Procedings of COBEM 2005
18th International Congress of Mechanical Engineering
November 6-11, 2005, Ouro Preto, MG

THE EFFECTS OF STIFFENERS MODELED BY SHELL ELEMENTS IN A FPSO 3D FEM HULL ANALYSIS

Henrique M. Gaspar
Escola Politécnica - University of São Paulo
henrique.gaspar@poli.usp.br

Gerson Machado
Escola Politécnica - University of São Paulo
gmachado@solido.com.br

Kazuo Nishimoto
Escola Politécnica - University of São Paulo
knishimo@usp.br

Abstract. *The structure modeling by finite element method of a ship structure takes several hours of engineering and computational process. This kind of structure is usually complex and subjected to many type of loads and the analysis should be approved by Classification Society. Simplifications in the model, such as coarse meshes and different element types could reduce the time spent in the model making, the computer process time and analysis time of the results. This paper shows how the use of shell elements to modeling horizontal and longitudinal stiffeners in the structural analysis of a FPSO unity affect on the value of stress and computational time. The result obtained from these models, as well as their process time, was tabled in this paper and shows how good is the proposal model. The conclusion of the paper is that the simplifications hypotheses adopted in these models improve the accuracy with an acceptable processing time.*

Keywords: *Finite Elements, Modeling and Meshing, Structural Analysis, FPSO*

1. Introduction

The prediction of ship structures stress could be very sophisticated work, especially if consider that structure arrangement change are always possible during the conceptual design stage. This change may lead to built a new model, that takes several hours of model editing and also more computational time for the analysis of the new structure.

The structural analysis of a FPSO (Floating Production Storage & Offloading) hull is a rather complex task (Parunov, 2003) - including a number of analysis that are usually not required for an ordinary sea going ship - especially for the new design concept like the hull of FPSOBR, in which main tank dimensions and hull shapes are not like conventional tanker ship.

For the structural analysis for new concept design as the FPSOBR, the use of some simplifications in the model can help the modeling process and can accelerate the process of vessel structures analyses.

Strong efforts have been made by Clssification Society to obtain reliable stress predictions of the ship & platform structures. Articles like "Simplified Stress Analysis of Ship Structures" (Che, Lee and Libby, 1999) and most recently "Guidance Notes on Finite Element Analysis of Hull Structures", written by ABS - American Bureau of Shipping, discuss about the FEM model simplifications for ships and showed the efficiency (ABS, 2004). The Classification Societies have been developing different type of codes, but based on similar concepts for structural analysis of FPSO's vessels as related in the recent paper by Paranoy (Parunov, 2003). Now, ABS uses Safe Hull, DNV uses Sesam, and Bureau Veritas uses VERISTAR (MARS) for FPSO or tanker ship hull (Parunov, 2003).

The accelerated evolution of the computer processors since 1999 allows the structure engineers the opportunity to use finer mesh density in an acceptable computational time.

Modeling horizontal and longitudinal stiffeners with shell elements could result in a entire shell element model, that can predict primary to secondary stresses, and even tertiary stresses at some areas of interest (Servis *et al.*, 2003).

The example presented in this paper is the structure analisys of the new concept of FPSO units, called FPSOBR, and shows the effects of different mesh sizes and elements in the evaluation of stress prediction of the hull structures in the concept design stage.

2. FPSOBR Unit

The oil produced in Brazil, especially in deep water (more than 1000 meters), is very heavy oil, with an API grade value higher than 20. This oil propriety makes the transport by oil pipelines difficult, and the use of FPSO units for the purpose of separating water and gas on the sea can solve the problem of the transportation even in the far from coast.

FPSOs are usually based on barges or existing tankers, converted to its nem function (Souza, Jacobe and Ellwanger, 1998). The actual FPSO unit working in the Brazilian sea, Campos Basin, is a converted tanker ship. This kind of ship has been presenting some problems along the years due to the operational process, for the design is not prepared to operate as a production platform . Another concern is the hull form, which was previously designed for transportaion system, and not to remain anchored as a stationary units. The vessel also operates on a same place for a long period of time, which demands a higher safety factor in the structure, because the FPSO does not have the regularity of oil tankers dry docking (Parunov, 2003). Therefore, the ship structure designed for dry docking condition is not adequate for stationary unit structure.

