28 November 2011 – Tallinn

THE NUMERICAL COMPUTATION OF VIOLENT LIQUID MOTION

Frédéric DIAS

School of Mathematical Sciences
University College Dublin
on leave from Ecole Normale Supérieure de Cachan

28 November 2011 - Tallinn
### Topics Covered in Today’s Talk

<table>
<thead>
<tr>
<th>Liquid Impact on a Wall</th>
<th>Wave Energy Converters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sloshel</td>
<td>LNG carrier</td>
</tr>
</tbody>
</table>

28 November 2011 - Tallinn
COLLABORATORS

- Jean-Philippe Braeunig (INRIA & CEA)
- Laurent Brosset (GTT)
- Paul Christodoulides (Cyprus University of Technology)
- Ken Doherty (Aquamarine Power Ltd.)
- Christophe Fochesato (CEA)
- Jean-Michel Ghidaglia (Ecole Normale Supérieure de Cachan)
- Emmanuel Reynaud (University College Dublin)

Postdocs
- Emiliano Renzi (University College Dublin)
- Roxana Tiron (University College Dublin)

PhD students
- Laura O'Brien (University College Dublin)
- Sarah Gallagher (University College Dublin)

Research funded by
- SFI (Science Foundation Ireland)
- GTT (Gaz Technigaz & Transport)

28 November 2011 - Tallinn
<table>
<thead>
<tr>
<th>LIQUID IMPACT ON A WALL</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Sloshel</td>
<td></td>
</tr>
<tr>
<td>LNG carrier</td>
<td></td>
</tr>
</tbody>
</table>
• Local phenomena involved during wave impacts are very sensitive to input conditions.

• The density of bubbles, the local shape of the free surface, the local flow make the impact pressure change dramatically even for the same experimental conditions.

• How does one extrapolate wave impact from model (small scale) to prototype (full scale)?

• In recent years, we have addressed the scaling issue by studying the various local phenomena present in wave impact one after the other in order:
  - to better understand the physics behind
  - to improve the experimental modelling

VIDEO 1 – Experiments in Marseille
(courtesy of O. Kimmoun)
SLOSHING IN TANKS OF LIQUEFIED NATURAL GAS (LNG) CARRIERS

Looking Forward where the Membrane Failed

Heavy weather and partially loaded LNG tank resulted sloshing damage in a LNG tank – 6 months and Millions $$$

GTT
### Various Scenarios of Wave Impact

<table>
<thead>
<tr>
<th>Scenario</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slosh</td>
<td>« Flip-through » condition with no air bubbles. The impact pressure has a single peak.</td>
</tr>
<tr>
<td>Collision</td>
<td>Collision of a plunging breaker with a thin air pocket.</td>
</tr>
<tr>
<td>Collision</td>
<td>Collision of a fully developed plunging breaker with a thick air pocket.</td>
</tr>
</tbody>
</table>

THE FLIP-THROUGH PHENOMENON

• Run-up of wave trough
• Forward motion of almost vertical wave front
• Jet flow generated at the wall

Bredmose et al. (2004), Water wave impact on walls and the role of air, *Proceedings ICCE 2004*

Brosset et al. (2011), A Mark III Panel Subjected to a Flip-through Wave Impact: Results from the Sloshel Project, *Proceedings ISOPE 2011*

28 November 2011 - Tallinn
PHENOMENOLOGY OF A LIQUID IMPACT

- **Global behaviour**
  - Global flow governed by Froude number

- **Local behaviour**
  - Escape of the gas between the liquid and the wall: momentum transfer between liquid and gas
  - Compression of the partially entrapped gas during the last stage of the impact
  - Rapid change of momentum of the liquid diverted by the obstacle
  - Possible creation of shock waves: pressure wave within the liquid and strain wave within the wall
  - Hydro-elasticity effects during the fluid-structure interaction

Braeunig et al. (2009), Phenomenological Study of Liquid Impacts through 2D Compressible Two-fluid Numerical Simulations, *Proceedings ISOPE 2009*
Locally, the impact process is not similar for similar inflow conditions

