Proceedings of the ASME 28th International Conference on Ocean, Offshore and Arctic Engineering
OMAE2009
May 31 - June 5, 2009, Honolulu, Hawaii, USA

OMAE2009-79053

WAVE-IN-DECK LOAD ON A JACKET PLATFORM, CFD-DERIVED PRESSURES AND NON-LINEAR STRUCTURAL RESPONSE

Bogdan Iwanowski*
FORCE Technology Norway AS
Claude Monets alle 5
1338 Sandvika
Norway
Email: boi@force.no

Rune Gladsø
FORCE Technology Norway AS
Email: rg@force.no

Marc Lefranc
FORCE Technology Norway AS
Email: mrl@force.no

ABSTRACT

The paper presents an industrial application of CFD and non-linear structural response codes in offshore technology. A Wave-In-Deck load due to an extreme wave, acting on a jacket platform, is studied numerically. Particular attention is given to details of local flow and local non-linear dynamical response of the structure.

A very detailed FEM model of the platform deck structure, composed of shell elements, is embedded into a non body-conforming CFD grid of computational cells. The applied CFD code is a Navier-Stokes equation solver with an improved Volume of Fluid (iVOF) method employed to displace and re-construct fluid's free surface and uses a simple, Cartesian grid. The two computational grids, FEM and CFD, are independent.

The challenge of a direct mapping of CFD-derived fluid pressures onto structural FEM shell elements is addressed. Then the non-linear dynamical response of the structure is found in time domain. The employed CFD code is ComFLOW while the FEM part is handled by the well-known commercial program LS-DYNA. The composed approach utilizes both robustness of VOF-based methods in tracking of the fluid's free surface and reliability of FEM structural codes such as LS-DYNA.

*Address all correspondence to this author.

INTRODUCTION

A positive air gap between design wave crest and platform's deck is no longer maintained for a number of installed offshore platforms of jacket type. Reasons for the insufficient air gap can be twofold: updated environmental wave data parameters and seabed subsidence at the platform's location.

It follows that such platforms can suffer a severe wave-in-deck impact with high inundation, understood here as the wave impact height. Structural reassessment of the platform requires an estimation of loads acting on the structure. In a preliminary analysis, the loads can be taken as total fluid forces acting on the entire platform deck. Such overall horizontal and vertical forces can be predicted either by methods based on fluid's momentum displacement considerations or by a more advanced CFD analysis. The momentum displacement methods are quick, but require a well-chosen set of coefficients (inertia, drag, shielding). The coefficients can be found and calibrated through expensive experiments [1], but otherwise their values are educated guesses only.

Detailed and very localised time-pressure histories cannot, in general, be obtained by the momentum displacement methods. However, one can apply a CFD code, where no inertia/drag/shielding coefficients are involved. This makes the CFD approach more robust and reliable, but one has to pay for the increased reliability by much longer calculations. A comparison of wave-in-deck fluid forces predicted by the momentum displacement method and with a very basic CFD calculations for a simple box structural model can be found in [2]. The agreement was encouraging for the incoming wave of Stokes 5^{th} order type. More recently, the wave-in-deck impact problem has been addressed in [3,4].

Motivation

Consequences of global wave-in-deck impact with respect to a jacket structure have been evaluated in the last 15 years. The deck structure itself has not been given a similar attention; even though a collapse of the topside super-structure would result in similar consequences for the platform.

Decks of the jacket platforms have not been designed for wave in deck loads since such load type was not anticipated. Hence it is nearly impossible to document ability of a jacket platform's deck to withstand this kind of wave load by traditional calculation according to currently existing regulations, such as NORSOK standards [5, 6], or by following the "best practice guidelines" such as an outcome of the ULTIGUIDE JIP [7].

A new approach for more accurate evaluation of structural integrity of the deck structure has been developed. The presented methodology uses the latest available possibilities with respect to load estimation and structural response calculation. Wave-in-deck loads are predicted with the CFD code and fluid pressures are directly mapped onto a structural Finite Element grid composed of shell elements. The following FEM analysis finds a non-linear dynamical response of the deck structure, and final results include strains within all modelled structural parts.

The goal is to document that the platform deck structure can withstand the wave-in-deck load, that the safety requirements are maintained, and that there is no high risk of pollution due to an oil spill. Depending of the strain utilization level of the deck's structural parts, a suitable local reinforcement can be proposed.

METHODOLOGY

The wave-in-deck problem for a jacket platform is schematically depicted in Fig. 1. Several CFD techniques can be used to find fluid flow around the platform's deck. The applied CFD method should be capable of finding complex fluid's free surface deformations. Examples of successful free surface tracking techniques are Volume of Fluid (VOF) and Smooth Particle Hydrodynamics (SPH). The former is an Eulerian formulation and requires a 3D fluid grid, which either conforms to the body shape or is generated separately, being independent from the object's geometry. The purely Lagrangian SPH approach is grid-free and in principle can handle arbitrarily complex body shapes.

The above methods are all computationally intensive. In the Eulerian formulations, the entire fluid domain must be gridded with a sufficient accuracy in order to resolve very complex flow features. Likewise, the fluid domain must be filled with a sufficient number of Lagrangian SPH particles.

Hereafter, this article will concentrate on the Eulerian approach. An application of the SPH to compute fluid flow around marine structures is described, for example, in [8].