Because of this situation, a new concept of FPSO's, called FPSOBR, has been developed by PETROBRAS, in association with research centers and universities, such as University of São Paulo. This unit has been designed to operate like a stationary oil production unit, in order to reduce the motions caused by waves, wind and sea current. With a real lower roll motion compared to converted FPSO's, the axial forces and bending moments acting on risers is mitigated.

The analysis presented in this study are based on the FPSOBR unity, and the main dimensions of the ship can be found in Table 1.

Table 1. FPSOBR Main Dimensions

Length	320m
Breath	50.76m
Depth	34m
Draft Design	27m
Oil Capacity	2.000.000 barrels

The general arrangement of the structure and the bulkhead positions of the FPSOBR is presented in the figure 1 and for the purpose of this paper, the main section of the mid structure was modeled for FEM as shown in the figure 2.

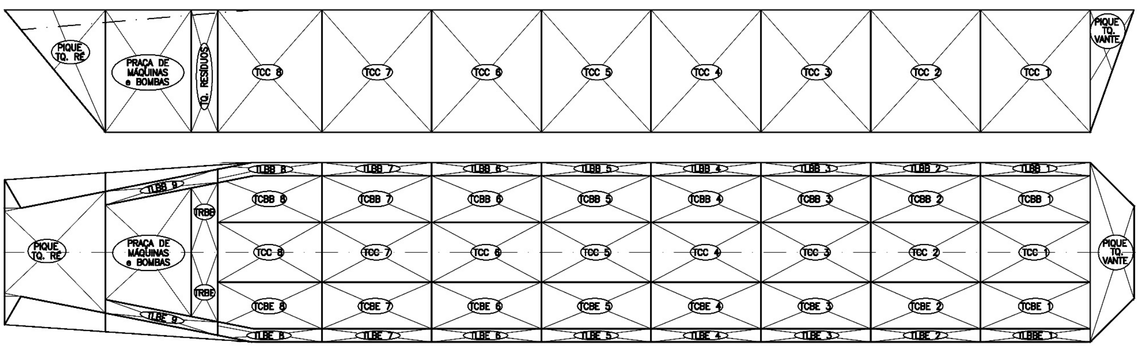

Figure 1. FPSOBR general arrangement of the hull

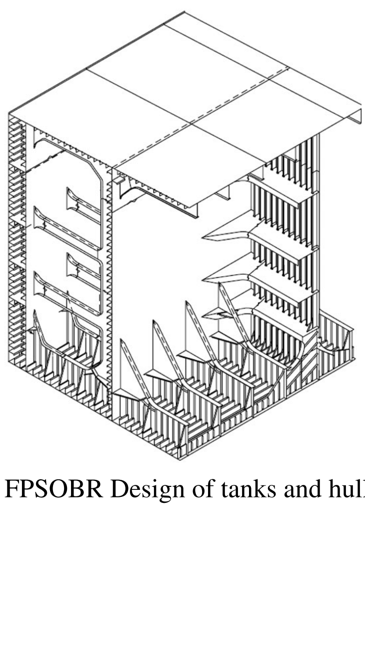

Figure 2. FPSOBR Design of tanks and hull form

3. FEM Modeling and Meshing

The use of hull models of three tanks, boundary conditions and symmetry axes is early proposed by Hughes (pp13, 1983), and discussed in ABS and Bureau Veritas (BV, 1995 and ABS, 2004).

The hull observed in Figure 1 and 2 was modeled and calculated using a finite element software, MSC Patran/Nastran, and their geometric model is presented in Figure 3 and 4. From this geometric model, several models were created and analyzed.

