From Hydrodynamic Loading to Global Strength Assessment of Offshore Vessels

Timo Wallnöfer

Author contact: timo.wallnoefer@gmail.com

S.M.I.L.E.-FEM GmbH
Heikendorf, Deutschland

Summary

This document presents the analysis procedure of the global strength assessment of a column-stabilized semi-submersible-unit, following the Mobile Offshore Drilling Units (MODU) code by the classification society American Bureau of Shipping (ABS). It is originated during a project for a Danish customer and aims at the classification of the structural steel drawings of the vessel.

First, a hydrodynamic diffraction analysis is carried out using ANSYS AQWA. The resulting response amplitude operators (RAOs) are compared to model test provided by the Danish model tank basin institute FORCE technology.

A short overview of the incorporation of mass items and their centre of gravity (COG) is given. This aims at the generation of section forces and moments of the vessel due to hydrodynamic pressure and acceleration loadings.

Then, the determination of the design waves is presented, five critical global hydrodynamic loads are identified and their parameters given.

Finally, the hydrodynamic loads are mapped on the global FE model for all critical load cases. The loaded models are solved in ANSYS and the stress results are evaluated according to the ABS rules. The critical parts and potentials for the improvement of the design are shown.

Keywords

Hydrodynamic analysis, AQWA, load mapping, Response Amplitude Operator, offshore

1 Introduction

1.1 Project Description

For an existing design of a column-stabilized semi-submersible unit, the global structural strength is assessed according to classification society rules and guidelines using ANSYS AQWA and Mechanical. The analysis procedure is divided into two main steps and followed by a detailed assessment of the results. First step is the hydrodynamic analysis, whose major results are the RAOs and the design waves. In the second step, the hydrostatic and hydrodynamic loads are mapped on the structural model and stress results are evaluated. For a complete global structural strength analysis, the buckling strength and the fatigue strength of the main connection is to be analysed according to classification society rules. In order to keep the extent of this paper reasonable, only the stress assessment is presented here.

1.2 Vessel Overview

Figure 1 shows the design of the vessel, which is divided into four main parts: the twin pontoon hulls below the water line, the eight columns piercing the waterline, the barge hull over the full breadth and the accommodation decks above the main deck.

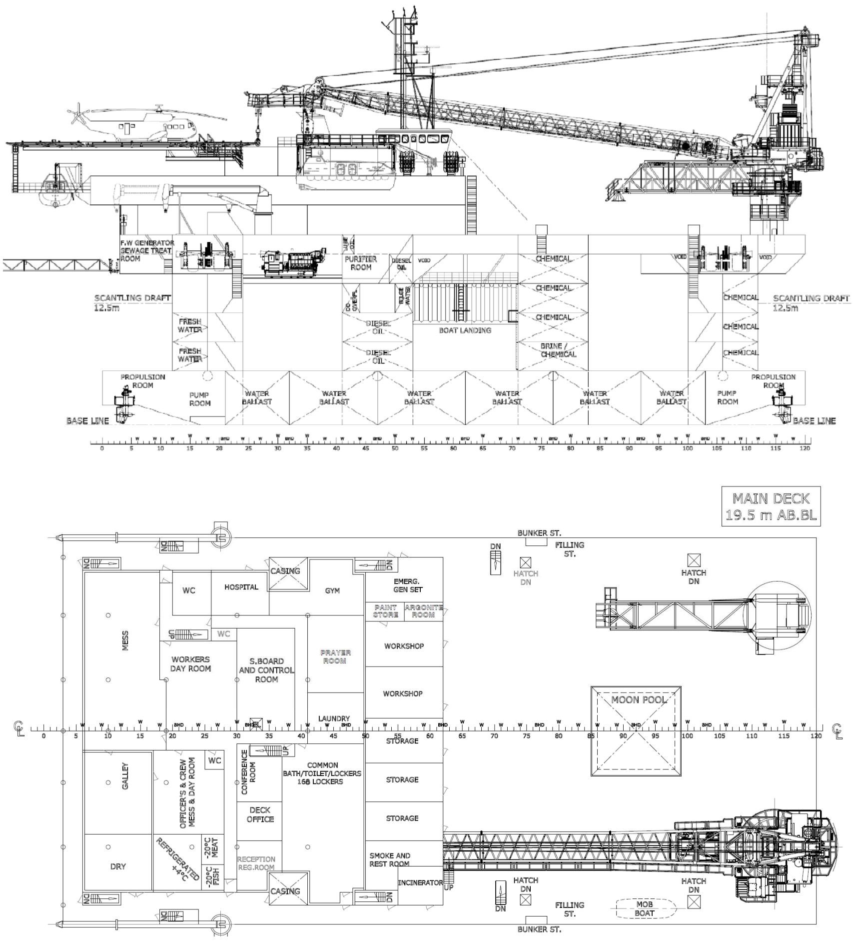


Figure 1 General Arrangement side and top view

1.3 Loading conditions

For classification, multiple loading conditions are to be verified, usually operational, survival and transit condition. Operational condition refers to one or several typical deep draughts with a large amount of ballast water in order to lower the centre of gravity and increase the total mass. This is done to improve the vessels motion characteristics while the daily work like operation of cranes etc. takes place. Survival condition is engaged during storm periods where no works are carried out. The vessel is ballasted in order to increase the air gap between the barge hull and the still water line, so no slamming loads are inflicted. Transit condition is the lowest draught where only the pontoons are immersed in order to decrease the drag and therefore the necessary propulsion power.

For this presentation the survival loading condition is chosen, as it is associated with the most severe wave environment.

