s = \begin{pmatrix} hu \\ hu^2 + g v^2/2 \end{pmatrix} \quad \text{and} \quad G_s = \begin{pmatrix} hv \\ hvu \\ hv^2 + g v^2/2 \end{pmatrix}
\]  
(5)

and
so that the system of three equations can be written in the following form:

\[
\frac{\partial F}{\partial t} + \frac{\partial G}{\partial x} + \frac{\partial G_{\zeta}}{\partial \zeta} = S(F)
\]

(7)

The Thompson method as implemented so far in Telemac-2D consists of considering a local system of coordinates based on a local normal vector \( \tilde{n} \) (normal to the boundary) and a local tangent vector \( \tilde{t} \). If the new system of coordinates is denoted \( \eta \) and \( \zeta \), we have

\[
\tilde{n} = \left( \begin{array}{c} \frac{\partial \eta}{\partial x} \\ \frac{\partial \eta}{\partial y} \end{array} \right) \quad \text{and} \quad \tilde{t} = \left( \begin{array}{c} \frac{\partial \zeta}{\partial x} \\ \frac{\partial \zeta}{\partial y} \end{array} \right)
\]

(8)

We keep these notations here, but the directions \( \tilde{n} \) and \( \tilde{t} \) may not be linked to the boundary. The components of velocity in the new system will be denoted \( u_\eta \) and \( u_\zeta \). We have

\[
\begin{align*}
(u_\eta) & = \left( \begin{array}{c} \frac{\partial \eta}{\partial x} \\ \frac{\partial \eta}{\partial y} \end{array} \right) (u) = \left( \begin{array}{c} \frac{\partial \eta u}{\partial x} + \frac{\partial \eta v}{\partial y} \\ \frac{\partial \eta u}{\partial y} + \frac{\partial \eta v}{\partial x} \end{array} \right) \\
(u_\zeta) & = \left( \begin{array}{c} \frac{\partial \zeta}{\partial x} \\ \frac{\partial \zeta}{\partial y} \end{array} \right) (v) = \left( \begin{array}{c} -\frac{\partial \zeta u}{\partial x} + \frac{\partial \zeta v}{\partial y} \\ -\frac{\partial \zeta u}{\partial y} + \frac{\partial \zeta v}{\partial x} \end{array} \right)
\end{align*}
\]

(9)

and

\[
\begin{align*}
(u) & = \left( \begin{array}{c} \frac{\partial \eta}{\partial x} \\ \frac{\partial \eta}{\partial y} \end{array} \right) u + \left( \begin{array}{c} \frac{\partial \zeta}{\partial x} \\ \frac{\partial \zeta}{\partial y} \end{array} \right) u_\zeta = \left( \begin{array}{c} \frac{\partial \eta u}{\partial x} + \frac{\partial \eta v}{\partial y} + u_\zeta \frac{\partial \zeta}{\partial x} \\ \frac{\partial \eta u}{\partial y} + \frac{\partial \eta v}{\partial x} + u_\zeta \frac{\partial \zeta}{\partial y} \end{array} \right)
\end{align*}
\]

(10)

We first want to put the system in the form:

\[
\frac{\partial F}{\partial t} + A_\eta \frac{\partial F}{\partial x} + B_\zeta \frac{\partial F}{\partial \zeta} = S(F)
\]

(11)

where \( A_\eta \) and \( B_\zeta \) are matrices. For this goal:

- In (2):
  \( \frac{1}{m} \frac{d}{dt} (g \frac{1}{m} \dot{v}) \) is written \( c^2 \frac{\partial \eta}{\partial x} \)
  \( \frac{1}{m} \frac{d}{dt} (hua) \) is written \( u \frac{\partial \eta}{\partial x} + h \frac{\partial \eta}{\partial x} = u^2 \frac{\partial \eta}{\partial x} + 2uh \frac{\partial \eta}{\partial x} \)
  which is also equal to \( u^2 \frac{\partial \eta}{\partial x} + 2u \frac{\partial \eta}{\partial x} - 2u^2 \frac{\partial \eta}{\partial x} \)
  \( \frac{1}{m} \frac{d}{dt} (hu) \) is written:
  \( v \frac{\partial \eta}{\partial x} + hu \frac{\partial \eta}{\partial x} = v \frac{\partial \eta}{\partial x} (hu) + u \frac{\partial \eta}{\partial x} - uv \frac{\partial \eta}{\partial y} \)

