SYNTHESIS REPORT

FOR PUBLICATION

CONTRACT N°: BREU-499

PROJECT BE-4354

TITLE: Experimental and Numerical Analysis of Sloshing and Impact Loads {EUROSLOSH)

PROJECT COORDINATOR: M. Dogliani (RINA)

PARTNERS:
Registro Italiano Navale (RINA)
British Aerospace (BAe)
Bureau Veritas (BV)
CETENA
SIREHNA

STARTING DATE: 1/11/91 DURATION: 40 MONTHS

PROJECT FUNDED BY THE EUROPEAN COMMUNITY UNDER THE BRITE/EURAM PROGRAMME

DATE: 26/04/1995

Experimental and Numerical Analysis of Sloshing and Impact loads

G. Casella Registro Italiano Navale (RINA) - Via Corsica 12-16128 Geneva - ITALY
J.P. Borleteau SIREHNA - Immeuble Atlanpole - Rue de la Noe 1-44071 Nantes Cedex 03 - FRANCE
R.H. Hughes British Aerospace Sowerby Research centre (BAe) - FPC 267 P.O. BOX 5- Filton Bristol BS127QW- UK
J.C. Le Hire Bureau Veritas (BV) - Rue Jacques Daguerre - Rueil Malmaison 92563- FRANCE
L. Sebastian CETENA - Via Savona - 16100- Geneva - ITALY

Abstract

This report presents the work done inside the BRITE/EURAM Project 4354: "Experimental and Numerical Analysis of Sloshing and Impact Loads", which started on 1st November 1991 and finished on 28th February 1995, The objectives of the project, as well as the strategy adopted to achieve them, are here illustrated, focusing on the technical results of the project.

Objective of the project was the study of sloshing through the definition, improvement and application of up-to date experimental techniques and sophisticated numerical tools for computational fluid dynamics (CFD).

The development and validation of an hydrodynamic software for the simulation of sloshing, the application of numerical and experimental techniques to a large selection of theoretically and engineering valid cases, as well as the incorporation of these techniques into an integrated multi-sectorial procedure for sloshing analysis, may be considered as the main results of the project.

Introduction

Liquid sloshing can be defined as "The large amplitude liquid response to external excitations which occur for partially filled tanks set in motion" /1/. Despite its resonant behaviour, the physics of sloshing is essentially of stochastic-chaotic nature. This nature, widely experimentally demonstrated, is implicit in the fluid-dynamics of turbulent liquid motions; in-fact, even if the sloshing response to an harmonic excitation appears to be macroscopically periodic, it is impossible to obtain two identical realisations of a sloshing event, even in a single slosh period duration, either by model tests or by numerical simulation.

Sloshing is a problem of practical interest for safety and economy in several industrial fields, such as sea (ships), in-land (trucks, trains) and aerospace (aircraft and space vehicles) transportation and civil engineering {nuclear plants, dams). The prediction of the sloshing induced loads on the tank walls and structural elements (in terms of both slowly-varying and impact pressure} and the determination of the most unfavorable conditions for the sloshing occurrence is very important for engineering purposes.

When dealing with sloshing, either experimental or sophisticated numerical analyses are used which need to be kept up to date with the developments in the experimental equipment and techniques as well as in computational fluid dynamics (CFD) theory and algorithms. The scope of the project was to incorporate these technologies into a multi-sectorial procedure for sloshing analysis. To this end, the following objectives have been considered:

- define the physical modelling, from the basic hydrodynamic equations (Reynolds Averaged Navier-Stokes), identifying the boundary conditions and investigating the physical parameters which influence the phenomenon;
- develop a fully-integrated software code for the time-domain simulation of sloshing, in terms of fluid pressure and velocity fields, free-surface orientation, impact loads;
- perform a set of experiments, highlighting the influence of physical parameters, aimed at validating the code, as well as at creating an experimental data-base for engineering applications;
- to perform relevant applications to cases of practical interest in the different engineering areas and compare the results of numerical calculations to test results for the purpose of validating the numerical procedure.

Besides the above objectives, two additional and very innovative goals of the project are worth being underlined:

- study the role of fluid-structure interaction in sloshing phenomenon and give the software the capacity to handle this effect;
- provide detailed guidelines explaining scopes, means and approach for a sloshing analysis to be made, as well as use of the results for a structural assessment.

These objectives and their implication in the different industrial fields have been investigated within the EUROSLOSH project by multi-disciplinary consortium, whose composition is summarised in Table 1. As it is clear from the scopes of the project, main emphasis was given to the experimental activity (which was carried out by SIREHNA) and to software development (which was mainly covered by British Aerospace) while theoretical development (addressed by CETENA) deserved a comparatively small effort. This is illustrated by the pie-chart in Figure 1 where each partner share of the total manpower of 200 man months is shown,

Partner number, Organisation Name and acronym (Country)	Activity	R&D Function in the Project
1 Registro italiano Navale - RINA (IT)	Classification society	Development of fluid/structure interaction, validation of software, development of methodology for extreme value assessment, development of guidelines (*general* and numerical aspects). Co-ordinator; end-user (pm-normative aspects).
2 British Aerospace - BAe {UK)	Aerospace *industry*	CFD expert, development of software (hydrodynamic aspects), development of visualisation suite, software debugging and validation. Partner; end-user (industrial applications in the aerospace field}.
3 Bureau Veritas - BV (FR}	Classification Society	Development of design scenario, development of pre and post processing, software validation, development of guidelines (experimental aspects). Partner; end user (pre-normative aspects).
4 CETENA - CETE (IT)	Ship Designer	Theoretical aspects for both hydrodynamics and fluid/structure interaction. Partner; end-user (ship design aspects).
5 SIREHNA - SIRE (FR)	Contract Research Organisation	Expert on model testing and experimental activity; validation of software. Partner; end user (experimental techniques].

Table 1- Role of the partnership

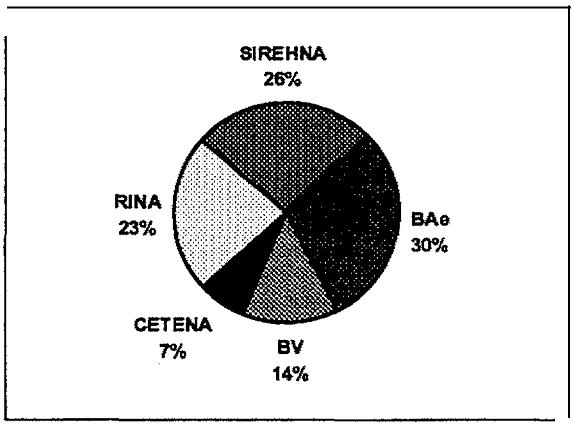

Figure 1- Man-power share

Sloshing experimental activity

Sloshing model tests resulted as a major activity in the project and one of the widest and most complete set of sloshing experiments ever made, ranging from experiments for scientific investigations, code validation and data collection on cases of high technical relevance in four different engineering fields.

Nine different models were tested by means of a Planar Motion Generator in SIREHNA laboratory. The characteristics of each model, and the scope of the relevant investigations are summarised in Table 2.

Model #	Shape	Dimensions (B x H x D - [m])	Scope	Notes
1	parallelepiped	0.8 x 0.4X 0.4	scientific investigation	*reference" model
2	"	0.8 x 0.4 x 0.4	·	flexible walls
3	"	0.4 x 0.2X 0.2	·	half scale with model 1
4	"	0.8 x 0.4x 0.4	·	distribution of spatial pressure
5	·	0.8 x 0.4 x 0.4	·	35 effects
6	fore tank of LNG earner ship	/	engineering interest	scaled 1:49 with respect to prototype
7	airbus A300 wing tank	/	·	scaled 1:5 with respect to prototype
8	cross section of tank trailer	∅ 0.6m, width 0.2m	·	also used in code validation
9	mid tank of double shell VLCC tanker ship	/	"	scaled 1:50 with respect to prototype, deck girders

Table 2- EUROSLOSH models

Tests for scientific investigation and code validation

Data collecting during these test series, such *as* peak pressure statistics, impact pressure time-histories, free surface location, wall acceleration, displacements and strains, synchronised with tank motion were processed to create a data base. Numerous visualizations of the tests performed with both normal and high speed video systems were collected on video tapes. The contents of this experimental database, having been developed on simply shaped structures, are particularly useful to get insight into the sloshing induced phenomena and for validation and testing of hydrodynamic software for sloshing **numerical simulation.**

In **order to have a well-defined basis** for comparisons, a "reference" test configuration was initially selected as Model 1 (see Figure 2):

- parallelepipeds tank 0.8x 0.4x 0.4 m,
- filling level 60% of tank height,
- harmonic horizontal excitation at the fluid resonance frequency.