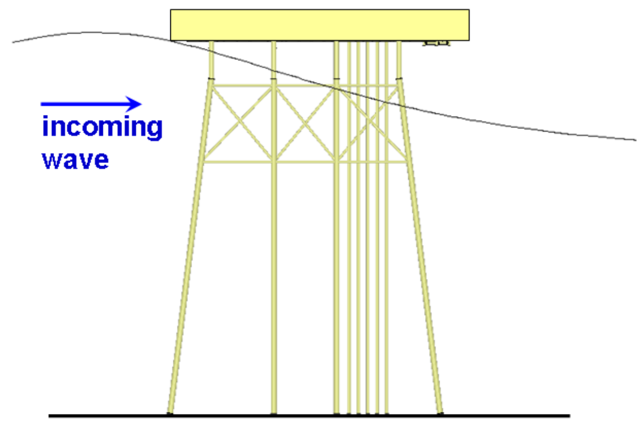

Figure 1. The wave-in-deck impact problem

CFD to Structure Interface

A structure of the jacket's deck can be relatively simple, being composed mainly of straight plates representing box-shaped containers and main girders, as shown in Fig.(2). But it can be very complex as well, Fig.(3), where there are pipes, gaps and various nooks and corners. It seems that a body-fitted grid could be constructed for the simpler structure. However, it would be a very tedious and time consuming task for the more complex object. It is very likely that the resulting 3D fluid grid (either of Finite Volume or Finite Element type) would be poor computationally-wise, with small cells neighbouring large cells. Or that there would be a very large number of (tetraheral) finite volumes or finite elements indeed. Usage of the un-structured, body-fitted grid for can be found in [9], where the hexahedral control volumes were used.

This paper will demonstrate a CFD-structure interface with following features:

(a) CFD computational grid is generated independently from the structural FE grid,
(b) CFD grid is composed of simple box-like computational cells (Cartesian grid),
(c) FE structural grid, composed of shell elements, can represent an object of arbitrary shape complexity,
(d) CFD-derived fluid pressures are directly mapped onto the structural FE shell elements.

Example of the Jacket Platform

The following discussion will use a complex jacket platform structure as an example. The entire structural model includes

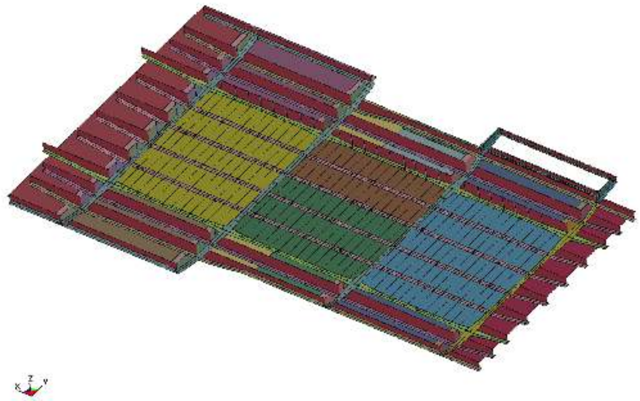

Figure 2. Deck of the jacket, simpler structure (view from below)

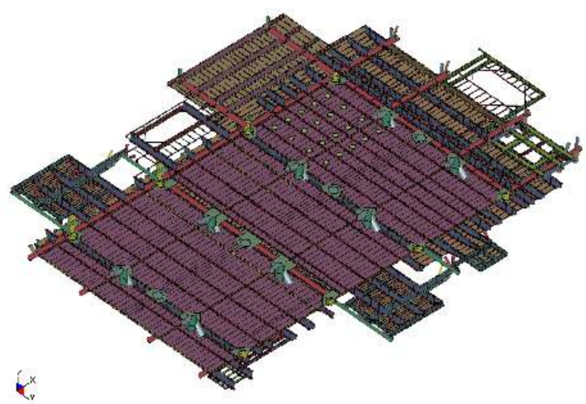

Figure 3. Deck of the jacket, complex structure, (view from below)

shell and beam elements, Fig. (4). The cellar deck structure is modeled with the shell elements. All relevant structural parts down to HP-stiffeners in the tank bottom are included.

Stiffness of the jacket structure below and above the cellar deck is accounted for by use of linear beam elements. Equipment on the deck is modelled by increasing material density in selected areas, mainly in the deck plating. Structure above the cellar deck is accounted for by applying nodal masses attached to beams. The masses are established based on information from the weight reports.

A more detailed description of the structural model and particulars of structural calculation are provided in a relevant section.

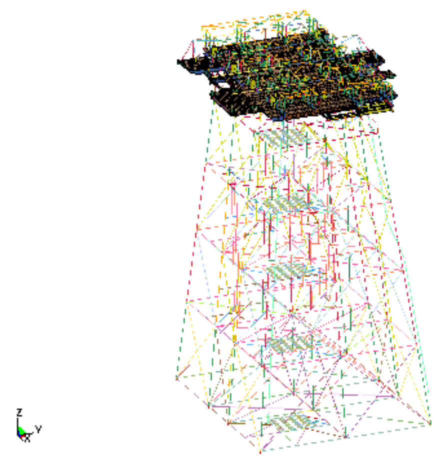

Figure 4. Jacket platform structure used as an example

CFD CALCULATIONS

The applied CFD code ComFLOW is a Navier-Stokes equation solver with an improved Volume of Fluid (iVOF) method employed to displace and re-construct fluid's free surface. Algorithms used in ComFLOW are described at length in [10, 11]. The code uses a staggered Cartesian grid as in the original VOF method [12], where control points for fluid velocity vector components are defined on faces of a computational cell, and where fluid's pressure control points are defined at the cell's centre.

A ComFLOW model of the entire jacket structure is shown in Fig. (5) and more details can be seen in Fig. (6). Traces of the applied computational grid are visible on bottom and side boundaries. The structured Cartesian grid is refined towards the body location and is nearly uniform within a virtual box containing the cellar deck object.

The incoming wave is a regular Stokes 5^{th} order wave. There is no wave crest height variation transversely to the propagation direction, which means that a 2D incoming wave model is used and that the wave impact height is constant along the platform leading edge.