2 Hydrodynamic Analysis

2.1 Model description

For the hydrodynamic analysis, a surface mode, consisting of the outer shell planes, is built in the 3-D modelling tool SolidWorks and imported to ANSYS Mechanical APDL via Parasolid. As the hydrodynamic diffraction model only needs the wetted surface of the vessel, the vertical extent of the model ends at the column to barge hull connection. No appendages like anchors, thrusters, fairleads and boatlanding are modelled and the horizontal and diagonal bracings are included as BEAM188 elements. The surface model is meshed into quadrilateral and trilateral panels using mapped meshing where possible. The element size is chosen in order to depict the geometry of the outer hull properly. This results to a highest wave frequency of f_{max}=2.2 Hz which corresponds to the shortest wave period of T_{min}=0.45 s. As the element shape is very important for the diffraction analysis, attention is paid to stay within the element growth rate and warping limits. Note that at the forward and aft ends of the pontoon hulls, this not always possible without an objectionable decreasing of the element size. Shape warnings are issued for a small amount of panels at this location, which is not deemed to degrade the analysis quality substantially and therefore ignored.

Figure 2 and Figure 3 depict the hydrodynamic model mesh. Note that the colouring indicates the panel normal direction which is set to pointing inward for all diffracting elements. This is of high importance for the algebraic sign of the pressure loading. Also note the global coordinate system, the origin is located on the midships section, on the centre plane at the still water line, with x-axis pointing forward, y-axis portside and z-axis upwards.

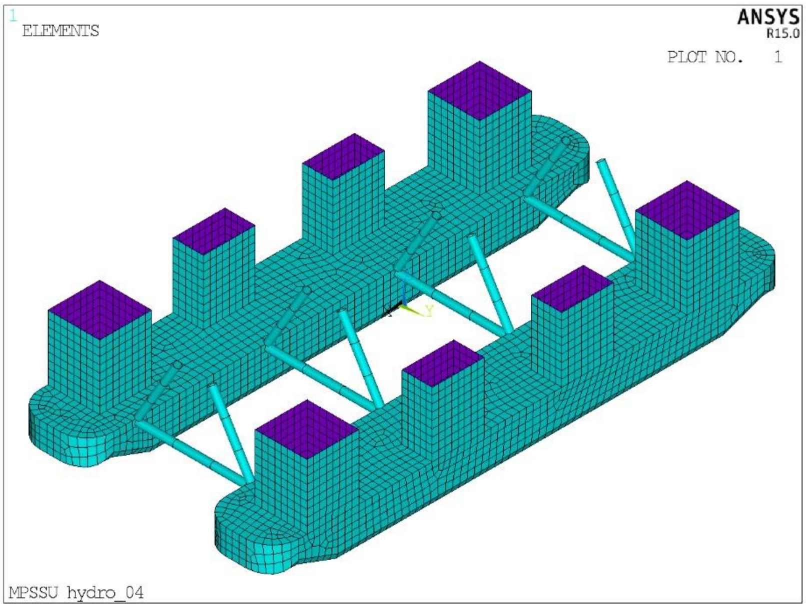


Figure 2 Hydrodynamic model top view isometric

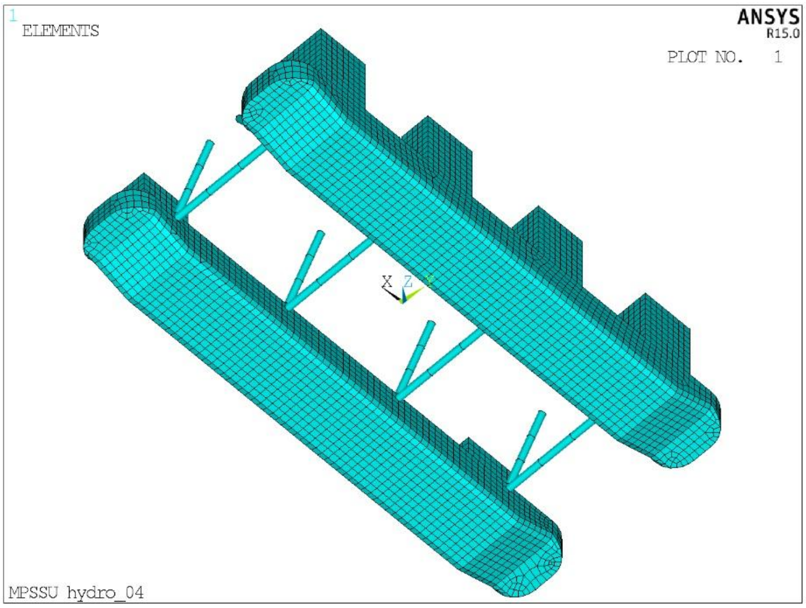


Figure 3 Hydrodynamic model bottom view isometric

2.2 Mass and COG Description

The mass of the vessel is incorporated differently in the stages of the hydrodynamic analysis. For the hydrodynamic diffraction analysis with AQWA LINE, the entire mass of the vessel is summarized as one point mass in the centre of gravity and assigned 6 moments of inertia.

For the design wave calculation and the following load mapping process, a more detailed separation of mass items is needed. Therefore, the vessel is subdivided into three items:

1. Structural steel parts, which are modelled in the FE model as beam and shell elements.
2. Major equipment items, which are modelled in the FE model as point masses in their respective COG, connected to the structural elements using link elements of infinite stiffness.
3. Tank items, which are modelled in the FE model as point masses on the nodes of the tank's boundary structural shell elements.

2.3 Global and Wave Parameter

The water depth is set to D=1000 m, the water density to ρ=1025 kg/m³. No current or forward speed is incorporated.