- In (3):
  \( \frac{1}{m} \frac{d}{dt} (g \frac{\dot{v}}{m}) \) is written \( c^2 \frac{\partial \eta}{\partial x} \)
  \( \frac{1}{m} \frac{d}{dt} (h\nu) \) is written \( -v^2 \frac{\partial \eta}{\partial x} + 2v \frac{\partial \eta}{\partial x} \)

\( \frac{\partial}{\partial \eta} (u) \) is written \( v \frac{\partial \eta}{\partial x} + hu \frac{\partial \eta}{\partial x} + u \frac{\partial \eta}{\partial x} - uv \frac{\partial \eta}{\partial y} \)

We effectively get to (11) with:

\[
\begin{align*}
A_\eta & = \begin{pmatrix} 0 & 1 & 0 \end{pmatrix} \\
B_\zeta & = \begin{pmatrix} uv & v & u \end{pmatrix}
\end{align*}
\]

(12)

(13)

Now we change the coordinates by writing that for every function \( f \) we have:

\[
\frac{\partial f}{\partial \eta} = \frac{\partial f}{\partial \eta} + \frac{\partial f}{\partial \zeta} \quad \text{and} \quad \frac{\partial f}{\partial \zeta} = \frac{\partial f}{\partial \eta} \frac{\partial \eta}{\partial \eta} + \frac{\partial f}{\partial \zeta} \frac{\partial \zeta}{\partial \zeta}
\]

It gives us a system in the form:

\[
\frac{\partial F}{\partial t} + A_\eta \frac{\partial F}{\partial \eta} + B_\zeta \frac{\partial F}{\partial \zeta} = S(F)
\]

(14)

with

\[
A_\eta = \begin{pmatrix} 0 & \frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial y} \end{pmatrix}
\]

\[
B_\zeta = \begin{pmatrix} \frac{\partial \zeta}{\partial x} & \frac{\partial \zeta}{\partial y} & 0 \end{pmatrix}
\]

(15)

and

\[
A_\eta = \begin{pmatrix} \frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial y} & \frac{\partial \eta}{\partial \zeta} \end{pmatrix}
\]

\[
B_\zeta = \begin{pmatrix} \frac{\partial \zeta}{\partial x} & \frac{\partial \zeta}{\partial y} & 0 \end{pmatrix}
\]

(16)

which gives

\[
A_\eta = \begin{pmatrix} \frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial y} & \frac{\partial \eta}{\partial \zeta} \end{pmatrix}
\]

(17)

and

\[
B_\zeta = \begin{pmatrix} \frac{\partial \zeta}{\partial x} & \frac{\partial \zeta}{\partial y} & 0 \end{pmatrix}
\]

(18)

or even, still denoting \( u_\eta \) as the normal component of velocity and \( u_\zeta \) the tangential component:

\[
A_\eta = \begin{pmatrix} \frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial y} & \frac{\partial \eta}{\partial \zeta} \end{pmatrix}
\]

(19)
We then get back to the diagonalized system:

\[ \frac{\partial F}{\partial t} + \Lambda \frac{\partial F}{\partial \eta} = S(F) \]  

An open question is: which part of \( S(F) \) should be kept in this equation? We discard \( \text{Sc} \), \( F_\varepsilon \) and \( F_\gamma \), and keep only the variations of bottom along the direction \( \eta \). It gives:

\[ S_\eta(F) = \begin{pmatrix}
\eta \frac{\partial}{\partial \eta} + \frac{\eta}{\partial \eta} \\
-\eta \frac{\partial}{\partial \eta} - \frac{\eta}{\partial \eta}
\end{pmatrix} \]  