In longitudinal direction (along wave propagation direction) the fluid domain extent is about 0.15 of the incoming wave length ahead of the platform's leading edge. There is also some space downstream of the platform, about 0.2-0.25 of the wave length. The downstream space is large enough to contain all created wave disturbances at the end of simulation. Extent of the domain in the wave propagation direction is related to inflow and outflow boundary conditions. The incoming wave parameters, kinematics and pressure, are specified at inflow and no reflections from the body reach the inflow boundary within the simulation time. Situation is obviously different on the downstream side. Although the incident wave is of monochromatic type, there will be a spectrum of diffracted waves due to wave-body interaction. However, the outflow boundary applied in the calculation

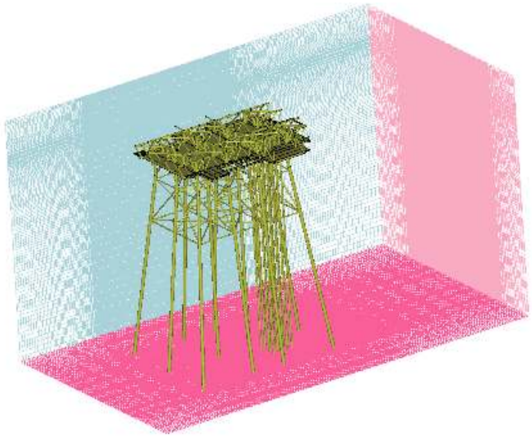

Figure 5. COMFLOW model of the entire jacket platform

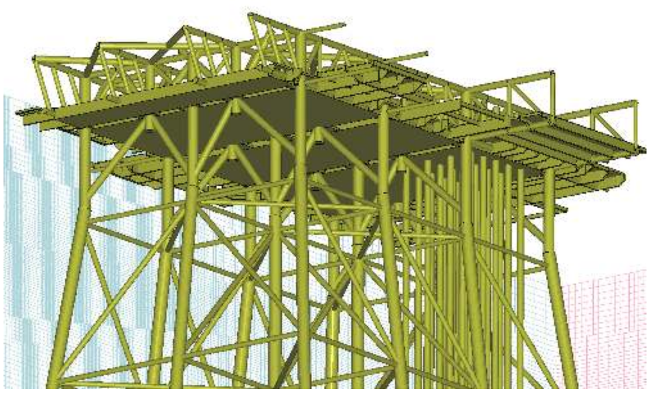

Figure 6. COMFLOW model of the jacket platform, close-up

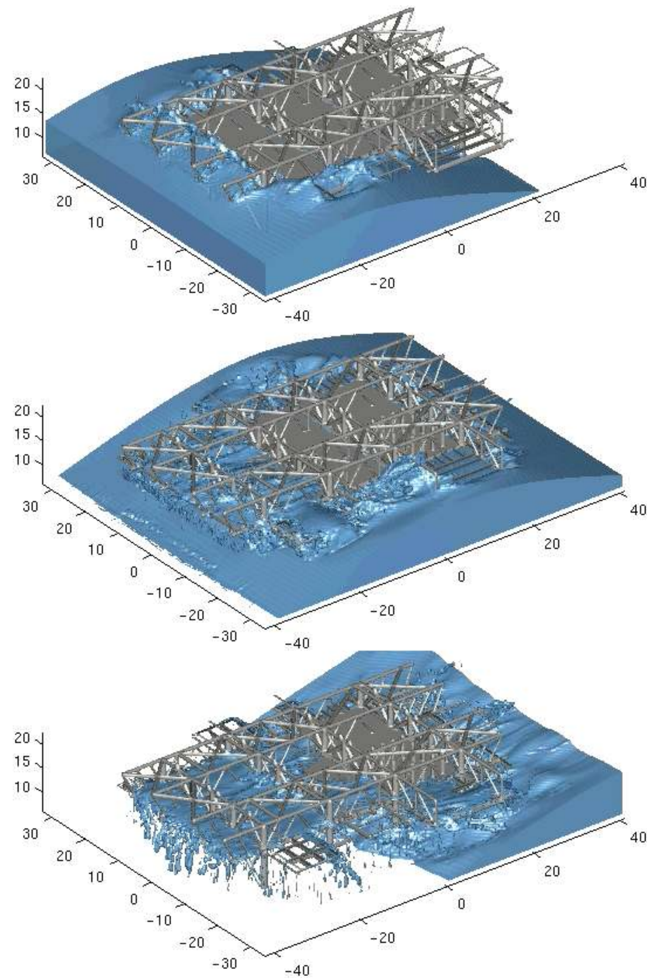

Figure 7. Wave flow past the platform deck, 3D view

is also monochromatic (of Sommerfeldt type) and other wave lengths present in the spectrum will be reflected back into the computational domain. Therefore, downstream part of the domain has been made large enough to keep the diffracted field in the domain. It is fortunate that the necessary simulation time is relatively short for the wave-in-deck problem, about one third of the incoming wave period. It is noted here that much more transparent inflow and outflow boundary conditions, capable of generating and absorbing a wide spectrum of wave frequencies, are present in ComFLOW and are in testing phase.

Boundary conditions imposed on sides of the domain are rigid walls and there should be enough space to prevent contamination of results by sideways reflections. Bottom boundary condition (the seabed) is of rigid wall type as well. Several further remarks are made:

(a) model of the structure is rigid and fixed in space during the CFD simulation,
(b) single-phase fluid flow physics is used. The computed fluid flow is that of incompressible and viscous sea water and no air cushioning is considered. The two-phase flow model is present in ComFLOW [13,14], but authors' experience from sloshing simulations is that various contributions from the air presence, like bubbles or pockets entrapped in corners are properly resolved only in small scale. This could be possible only with very fine CFD computational grids (or with embedded local grids).
(c) no turbulence model is employed. The considered problem is dominated by inertia effects and it is unlikely that viscous stresses contribute much to wave-in-deck loads.

The ComFLOW code was validated against experimental results during the ComFLOW-2 Joint Industry Project, [15].