The analysed wave parameters are chosen to be a period range of 50 periods, starting at T_1=62.83 s to T_2=2.86 s and 12 directions from -180° to +180° in steps of 30°. The following table comprises details of the wave parameters:

NUMBER	**FREQUENCY ω**	**PERIOD T**	**WAVE NUMBER K**	**WAVE LENGTH λ**	**MAX ELEM SIZE**	**DEPTH**	**RATIOS**
[-]	[rad/sec]	[s]	[-]	[m]	[m]	D/λ	K*D
1	0.100	62.83	0.00122	5167.8	738.3	0.19	1.22
2	0.113	55.85	0.00144	4355.1	622.2	0.23	1.44
3	0.125	50.27	0.00170	3691.1	527.3	0.27	1.70
4	0.138	45.70	0.00200	3142.8	449.0	0.32	2.00
…	…	…	…	…	…	…	…
45	1.829	3.43	0.34107	18.4	2.6	54.28	341.07
46	1.903	3.30	0.36928	17.0	2.4	58.77	369.28

47	1.978	3.18	0.39862	15.8	2.3	63.44	398.62
48	2.052	3.06	0.42909	14.6	2.1	68.29	429.09
49	2.126	2.96	0.46067	13.6	2.0	73.32	460.67
50	2.200	2.86	0.49337	12.7	1.8	78.52	493.37

Table 1 Wave parameter details

2.4 Additional Input

In order to improve the diffraction analysis results, several additional effects are accounted for via AQWA LINE commands.

2.4.1 Free Surface Effects

In order to incorporate the free surface moments of partly filled tanks, the hydrostatic stiffness matrix of the AQWA LINE database is modified manually. The program first calculates the hydrostatic stiffness based only on the cut water plane and displaced volume properties. It then adjusts the second moments of area I_{XX}, I_{YY} and recalculates its associated properties, PHI (principal axis), GMX/GMY, BMX/BMY etc. to give the required GM values. The associated additional hydrostatic stiffness is calculated automatically and stored in the hydrodynamic database.

2.4.2 Additional Damping

In potential flow analysis, no viscous effects are included. In order to account for this, additional damping is introduced as frequency independent, linear added damping. The amount of additional damping is determined based on the results of an initial decay test, performed during the model tests. As result of these tests, the mean damping to critical damping ratio ζ and the Eigen periods T_E are evaluated for the six degrees vessel motions of surge x, sway y, heave z, roll φ, pitch θ and yaw ψ.

Of the six first Eigen periods evaluated in these test, only the values of heave, roll and pitch motion are within a range where possible resonance with the exciting wave periods seems possible. Therefore, damping effects only on these motions are accounted for, using the FIDD command as manual input of additional damping.

In order to determine the absolute values of the added damping, first the critical damping of the motion concerned is calculated with $c_{crit} = \sqrt{2 \cdot k \cdot m}$. Here, the stiffness k and the mass m are both dependent on the period. They are calculated with a preceding AQWA LINE run at sampling points where the wave period matches the particular Eigen periods. Furthermore, the actual damping in the potential flow model is read out at these periods. The added damping value can then be determined as difference between the target damping and the existing damping.

2.4.3 Suppression of Irregular Frequencies

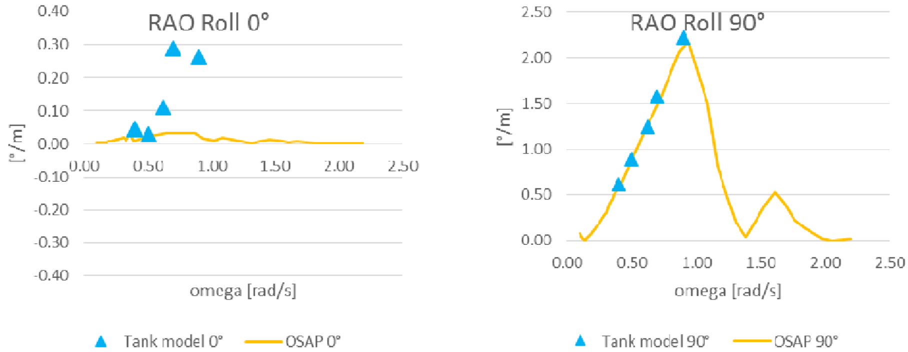
As suggested by the AQWA LINE reference manual for double hull structures, additional lid elements are introduced. These horizontal elements are created by the software on the water plane level in order to suppress irregular frequencies which occur as a result of standing waves between the two hulls. Note that in this analysis, the effect of irregular frequencies was observed in a reference study to be not identifiable in the RAOs. Nevertheless, the elements are to be introduced, since the effect on second order forces might be more distinct.

2.5 Response Amplitude Operator Results

In this section, the results of the hydrodynamic diffraction analysis are presented as graphs of the RAOs over the wave angular frequency ω. They are compared to the results given in the tank model tests. On the diagrams, the tank model test results are denoted as blue triangles and the results of this analysis as yellow line. For this loading condition, the results generally are in good agreement with the measured data. Especially for the relevant motions of heave, roll and pitch, the agreement is excellent. Some exceptions are found at inappropriate combinations of motion and wave direction:

- RAO surge 90°: low amplitudes, as expected, with high variation
- RAO yaw 90°: very low amplitudes, high variation due to the sensitivity for yaw moment
- RAO surge 60° and 120°: over-prediction of motion
- RAO roll 30°: outlier of a single measurement data point