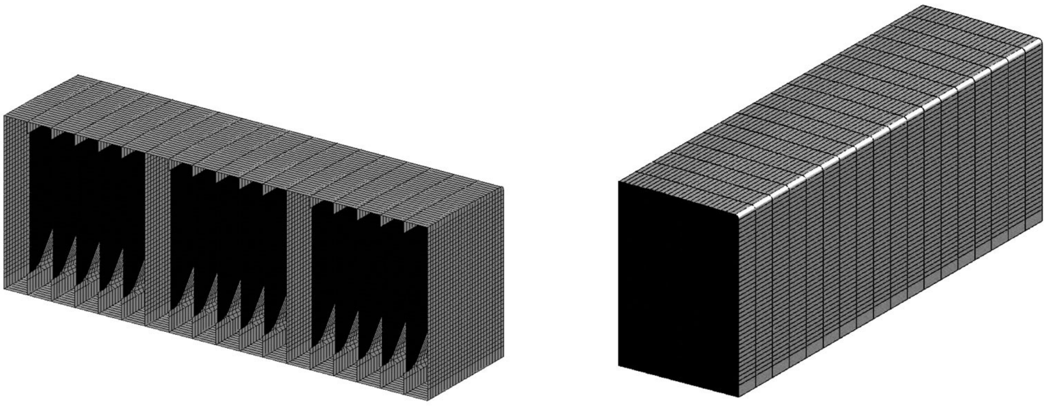

Figure 3. Two isometric views of three cargo hold surface model

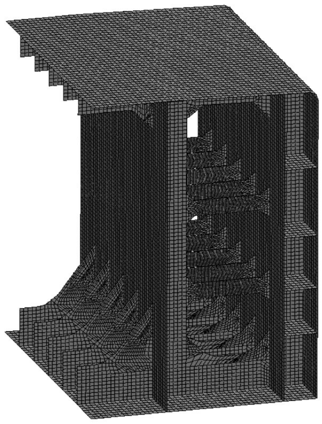

Figure 4. Detail on transversal frames surface model

The two main important differences observed for the purpose of comparing horizontal and longitudinal stiffeners models in the hull analysis are the mesh density and element type. The first is related to the size of each element. The use of a higher mesh density can result in a more accurate stress prediction, but requires more computer resources (Che, Lee and Libby, 1999). However, if an insufficient number of elements are used, the structure will lack the right elasticity, leading to inaccurate results.

The second one is the use of beam and rod elements in lieu of shell elements to model longitudinal and horizontal stiffeners. As recommended by Che, Lee and Libby (1999) and, recently by ABS (ABS, 2004), the use of rod and truss elements should be carefully studied, because this kind of elements lead to a large difference in local stress calculation, since rotational displacements are not computed. Besides, the computational time reduction achieved using these elements instead of beams is minimum.

Based on this assumption, in this work we did not adopted rod or truss elements. As suggested by ABS (ABS, 2004), only shell and beam elements were adopted in our study. The use of shell element to model, the stiffeners can also be used to calculate the tertiary stress in some points, depending on the size of the mesh (Servis *et al.*, 2003). The same does not occur in models in which beam elements are used, for these cases a separate model is always required.

3.1 Traditional Models

There are two main approaches to analyze a ship structure (ABS, 2004). The first is a unique finer 3D mesh model, with a fine mesh size that possibilities well done details and curves to be modeled, mainly in critical areas, and the response of the structure is made with only one model.

The second is a Global 3D FE model with a coarse mesh, which identifies the global displacements and critical strength locations. Together with the global coarse mesh, local models with fine mesh shall be prepared to study hot spot stresses areas. The displacements of the global model are inputted in the local models. A certain amount of engineering judgment must be used to decide the level and detail in which the analysis should be carried out, and one of the aims of this study is to reduce the number of required local models.

3.2 Proposed Model

An efficient simplificated model can be defined as a 3D global FEA model with mesh density enough to evaluate both global hull girder and local main supporting structural yielding strengths (ABS, 2004). A local very fine mesh model may be required when a detail design or critical structural areas demand to be analyzed (ABS, 2004).

The usual mesh size for coarse models should be at least on longitudinal frame space (900mm in the studied FPSOBR). In order to analyze the effect of finer meshes, models with three others mesh sizes were studied: two, three a four elements per longitudinal frame space - respectively 450, 300 and 225mm. The same size was used between transversal frames. All the models were analyzed in six different load cases, approaching static and environmental loads, such as (Machado *et al.*, 2005):

- Hydrostatic pressure of sea
- Hydrostatic pressure of cargo and ballast
- Wave induced hydrodynamic
- Structural weight
- Stools and deck loads

Since the analysis is not a complete dynamic analysis, but dynamics loads is considered as a quasi-static loads, some discussion about sea loads and mooring forces on a FPSO vessel, such as the effect of a correct combination of sagging wave bending moments, topside loads and others is overviewed in "Hydrodynamic and Structural Analysis of a FPSO Ship" (Parunov, 2003).

An abbreviated explanation of load equilibrium and boundary conditions to maintain the model adjusted as a free body is obtained in "Guidance Notes on SAFEHULL Finite Element Analysis of Hull Structures" (ABS 2004). Spring elements in both vertical and horizontal directions are used in this study.