The influence on sloshing of a number of physical parameter was assessed and some open questions were addressed leading to the following conclusions based on flow observations and on statistical values of peak pressure:

- the first natural sloshing frequency in rectangular tanks is correctly predicted by the 2D linear theory (discrepancies less than 1% between theoretical and experimental resonant frequencies for filling level equal to the 60% of tank height),
- irregular motions, having a triangular mean square density spectrum centred **on resonant frequency,** generate impacts which are less numerous but of higher magnitude than equivalent (in a mean square sense) sinusoidal motions,
- a particular gyration axis location giving maximum pressure loads appears to exist, placed immediately below the free surface level,

- an important growth of peak pressure value (up to 400%) is detected as the ullage pressure is reduced from atmospheric value down to 35 mbar,
- increasing the kinematic viscosity, a small decrease in the first natural frequency may be observed (about 1%, considering a mixture of water and glycerol),
- baffles presence and location have an important effect on sloshing behaviour,
- the Froude scaling law appears to be appropriate for first natural frequency evaluation,
- the extrapolation of pressure values from small scale to a larger scale using Froude scale-law appears to be conservative.

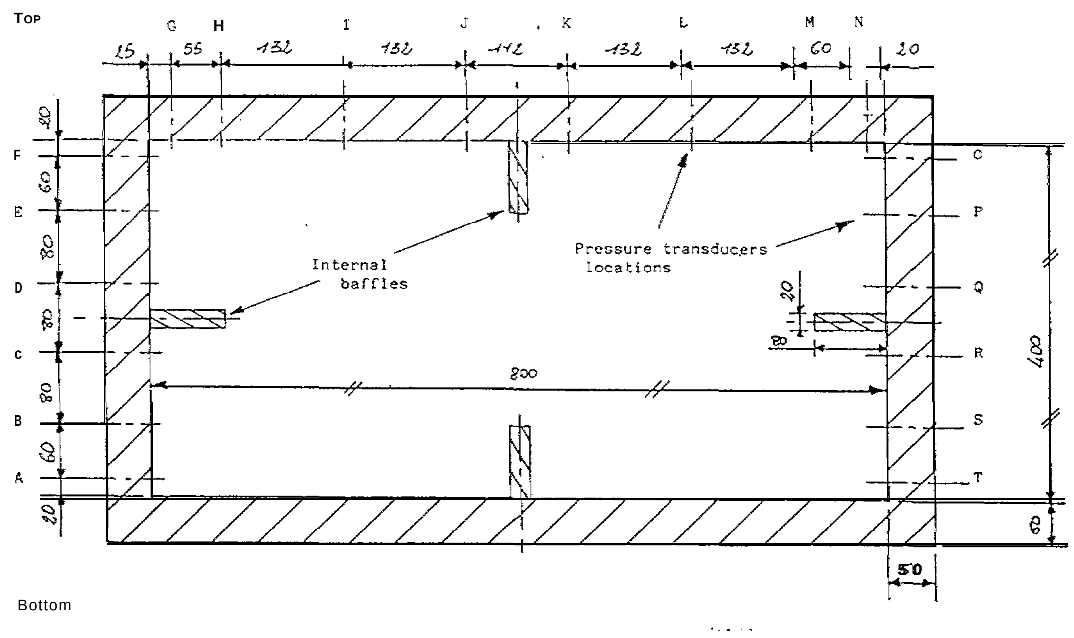

Figure 2- The "reference" model (Model 1)

Particular scientific and practical interest was reflected by the experiments aimed at investigating the fluid-structure interaction, as in practice there is no availability of such results in literature. Two flexible bulkheads consisting in rectangular plates of different thickness and material (in the following the “rigid” and the “flexible” plates} were alternately mounted on the “reference” model as shown in Figure 3. The pressure loads (on the plates and on the other walls of the tank) and the accelerations of the plates were measured; in addition, the displacements and the strains of the more flexible plate were measured together with the motion of the tank.

The following conclusions may be drawn from these tests:

- the flow inside the tank was not significantly altered by the presence of the rigid plate, nevertheless, on the plate the pressure loads were lower than the ones obtained with fully rigid condition in the “reference” test series;
- no important displacements of the rigid plate were observed under pressure peaks, with typical accelerations on the order of 4 g. Yet these accelerations measured at particular points reasonably well related to the pressure load recorded at the same locations.
- an important fluid-structure interaction took place with the flexible plate; in particular, large differences, compared to the first results, were observed in the pressure loads measured on the plate and on the other walls of the tank.

As an example of the level of sophistication achieved in the experimental activity within the project, the tests for the determination of the space-time distribution of impact pressures may be considered. The time traces of the pressure peaks, in the 2D reference experimental conditions, were simultaneously recorded in 12 locations by piezo transducers at the sampling rate of 200 kHz (see Figure 4) to obtain a map of the space-time distribution of the peaks of the pressure in a localised sloshing impact (see Figure 5).

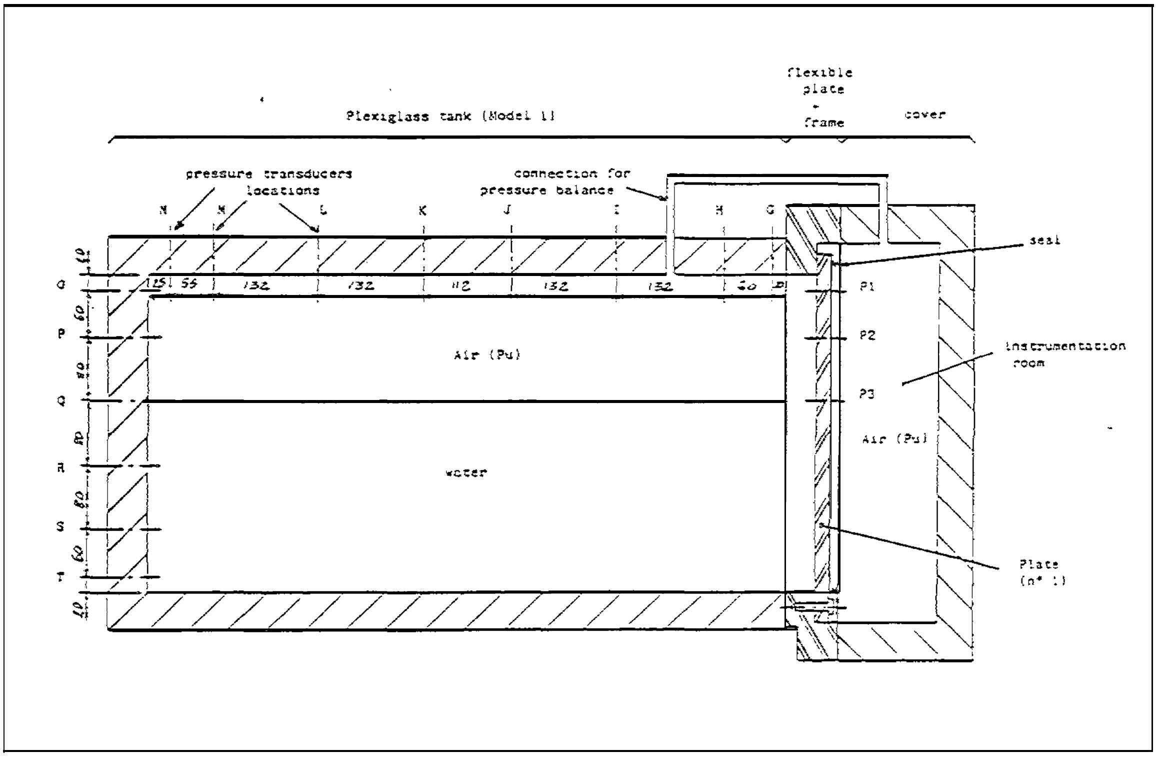

Figure 3 - Middle plane section of Model 2 used for wall flexibility influence analysis