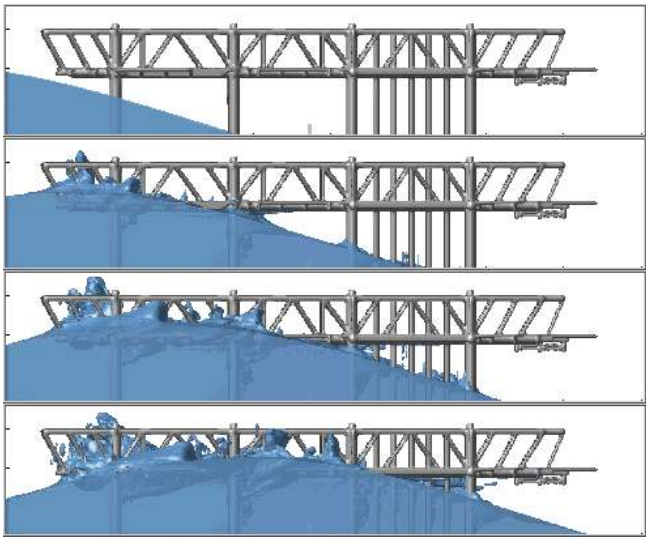

Figure 8. Wave flow past the platform deck, side view, part I

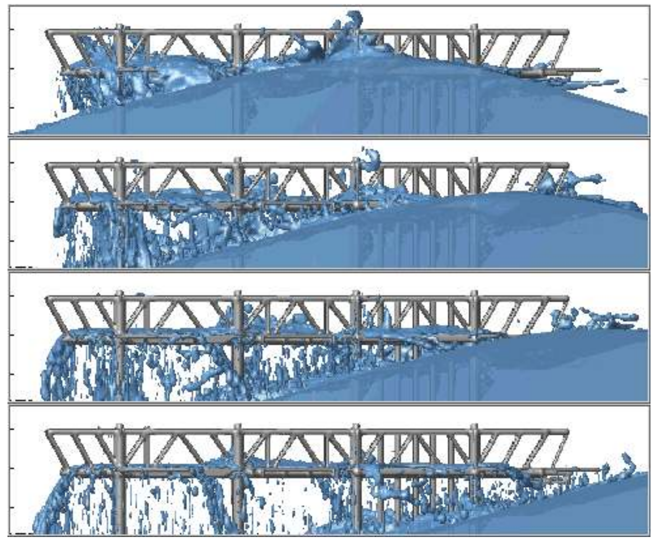

Figure 9. Wave flow past the platform deck, side view, part II

RESULTS OF CFD SIMULATION

Visualization of the CFD-computed fluid flow past the platform deck is shown in a number of snapshots taken from fluid flow animation movies. The three frames shown in Fig. (7) display stages of a simulation viewed from 3D perspective. It can be seen that some amount of sea water overtops the platform's deck edge and enters the deck. Then, droplets of water running out of the deck are visible in the last frame.

A more detailed fluid flow visualization is presented in Figs. (8) and (9). There are 8 consecutive stages of the simulated wave-in-deck phenomenon shown from a side view.

Mapping of CFD-derived Pressures

Local fluid pressures obtained from the CFD simulation are available at centers of cells. A FE representation of the structure is simply submerged into the CFD computational grid at exact location of ComFLOW representation of the body, as shown in Fig. (5). The first task is selection of CFD pressure points that are the closest neighbours of structural elements. The submerged structure is depicted in Fig. (10), and the selected CFD pressure points are shown as small dark squares in centres of CFD computational cells (the CFD grid is shown in background).

Each structural shell element has always a geometrical centre and a normal vector defined. Only one more bit of information must be associated with an element. It must be explicitly stated which face of the shell element is exposed to the fluid flow and relation of the exposed face to the element's normal. The three obvious possibilities are no exposed element faces (an internal stiffener), one exposed face (wall of a bottom tank), two exposed faces (an external girder). If the above information is available, then choice of the CFD pressure point relevant to a given shell element can be fully automated. An example is presented in Fig. (11), where all exposed shell elements of a complicated part of the jacket's cellar deck structure have appropriate CFD pressure points assigned. It is noted that:

(a) rather large data sets must be processed. For a problem at hand, the typically used CFD grid has 10-20 million of computational cells an there are typically 200-500 thousands of FE shell elements,
(b) number of created time-pressure functions was, for the tested cases, in range of 100-200 thousands. Only a very limited part of these functions can be eye-inspected,
(c) it helps that the used CFD computational grid is Cartesian and structured. But the pressure selection algorithm will work with unstructured grids as well (and, although this was not tried, should work with the SPH particles).

Removal of Pressure Peaks

The iVOF method used in ComFLOW enforces mass conservation principle in grid cells that contain fluid's free surface by a scheme which involves velocity extrapolation. This procedure is described in [10, 11] and typically performs very well. However, some pressure spikes occassionally do appear in computed time-pressure signals, especially in cells that are partially occupied by the immersed body and where fluid's free surface has just appeared or just vanished.

For the considered problem, cells of such type are present everywhere in vicinity of the jacket structure as the incoming wave passes through the platform's deck location.