For the time being, we call it \( S_\eta(F) \) whatever its value and go on with the diagonalization of \( \Lambda_\eta \). Now \( \Lambda_\eta \) is diagonalized as \( \Lambda_\eta = L_t^\dagger \Lambda L_t \) with:

\[ L_t = \begin{pmatrix}
u & \frac{\eta}{\partial \eta} & -\frac{\eta}{\partial \eta} \\
\frac{\eta}{\partial \eta} & c - \eta & \frac{\eta}{\partial \eta} \\
\frac{\eta}{\partial \eta} & \frac{\eta}{\partial \eta} & c + \eta
\end{pmatrix} \]  

and

\[ \Lambda_t = \begin{pmatrix}\eta & 0 & 0 \\
0 & \eta + c & 0 \\
0 & 0 & \eta - c
\end{pmatrix} \]  

This can be controlled by checking that \( \Lambda_\eta = L_t^\dagger \Lambda L_t \). By stating that \( dW = LdF \), we then get back to the diagonalized system:

\[ \frac{\partial W}{\partial t} + \Lambda \frac{\partial W}{\partial \eta} = L_s \eta \]  

Each of whose lines is a simple transport equation with source term. Thompson proposes to consider that \( L \) is constant in the vicinity of a boundary point, and to write \( W = L F \), where

\[ \bar{L} = \begin{pmatrix}
u & \frac{\eta}{\partial \eta} & -\frac{\eta}{\partial \eta} \\
\frac{\eta}{\partial \eta} & c - \eta & \frac{\eta}{\partial \eta} \\
\frac{\eta}{\partial \eta} & \frac{\eta}{\partial \eta} & c + \eta
\end{pmatrix} \]  

the over-bar values being considered as constant (these are the values deduced from the local conditions: \( h, u \) and \( v \) at the original starting point of the characteristics). The Riemann invariants of the vector \( W \) are thus:

\[ W_1 = h(\bar{u}_\eta - \bar{u}) \]  
\[ W_2 = h(\bar{c} + u - \bar{u}) \]  
\[ W_3 = h(\bar{c} - u + \bar{u}) \]  

and to which can be added, if a tracer \( T \) also has to be considered:

\[ W_4 = h(T - \bar{T}) \]  

Pure advection is treated with the method of characteristics. To be more precise, a first advection is done with velocity \( u_\eta \). This is done backwards in time. For every boundary point of Thompson type, we compute the backward trajectory and find, at what is called the foot of the characteristic curve (starting point of the trajectory which will arrive at the boundary point after \( \Delta t \)), the values of depth and components of velocity which we call \( \bar{h}, \bar{u}, \bar{v} \), etc. If we neglect the source terms and take the invariants at this foot of characteristic path-line, we have:

\[ W_1 = h(\bar{u}_\eta - \bar{u}) \]  
\[ W_2 = h(\bar{c} + u - \bar{u}) \]  
\[ W_3 = h(\bar{c} - u + \bar{u}) \]  

Then, after an advection with velocity \( u_\eta + c \), i.e. with results now called \( \bar{h}, \bar{u}, \bar{v} \):

\[ W_2 = h(\bar{c} + u - \bar{u}) = W_2 \]  
\[ W_3 = h(\bar{c} - u + \bar{u}) \]  

and \( \bar{u}_\eta = \bar{u}_\eta + c \). with yet other values denoted \( \bar{h}, \bar{u}, \bar{v} \):

\[ W_3 = h(\bar{c} - u + \bar{u}) \]  
\[ W_3 = h(\bar{c} - u + \bar{u}) \]  

All this is valid only if the backwards characteristic goes inside the domain. This can be checked by the fact that \( \bar{u}_\eta ~ u > 0 \), where \( \bar{u}_\eta \) is the advection velocity field (i.e. based on \( u_\eta \), \( u + c \) or \( u - c \), respectively for \( \bar{W}_1, \bar{W}_2 \) and \( \bar{W}_3 \)). If \( \bar{u}_\eta \leq 0 \), all variables with a tilde will be based on the boundary conditions prescribed by the user. For example, \( \bar{u}_\eta \) may be taken equal to:

\[ -u_{\text{bar}} \frac{\eta}{\partial \eta} + v_{\text{bar}} \frac{\eta}{\partial \eta} \]  

where \( u_{\text{bar}} \) and \( v_{\text{bar}} \) are the prescribed components of the velocity field. Source terms will be considered later. Once
the Riemann invariants are known, the primitive variables can be restored by the following formulae:

\[ h = \frac{W_i + W_j}{2c} \]  