- Gas compressibility bias: the equations of state should be scaled
- Liquid compressibility bias: speed of sound should be scaled

Biases are different for different impacts: no unique scaling law!
Fluid equations:

\[ u_t + uu_x + vu_y + wu_z + \frac{1}{\rho} \frac{\partial p}{\partial x} = 0 \]
\[ v_t + uv_x + vv_y + wv_z + \frac{1}{\rho} \frac{\partial p}{\partial y} = 0 \]
\[ w_t + uw_x + vw_y + ww_z + \frac{1}{\rho} \frac{\partial p}{\partial z} = -g \]
\[ u_x + v_y + w_z = 0 \]

Boundary conditions:

\[ u \cdot n = 0 \]
\[ h_t + uh_x + vh_y = w \]
\[ p = 0 \]

\( (x,y,z) \) : spatial coordinates
\( u = (u,v,w) \) : velocity vector
\( p \) : pressure     \( \rho \) : density
\( g \) : acceleration due to gravity

\( h(x,y,t) \) : elevation of the interface

kinematic and dynamic conditions on the interface
Froude scaling \[ \frac{u_{fs}^2}{gD_{fs}} = \frac{u_{ms}^2}{gD_{ms}} \]

\( fs \) stands for full scale, \( ms \) for model scale

\[ D_{fs} = \lambda D_{ms} \]

\[ u_{fs} \left( \lambda x_{ms}, \sqrt{\lambda} t_{ms} \right) = \sqrt{\lambda} u_{ms} \left( x_{ms}, t_{ms} \right) \quad t_{fs} = \sqrt{\lambda} t_{fms} \]

\[ \dot{u}_{fs} \left( \lambda x_{ms}, \sqrt{\lambda} t_{ms} \right) = \dot{u}_{ms} \left( x_{ms}, t_{ms} \right) \quad p_{fs} = \lambda p_{ms} \]
TWO-FLUID COMPRESSIBLE EULER EQUATIONS WITH AN INTERFACE IN PHYSICAL VARIABLES

Fluid equations

\[ \frac{\partial \rho_k}{\partial t} + \nabla \cdot (\rho_k \vec{u}_k) = 0 , \]

\[ \rho_k \left( \frac{\partial \vec{u}_k}{\partial t} + (\vec{u}_k \cdot \nabla) \vec{u}_k \right) + \nabla p_k = \rho_k (\vec{g} - \vec{\gamma}_k - 2 \Omega \times \vec{u}_k) , \]

\[ \rho_k \left( \frac{\partial e_k}{\partial t} + (\vec{u}_k \cdot \nabla) e_k \right) + p_k \nabla \cdot \vec{u}_k = 0 , \]

Boundary conditions

\[ \vec{u}_k \cdot \vec{n} = 0 \]

\[ \frac{\partial f}{\partial t} = -\vec{u}_1 \cdot \nabla f = -\vec{u}_2 \cdot \nabla f , \quad \text{on} \quad f(x_1, x_2, x_3, t) = 0 , \]

\[ p_1 = p_2 , \quad T_1 = T_2 , \quad \text{on} \quad f(x_1, x_2, x_3, t) = 0 . \]

Equation of state

\[ \rho_k = R_k (p_k, e_k) . \]
INVARINANCE OF TWO-FLUID COMPRESSIBLE EULER EQUATIONS WITH AN INTERFACE

Froude scaling
\[ \frac{u_{fs}^2}{g D_{fs}} = \frac{u_{ms}^2}{g D_{ms}} \] (including the speeds of sound)

\[ \mathbf{u}_{fs} \left( \lambda \mathbf{x}_{ms}, \sqrt{\lambda} t_{ms} \right) = \sqrt{\lambda} \mathbf{u}_{ms} \left( \mathbf{x}_{ms}, t_{ms} \right) \]

\[ \mathbf{u}_{fs} \left( \lambda \mathbf{x}_{ms}, \sqrt{\lambda} t_{ms} \right) = \mathbf{u}_{ms} \left( \mathbf{x}_{ms}, t_{ms} \right) \]