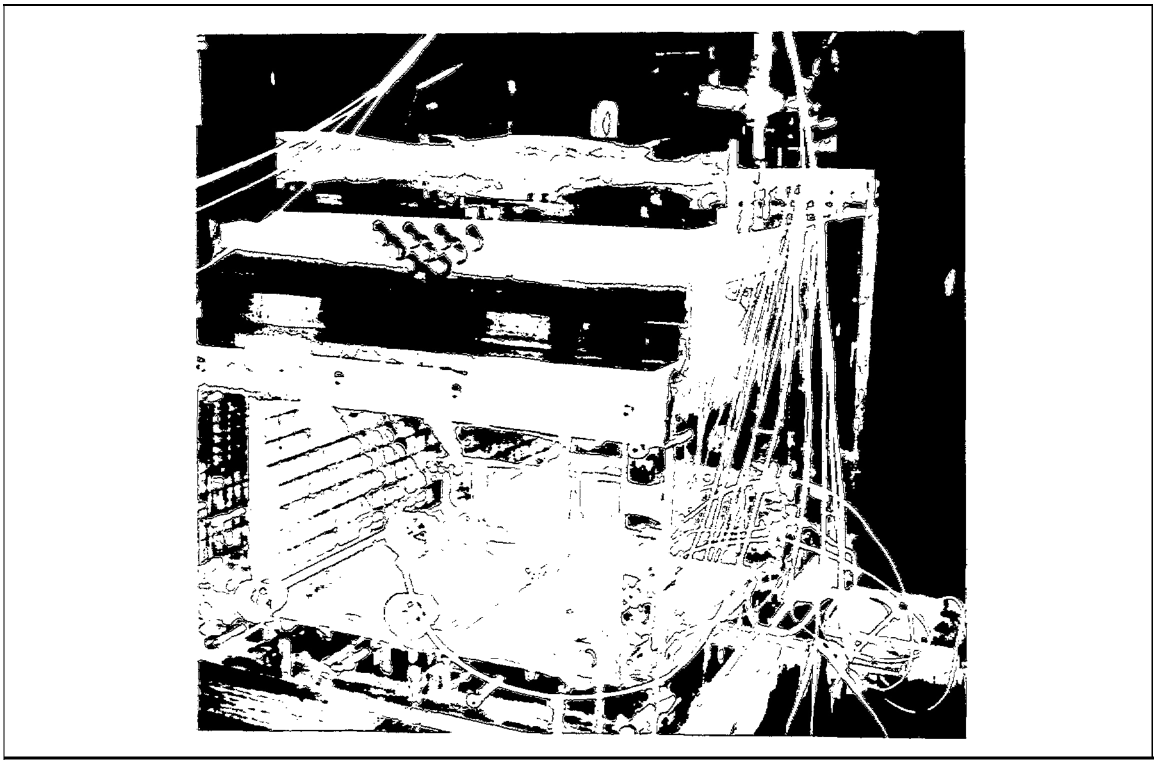

Figure 4 - View of Model 4 and space distribution of the pressure probes

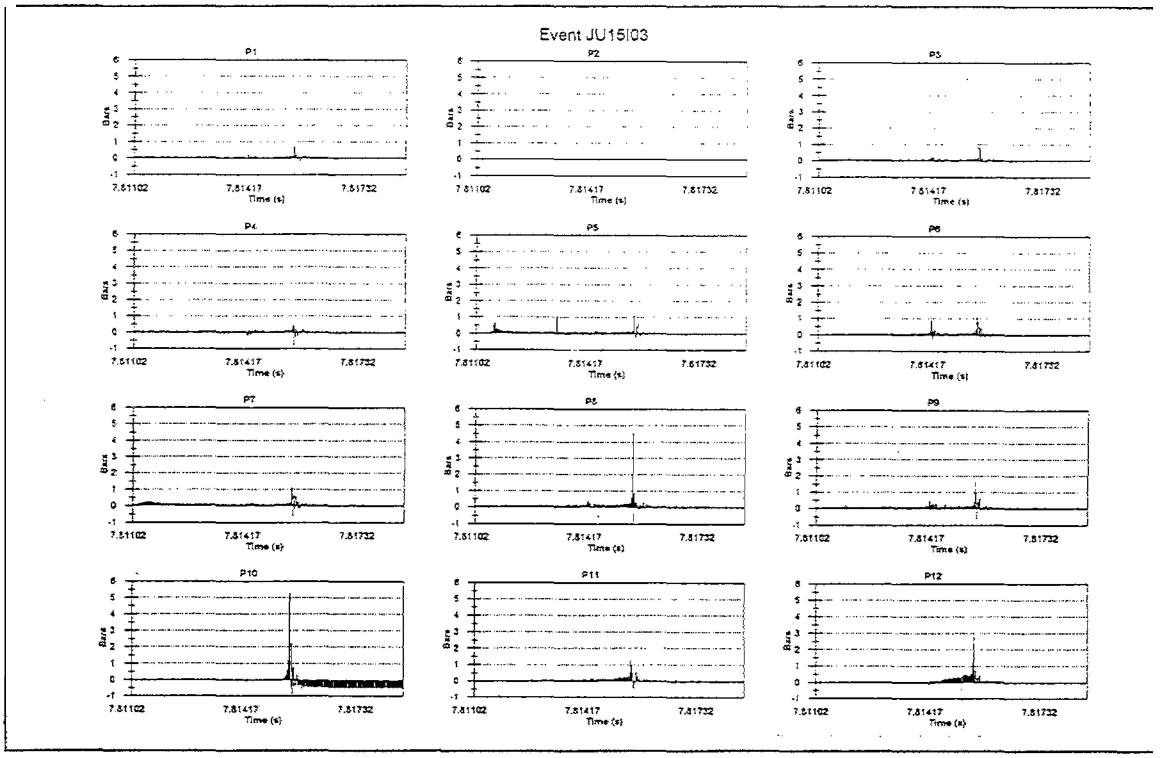

Figure 5- Pressure time traces in Model 4 experiments

Tests for engineering applications

As regards the engineering applications, the objective of this part of the project was to collect data concerning sloshing induced phenomena for four engineering test cases corresponding to Marine (Models 6 and 9), Aerospace (Model 7) and Automotive (Model 8) transportation.

Model 6: this model reproduced, at a scale 1:49, a 3D shaped LNG fore-tank, designed by Chantiers de l'Atlantique (CdA) (see Figure 6). It was tested at three different filling levels (90% and 80% of tank internal height, 10% of tank internal length). The model was subject to excitation in a vertical plane as combination of either surge+ heave+ pitch or sway+ heave+ roll, The motion time-series were derived from mean square density spectra of the ship motions, evaluated for different sea-states and ship-wave incidence in order to have a peak frequency as close as possible to the tank resonant frequency in the different test conditions.

Measurements consisted in pressure peaks detection using five piezo transducers flush-mounted in the tank walls connected to the peak detector. The locations of the transducers were chosen with respect to the filling level, For each configuration, data collection consisted in statistical results {number N of peaks detected, average of the 10, N/10 and N/3 highest peaks) and in a list of pressure peaks. For the most interesting configurations, further analyses were performed for best fitting with distribution laws, and long duration simulations (40 minutes) were also conducted for investigations on the statistical stability of results.

The maximum pressure loads were detected for the filling ratio of 90% with surge+heave+pitch motions, and 10% for sway+ heave+roll motions. For the former, they were measured on the ceiling close to the forward bulkhead, and for the latter, in a comer close to the aft bulkhead and in the vicinity of the still free surface level.

Model 7: the purpose of tests on Model 7 was to collect data on fuel motions in a moving aircraft wing tank, for the experiments performed by BRITISH AEROSPACE until then only dealt with static tanks. A model of A300 wing tank at scale 1/5, provided by BRITISH AEROSPACE (see Figure 7) was partially filled with water (five levels: 90%, 80%, 60%, 50% and 40% of the tank capacity} and was subject to three realistic wing motions (roll up, roll down, pitch) of five to ten minutes duration. Because of the overall dimensions of the tank (about 4 m long), the model was mounted below the Planar Motion Generator which was itself installed on a rig above a basin.