For the calculation of wave induced dynamic pressures to input the pressure field in the structure analysis, the software WAMIT (WAMIT, 2003) was used.

The model used in this work is composed by several plates, bulkheads and horizontals as shown in the figure 2. However, the difference between the traditional and present model is that the stiffeners are usually modeled with beam elements, which is the Type 2 models. In the Type 1 model, all the stiffeners were modeled with shell elements.

Table 2 presents these two main types of models studied in the analysis. The "1 element" information represents that the model has one element per longitudinal spam. The model named Type 1 consists of shell elements only (proposed model). The Type 2 represents the same structure modeled with plate elements and beam elements as stiffeners (traditional model). This model has a lower number of DOF.

Table 2. Element sizes and number of DOF for each model

Type 1 - Only Shell Elements	
Elem. sp	Number DOF
1 element	192882
2 elements	552618
3 elements	1185828
4 elements	2083608
Type 2 - Beam and Shell Elements	
1 element	82794
2 elements	334896
3 elements	752012
4 elements	1406070

Both models were meshed with four different mesh sizes, and the aspect ratio of the mesh is observed in Figure 5.

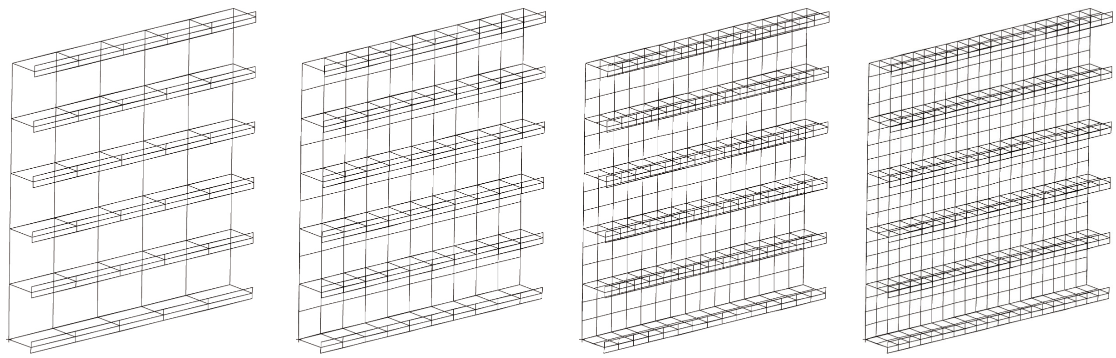

Figure 5. Four different mesh sizes, with respectively one, two, three and four elements per longitudinal spam

4. Discussion & Analysis

4.1 Computational time - CPUs

To evaluate the performance in different machines, the models where analyzed in four different CPUs configurations:

1. Xeon 64 bits, Dual Processor 3.0GHz, 1024KB cache size, 2.0Gb RAM
2. Xeon 32 bits, Dual Processor 3.06GHz, 512KB cache size, 1.0Gb RAM
3. Pentium 4, Processor 2.4GHz, 512KB cache size, 256KB RAM
4. SGI Octane, Processor 300Mhz, 256KB cache size, 512 RAM

The software used to modeling and analysis was MSC Nastran/Patran 2005.

To compare the computational time each model has spent in the different CPU's, a table with the results was drawn. The time value in seconds can be found in Table 3.

Table 3. Analysis computational time for each model in each CPU - time in seconds

Model	Number DOF	Computational Time (s)			
		1. Xeon 64	2. Xeon 32	3. Pentium 4	4. SGI Octane
Type 1 - 1sp	192882	412	443	612	547
Type 1 - 2sp	552618	547	1020	1260	2940
Type 1 - 3sp	1185828	1994	4196	6360	9840
Type 1 - 4sp	2083608	8374	11040	16780	*
Type 2 - 1sp	82794	114	127	230	212
Type 2 - 2sp	334896	496	750	837	1450
Type 2 - 3sp	782012	1354	3342	4890	7542
Type 2 - 4sp	1406070	3894	9630	13080	*
* - Not enough space in hard disk - no analysis computed.					

4.2 Yielding factors

In order to facilitate the interpretation of the yielding result, a frame arrangement of the FPSOBR structure showed in Figure 2 was drawn, which can be observed in Figure 6.

A total of six load cases are applied to these eight models, in which each load condition is observed in Table 4.