Differences with the one-fluid incompressible case

\[ \mu = \frac{\rho_{ms}}{\rho_{fs}} \]

\[ R_k^{\lambda} (p, e) = \mu R_k \left( \frac{\lambda}{\mu}, \lambda e \right) \]

\[ \frac{\rho_{fs}}{\rho_{fs}} = \frac{\rho_{ms}}{\rho_{ms}} \]

\[ p_{fs} = \lambda \left( \rho_{liq,fs} / \rho_{liq,ms} \right) p_{ms} \]

Scale the equations of state

Keep the same density ratio

28 November 2011 - Tallinn
THE ROLE OF NUMERICAL STUDIES

• For sloshing inside the tank of a LNG carrier or for the motion of a wave energy converter, numerical simulations can provide impressive results but the question remains of how relevant these results are when it comes to determining impact pressures!

• The numerical models are too simplified to reproduce the high variability of the measured pressures. NOT POSSIBLE FOR THE TIME BEING TO SIMULATE ACCURATELY BOTH GLOBAL AND LOCAL EFFECTS! (see ISOPE 2009 Numerical Benchmark)

• However, numerical studies can be quite useful to perform sensitivity analyses in idealized problems (see ISOPE 2010 Numerical Benchmark)
COMPARATIVE NUMERICAL STUDY (2010)

Length (m)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>H</td>
<td>15</td>
</tr>
<tr>
<td>h</td>
<td>8</td>
</tr>
<tr>
<td>h₁</td>
<td>2</td>
</tr>
<tr>
<td>h₂</td>
<td>5</td>
</tr>
</tbody>
</table>

Organizers: GTT and UCD

(compressible bi-fluid software was required)

1D case

LNG = Liquefied Natural Gas
NG = Natural Gas

<table>
<thead>
<tr>
<th>Case #</th>
<th>Scale</th>
<th>Liquid</th>
<th>Gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1:1</td>
<td>LNG</td>
<td>NG</td>
</tr>
<tr>
<td>2</td>
<td>1:40</td>
<td>LNG</td>
<td>NG</td>
</tr>
<tr>
<td>3</td>
<td>1:40</td>
<td>Water</td>
<td>Air</td>
</tr>
<tr>
<td>4</td>
<td>1:40</td>
<td>Water</td>
<td>SF₆+N₂</td>
</tr>
<tr>
<td>5</td>
<td>1:40</td>
<td>1:40-scaled LNG</td>
<td>1:40-scaled NG</td>
</tr>
</tbody>
</table>
1D SURROGATE MODEL OF AIR-POCKET IMPACT

Piston model

- Perfect gas
  \[ p = \rho RT \]

- Adiabatic process
  \[ p \left( \frac{1}{\rho} \right)^\gamma = \text{constant} \]

Initial conditions:
\[ p = p_0, \quad z(0) = h_1, \quad \dot{z}(0) = 0 \]

\[ \ddot{z}(t) = -g - \frac{p_0}{\rho_i h} \left( \left( \frac{h_2}{H - h - z} \right)^\gamma - \left( \frac{h_1}{z} \right)^\gamma \right) \]
Pressures and times are Froude-scaled for cases 2, 3, 4, 5 with $\lambda = 40$, $fs = \text{full scale}$, $ms = \text{model scale}$.

$P_{fs} = \lambda \left( \frac{P_{liq}^{fs}}{P_{liq}^{ms}} \right) P_{ms}$

$t_{fs} = \sqrt{\lambda} t_{ms}$

**Warning:**
- Compressibility matters, when comparing at same scale.
- The 5 cases are very smooth compression cases.

28 November 2011 - Tallinn
CONCLUSIONS FOR THE 1D CASE

- The different numerical methods are able to simulate adequately a simple smooth compression of a gas pocket without escape of gas.
- Very good agreement on the maximum pressure.