Table 2: RAO results heave motion



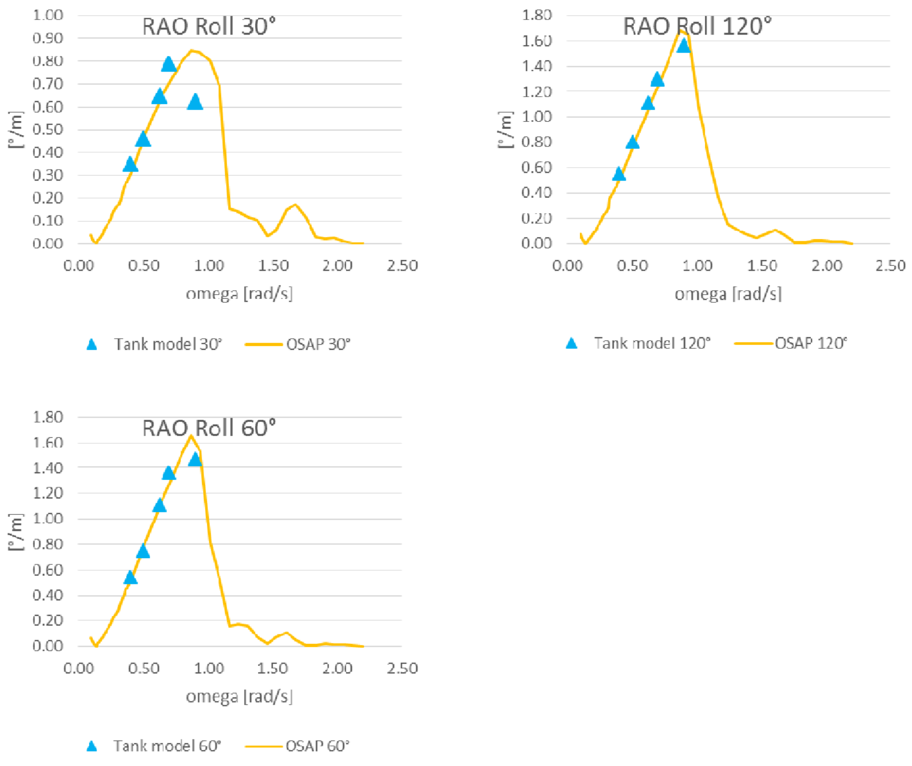


Table 3 RAO results roll motion

2.6 Design Wave Calculation

As described in the MODU code 3-2-A2/7 [2], the design waves for the vessel are selected following the deterministic approach. For this, the steps i) to v) of the code are consequently processed. First, the critical wave headings and lengths are determined and seven critical global hydro dynamical loads are identified. These are:

- Split force between pontoons, denoted FY1
- Twisting pitch moment about transverse horizontal axis, denoted MY1
- Longitudinal shear force between pontoons, denoted FX1
- Vertical wave bending moment on the pontoon, denoted MY2
- Inertia forces induced by longitudinal, transverse and vertical accelerations of deck mass, denoted ACCX, ACCY and ACCZ

Then, the owner selected design wave environment is set to be the sea sate of 100-year typhoon in the South China Sea, with the following parameters: peak period T_p=9.7 s and significant height H_{100}=6.87 m. The wave steepness S is selected following the recommendation in [1]. Here, the steepness is derived as function of the period with the following equation:

$$S = \begin{cases} \dfrac{1}{7} & \text{for } T \leq 6\text{s} \\ \dfrac{1}{7 + \dfrac{0.93}{H_{100}}(T^2 - 36)} & \text{for } T > 6\text{s} \end{cases}$$

Following the steepness limit, the corresponding wave height is derived with the following equation:

$$H = \begin{cases} 0.22T^2 & \text{for } T \leq 6s \\ \dfrac{T^2}{4.5 + 0.6/H_{100}(T^2 - 36)} & \text{for } T > 6s \end{cases}$$

Applying this relation, the limiting regular wave heights are calculated for wave periods ranging from 3 to 15 s with steps of 0.25 s, which results in 49 sea states as input for the design wave analysis.

Finally, the RAOs calculated in section 2.5 are used for the calculation of the response load by multiplication of the limiting regular wave height at each period with the RAO. The resulting accelerations are applied on the distributed masses of a sectional model in an external software in order to compute the section forces and moments for all wave periods.

The maximum results of this calculation are found by the software and chosen to be the design waves. Careful examination of the resulting design wave and comparison to the proposed design wave cases in [1] and [2] is recommended. The following table comprises the design waves for this loading condition.

#	Direction	Period	Height	Response	Description
[-]	[°]	[s]	[m]	[-]	[-]
1	-180	7.50	8.973	FX01	iii) longitudinal shear force
2	90	6.00	8.000	FY01	i) split force
3	-45	7.50	8.973	MY01	ii) twist pitching moment
4	-45	7.50	8.973	MY02	v) vertical wave bending moment
5	-180	7.00	8.695	ACCX	iv a) longitudinal acceleration
6	90	6.00	8.000	ACCY	iv b) transverse acceleration
7	0	8.25	9.323	ACCZ	iv) c) vertical acceleration

Table 4 Design waves for survival condition

The direction of the design wave, as well as the wave length, correspond to the nature of the load. This correlation is described in detail in [1], section 4.6.

3 Global Strength Assessment

3.1 Model Description

The model is built based on structural steel drawings provided by the customer, where the scantling was pre-estimated. The structure is modelled by surface and line bodies in Solidworks. Here, several parts are combined to sub-components and components according to the mentioned subdivision of the vessel. The calculation model for the global strength analysis is established by importing the geometry model into the ANSYS software. The geometric properties of the structural members, like plate thickness and cross section dimension are read in at this point and a finite element mesh is generated using quadrilateral, 8-node elements (SHEL181) with half the frame spacing of 300 mm as maximum size. Note that this element size is further reduced at geometrical transitions. For line elements, 2-node beam elements (BEAM188) and 2-node truss elements (LINK180) are used.

The two following figures depict the global mesh.