In order to study liquid motions inside the tank, visualizations of the liquid were performed over the entire model by means of two video cameras fixed with respect to the tank. A special care was put in the lighting and shooting techniques to visualise the trajectories of air pockets

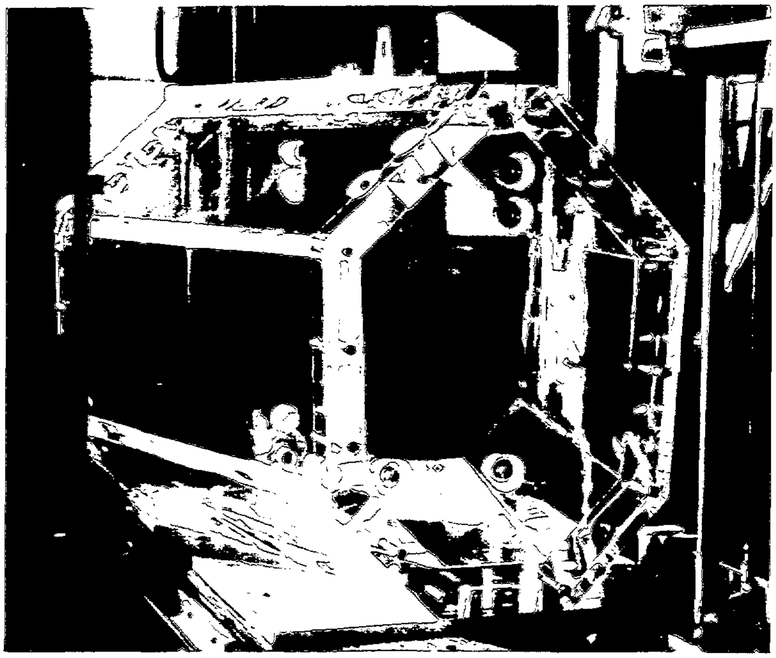

Figure 6 - Model 6 (LNG fore-tank by CdA)

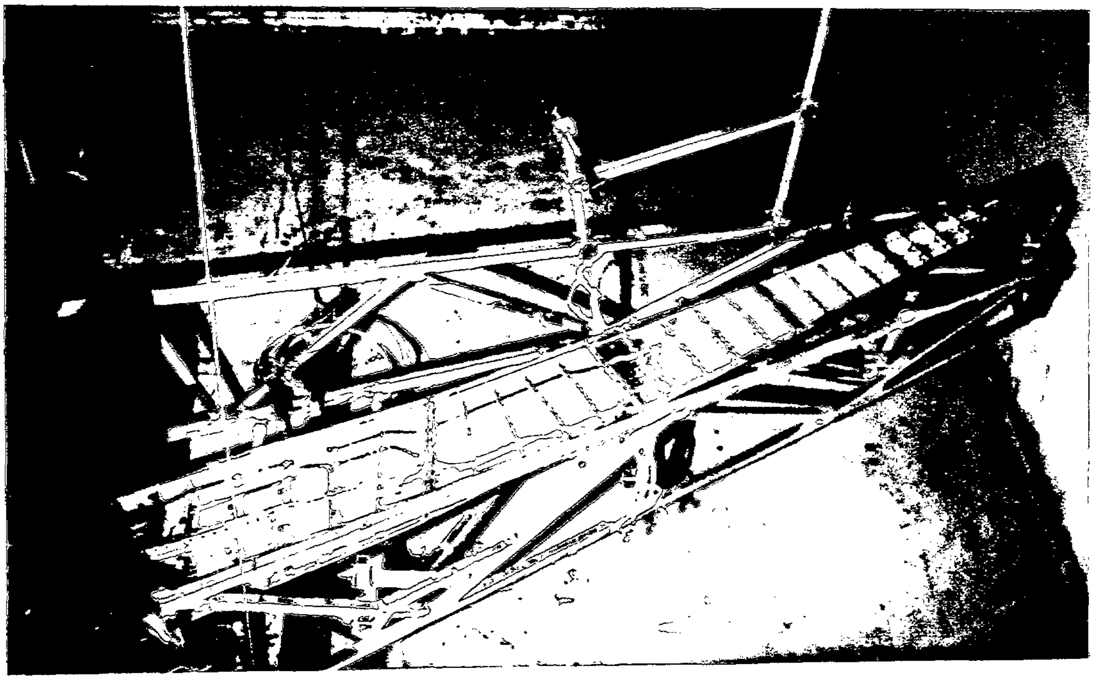

Figure 7 - Model 7 (aircraft wing tank, by BAe)

Model 8: this model was related to automotive applications and had the shape of a flat circular cylinder (diameter: 0.6 m, width: 0.2 m) to model a cross section of a tank trailer. It was tested at three different filling levels (20%, 50% and 80% of the diameter), subject to transverse horizontal motions.

Global forces on the tank (horizontal/vertical force, moment in the vertical plane of motion) as well as dynamic pressures at five non-impact locations were measured during the tests. Free surface visualizations were also recorded. Some special tests on Model 8 were addressed to the determination of the damping curve of the fluid motion with the tank at rest after an initial excitation, in order to collect data for the evaluation of numerical dissipation in the validation of the hydrodynamic code.

Model 9: this model reproduced, at a scale 1/33, a double hull VLCC ship tank. Six removable baffles, reproducing the deck girders, were mounted below the tank ceiling (see Figure 8).

Measurements consisted in pressure time-histories recorded at five different locations, with a sampling rate of 200 Hz. Simultaneously, the peak detector was used, which provided statistical values on the pressure peaks. Besides, the most significant tests were video recorded.

The tests showed that the number of peaks detected was larger at 85% than 60% filling, with similar values for the maximum peaks. The main difference was in the locations of the occurrence of the maximum peaks, on the ceiling, close to the side baffles, for 85% level and on the side walls for the 60%. Besides,' larger values of the pressure peaks generally appeared in the tests with the baffles.

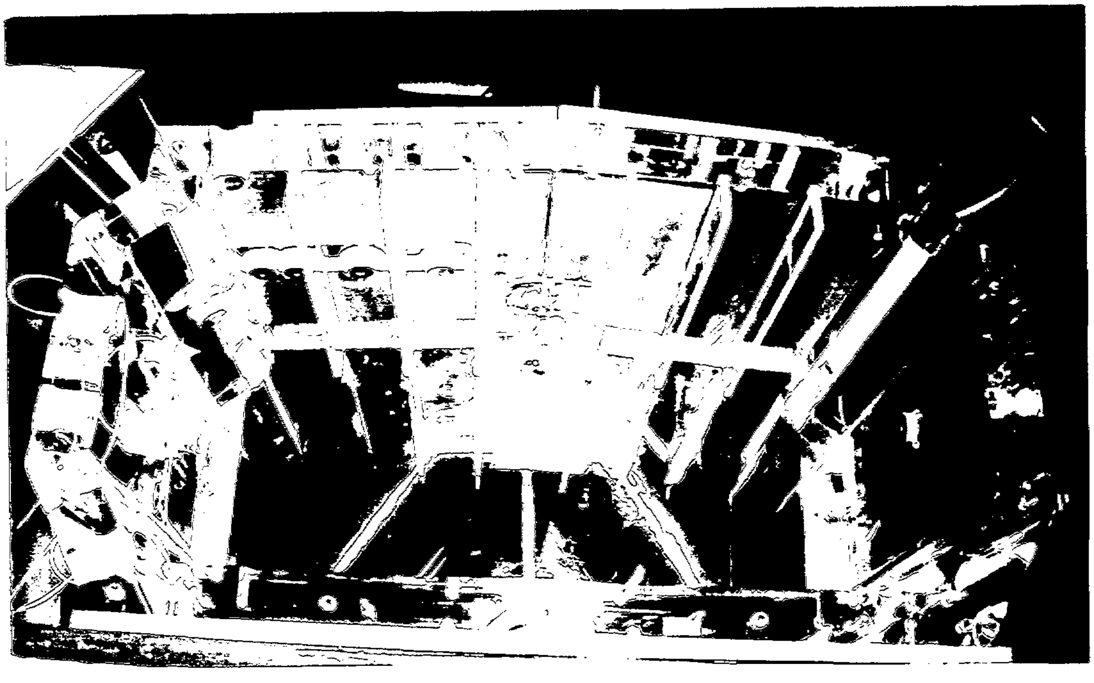

Figure 8 - Model 9 (VLCC centre-tank with deck girders)

Statistical analysis of the experiments: the test results were analysed in order to perform a statistical characterisation of sloshing pressures. Due to the highly non-linear nature of sloshing, the traditional probability distributions adopted for wave induced global loads (e.g. the hull vertical bending moment) are in-fact not directly applicable to the problem. Furthermore, the length of simulation to allow reliable statistical estimates (either experimental or numerical), as well as the selection of a subset of scenarios representative of the infinite different possible situations that a ship could encounter in her operating life were carefully investigated.

More than 100 different pressure time histories resulting from experiments carried out on Model 6 and Model 9 were processed in statistical terms and analysed, leading to the following conclusions:

- *suitable distribution for extremes:* the extreme distribution is well fitted by a Weibull law; in particular when the impulsive nature of the pressure is dominant, the slope of the Weibull law approaches 1 (exponential distribution). On the other side, when the dynamic non-impulsive nature is more important, a slope 2 is appropriate (Rayleigh distribution). This seems to be independent from the value of the ullage pressure. In Figures 9 and 1 0 the probability distribution of the sloshing pressure peaks in two different locations is shown, the first case is clearly impulsive dominated, while in the second a dynamic non-impact behaviour is more evident. In both cases a significant difference between the highest and the second highest value exists (a logarithmic scale is adopted for the abscissa axis).