- For all methods:
  - Complete Froude Scaling (CFS) works (same result for cases 1 & 5).
  - Partial Froude Scaling (PFS) generates a bias.
2D case

<table>
<thead>
<tr>
<th>Length</th>
<th>(m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>H</td>
<td>15</td>
</tr>
<tr>
<td>h</td>
<td>8</td>
</tr>
<tr>
<td>h₁</td>
<td>2</td>
</tr>
<tr>
<td>h₂</td>
<td>5</td>
</tr>
<tr>
<td>L</td>
<td>20</td>
</tr>
<tr>
<td>l</td>
<td>10</td>
</tr>
<tr>
<td>l₁</td>
<td>5</td>
</tr>
</tbody>
</table>

28 November 2011 - Tallinn
## COMPARATIVE NUMERICAL STUDY (2010) – 2D CASE

<table>
<thead>
<tr>
<th>Case #</th>
<th>Scale</th>
<th>Liquid</th>
<th>Gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>1:1</td>
<td>LNG</td>
<td>NG</td>
</tr>
<tr>
<td>7</td>
<td>1:40</td>
<td>LNG</td>
<td>NG</td>
</tr>
<tr>
<td>8</td>
<td>1:40</td>
<td>Water</td>
<td>Air</td>
</tr>
<tr>
<td>9</td>
<td>1:40</td>
<td>Water</td>
<td>SF$_6$+N$_2$</td>
</tr>
<tr>
<td>10</td>
<td>1:40</td>
<td>1:40-scaled LNG</td>
<td>1:40-scaled NG</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Participants</th>
<th>Software</th>
<th>Method</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS</td>
<td>Fluent</td>
<td>Finite Volume/VOF</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Principia</td>
<td>LS-DYNA</td>
<td>FEM Euler/Lagrange</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>ENS-Cachan</td>
<td>Flux-IC</td>
<td>Finite Volume/NIP</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Lloyds Register</td>
<td>OpenFOAM</td>
<td>Finite Volume/VOF</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Force</td>
<td>Comflow</td>
<td>Finite Volume/VOF</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>UoSFSI</td>
<td>in House</td>
<td>Finite Differences/VOF</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
</tr>
</tbody>
</table>

28 November 2011 - Tallinn
A totally Eulerian finite volume solver for multi-material fluid flows

J.-P. Braeunig\textsuperscript{a}, B. Desjardins\textsuperscript{b}, J.-M. Ghidaglia\textsuperscript{c,}\textsuperscript{*}

\textsuperscript{a} CEA/DIF Bruyères-le-Châtel, 91297 Arpajon Cedex, France
\textsuperscript{b} ENS Paris, DMA, 45 rue d'Ulm, 75230 Paris cedex 05, France
\textsuperscript{c} CMLA, ENS Cachan and CNRS, Université and IRC M6SO, ENS Cachan and CEA/DIF, 61 avenue du Président Wilson, F-94235 Cachan Cedex

\textbf{ARTICLE INFO}

\textit{Article history:}
Received 28 April 2008
Received in revised form 19 March 2009
Accepted 19 March 2009
Available online 26 March 2009

\textbf{ABSTRACT}

The purpose of this work is to present a new numerical scheme for multi-material fluid flows in dimension \(d \geq 1\). It is a totally Eulerian conservative scheme that allows to compute sharp interfaces between non-miscible fluids. The underlying flux scheme in single material cells is the so-called FVCF scheme, whereas interface reconstruction and directional splitting is used in multi-material cells. One of the novelty of our approach is the introduction of the concept of “condensate” which allows to handle mixed cells containing two or more materials. Moreover, it has been designed to allow free sliding of materials on each others, thanks to a material volume centered computation of variables in mixed cells.