Since origin of the numerically induced pressure peaks is known and explainable, it is justified to use a reasonably de-

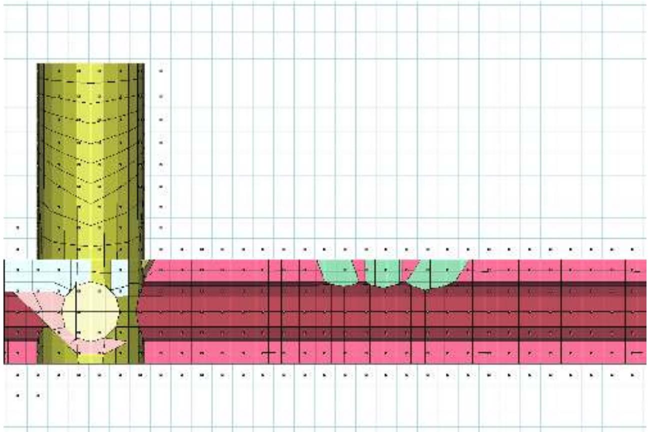

Figure 10. FE structure representation submerged into the CFD grid

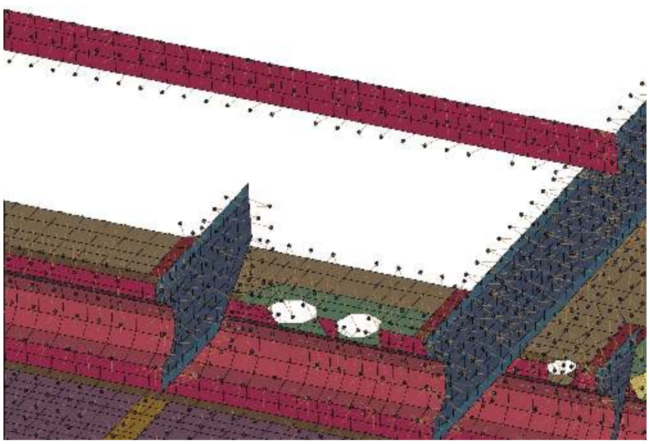

Figure 11. Structural shell elements and assigned CFD pressure points

signed peak-removal and curve smoothing procedure. It has also ben found that short-lasting pressure peaks have adverse effects on the transient structural response simulation.

The applied peak-removal and/or smoothing technique should be rather delicate to avoid smoothing-out the truly existing flow features. The procedure must be also fully automatic, since it is not possible to visually inspect all the derived time-pressure functions.

All time-pressure curves obtained from the ComFLOW simulation have been post-processed by a multi-pass peak-removal and smoothing procedure. The data processing passes included:

1. re-sampling,
2. removal of standalone peaks (pressure peaks with a very short duration, like one CFD time step. Such peaks are considered non-physical),
3. median filter.
4. moving average filter with a Gauss or Kaiser window.

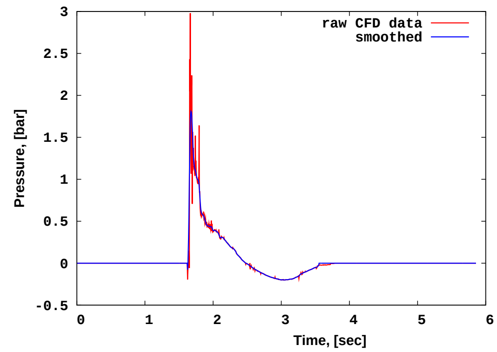

Figure 12. Smoothing of a CFD pressure signal, example A

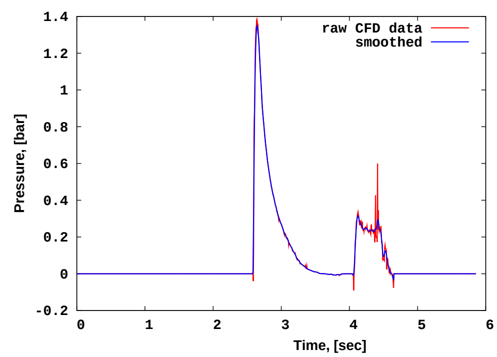

Figure 13. Smoothing of a CFD pressure signal, example B

Two arbitrarily chosen examples of such data post-processing are displayed in Fig.(12) and Fig.(13). It can be seen that the original time-pressure signals (red colour curves) are somewhat shaky and that there are spikes. The post-processed pressure curves are displayed in blue colour.

CFD Results, Pressure

Graphs of the CFD-derived time-pressure functions are presented for two selected parts of the cellar deck structure. Shell elements within the deck's leading edge and within external girders are shown in Fig.(14) and Fig.(15), while associated pressure load curves are presented in Fig.(16) and Fig.(17), respectively. The graphs display rather typical impact-like time-pressure curves.

The loading pressures can be displayed over any part of the structure as well. Spatial distribution of the fluid pressure on the entire deck's bottom plating at some time instant is displayed in Fig.(18).

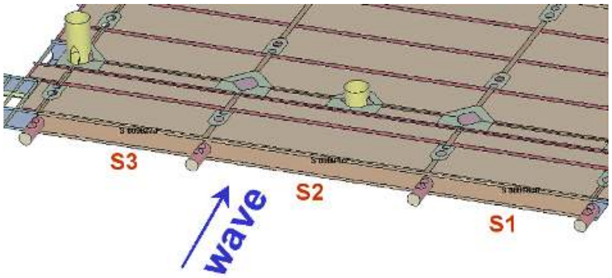

Figure 14. Control shell elements, group A

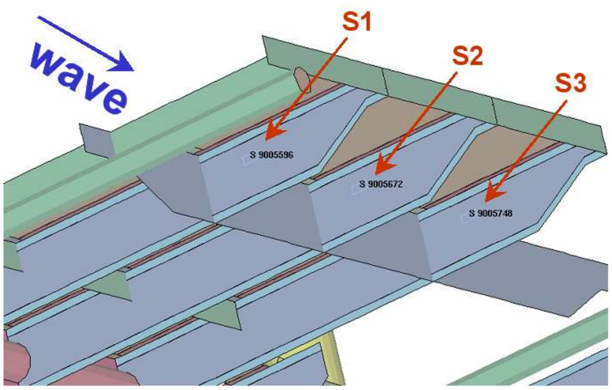

Figure 15. Control shell elements, group B

STRUCTURAL RESPONSE SIMULATION

The structural response analysis was carried out with the FE-code LS-DYNA, [16]. This code is well known and highly regarded by the FEM community and results are typically reliable and trusted, provided that a good engineering practice is followed during the structure modelling and setting of time simulation parameters.