Table 4. Load cases applied in each model

LC1	Still Water - full cargo
LC2	Still Water - Empty cargo
LC3	Sagging wave - full cargo
LC4	Sagging wave - empty cargo
LC5	Hogging wave - full cargo
LC6	Hogging wave - empty cargo

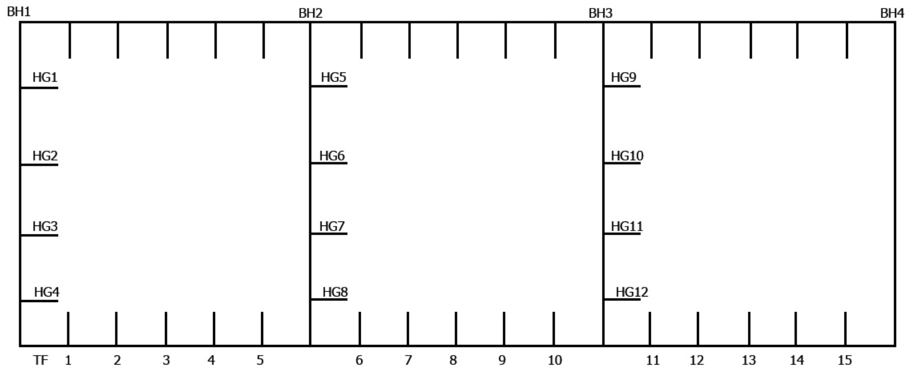

Figure 6. Frame arrangement of the FPSOBR vessel

Yielding evaluation is performed and represented by yielding factor, which is the ratio of baseline case value at each location for the FPSOBR model with two element type combinations (Che, Lee and Libby, 1999). The Type 1 - 4sp were the model used as baseline. Five different places are observed in Table 5, in accordance to the frame arrangement numeration from Figure 6.

4.3 Discussion on Results

All stress values were normalized with the Type 1 - 4sp value for each load case and the ratio for the yielding factors can be found in Table 5.

Table 5. Comparison of yielding Factors for the FPSO with eight different models

	Deck (TF8) LC2	Side (TF8-9) LC5	Hot Spot (Frame) (TF8) LC1	Hor. Girder (HG6) LC4	Bulkhead (BH2) LC6
Type 1 - 1sp	0.91	0.91	0.71	0.88	0.98
Type 1 - 2sp	0.92	1.00	0.85	0.94	0.99
Type 1 - 3sp	0.94	0.96	0.97	0.95	0.99
Type 1 - 4sp	1.00	1.00	1.00	1.00	1.00
Type 2 - 1sp	0.90	0.93	0.70	0.84	0.98
Type 2 - 2sp	0.92	1.01	0.84	0.96	0.99
Type 2 - 3sp	1.03	1.07	0.96	1.02	0.99
Type 2 - 4sp	1.04	1.08	1.13	1.06	0.99

Is assumed that Type 1 - 4sp model have a finer mesh, and the stress value converge to a more accurate result. The accuracy of the solution will increase continuously as we continue to refine the finite element mesh. As the mesh refinement was performed by subdividing a previous used element in two or more elements, the old mesh will be "embedded" in the new mesh and the dimension of the finite element solution will be continuously increased to contain ultimately the exact solution (Bathe, pp 229, 1996).

The element stresses are calculated using derivatives of the displacements, and the stress obtained at an element edge (or face) when calculated in adjacent elements may differ substantially if a coarse mesh is used (Bathe, pp245, 1996) and the stresses were redistributed throughout the plate panels (Hansen, 1996), leading to a lower result than expectated. This effect was observed in both models with an element size of one longitudinal frame space (Type1 - sp1 and Type2 - sp1). The stress differences at the element boundaries decrease as the finite element mesh is refined. This effect could be observed in Table 5, with the refinement of the mesh at both models, leading to a convergence in the result.

One of the benefits of applying only shell elements is the facility of using only one type of element, simplifying the model. Also, it is possible to observe the effects, like wrapping and buckling, in local areas and stiffeners. Certainly, local analysis should be required in very critical local areas to yielding assessment, but this kind of model can decrease the number of required local analysis, especially in hull design processes. The negative effect is the increasing of the computational time.

The element with size with 1/4 of space is adequate to determine stress distribution in local areas, and the Type 1 - 4sp model has mesh size enough to reduce the need of local model. However, for the observation of hot spot stress areas and small openings, which are free from structural discontinuity and/or weld based, this mesh space is not adequate. Local models with 1/16 and 1/64 size are recommended (ABS 2004).

