# EXTREME WAVE LOADING ON OFFSHORE WAVE ENERGY DEVICES USING CFD: A HIERARCHICAL TEAM APPROACH

**D.M. Greaves [1], C.J.K Williams[1], D.M. Causon[2], C.G. Mingham[2],**
**P.K. Stansby[3], D.R. Laurence[3], P.H. Taylor[4]**

*1 Department of Architecture and Civil Engineering, University of Bath*
*2 Department of Computing and Mathematics, the Manchester Metropolitan University*
*3 School of Mechanical, Aerospace and Civil Engineering, University of Manchester*
*4 Department of Engineering Science, University of Oxford*

## PART 1: PREVIOUS RESEARCH AND TRACK RECORD

### The Academic Team

The investigators have extensive experience of the prediction of wave hydrodynamics, the modelling of coastal processes, and Computational Fluid Dynamics (CFD).

### Summary of Recent Research Activities and Specific Expertise

The CFD Research Group at the University of **Bath** is concerned with developing numerical techniques for modelling viscous and inviscid fluid flow problems with the use of adapting hierarchical meshes [see e.g. 1-4]. Applications currently under consideration include: free surface waves; blood flow in flexible vessels; separated flow and vortex shedding; wind interaction with flexible roof structures. Funding for this work has been provided by the EPSRC and The Royal Society. The **MMU** group has been developing CFD methods for aerospace, process and hydraulics applications for over twenty five years. Applications include engine surge in twin aero-engine configurations, blast waves, underwater shock waves, lithtotripsy devices, river/coastal hydrodynamics including wave diffraction and overtopping at sea walls and wave power devices. The AMAZON suite of CFD codes has been developed using advanced Riemann-based finite volume methods on Cartesian cut cell meshes [5-9] with funding from EPSRC and EU. There is considerable CFD activity at the University of **Manchester** on aerodynamics and heat transfer as well as environmental flows with Research Council and industry funding. CFD vendors STAR-CD/COMET, CFX and Fluent provide their codes to enable the latest research on various aspects to be incorporated. Research on Smoothed Particle Hydrodynamics (SPH) of relevance to this project is supported by Electricité de France as well as EPSRC. Recent work in **Oxford** has concentrated on the mechanics of extreme waves, both direct numerical simulation and analysis of field data which demonstrates that NewWave is a suitable model for extreme wave events, and is supported by Shell as well as EPSRC.

**Deborah Greaves** was appointed as a Royal Society University Research Fellow in the Department of Architecture and Civil Engineering at the University of Bath in October 2000. Before that, she was a lecturer in the Department of Mechanical Engineering at UCL for five years after completing her DPhil at Oxford University. Her research interests are in adaptive hierarchical mesh generation for viscous and inviscid fluid flow modelling and she has published over 20 articles in refereed journals and international conference proceedings. She was a co-investigator on EPSRC Grant GR/L24083, is a Chartered Engineer, a member of the Peer-review College of the EPSRC and a reviewer for several journals.

**Derek Causon** is Professor of Computational Fluid Dynamics and Head of the Department of Computing and Mathematics at MMU. His early research interests were in shock waves and blast waves in aerospace and process engineering. Since 1995 he has been involved in free surface flows using shallow water and surface capturing finite volume methods and adaptive Cartesian cut cell techniques for applications to sub/trans and supercritical hydraulic flows including wave attack on coastal structures wave power. He was PI on related EPSRC grants GR/N2416 (VOWS project) and GR/M42428 on free surface capturing methods and is Co-I on current projects GR/S12333 and GR/T18622 on wave power and wave overtopping. He has 30 years experience in CFD and has published over 85 articles in refereed journals and international conference proceedings. He has been a member of several EPSRC Steering Groups, is a member of the EPSRC College of Peers and a reviewer for several journals.

**Peter Stansby** is Professor of Hydrodynamics at Manchester. His early research interests were in wave and current loading on offshore structures, including vortex-induced vibrations. Since around 1990 he has also been involved with coastal hydrodynamics, mainly in relation to shallow-water flows and surf zone waves. In recent years he has become involved with wave energy devices, developing a particular robust point absorber, the ‘Manchester Bobber’, for commercial exploitation. This has yet to be publicized and its development is being supported by the Carbon Trust. He has been an investigator on over 30 grants and contracts, mainly from EPSRC. He is an associate editor for the ASCE Journal of Hydraulic Engineering and on the editorial board of Applied Ocean Research. He is a Fellow of the Royal

Academy of Engineering and the Institution of Civil Engineers. He has published more than 65 papers in international refereed journals.