LS-DYNA is an explicit non-linear Finite Element Method code for analysis of large deformation, dynamic response of structures. The analysis takes full account of both material and geometric non-linearities. Material non-linearities include effects of yield strength, strain hardening and failure of steel material. Geometric non-linearities include effects of large deformations, large rotations, membrane stretching of shell elements and local and global instability/buckling.

Particulars of the Structural Model

A 4-node Belytschko-Tsay shell element (5 degrees of freedom at each node) was used to model the cellar deck structure. This type of element is very effective computationally and therefore widely used. In order to obtain correct stress distribution through the wall thickness in elasto-platic range, the shell elements were defined with 5 integration points through the thickness.

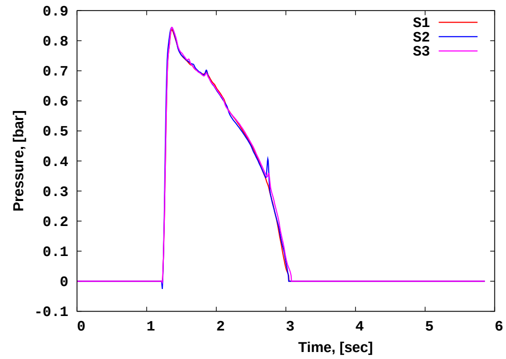

Figure 16. Shell-loading time-pressure graphs, elements in group A

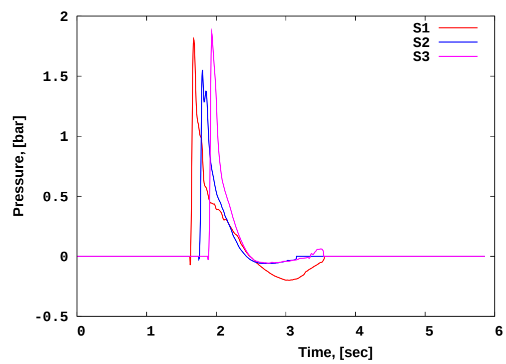

Figure 17. Shell-loading time-pressure graphs, elements in group B

Parameters of structural materials were taken from material specification on drawings. A constant kinematic hardening model was chosen, since the simulation should also account for a shake down (repeated wave load).

A possible fracture in the steel material is included with use of a "layer removal model". In this approach it is checked at every integration point through thickness of the shell elements whether strain reaches a predefined, critical level and then the layer is deleted (removed from further calculations). This is equivalent to reducing of the thickness. It is not a specialized damage model as it does not distinguish between strain caused by tension, compression or bending. However, it finds fractures in a quite realistic way. For example, a shell element working in bending will fracture close to its outer surfaces, but will still be maintained in the calculation as long as the strain in mid-surfaces is below the predefined critical strain.

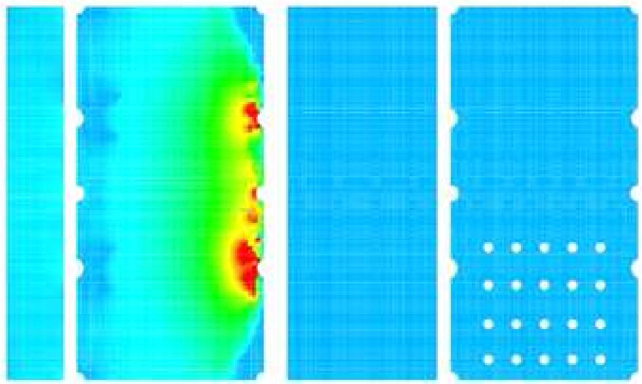

Figure 18. Instantaneous fluid pressures on the deck bottom plating

Time-Load Scenario

Load sequence in the structural simulation was applied as follows:

1. Gravity and wind loads were gradually applied in a quasi-static manner during the first second and then were kept constant throughout the simulation.
2. Detailed time-pressure loads derived from the CFD solution were applied on corresponding shell elements. A load factor (multiplier) can be used if required by regulations.
3. If a shake down calculation is necessary, then the simulation proceeds with time-pressure loads due to the second wave passing through the platform's location. Parameters (height and period) of the second wave depend on the primary wave and are set according to regulations. The shake-down pressures are found with a separate CFD calculation.

Typically, duration of one wave load stage is 4-5 seconds depending on wave height, inundation of deck, and physical size of deck along the wave direction. With the preceding quasi-static gravity and wind load stage and with a possible shake down stage, the entire structural response simulation time can be in range of 5-11 seconds. A typical time step in the structural simulation is 2×10^{-5} [sec], which leads to an overall number of computational cycles in range of 250,000-500,000. The very small time step is determined by the FE-program, as the stress wave should travel less than an element length within a cycle in order to avoid numerical instabilities of the explicit time integration scheme. The time step value used in structural simulation is much smaller than that of CFD (where it is around 2×10^{-3} [sec]), which ensures that the fluid pressure curve shapes are preserved.

Critical Strain Value

Depending of the cellar deck geometry and its physical size, the structure is made up of approximately 200,000-500,000 shell elements. Element sizes are (typically) in range of 50-200 millimeters. Such an element size is adequate for accurate estimation of local and global buckling modes; however, it is not suitable for notch strain capture on a very detailed level. It is necessary to use elements with sizes close to the plate thickness in order to estimate the notch strain towards a weld. This would scale the element sizes down to (for example) 8 mm and, for large structures, would lead to structural models with many millions of shell elements. Predefining a reasonable value of the critical strain is therefore a challenge. According to NORSOK [6], the following value of critical average strain in the material (for an axially loaded plate) may be used in conjunction with nonlinear Finite Element analysis or simple plastic analysis:

$$\varepsilon_{cr} \;=\; 0.02 + 0.65\,\frac{t}{l} \tag{1}$$

where

t – plate thickness
l – length of plastic zone (minimum $5t$)

Using minimum length of the plastic zone gives $\varepsilon_{cr}^{(5t)} = 0.15$. For the discussed structural simulation purposes the failure strain has been set to:

$$\varepsilon_{cr}^{used} = 0.12 \tag{2}$$

In performed simulation, strain in the parent material was mainly caused by local buckling.