(27)

\[ h(u - \bar{u}) = \frac{\partial \eta}{\partial y} W_i + \frac{\partial \eta}{\partial x} (W_j - W_i) \]  

(28)

\[ h(v - \bar{v}) = \frac{\partial \eta}{\partial y} (W_i - W_j) - \frac{\partial \eta}{\partial x} W_i \]  

(29)

\[ h(T - \bar{T}) = -W_i \]  

(30)

Equation (14) can be used to eliminate \( h \) from the three others, yielding

\[ h = \frac{W_i + W_j}{2c} \]  

(31)

\[ hu = \frac{W_i + W_j}{2c} - u + \frac{\partial \eta}{\partial y} W_i + \frac{\partial \eta}{\partial x} (W_j - W_i) \]  

(32)

\[ hv = \frac{W_i + W_j}{2c} - v + \frac{\partial \eta}{\partial y} (W_i - W_j) - \frac{\partial \eta}{\partial x} W_i \]  

(33)

\[ hT = \frac{W_i + W_j}{2c} \bar{T} - W_i \]  

(34)

This form is not the most practical but readily gives, if necessary or for checking:

\[
\mathbf{L}^{-1} = \begin{pmatrix}
0 & \frac{1}{2c} & \frac{1}{2c} \\
\frac{\partial \eta}{\partial y} & \frac{\partial \eta}{\partial y} + \frac{\partial \eta}{\partial x} & -\frac{\partial \eta}{\partial x} - \frac{\partial \eta}{\partial y} \\
\frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial x} & \frac{\partial \eta}{\partial x}
\end{pmatrix}
\]  

(35)

We will favour the following formulae for the implementation:

\[ h = \frac{W_i + W_j}{2c} \]  

(36)

\[ u = \frac{\frac{\partial \eta}{\partial y} W_i + \frac{\partial \eta}{\partial x} (W_j - W_i)}{h} + u \]  

(37)

\[ v = \frac{\frac{\partial \eta}{\partial y} (W_i - W_j) - \frac{\partial \eta}{\partial x} W_i}{h} \]  

(38)

\[ T = \frac{W_i}{h} + \bar{T} \]  

(39)

If we do not neglect source terms, they have to be integrated along the characteristic curve. Assuming a constant \( \mathbf{L} \) as done before we have

\[
\begin{pmatrix}
W_i \\
W_j \\
W_i + \text{Sce} \Delta \end{pmatrix} = \begin{pmatrix}
\bar{u}_c \\
\bar{v}_c \\
\frac{\partial \eta}{\partial x} \bar{u}_c + \frac{\partial \eta}{\partial y} \bar{v}_c \\
\frac{\partial \eta}{\partial x} \bar{u}_c - \frac{\partial \eta}{\partial y} \bar{v}_c \\
\bar{v}_c \end{pmatrix}
\]  

(40)

Though the source terms could be treated in an explicit way, it appears very strange that characteristics of the same family its normal vector must be exported to all sub-domains. It also whole domain, which implies that for every Thompson point, parallel because these advection fields should be built for the same advection field. It becomes even more heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment. This was heavy in scalar mode, where points with the same advection fields depend on the boundary point under treatment.
stemming from two different points may cross because they have a different original direction.

The new theory consists of choosing advection fields that would not depend on a given boundary point. It seems very natural to choose, instead of the outward normal vector $\hat{n}$, the direction of the velocity field itself. We have then:

$$\hat{n} = \frac{\nabla n}{\|
abla n\|} = \left(\frac{u}{\sqrt{u^2 + v^2}}, \frac{v}{\sqrt{u^2 + v^2}}\right)$$

An important consequence of this choice is that the velocity $u^e$ is always 0 by definition, which would lead to $W = 0$. This is true in fact only if we consider that the direction $\hat{n}$ changes along characteristics, it is false if we keep the original $\hat{n}$, which would be consistent with the linearisation leading to $L$. Tests show that it is better to consider that $u^e$ is indeed not 0, thus sticking to the linearisation. A possibility that remains to be tested would be considering that $u^e$ is indeed 0, and taking the norm of velocity for the component $u^e$.