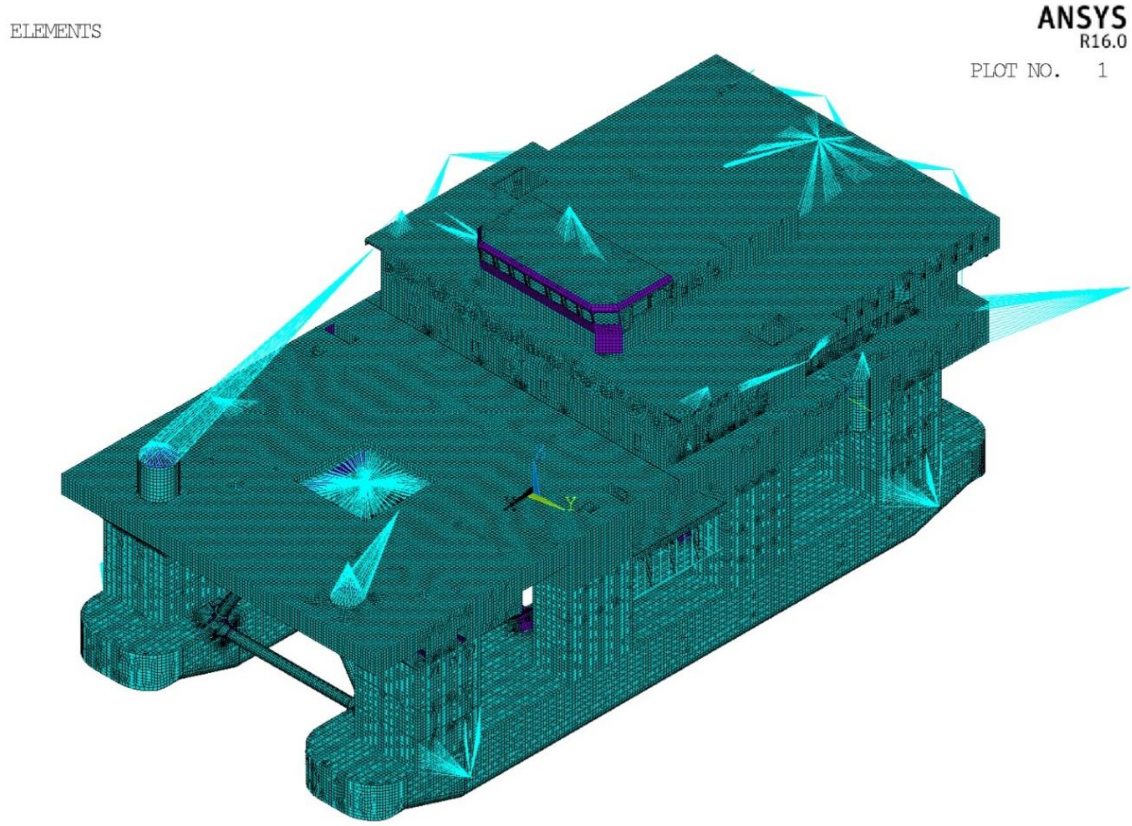


Figure 4 Global model mesh depiction, top view, isometric

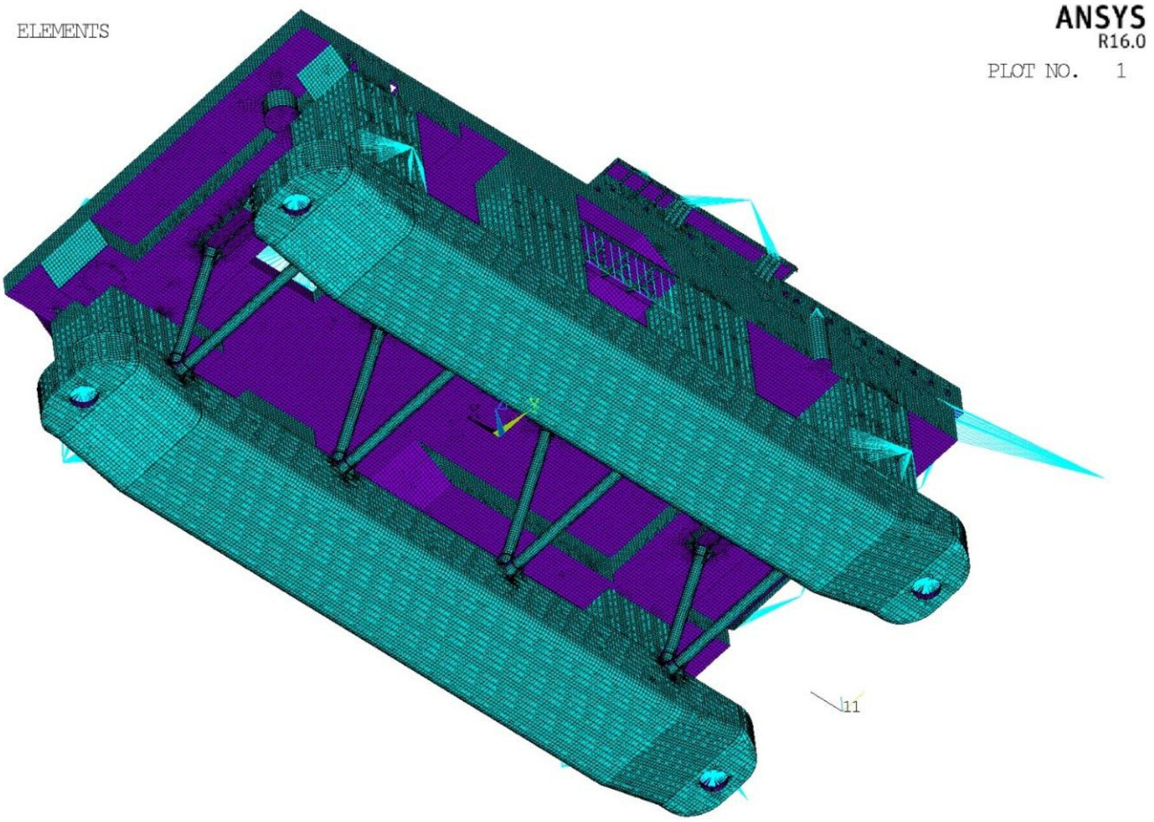


Figure 5 Global model mesh depiction, bottom view, isometric

3.2 Mass and COG Description

The structural elements of the FE model have a defined mass given by their volume and specific gravity, whereas all non-structural masses like equipment and outfitting as well as all tank loads have to be taken into account separately. In this case, the additional masses are accounted for in three different ways:

1. The additional masses of particular equipment are modelled as point mass elements at their respective COG and connected to the affected structural members. The connection to parts like foundations, attachment points, deck girders etc. is done using mass-less link elements of infinite stiffness. This way, the mass properties of the items are incorporated comprehensively. Moments of inertia of the items are also incorporated, as far as they are known at this stage.
2. Some additional masses, which cannot be assigned to a particular COG, are distributed on structural elements via an adapted density. This is done to account for heavy bulk items like the interior of the accommodation structure, provision stores, personal & effects etc.
3. The additional mass of tank fillings are incorporated as point mass elements on the nodes of the surrounding structural elements. The mass of one element is given by the total mass of the tank divided by the number of tank nodes.

All structural and non-structural mass elements have to be in equilibrium of moments with the single mass element taken into account in section 2.2 in order to depict the same loading condition, see section 3.4 for additional information.

3.3 Load Mapping

The process of load mapping applies the hydrostatic and hydrodynamic pressures on the diffracting panels to the pressure loading of the shell elements in the structural model. Additionally, rotational and translational acceleration loads are applied on all mass elements based on the vessels motion. This process is done for loads due to the calculated design waves and loading conditions and results into a number of load cases for the global finite element model. Two options for this load mapping process exist in ANSYS AQWA and Mechanical, which are presented in the following.

3.3.1 First Approach: AQWA WAVE and ASAS model

As part of the AQWA suite, AQWA WAVE is used in this analysis for load mapping of hydrodynamic and hydrostatic loads based on an AQWA LINE database. For this, the structural model is first to be converted into the ASAS format, using the ANSTOASAS command. This command creates an ASAS input file from the current ANSYS model with all structural and non-structural elements included. In order to identify the elements on which hydrostatic and hydrodynamic loads are to be mapped, a surface element load of irrelevant magnitude is mapped on the wetted surface elements. This is also used for the pressure direction control, which is generated by the element surface normal vector and the surface pressure load algebraic sign.

In the run file for AQWA WAVE, the load cases are identified by giving the following wave parameters as input: height H, period T, direction μ and phase angle ε. In this approach three load cases are generated for each design wave: the first at ε=0°, which refers to the wave crest at the origin of the coordinate system, the second at ε=90° and the third at ε=180° to cover the phase progression of the ocean wave. The disadvantage of this approach is, that the phase progression is only depicted rudimentary and possible maxima between the three sampling points may be ignored. Of course additional sampling points may be introduced, but each of these has to be load mapped, balanced and solved in the FE analysis.

An improvement of this method is the usage of complex pressure definition. Here, two load cases are defined with ε=0° and ε=90° as real R and imaginary I part of the complex pressure. A linear load effect E at any time instance in the wave can then be obtained by combination of the real and imaginary part using the following formula:

$$E(t) = R \cdot \cos(\omega t) + I \cdot \sin(\omega t)$$

This procedure is implemented in ANSYS in the LCOPER command with the CPXMAX option enabled.

3.3.2 Second Approach: Harmonic Ocean Wave Procedure HOWP

A different method of mapping the loads from a hydrodynamic analysis to a structural model is outlined in the harmonic ocean wave procedure. Here, the load mapping process is performed in ANSYS using SURF154 elements for shell pressure application and PIPE288 for line bodies. The hydrodynamic data is imported with the OCDATA command and the wave parameters identified with OCTABLE. The number of phases at which the hydrodynamic loads are calculated is set via the HROCEAN command.

This procedure is found to be very promising and a verification for the global model is planned for future work.

3.4 Load Balancing

The next step is the definition of boundary conditions in the global FE model. As the hydrodynamic loads include hydrostatic pressure, the loads on the hull surface from static and dynamic pressure should be in equilibrium with the inertia loads from gravity and the vessel's motion. As the wetted hull panel elements in the hydrodynamic model and the shell elements in the structural model are not of the same size or number, extrapolation is necessary and therefore the integrated pressure on the two models is not equal. Furthermore the accelerations in the hydrodynamic model are derived using one mass element with 6 moments of inertia, whereas the structural model constitutes this total values from a multiplicity of structural and point mass elements. As result, the inertial loads are not equal, too. All this constitutes the need of additional boundary conditions, where residual forces are evaluated as reaction. The magnitude of these reaction forces is a measure for the balancing status of the loaded model and is presented as fracture of the total weight. Attention has to be paid to not influence the global deformation of the model it in an unrealistic way. In order to achieve that, three nodes are specified as boundary conditions, following the recommendations in [1], section 4.4.8:

1. One node at the port, most aft column, at pontoon deck elevation is constrained in all three translational degrees of freedom (DOF).
2. One node at the port, most forward column, at pontoon deck elevation is constrained in the translational DOFs in y and z direction.
3. One node at the starboard, most forward column, at pontoon deck elevation is constrained in the translational DOF in z direction.

As evidence for a balanced model, the total residual horizontal forces on all three nodes in one direction shall not exceed 2% and the total residual vertical force 4% of the total displacement force of the vessel. In case of an unbalanced model, the weight distribution has to be adapted in order to represent the input values of the hydrodynamic analysis in a more congruent way. In addition a more congruent elementation of the hydrodynamic and the structural model can help with the balancing, but for this both models have to be recalculated.