When it comes to predefining a critical strain value in proximity of welds, the guidelines given in ULTIGUIDE [7] have been used in the simulation. ULTIGUIDE suggest a limit of 5% nominal tension strain averaged over a length of 20× the plate thickness. This applies to welded connections without cracks, provided that fabrication was performed to obtain overmatching welds. Otherwise, the cracked region should not exceed critical strain of the crack as defined by fracture mechanics assessment. Element size in important parts of the structure is normally far less than 20× the plate thickness, indicating that failure criteria of 5% in the element can be used directly as a conservative assumption. A possibility of pre-defining the failure strain value allows to continue the structural simulation even though some elements fail.

Wave-in-deck accident is normally an issue for old jacket platforms subjected to a seabed subsidence. Typically, condition of welds in the platform deck is not known, and inspection of the entire cellar deck is an enormous task. However, the cracks are almost certain to exist. Plastic strain in proximity of the welds should therefore be evaluated based on engineering judgment of the region importance where this strain level has been reached.

Areas considered to be vital for structural integrity of the platform should therefore be subjected to inspection for cracks if plastic strain is found in these areas. If cracks are found, then the critical strain value can be defined by a fracture mechanics assessment.

Large Displacements and Local Structural Failure

Very large displacements or separation of larger parts of the structure (after a local material failure) cannot be however accepted. Firstly, the CFD simulation is performed on a rigid and fixed in space structure. A large displacement or separation of a large structural part would influence the fluid flow pattern and thus the CFD-calculated pressures would be no longer valid. And secondly, the separated, falling parts may potentially damage the jacket structure.

A structural reinforcement needs to be designed and implemented in the FE-model, and the structural simulation must be re-run, perhaps several times, until an acceptable result is achieved. An evaluation of how the reinforcement influences the fluid flow must be done before the structural re-calculation is performed. If the added parts are flow-exposed and changes to the local flow pattern are likely, then a new CFD re-calculation must be done prior to the structural re-calculation.

Estimation of local notch stresses with use of shell elements comparable in size to the plate thickness can be done by either analysis of a refined cut-out of the global model or re-running the global model with appropriate local areas refined. The later option is preferred, as flexibility conditions of surrounding structure will be much more accurate. Such analysis was performed on bottom plating of the example jacket, as it carried the load mainly by a membrane action.

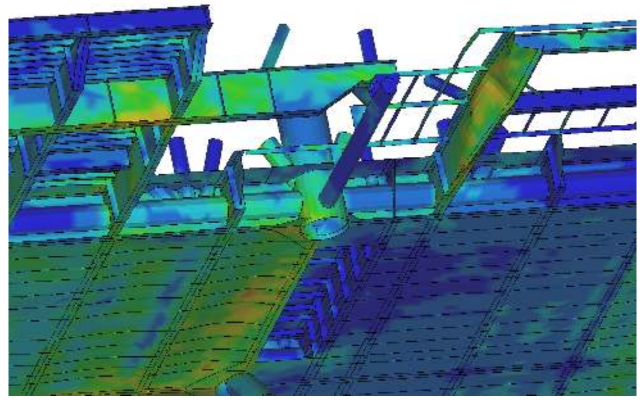

Figure 19. Part of the jacket's deck, von Mises stress

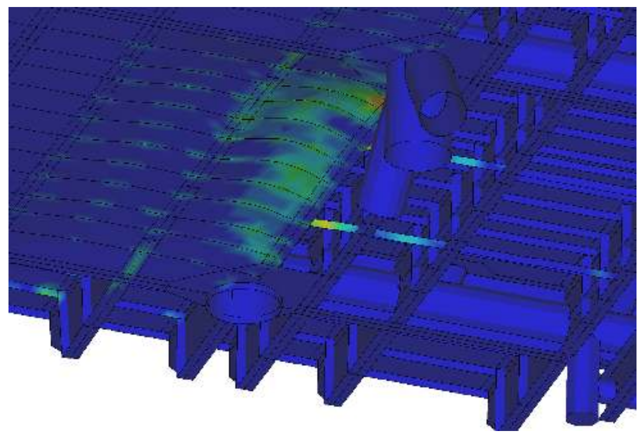

Figure 20. Plastic strain utilization, 5% cut-off

Results of the Structural Response Simulation

A few pictures illustrate results from the structural response simulation. The resultant stress (von Mises) distribution at some time instant is shown in Fig. (19), where substantial deformations of structural parts can be observed. Plastic strain utilization in the bottom plating is shown in Fig. (20), where colours have been scaled to 5 % cut-off in order to improve legibility (red colour means a plastic strain of 5 % or more). Significant deformations of bottom plates are visible again. An example of fractured structural part is displayed in Fig. (21).

Structural Response Simulation, Summary

Numerical simulation of wave-in-deck structural response is nowadays performed mostly with non-linear frame models. Results from the approach presented in this paper indicate that using of a frame model could lead to incorrect conclusions. A detailed mapping of the loading fluid pressure, which varies locally over the structural parts, has proven to be important for response calculations.

Failure modes are often found as twisting of the H-beams, as such beam is restrained against lateral deflection at top-flange due to connection with deck plating. Frame model usually cannot reveal the torsion buckling. Frame model will also estimate capacity of structural members as class 1 sections, independent of their actual slenderness. Typically, the frame-based simulation must therefore be supplemented by a large amount of hand-calculations in order to account for local failure modes.