© 2009 Elsevier Masson SAS. All rights reserved.
Pressure at P1 for three different meshes

- **Warning:**
- High refinement is required for capturing the pressure peak
Velocities and times are Froude-scaled for cases 7, 8, 9, 10
\[ v^s = \sqrt{\lambda} \cdot v^ms, \quad t^s = \sqrt{\lambda} \cdot t^ms \]
with \( \lambda = 40 \), \( v^s = \) full scale, \( v^ms = \) model scale

28 November 2011 - Tallinn
MAXIMUM PRESSURE AT $P_1$ : ALL PARTICIPANTS

$P_{\text{max}}$ (bar) - all participants

- Case 6: scale 1 - LNG/NG
- Case 10: scale 1:40 - scaled LNG/NG
- Case 7: scale 1:40 - LNG/NG
- Case 8: scale 1:40 - water/air
- Case 9: scale 1:40 - water/(SF6+N2)

28 November 2011 - Tallinn
• Absolute values for $V_{\text{max}}$ and $P_{\text{max}}$ are very scattered
• The meshes are not refined enough to capture sharp peak pressures
• After some work on the models, results should be much less scattered
  (work in progress)

• Such a simple test should be passed adequately before attempting to calculate more complex impacts

• For all methods, whether relevant or not:
  > Complete Froude Scaling (CFS) is satisfied
  > Partial Froude Scaling generates a bias
• Strategy used to compute wave impact: couple potential flow solver with two-fluid compressible flow solver

• Potential flow solver computes the wave all the way to overturning (Fochesato & Dias 2006)

• Two-fluid (gas + liquid) compressible flow solver computes the liquid impact on the wall

work in progress
A fast method for nonlinear three-dimensional free-surface waves

By Christophe Fochesato and Frédéric Dias*

Video 5 – Experiments in Nantes

- Evidence of directional wave focusing in a « numerical » wave tank (Fochesato, Grilli, Dias, Wave Motion, 2007)
- High-order three-dimensional boundary element method combined with mixed Eulerian–Lagrangian time updating, based on second-order explicit Taylor expansions with adaptive time-steps
- Accelerated by the Fast Multipole Algorithm
<table>
<thead>
<tr>
<th>OCEAN WAVE ENERGY : AN ASSET</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>WAVE ENERGY CONVERSION</strong></td>
</tr>
<tr>
<td>Oyster Aquamarine Power 2009</td>
</tr>
<tr>
<td>Oyster Aquamarine Power 2011</td>
</tr>
</tbody>
</table>

28 November 2011 - Tallinn
Aquamarine Power is a technology company that has developed a product called Oyster which produces electricity from ocean wave energy.

UCD and Aquamarine Power are collaborating to deliver the next-generation Oyster 800.
• Large mechanical ‘flap’ moves back and forth with motion of waves
• Two hydraulic pistons pump high pressure water via pipeline to shore
• Conventional hydroelectric generator located onshore
• Secured to seabed at depths of 8 – 16m
• Located near shore, typically 500 – 800m from shoreline
• **Oyster 1 Project** – 315kW demonstrator successfully installed and grid-connected at European Marine Energy Centre (EMEC) in Orkney, October 2009 – Spring 2011 (finished)

• **Oyster 2 Project** – 2.4MW project (3 Oyster 800 WEC) – on schedule for 2011 (1 Oyster 800 installed) / 2012 (two more to be installed)

• **Oyster 3 Project** – 10MW development on track – commissioning 2013 or 2014

• First commercial wave farm off the west coast of Ireland (Westwave project) – 2015
WHAT DOES MATHEMATICS BRING?
High end computational modeling for wave energy systems

1. Wave impact and pressure loads on a single Wave Energy Converter

2. Optimal device spacing for an array of Wave Energy Converters

3. Preferred geographical locations for near shore Wave Energy Converter sites in Ireland

4. Biofouling (biological growth on surfaces in contact with water)
• One of the most commonly acknowledged difficulties of conducting experiments with Wave Energy Converters: presence of scale effects (Reynolds much larger at full scale than at small scale – for example, at scale 1/40, viscous forces on the model are multiplied by a factor 253 if only Froude scaling is satisfied)

• This makes mathematical and numerical modelling a particularly valuable tool in the development of Wave Energy Converters
Wave impact and pressure loads on a single Wave Energy Converter

Fully nonlinear potential flow solver (for example WSIM – Boundary Element Method) combined with a Navier-Stokes solver (for example OpenFOAM – vortex shedding at the edges of the flap)

Alternative approach (dissipative surfaces)

Oyster 800 prototype is equipped with a number of pressure sensors
Fairly perfect fluid (joint work with X.B. Chen – Bureau Veritas)

Fluid domain with a body and **dissipative surfaces**
WHICH TOOLS?