**Dominique Laurence** is Professor of Computational Fluid Dynamics (CFD) and Head of the CFD group. For 20 years he has been the turbulence expert at Laboratoire National d'Hydraulique (Paris) and lectured in hydraulics, environmental flows and turbulence at Ecole Nationale des Ponts et Chaussees, the "grande Ecole" (leading University) for Civil Engineering. Although he spent most of his career in Industry (Electricite de France), his contribution there has been essentially scientific research in CFD and turbulence modelling. His University research has involved 8 PhD students with funding from two Aerospace-CFD EU projects, a TCS scheme with CD Ltd (STAR-CD/COMET software) for two RAs on large eddy simulation (LES), and further LES contracts from British Energy, EDF, French Rail. He has visiting professorships at the Technical University of Delft, Kyoto University, Tokyo University and several invitations at Stanford University. He manages activity in www.ercoftac.org, the leading European organisation in Flow Turbulence and Combustion .

**Clive Mingham** is Reader in Hydroinformatics in the Department of Computing and Mathematics at MMU. Since joining MMU from industry in 1989 his research interests relate to both structured and Cartesian cut cell shallow water, Boussinesq and Navier Stokes models for coastal and estuarine flows including finite volume, interface capturing, spectral and Lattice Boltzmann methods. He has over 40 publications in refereed journals and international conference proceedings. He was Co-I on related EPSRC grants GR/N24162, GR/M42428, currently PI on GR/S12333 on numerical modelling of a novel wave power device and Co-I on wave overtopping grant GR/T18622. He is an EPSRC College of Peers member, a reviewer for Science Foundation Ireland and referees for several academic journals.

**Paul Taylor** is a University Lecturer in Engineering Science at Oxford. Appointed in 1997 he was previously a Senior Research Engineer with Shell for 17 years. He has worked extensively on water wave theory and the analysis of field measurements, the statistics of waves and fluid loading on space-frame structures and wave scattering from large surface-piercing columns, and has published widely. He was the principal investigator on grants GR/M98814 and GR/N22595.

**Chris Williams** is a senior lecturer in structures and numerical analysis in the Department of Architecture and Civil Engineering at the University of Bath. His main research interests are the analysis of fabric and cable net structures in wind. He has written software for a number of engineering and architectural practices, recent projects include the British Museum Great Court Roof and the Millennium Dome, on which he was consulted regarding the aeroelastic stability of the fabric.

**Project Partners**

David Pizer, OPD, Rod Rainey, WS Atkins, Demetris Clerides, adapco (STAR-CD/COMET), Ian Jones, ansys (CFX). The project partners are all making significant contributions to the project and together with the Investigators will form the Management Group.

## PART 2: DESCRIPTION OF THE PROPOSED RESEARCH AND ITS CONTEXT

### A: Background

Renewable energy is now recognised to be a key component of UK energy policy. While wind turbines are an established technology and tidal stream turbine prototypes are being tested, *offshore* wave energy provides a concentrated source of renewable energy which can, in principle, make a substantial contribution to the UK consumption of 30-40GW in average conditions. Numerous devices have been proposed and, after the need to generate energy at competitive economic rates in low-average sea states, a further crucial design consideration is the need to assess each device for survivability in extreme waves. Here we intend to apply three different CFD approaches through research students in Groups with appropriate expertise. The students and their supervisors/investigators will meet regularly with the management board involving industry supporters to engender a supportive team ethic. We consider two promising classes of device.

The Ocean Power Delivery device **Pelamis** is a freely floating device, composed of four tubular cylindrical sections of 3.5m diameter connected by hinges, having a total length of 150m. Pelamis naturally heads into the oncoming waves and the sections move relative to one another as the wave propagates along its length. The energy is extracted via hydraulic rams at the hinges, which drive electricity generators. The Prototype Pelamis has been installed at the European Marine Energy Centre in Orkney and successfully generated electricity into the grid. Three more machines are currently being constructed for export to Portugal (8m Euro) to take advantage of their premium tariff of 0.24 euro/kWh. As was demonstrated by the wind power industry, significant cost reductions are anticipated for the Pelamis as further technical advances and economies of scale take effect.

The Manchester **Bobber** is a heaving point absorber comprising a float with hemispherical base generating oscillatory shaft motion which is converted to unidirectional rotation through a freewheel/clutch which in turn drives an electricity generator. This is a particularly robust system with only the float in contact with water and all the electro-mechanical components housed on a structure above water. The intention is that a wave farm would consist of multiple devices generating at least 10MW in low-medium sea states. The device has been studied experimentally in the laboratory, verifying a single-degree-of-freedom mathematical model of its behaviour [10, 11] and at intermediate (1/10th) scale in September 2005 at NaREC in Blyth. Both devices take advantage of resonance of the mechanical system, tuning their natural frequency to the wave frequency to amplify power output in low-medium seas. The Bobber represents a generic class of floating buoy device; other similar devices that the proposed technical developments would be directly transferable to include AquaBuOY (AquaEnergy Group, Ltd), WaveBob (Clearpower Technology) and the Seavolt Technologies point absorber.