Even though, the FPSOBR model during conceptual design stage has frequently changed, and a quick and reliable analysis is required from the engineer. As observed in Table 3, the computational time for a model with more than 2.000.000 DOF can be rather long time processing, with more than four hours length in a Pentium 4 machine. However, with the modern techniques of parametric modeling, the use of a global geometry with only one type of element can decrease the model assembly time.

If computer resource is available, the FE model with at least an element size of 1/4 of longitudinal spam is recommended, because this approach will reduce the need for further local analysis. Otherwise, an efficient model can be done with coarse shell mesh or beam elements instead of shell elements to model longitudinal stiffeners and flat bars. This combination increases the modeling efforts but decreases the computational time. Extra additional local detail analysis may be required in comparison to a Type1 4sp model. However, the use of shell elements as stiffener lead to a better distribution of the strain energy, increasing the displacements. When stiffeners are modeled with beam elements, the hypotheses of the beam theory are used, leading the structure to a higher stiffness, increasing the stress value.

5. Conclusion

Through the present work, the following conclusions can be summarized:

. The use of shell elements to modelling horizontal and longitudinal stiffeners, resulting in an entire model with one elemet type, is an efficient method for calculating the stresses values in a FPSO hull structure.

. Stress values may differ if a coarse mesh is used, especially in hot spot areas. The calculated value in coarse meshes are substantially lower than the real value.

. The accuracy of stress results will increase continuously as the finite element mesh is refined.

. The use of an element size equal to four elements per longitudinal frame space (Type 1 - sp4) can reduce the number of required local models to analyse hot spot areas, with the negative effect of increasing computational time.

. An efficient model can be done with the combination of shell and beam elements. This combination increases the modeling efforts but decreases the computational time.

. Extra additional local detail analysis may be required in coarse mesh models and when is used beam elements as stiffeners.

6. Acknowledgments

The authors would like to thank PETROBRAS for their contribution on this study. Appreciation is given to Dr. Marcos Donato from PETROBRAS/Cenpes, and Fábio Okamoto from USP, for their comments and suggestions. The National Agency of Petroleum, Natural Gas and Biofuels (ANP) and Numerical Offshore Tank (TPN) also supported this research.

7. References

ABS, 2004, "Guidance Notes on Finite Element Analysis of Hull Structures - Local 3D Model Analysis", American Bureau of Shipping Rules and Guides, http://www.eagle.org/rules/downloads.html.

ABS, 2004, "Guidance Notes on Safehull Finite Element Analysis of Hull Structures", American Bureau of Shipping Rules and Guides, http://www.eagle.org/rules/downloads.html.

Bathe, J.K., 1996, "Finite Element Procedures", Prentice Hall.

BV, 1995, "Guidelines for Structural Analysis of Oil Tankers", Bureau Veritas, NR 399 DJM R00E, Tentative Rules, February 1995.

Che, X., Lee, F.K. and Libby, D.H., 1999, "Simplified Stress Analysy of Ship Structures", American Bureau of Shipping - Technical Papers, Marine Issues.

Hansen, A. M., 1996, "Strength of midship sections", J. of Marine Structures Vol. 09, pp 471-494..

Hughes, O.F., 1983, "Ship Structural Design", Ed. John Wiley & Sons, Sydney, Australia, 570p.

Machado, G., Nishimoto, K. *et al.*, 2005, "RT-005-06-06 Memória de cálculo estrutural - FPSOBR (in portuguese)" Technical Report, USP-PETROBRAS.

Parunov, J., 2003, "Hydrodynamic and Structural Analysis of a FPSO Ship", Journal of Naval Architecture and Shipbuilding Industry, 4-2003.

Servis, D., Voudourious, G., Smuelides, M. and Papanikolau, A., 2003, "Finite element modelling and strength analysis of hold No. 1 of bulk carriers", J. of Marine Structures Vol. 16, pp 601-626.

Souza, M. J., Jacob, B. P. and Ellwanger, G. B., 1998, "Structural design of process decks for floating production, storage and offloading units", J. of Marine Structures Vol. 11, pp 403-412.

WAMIT, 2003, "WAMIT - the state of art in wave interaction analysis - User's Manual", WAMIT Incorporations.

8. Responsibility notice

The authors are the only responsible for the printed material included in this paper.