- *possible "loner" effects:* the highest pressure value if very often two to three times larger than the second higher value; this "loner" effects is clearly detectable is most of the experimental results and is more pronounced when the ullage pressure is low.

- *effects of baffles:* some of the experiments on Model 9 were performed on both an unbaffled and a baffled configuration. From the analysis of results a considerable difference in the value of the pressure is evident as the result of the presence of baffles. This is clearly due to the baffle creating a solid wall which does not allow the fluid to flow away: as a result, the fluid buildsup and a higher pressure is detected in the neighbour of the baffle closest to the side walls. Despite the very large difference in absolute value, the shape of the probability distribution is pretty similar in the cases with and without baffles.

- *statistical stability:* the statistical stability of experimental results have been analysed with the aim of assessing the minimum time duration of a sloshing time history needed to get stable statistical results and of evaluating the effects of both waves and motions spectral shapes on the statistical parameters. The first issue was investigated by plotting the time history of the sloshing pressure variance: after order of 100 slosh cycles, oscillations in the value of the variance were found within a 10% band which can be taken as an indication of a good stochastic stability of results if the highly non-linear nature of sloshing is considered.

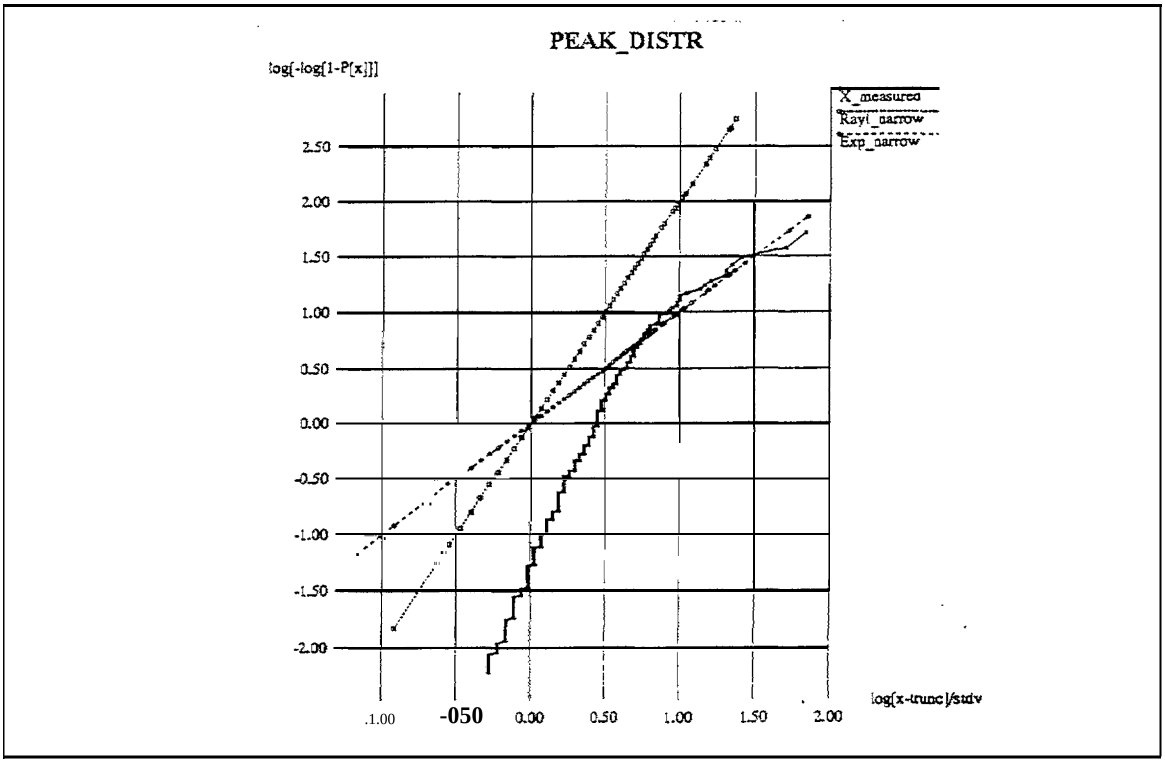

Figure 9- Impulsive dominated sloshing pressure, probability distribution of the peaks plotted on Weibull paper (slopes of the exponential and Rayleigh distributions shown for the sake of comparison)

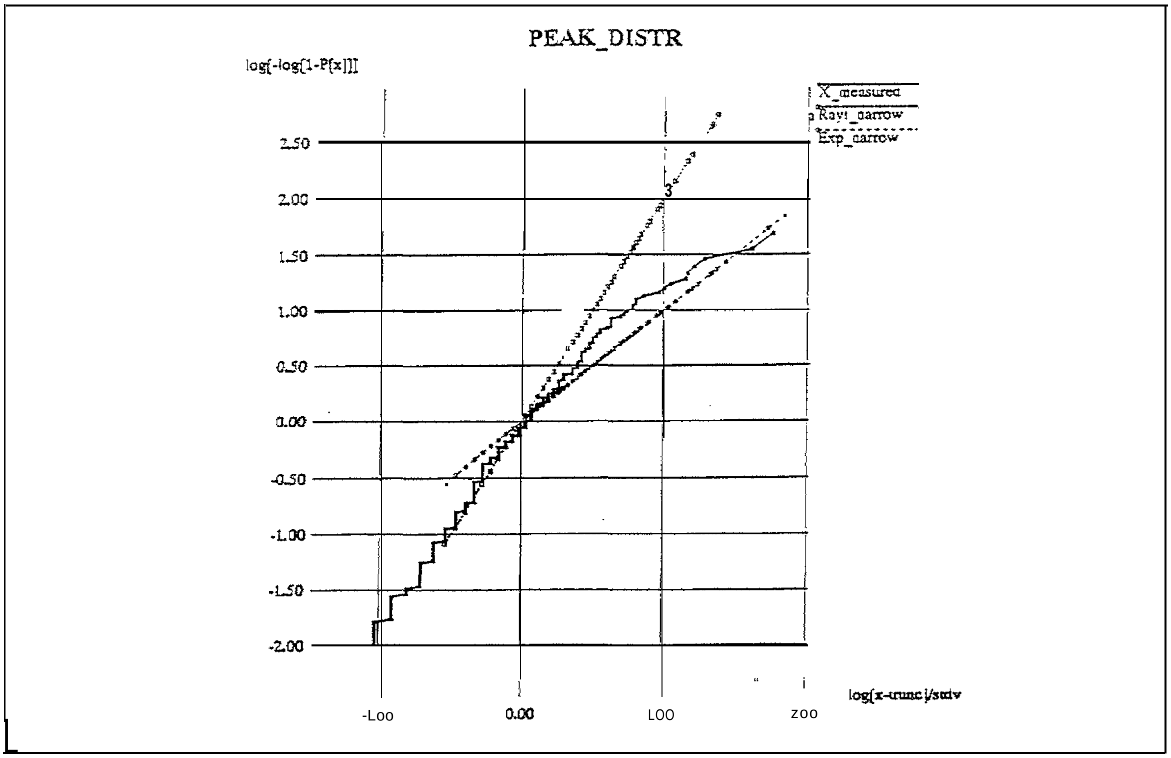

Figure 10 - Dynamic non-impulsive dominated sloshing pressure, probability distribution of the peaks plotted on Weibull paper (slopes of the exponential and Rayleigh distributions shown for the sake of comparison)

CFD Sloshing Software

As a result of an analysis on the possible design scenarios, a wide range of applications for free surface hydrodynamics was highlighted and was demonstrated the need for software able to model a range of complex geometries with multiple free surfaces.

A 3D and a 2D version of the 'slosh' hydrodynamic code (in the following 'slosh2d' and 'slosh3d') were released, the latter being a cut down version of the former using an improved genuinely 2D pressure solver. The 20 version, sensibly faster and easier to be used than the 30 one, is particularly addressed for the analysis of two dimensional configurations. Moreover, in the 2D version, an innovative fluid-structure interaction scheme is implemented which takes into account the effects of wall flexibility on the sloshing impact pressure.

Having considered the various current research and commercial software available, the Volume of Fluid (VOF) methodology was adopted to track the free surface and it was decided that geometry modelling would be handled by overlaying the geometry on a Cartesian Mesh /2/. This removes one of the major drawbacks in Computational Fluid Dynamics (CFD) which is the grid generation problem for complex geometries. From a users point of view, once the geometry is defined it is as easy to run a complex geometry as a simple one. This approach also allows the modelling of baffles, although the user has to be careful about the scales/thicknesses ifs/he does not wish to have very locally refined grids, with a correspondingly small time-step.