High end computational modeling for wave energy systems

Optimal device spacing for an array of Wave Energy Converters

Analytical approach for the time being
POTENTIAL FLOW THEORY: $\nabla^2 \Phi = 0$

Wave train incident upon a moving structure

$\Phi(x, y, z, t) = \Phi^S(x, y, z, t) + \Phi^R(x, y, z, t)$

Scattered waves  Radiated waves
Free-surface elevation front side

Free-surface elevation back side

Reflected waves

Free-surface density plot

$T = 7 \text{ s}$
3D LINEAR THEORY

Scattering problem – Validation

- Comparison with numerical model (WAMIT data by J. van’t Hoff) very satisfactory (time mesh should be refined)
- Thin-plate approximation validated (good also for thicker plate)

OWSC width 18 m
Dist. btm-hinge 1.5 m
Water depth 10.9 m
Ampl. inc. wave 0.3 m
3D LINEAR THEORY

Scattering problem - Discussion

For a given modal order \( m \), the modal interaction factor is max when

\[
\nu = \nu_m = \frac{b}{2m}
\]

Resonant width for mode \( m \)

<table>
<thead>
<tr>
<th>Mode</th>
<th>Resonant width (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>45.8</td>
</tr>
<tr>
<td>2</td>
<td>22.9</td>
</tr>
<tr>
<td>3</td>
<td>15.27</td>
</tr>
<tr>
<td>4</td>
<td>11.45</td>
</tr>
<tr>
<td>5</td>
<td>9.16</td>
</tr>
<tr>
<td>6</td>
<td>7.6</td>
</tr>
</tbody>
</table>

(Too large!)

18 m Oyster

12 m Oyster

6 m Oyster

\[ \bullet \text{4 November 2011 - Tallinn} \]
3D LINEAR THEORY

Scattering problem  - Discussion

An optimum width (in this channel!) for which the Oyster interacts mostly with the first two sloshing modes:

\[ w_{opt} \approx \frac{46 + 23}{2} \text{ m} \approx 34 \text{ m} \]

Constructive interference of sloshing waves increases the 3D performance

Large range of periods (7 s - 11 s) for which the torque is close to max value
3D LINEAR THEORY

Scattering problem – Discussion

In open ocean this means:
Preferred geographical locations for near shore Wave Energy Converter sites in Ireland

Variety of spectral wave models available to scientists and engineers, including SWAN and WAVEWATCH III™ (one solves the random phase spectral action density balance equation for wavenumber-direction spectra). Is it enough for wave prediction or is it necessary to couple such spectral wave models with shallow-water type models in very shallow water?
Biofouling (biological growth on surfaces in contact with water)
WAVE POTENTIAL AROUND THE WORLD

Numbers show the available power in kW/m (Source: World Energy Council)

28 November 2011 - Tallinn
Source: Kelp data: Kelp forest ecosystems - biodiversity, stability, resilience and future, Robert S. Steneck, Michael H. Graham et al.

Temperate Shelf and Seas
seasonal variability + freshwater influx from coastal streams and tidal action
+ very heterogeneous habitats
= high diversity of organisms: algae, invertebrate, fish, marine mammals, ...
A tidal device which spent one year in water on Ouessant island in Brittany ...

Modeling of growth done in collaboration with E. Reynaud (biologist at UCD) and C. Pinck

work in progress
EXPECTED SETTLEMENT ON THE FLAP

Some indications obtained from videos during cleaning operations

Mechanical effects on the flap
Flaps might be simple devices from the engineering point of view. But they are really challenging from the fluid mechanics and numerical simulation points of view.

- Intermediate water depth (kh of order unity)
- 3D problem
- Neither laminar flow nor potential flow (vortex shedding)
- Moving solid boundary (large motion) + free surface
- Intermediate size structure (no simplification)
- Nonlinear waves
- Coupling with biological growth
THANK YOU FOR YOUR ATTENTION

Howth, Ireland

28 November 2011 - Tallinn