CONCLUSIONS

1. A solution methodology has been presented for a problem of practical importance. Wave-in-deck loads for a jacket platform have been predicted by a CFD simulation and non-linear transient response of the structure by a FE-code.
2. The applied CFD code ComFLOW uses the VOF method to resolve fluid free surface movement. The employed CFD

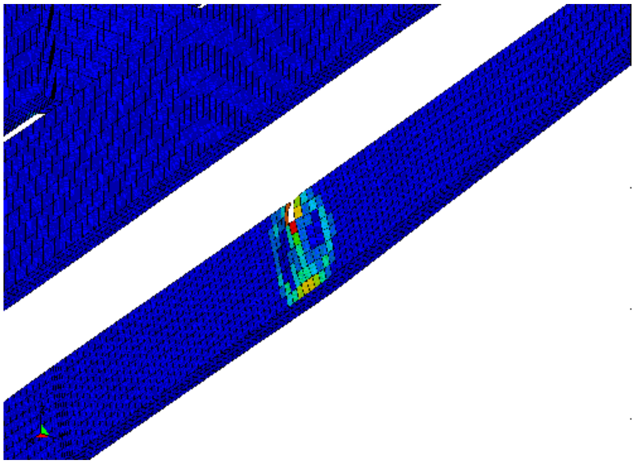

Figure 21. Fracture of an external beam

computational grid is of Cartesian type and is very easy to generate.
3. A structure of arbitrarily complex shape, modelled as FE shell elements, can be submerged into the Cartesian CFD computational grid. The two grids are independent. Fluid pressures obtained from the CFD simulation are directly mapped onto the FE shell elements as time-pressure curves. The structural response calculation is carried out by LS-DYNA FE-analysis code.
4. The pressure mapping procedure is to a large extent automated. Further automation possibilities exist and will be pursued (i.e., assignment of shell element's faces exposed to fluid flow).
5. It is noted that the CFD object is modelled as a rigid and fixed in space body. Body movement caused by the fluid flow and hydro-elasticity are not yet addressed. Improvements in this area are possible and feasibility of their implementation will be explored.
6. An assessment procedure of a deck structure exposed to large dynamic loads has been developed. The method gives reasonable results. It is noted that the calculated strains must be further evaluated against actual status of welds (presence of cracks and type of the weld) and against corrosion of the structural parts.

REFERENCES

[1] Kaplan, P., Murray, J.J., and Yu, W.C., 1995. "Theoretical Analysis of Wave Impact Forces on Platform Deck Structures". In Proceedings, 14-th International OMAE Conference, Copenhagen, Denmark.

[2] Iwanowski, B., Grigorian, H., and Scherf, I., 2002. "Subsidence of the Ekofisk Platforms: Wave in Deck Impact Study. Various Wave Models and Computational Methods". In Proceedings, 21-st International OMAE Conference, Oslo, Norway. Paper OMAE2002-28063.

[3] Brodtkorb, B., 2008. "Prediction of Wave-In-Deck Forces on Fixed Jacket-type Structures Based on CFD Calculations". In Proceedings, 27-th International OMAE Conference, Estoril, Portugal. Paper OMAE2008-57346.

[4] Brodtkorb, B., Nestegård, A., and Johansen, A., 2008. "Prediction of Increased Jacket Substructure Loads Due to Wave-In-Deck Dffraction Based on CFD Calculations". In Proceedings, 27-th International OMAE Conference, Estoril, Portugal. Paper OMAE2008-57361.

[5] NORSOK Standard N-003. *Actions and action effects*. Edition 2, September 2007.

[6] NORSOK Standard N-004. *Design of steel structures*. Rev. 2, October 2004.

[7] Det Norske Veritas, SINTEF and BOMEL, 1999. *ULTIGUIDE, Best Practice Guidelines for Use of Non-linear Analysis Methods in Documentation of Ultimate Limit States for Jacket Type Offshore Structures*.

[8] Rudman, M., Cleary, P., Leontini, J., Sinnott, M., and Prakash, M., 2008. "Rogue Wave Impact on a Semi-submersible Offshore Platform". In Proceedings, 27-th International OMAE Conference, Estoril, Portugal. Paper OMAE2008-57705.

[9] El Moctar, O., Schellin, T.E., and Peric, M., 2007. "Wave Load and Structural Analysis for a Jack-up Platform in Freak Waves". In Proceedings, 26-th International OMAE Conference, San Diego, CA. Paper OMAE2007-29434.

[10] Kleefsman, K.M.T., 2005. "Water Impact Loading on Offshore Structures - A Numerical Study". PhD Thesis, University of Groningen, The Netherlands, November.

[11] Kleefsman, K.M.T., Fekken, G., Veldman, A.E.P., Buchner, B., and Iwanowski, B., 2005. "A Volume-Of-Fluid Based Simulation Method for Wave Impact Problems". *Journal of Computational Physics,* **206**, pp. 363–393.

[12] Hirt, C.W., and Nichols, B.D., 1981. "Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries". *Journal of Computational Physics,* **39**, pp. 201–225.

[13] Wemmenhove, R., 2008. "Numerical Simulation of Two-Phase Flow in Offshore Environments". PhD Thesis, University of Groningen, The Netherlands, May.

[14] Wemmenhove, R., Luppes, R., Veldman, A.E.P., and Bunnik, T., 2008. "Application of a VOF Method to Model Compressible Two-Phase Flow in Sloshing Tanks". In Proceedings, 27-th International OMAE Conference, Estoril, Portugal. Paper OMAE2008-57254.

[15] Iwanowski, B., 2008. ComFLOW-2 JIP: Benchmarks of the ComFLOW Program. Technical Report TR-2008-0049, FORCE Technology Norway AS, Sandvika, Norway, October.

[16] Livermore Software Technology Corporation (LSTC), 2007. *LS-DYNA Keyword User's Manual, Version 971*.