The 'slosh' codes model time-varying incompressible flow (for example water or fuel) in two or three dimensional containers of arbitrary shapes under externally applied forces. The configuration geometry is modelled as a set of connecting triangles about which the code automatically places a Cartesian mesh and derives intersection points. The externally applied forces are assumed to be the sum of constant, sinusoidally varying pitch forces and sinusoidally varying forces in any of the co-ordinate directions. The Navier-Stokes equations of motion are solved with walls assumed to be 'reflecting' and with the free surface modelled using a volume of fluid approach.

The volume of fluid method assigns a fill value between O and 1 to each computational cell and convects this value as the flow develops. A donor-acceptor algorithm is used to convect this free surface value to ensure that sharp surface shapes are maintained as time progresses.

The boundary conditions are implemented using 'image' points whereby points on the computational Cartesian mesh outside the user defined geometry are assigned flow values such that the average of these values and the neighboring 'real' values give rise to the desired flow quantity on the geometry. In this way a Cartesian mesh structure is maintained throughout the code leading to an efficient means of solution without the restriction of a small time step that would result from small cells formed by direct boundary intersection.

The software is automatic and user friendly. The user is required to input a geometry specified by points and connectivities which form triangles. This system is identical to the system used by a number of structural finite element packages. The user is asked for physical parameters to define the "sloshing input" and asked for the required resolution in terms of the number of cells in each direction. The calculation will then take place on this grid unless part of the geometry cannot be modelled, i.e. it occurs on two line segments. In this case the code performs a one-dimensional refinement so that the grid correctly recovers a representation of the geometry. The hydrodynamic code runs, and automatically adjusts the time-step to preserve stability limits. The most computer intensive part of the algorithm is the pressure solver which lies at the core of the code. The monitoring output indicates the major stages on each time-step and warns the user of any problems as well as informing him/her of the current state of the calculation.

The output is also user controlled and can generate both static and animation output. The frequency and location of the output can be changed. It can be inspected directly or by the use of commercial packages.

Several applications of the code were made during the project, including simulation of sloshing in Model 9 (double hull tanker) under a forced harmonic motion of frequency 5.227 rad/s and amplitude 0.01 m {model scale).

This geometry represents a typically complex section of a ship tanker. Two fill levels were considered; 60% and 85%. Predictions are shown at certain times in Figures 11 and 12.

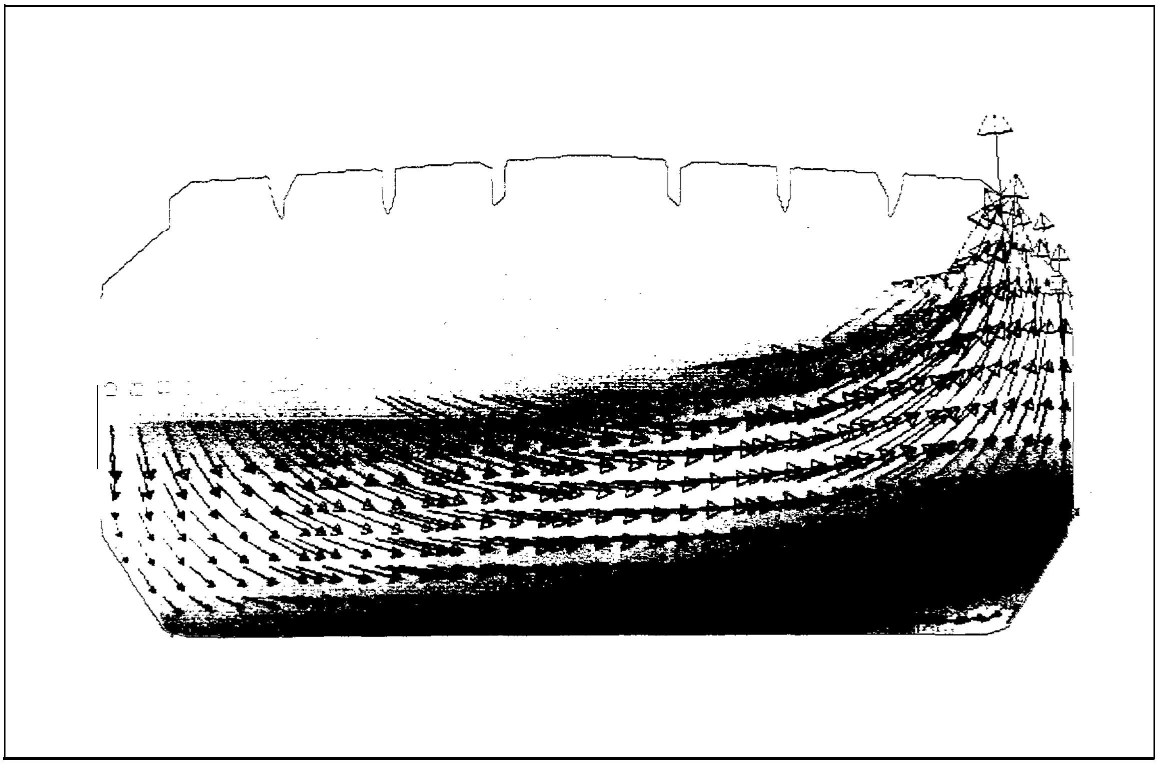

Figure 11 - Numerical simulations on Model 9 - 60% filling level

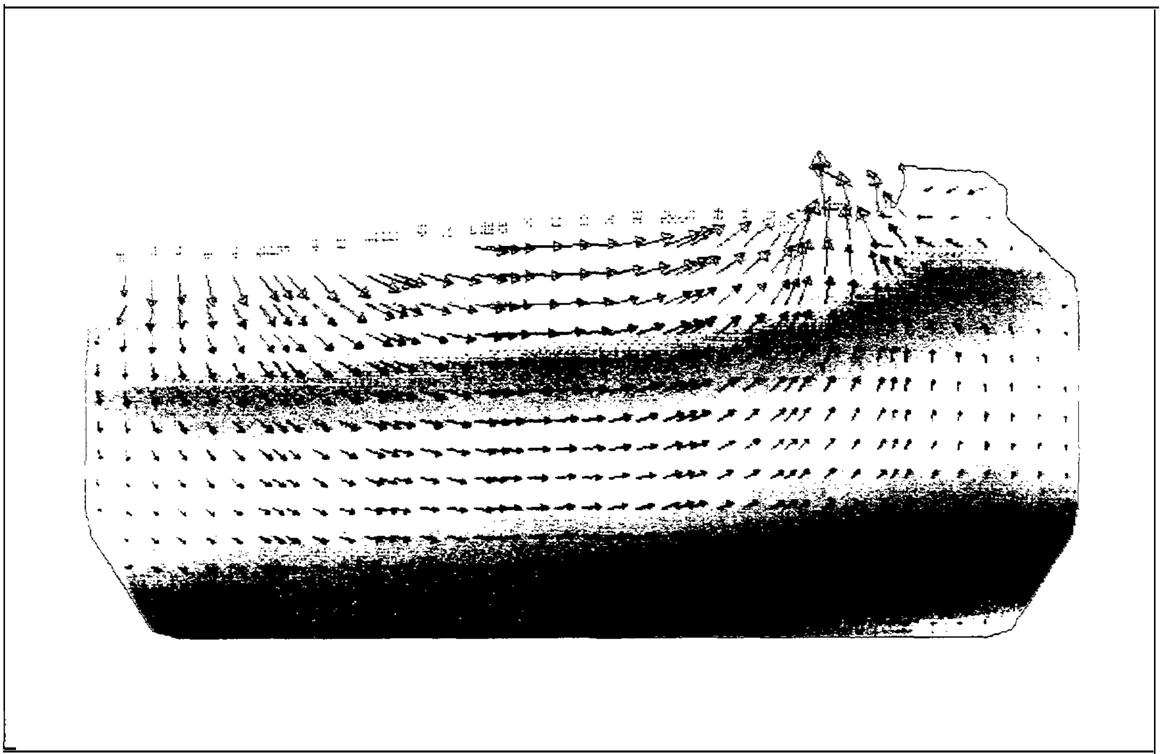

Figure 12 - Numerical simulations on Model 9 - 85% filling level

Although the meshes used are relatively coarse it can be seen that the main features of the sloshing motion are captured well. Indeed, " in the 85% fill case small re-circulation regions develop as fluid is entrained between the end most baffles and the tank walls. It is clear from the experimental measurements, where the fluid sloshes against the tank ceiling after 6 seconds that some damping is inherent within the code. Effort has been made to reduce this damping and it is believed that 'slosh3d' and 'slosh2d' are comparable, if not better, with existing available software.