3.5 Stress Results Evaluation

As described in section 3.3.1 three load steps are solved for each design wave. Each of this load steps is defined as load case and read in the database using the LC*** command family. Subsequently, a search for the maximum equivalent stress result for each element is performed using the LCOPER command. After that, the load case combination of the maximum equivalent stress results are appended as additional load step for later reference using the RAPPND command.

The results are compared to the maximum allowable stress in the beam and plate elements. See the following figures for a depiction of the maximum VON MISES equivalent stress in Pascal. Note that the wave parameters are given with H, T, D and P for the first load step of the design wave, while the step is denoted 9999, because of the load case combination.

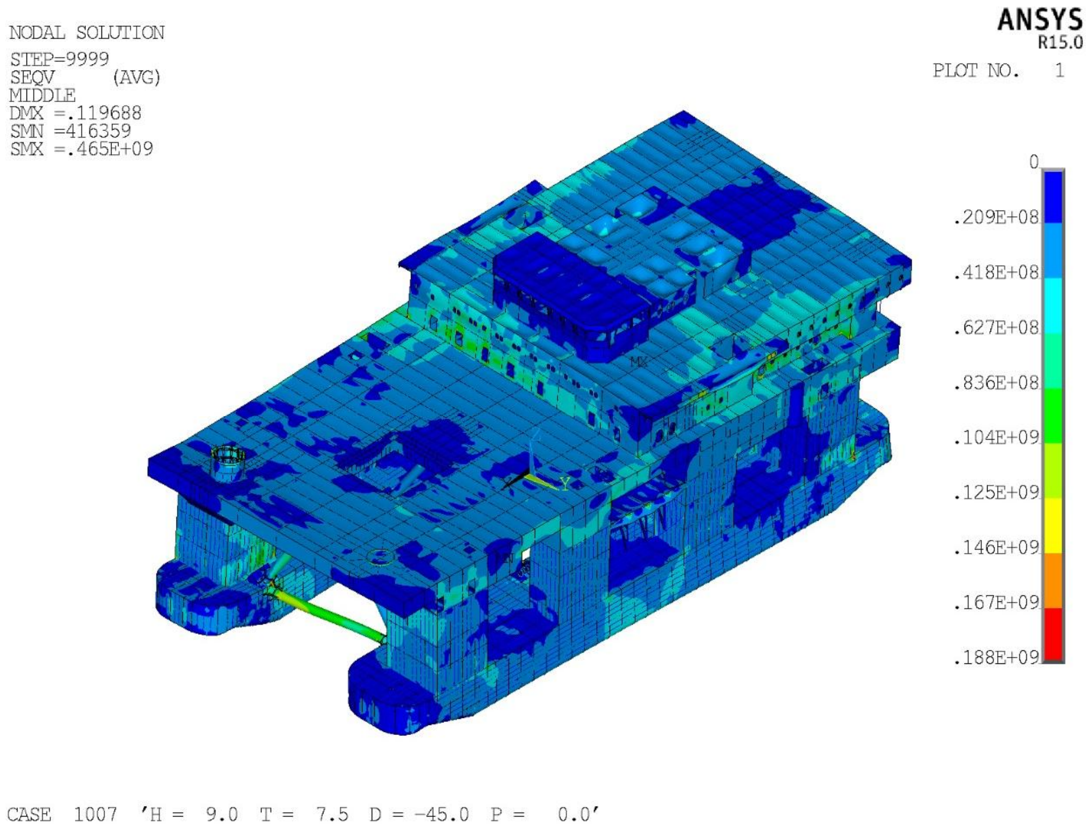


Figure 6 Design wave 3 equivalent stress results global model, top view isometric