In any case there is an obvious problem when there is no velocity, the direction where to apply the celerity $c$ is then undefined. A first idea is to cancel also the celerity $\zeta$. In a corner the average characteristics itself is able to check if the path-line goes out of the domain, and in this case it stops and interpolates at this exit point. In a corner the average characteristics itself is able to check if the path-line goes out of the domain, and in this case it stops and interpolates at this exit point. In a corner the average characteristics itself is able to check if the path-line goes out of the domain, and in this case it stops and interpolates at this exit point.

**B. Depth and velocity for interpolation**

An unexpected problem occurred in the results, showing that the tests $\tanh \hat{n} > 0$ to decide whether we should take e.g. the depth $h$ or the prescribed depth $h_{\text{conv}}$ for computing $h$, could happen to be wrong. As a matter of fact the method of characteristics itself is able to check if the path-line goes out of the domain, and in this case it stops and interpolates at this exit point. In a corner the average $\hat{n}$ of the corner point may lead to a different decision, thus leading to wrongly choose for example $h$ instead of $h_{\text{conv}}$. Any case where the value $\tanh \hat{n}$ is very close to 0 will lead to a random choice, and then to large differences if $h$ and $h_{\text{conv}}$ are very different. It was thus decided to discard the tests $\tanh \hat{n} > 0$ and to use interpolation fields of $h, u$ and $v$ that already contain the prescribed boundary conditions. A characteristic path-line that exits a Thompson boundary will thus find naturally that $h = h_{\text{conv}}$, without resorting to testing $\tanh \hat{n} > 0$. A drawback is that for small Courant numbers, when the characteristics path-lines will not go far from boundaries, their interpolated values will be influenced by the prescribed values of the boundary. When prescribed values are correct, which is generally the case with box models and measurements, this could be also an advantage. With this new approach there can be no discontinuity of choice due to a truncation error.

**C. Tests**

The more convincing test is the Gaussian hill test, if we consider that all the boundaries of the square domain are open (test thompson in folder testgb in telemac-2D release). In this case no information is given on the boundaries. The circular wave spreading in the square domain is supposed to exit freely the domain, without any reflection on the boundaries. The results are shown on Figure 1. The new method, on the right, gives slightly more circular iso-lines of depth.

Test-case number 2 checks a boundary with prescribed elevation and free velocity. In a channel 600 m long and 6 m large, a solitary wave is imposed at the left entrance ($x = 0$). The original depth is 10 m and the wave height 2 m. The original position of the wave at the beginning of the computation is $x_0 = -80$ m, so that velocity and depth in the channel are undisturbed. With A the amplitude and $c = \sqrt{gh}$, the celerity of the wave, depth and velocity are:

$$h(x,t) = h_0 + \frac{A}{\cosh \left[ \frac{3A}{4h_0} (x + x_0 - ct) \right]^2}$$

$$u(x,t) = c \left( \frac{h}{h_0} - 1 \right)$$

Two cases are considered, both with elevation declared as imposed at the entrance and velocity free at both ends, but the first one (top of Figure 2) with Thompson boundary conditions and the second one (bottom of Figure 2) without. The arrival of the wave is equally well treated, but the exit is correct only with Thompson conditions, a totally free output yielding a spurious reflection. In this solitary wave case we have solved the Boussinesq equations, knowing that the solitary wave employed here is a first order solution of Navier-Stokes equations, which is rather badly treated by Saint-Venant equations. The drawback is that Boussinesq equations will perhaps not comply with the theory of characteristics underlying Thompson conditions... however the result clearly shows the improvement brought by Thompson.

**REFERENCES**


Figure 1 – A circular wave exiting through a square open boundary. Comparison of version 6.0 and 6.1 of Telemac-2D.

Figure 2 – A solitary wave with Boussinesq equations, with Thompson boundary conditions (top) and without (bottom).