The validation of the software, carried out by comparing numerical and experimental results, showed that, despite the efforts made, some problems still exists which are related to the numerical damping of the numerical scheme, in order to assess its effects, on the predicted results a rectangular box (0.8m×0.4m) was put under forced sloshing motion for 3 seconds. After 3 seconds this forcing was switched off. The code was run 'inviscid' and hence any decay in amplitude of the motion has to be attributed to the 'slosh' algorithm. As a result of this test, it can be seen that damping is evident, but not excessive. The main cause of this damping is due to the fluxing or moving of the fluid across the computational mesh. Movement in any of the three (or two) co-ordinate directions is accurate but movement diagonally across grid cells introduces inaccuracies (essentially since diagonally fluxing is approximated by a sum of two fluxes in different directions).

Finally, as far as computing requirements are concerned, the general size of the 3D code is around 4.5x 10^{-4} megabytes per grid cell, giving a code of around 5.5 Megabytes for 12,000 cells {say 50×30×8). The time taken for the 3D code to complete a cycle is very dependent upon the type of flow being considered. For the Model 9 case discussed above, on a 4.5x20x3 mesh takes around two hours to complete 1.0 seconds of sloshing on a DEC ALPHA 3000. For the 2D code then it will take approximately 15 minutes.

Together with the hydrodynamic software, the user friendly graphical interface animation suite 'anim3d' was produced. Various graphics languages were considered but only the Silicon Graphics language GL(™ Silicon Graphics Inc) offered the performance and versatility necessary for animation of the large, complex, data sets produced from the code.

The software has been developed to allow the user to interact as intuitively as possible, A number of graphics features have been included to allow the user to visualise the data in certain ways. The interface has been designed to reflect initially a video player, with a number of sub-menus which control the viewpoint, what is presented and how it is presented.

Interaction with the system is provided through a user interface based on OSF/Motif (™ Open Software Foundation), a system available on a wide range of Workstations. A screen dump of a representative The program is written in C and uses the xdr library to read and write data. The xdr libraries dramatically reduce the size of the data files compared to an ASCII representation but allows data to be ported across a number of computer platforms unlike binary files.

Fluid-structure interaction

This is one of the most innovative parts of the whole project as, to our knowledge, no existing sloshing package has in-fact the capability to consider the effects of the fluid structure interaction. This aspect is particularly important in the case rigid structures are adopted as tank walls (see e.g. the side **walls** and the bulkheads of a tanker ship, made of stiffened steel plates): the magnitude of the peak values of the pressure impact is in-fact substantially influenced by the fluid-structure interaction effects and this is to be taken into account in modelling the boundary conditions at the wall when performing a numerical simulation (see Figure 13) for a typical pressure peak time-trace].

Fluid-structure interaction should be **also** carefully considered when analysing model scale experiments results, in particular when pressure peaks are concerned. The effective inertial and elastic properties of the tank walls should, in-principle, be taken into account in designing model scale experiments. Nevertheless, in the shipbuilding field it is unpractical to scale the ship structural characteristics. Fluid-structure interaction effects could be taken into account in a correct scaling up of the experimental results, by considering the actual characteristics of both the ship and the model structure.

In the development of the theory for the fluid-structure interaction, two different cases were considered since the beginning of the Project:

- *flexible walls case:* in which very flexible wails are treated;
- *semi-rigid walls case.'* in which rigid structures, such as ship tank boundaries are treated. The prefix "semi" stands to indicate the difference with the case of "infinitely" rigid walls {a purely mathematical abstraction), where no fluid-structure interaction can occur, which in fluid-dynamics is currently just named "rigid".

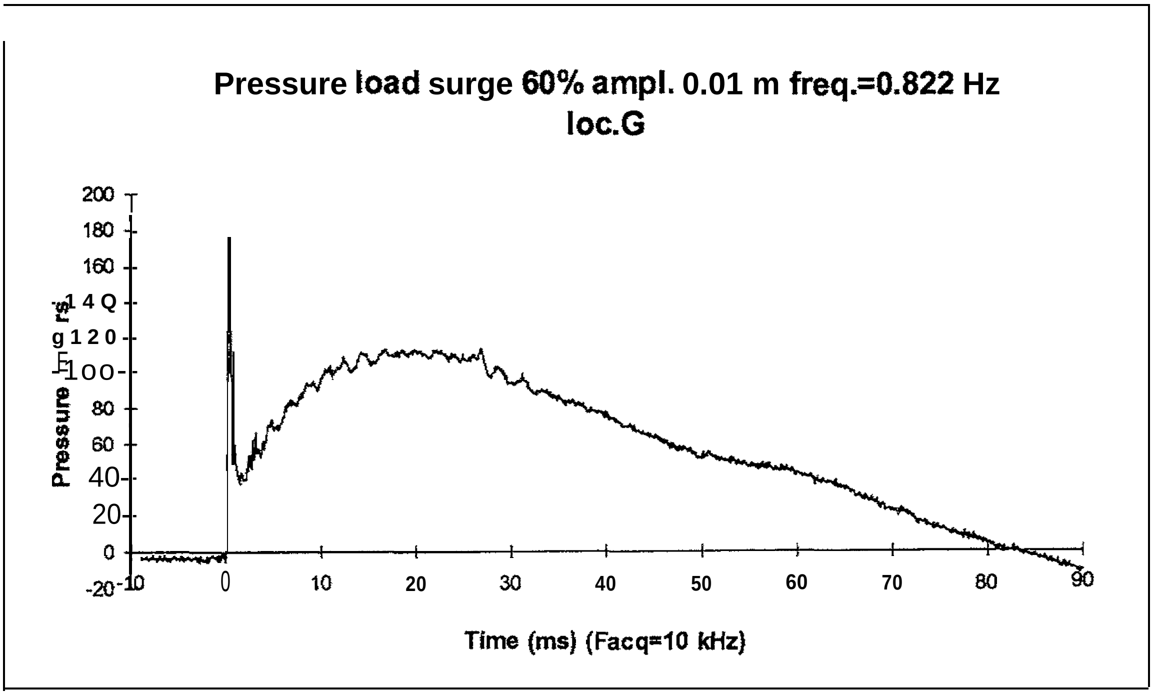

Figure 13 - Typical impact pressure experimental time trace.

Both cases were considered, the latter under a speculative point of view leading to a prototype software development while the former, for which further details are given in the following, was implemented into the following, was implemented into the 2D CFD code.

The aim of the semi-rigid boundary conditions is to replace the ordinary hydrodynamic 'rigid' boundary conditions (zero normal velocity and zero pressure normal derivative at the wall) in order to take care of the structural response of the wall. The conditions prescribe the fluid velocity normal to the wall equal to the wall normal velocity and a pressure normal derivative at the wall proportional to the normal wall acceleration.

As "semi-rigid" conditions are of relevance only during sloshing impacts, the modified boundary conditions are applied only when "severe pressure changes" are detected.

The method adopted for the coupling of the structural and the hydrodynamic system is named 'Implicit method', it is in-fact based on the direct integration of the structure equations inside the pressure solver of the hydrodynamic module. This way no structural solver is needed with the advantage of a tighter link between the fluid and structure equations that leads to a faster convergence for the solution of the pressure field,

The following equation summarises the considered boundary condition for the pressure problem, i.e. the dependence of the pressure normal derivative at the wall, from the wall mass, damping and stiffness matrices and the pressure itself at the wall. The normal derivative of the pressure at the wall depends on the wall mass, on the structural damping and stiffness matrices and on the value of the pressure itself at the wall; this dependence can be recast in a Neumann condition as given by the following equation:

$$\left(\frac{\partial p}{\partial n}\right)_{wall} = f\left(F(M,D,K), p_{wall}\right)$$

where f() is the expression of the wall acceleration due to water impact, obtained as a direct consequence of the pressure itself. This way the pressure is implicitly modified by the wall condition. Once the pressure field has reached convergence, wall velocity is calculated and adopted as boundary condition for the velocity field, which in the operator splitting method is solved after the pressure one.

The fluid-structure interaction algorithm is devoted to the correction of pressure peaks by a modification of the pressure boundary conditions induced by the acceleration of the structure subject to these peaks, a peak-detecting routine in the main code pilots the start and the stop of the interaction algorithm.

Structural characteristics are modelled by means of the FEM technique through the mass matrix **M**, used in the structure equation as a multiplier of the structure nodes acceleration vector, and the elastic stiffness matrix K, which is a multiplier of the vector of the wall displacements.