At present, the effect of extreme waves must be evaluated through physical experimentation and sea trials. Linear, or second-order, wave diffraction theory is the standard approach for such structures of large dimensions and does not (by definition) represent highly nonlinear effects associated with extreme waves. The alternative approach to be developed in this project is through CFD (computational fluid dynamics) solving the full Navier-Stokes and continuity equations. This has had many successes in other areas of engineering, e.g. turbomachinery, aeronautics and combustion; however, the problem inevitably pushes the limits of the methodology and computer processing power requirements. The problem will be addressed in three ways: via commercial codes, where there has been considerable recent investment in free-surface problems; by recent advanced surface-capturing codes developed at MMU; and, by the novel SPH (smoothed particle hydrodynamics) method being developed at Manchester which is a fully Lagrangian, grid-less method well suited to highly accelerating, distorted free-surface flows.

The benefits of CFD are well known. Design cases may be set up and analysed in relatively short times. The input conditions, particularly important for extreme waves, may be precisely controlled. A full set of flow variables can be output, e.g. pressure and velocity fields, as well as integrated effects like forces and response which are usually the only output from physical experiments. However, a proper assessment must be made of the accuracy of the predictions, numerical convergence and computational efficiency. In general, 3-D CFD calculations require high performance computing (HPC) involving massively parallel processing. Parallel versions of the commercial codes STAR-CD/COMET and CFX are available; the MMU code is currently being parallelised by Manchester Computing (MC) on EPSRC Grant GR/T18622; the SPH methodology is less mature but parallelisation on PC clusters is in hand and there are discussions about using HPC with MC who have experience in techniques and algorithms for the parallelisation of particle-based methods in the field of molecular dynamics (see http://www.fisica.uniud.it/~ercolessi/md/).

In order to assess the accuracy of the CFD predictions, the basic requirement is a reliable set of experimental data for comparison. Data will be available for both the Bobber and Pelamis, as well as bench-mark test cases with fixed bodies (Table 2). All predictions can be adversely affected by inadequate numerical convergence or failure to represent important physical processes. These two requirements are in opposition as resolving detailed flow characteristics generally requires mesh refinement which in turn requires greater computer resource. For this reason, we propose the following *hierarchy* of increasing physical complexity:

- An inviscid single fluid approach (with slip solid boundary conditions). This will represent inertia effects only which will dominate in high accelerating flows associated with steep waves.
- A viscous single fluid approach (with no-slip solid boundary conditions or wall functions and an appropriate turbulence model). Thus boundary layers will be resolved giving skin friction.
- A two-fluid (water/air) viscous approach which will allow air entrainment in breaking waves, potentially important for slam forces.

In turn a *hierarchy* of increasing computational novelty will be applied:

- Commercial codes based on the Volume of Fluid (VoF) method.
- Surface-capturing adaptive multi-fluid Godunov type codes from MMU.
- Fully Lagrangian SPH methods from UoM,

whilst this project draws upon data from two classes of wave energy device with significant dynamic response, which, due to their small size are particularly sensitive to wave nonlinearity. The resulting CFD methodology will benefit analysis of extreme wave interaction with other devices, ships, other marine vehicles and structures in general. For example, interaction with freak waves and the ‘green’ water problem have yet to be fully resolved.

## Programme and Methodology for SPH

In SPH a fluid is represented by a large number of particles with force interactions defined by a kernel function chosen to enable the particles to behave in a statistical sense representing the continuum defined by the Navier-Stokes and continuity equations. The method is grid-less, wholly Lagrangian, and is thus well suited to problems with distorted, in the extreme disconnecting, free surfaces avoiding the numerical stability problems of some continuum methods. In most approaches to date, pressure is determined from an equation of state for a weakly compressible fluid. The method is well suited to flows with weak viscous effects, dominated by inertial effects. Because of its flexibility for distorted free surfaces it has been successfully applied to surf zone waves which break and splash up and sometimes collide, essentially treating the flow as a mass and momentum transfer process [14].

A particularly attractive aspect of SPH is that it is very simple to program. The downside is that many particles are needed for accurate simulation, e.g. $10^4 – 10^7$. Particular developments in a current EPSRC grant GR/S28310 are to incorporate viscous and turbulence effects effectively and also to provide a formally incompressible approach, both having been given preliminary analysis in Issa’s PhD thesis [15]. It should be mentioned that SPH in both weakly compressible and incompressible forms is advancing rapidly, producing encouraging results for violent sloshing, slamming and ship motion, for example in the recent Workshop on Water Waves and Floating Bodies, Spitzbergen 2005. We have found a particular problem in the context of body forces that pressure is very noisy with the weakly compressible approach but this is much reduced in the incompressible formulation. There is a need to optimise the methodology for simple well known flows around square and circular cylinders.

SPH is thus potentially well suited to simulate dynamic body motion in steep or extreme waves. Its motion is determined from the pressure distribution on its wetted surface and its overall dynamic characteristics. Following progress on turbulence modelling on the present SPH Grant, a no-slip body condition or wall functions will be incorporated with turbulence modelling. Finally treatment as a two-phase air/water will be attempted. In this hierarchy the risk is low for the inviscid problem and increases for the viscous cases.