Under the semi-rigid regime, at the very moment of a severe fluid impact, a large acceleration is detected together with small wall displacements; this was confirmed experimentally. These latter may then be neglected without loss of precision of the results as the focus is on the correction of the pressure value rather than to the evaluation of structural stresses and strain. This means that the mass matrix only plays a role and the stiffness matrix is not necessary; damping terms are also neglected in the present implementation leading to the following simplified condition:

$$\left(\frac{\partial p}{\partial n}\right)_{wall} = f\left(F(M), p_{wall}\right)$$

The effectiveness of these hypotheses was also tested by means of a number of dynamic finite element calculations on typical ship structures

A lumped mass scheme was considered as a further simplification for both the user and for the computer time demand of the code. This scheme is based on assigning a single structural node for each hydrodynamic cell and allows a diagonal mass matrix to be considered.

It is to be stressed that the adoption of the semi-rigid hypothesis, valid only in presence of a wall which responds to a pressure shock mainly with its inertial characteristics, limits the application of the algorithm to few time-steps after the impact (before the wall starts to sensibly change its position), thus allows only the correction of the pressure peak value to be calculated. On the other side the main advantage of the semi-rigid hypothesis is that it is straightforward to implement it by the implicit method, since the boundaries are fixed. This means: no fluid re-orientation is needed after the structure response has been applied and one single module for the pressure solution may be easily adopted.

As a first example a 0.4x 0.8 [m] tank, filled by water at 60% of its height and subject to an harmonic surge motion of 0.03 m amplitude at the linear resonant frequency of 0.848 Hz is considered; the semi-rigid condition is applied to the upper five cells of the vertical right wall of the tank varying the value of the lumped mass associated to each of the considered hydrodynamic grid points (adopted hydrodynamic grid is 40 x 20 cells).

[n Figure 14 the pressure time history in the upper right corner of the tank, under peak condition, is plotted for different values of the lumped mass. The case under consideration is the one given in the example: (see also the input file shown in the above). The results show that, as expected, greater values for the reduction of the peak pressure are found for the lower values of wall mass. For a very lame mass value (e.q. 1.OE+ 10) the condition of infinitely rigid wail is reproduced. The condition of infinitely rigid walls is obtained by setting the threshold value of the pressure time derivative to a very high value (e.g. 1.0E+10)

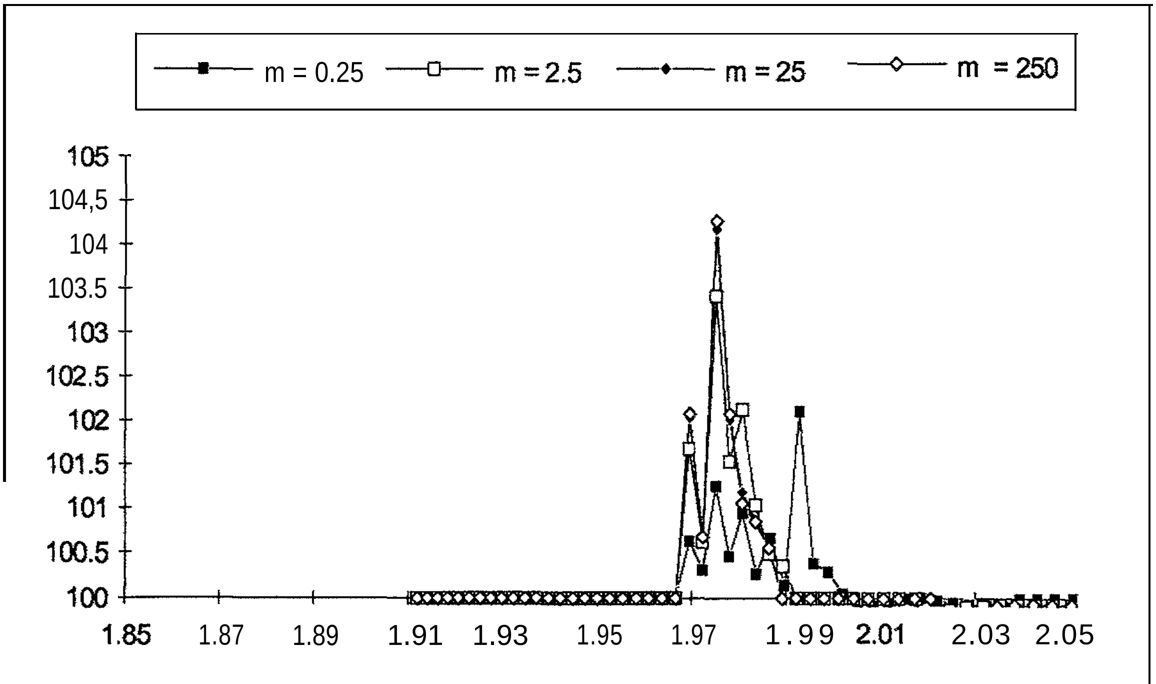

Figure 14- Impact pressure time histories (non-dimensional] for different lumped mass at the wall

Conclusions and recommendations

Scientific aspects

The EUROSLOSH project has produced significant scientific results, in particular by clarifying several aspects among which the most outstanding are:

- *an extensive analysis of/he effects of different physical parameters on the sloshing phenomenon:* these results, summarised in the experimental database, are of particular value as they refer to tanks of simple geometry and are therefore not polluted by side effects which makes interpretation questionable; besides this, the experimental activity carried out within the project is believed to have produced one of the most massive, complete and coherent set of model tests on sloshing performed so far; finally, it is worth underlining that a PhD thesis was developed on these aspects /3/;

- *an investigation into scale effects:* although this was made within the limits imposed by the use of models, which reduced to a certain extent the interest in the results, the investigation can be considered as an important starting point for a future research into the scale effects in the sloshing phenomenon;
- *a deep analysis of the fluid-structure interaction problem:* the "implicit" algorithm developed has a major advantage in being very simple and therefore particularly suited to cope with the VOF method;;
- *a significant progress in CFD applied to sloshing:* several different algorithms and numerical techniques have been implemented and related pros and drawbacks tested on the field, resulting in a considerable experience and step forward.

Technological aspects

From a technological point of view, the consortium has significantly progressed in terms of:

- *experimental techniques.'* a big effort was made, mainly by SIREHNA with the assistance of BV, resulting in a considerable increase in the know-how on the equipment as well as in a strong progress in the techniques to be adopted to carry out *model* tests on sloshing, with particular emphasis on impact pressure and peak detection for which ad-hoc high speed acquisition systems were developed;
- *understanding of the effects of sloshing on structures:* by means of evaluation of the structural response to sloshing impact as well as dynamic pressure loads;
- *analysis of relevance of sloshing in different industrial areas.'* by means of the development of an experimental data base covering 4 different industrial areas (road transport, civil engineering, ship and aeronautics) for each of which selected relevant applications were considered having shapes among the most complex ever considered in model tests.

Recommendations

The following recommendations are worth being made:

- *scientific* aspects: although the effect of some physical parameters would need further investigations (an example is air entrainment] and any investment **in this area** would be beneficial, it is believed that the highest benefit / cost ratio can be achieved by carrying out a research into scale effects which should involve full scale (at sea) measurements, model tests {possibly on two models having different scales) and numeric-al simulations of the same tanks;
- *numerical aspects:* two outstanding problems are related with the EUROSLOSH code as it stands at the end of the project, namely some numerical dissipation and the high computer time demand. [n order to take full advantage of the progress in numerical algorithms, it is very important to exchange views and results between different CFD users from different fields; in this respect, the implementation of a concertated action on CFD applications and developments would be appropriate;
- *training:* training on sloshing and its effects on structures {ships, offshore, civil engineering etc.) as well as on aeronautic sloshing related aspects would be appropriate to disseminate the results of the project. Part of the documentation of the project (videos, reports on sloshing calculations, guidelines} is very well suited to be converted into lecture notes and documentation to be included into training courses.

Acknowledgements

The present work was carried out with the support of the European Community under the BRITE/EURAM Project BE-4354, Contract BREU-499, "Experimental and Numerical Analysis of Sloshing and Impact Loads" (EUROSLOSH).

References

/1/ N.A. Ham[in, Liquid slosh loading in slack ship tanks; forces on internal structure & pressures, SSC report n. 336, Nov. 1986.

/2/ C.A.J. Fletcher, "Computational Techniques for Fluid Dynamics", Second edition, Vols. 1 and 2, 1990.

/3/ P. CORRIGNAN - "Analyse Physique des Phenomenes Associes au Ballottement de Liquide clans des Reservoirs (Sloshing)", in French, Ecole Centrale de Nantes, thesis of PhD, Nov. 1994.