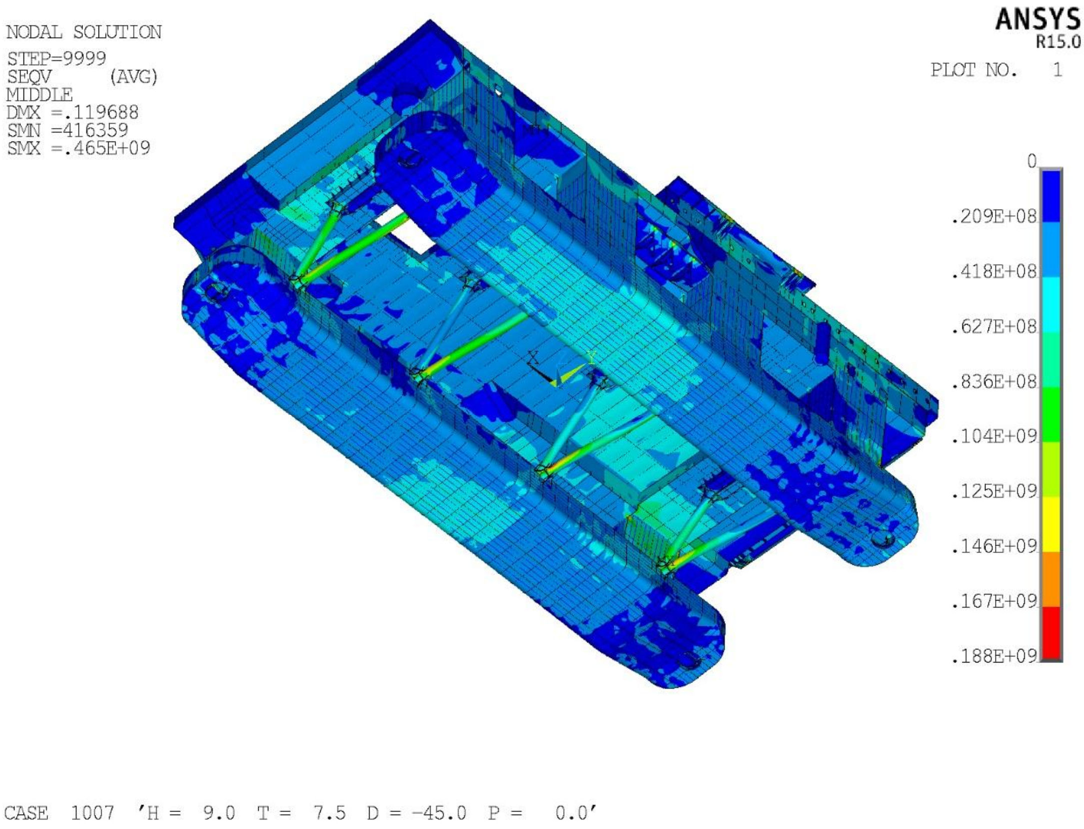


Figure 7 Design wave 3 equivalent stress results global model, bottom view isometric

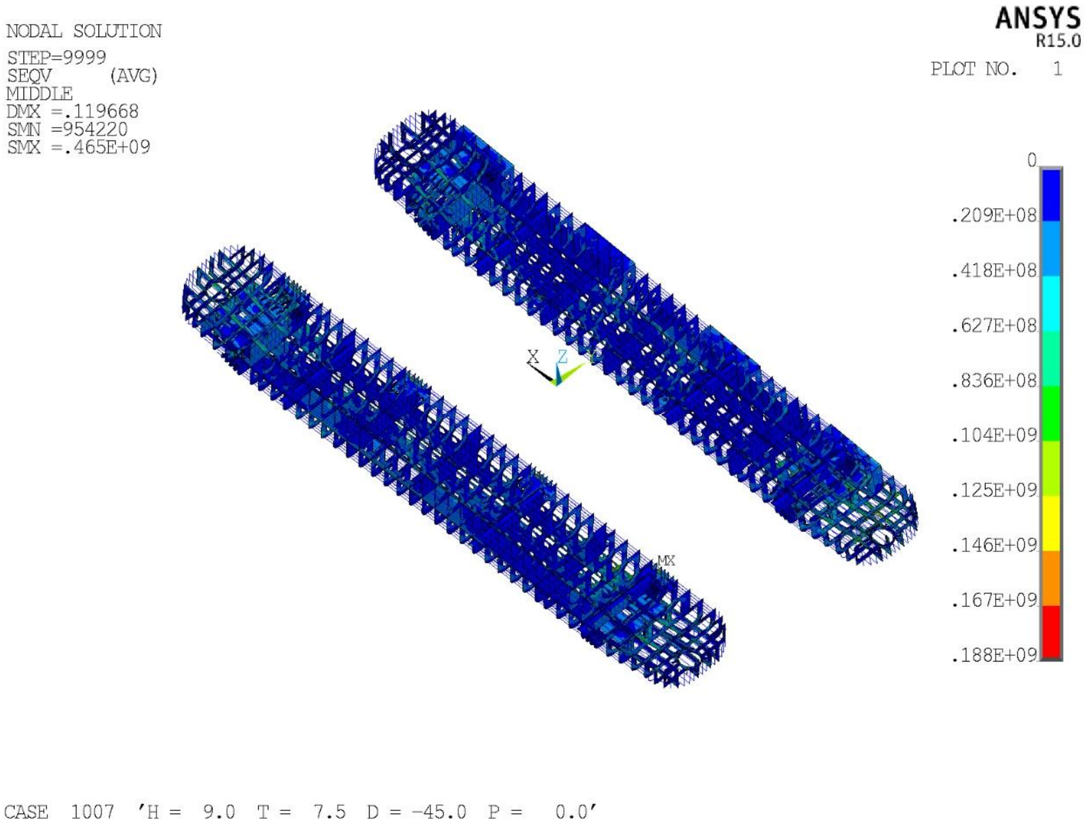


Figure 8 Design wave 3 equivalent stress results, pontoon hulls girder grid, bottom view isometric

As the classification rules demand buckling analysis for the global strength assessment, the combined stress results may be used for a linear buckling analysis in ANSYS. The alternative is the buckling assessment by determination of the maximum allowable stress results for buckling in girders, stiffened plate panels and columns according to [3]. This is most easily done by using a software tool provided by the classification society. The implementation of the rules into ANSYS using MAPDL macros seems also feasible, but is not performed at this stage.

The stress results are also the basis for a fatigue analysis, which is not part of this article, in order to keep the extent of this paper reasonable

4 Summary

The assessment procedure starts with the hydrodynamic analysis description. The model and elementation is shown, the mass and COG implementation explained and global and wave parameters set. Additional options are given and the results are presented as RAOs. The computation of design waves according to the rules is presented.

The second step is the global strength assessment, again where the model and its elementation is explained. Mass and COG properties are defined and compared. Then, two methods of load mapping are described and their benefits shown. For this analysis, the first method is chosen for its simplicity. The importance of the load balancing of the model is presented. Finally, stress results are show and assessed.

5 Conclusions

It can be stated that the combination of the modelling tool Solidworks with the ANSYS AQWA suite and Mechanical / MAPDL is used very successfully for the global strength assessment of floating offshore structures. As the integration of AQWA into the workbench is not compatible with all AQWA LINE commands, the usage of MAPDL scripts for the generation of the hydrodynamic model is recommended.

The results from the hydrodynamic diffraction analysis are in exceptional agreement with the model test results, so the analysis procedure itself is deemed to be reliable and viable for other geometries and loading conditions. As the classification rules do not demand model test explicitly, a numerical seakeeping analysis may be sufficient for design. Additional model tests are recommended anyhow.

The results of the global strength analysis are easily post processed as stress results with ANSYS. The buckling analysis is also available in ANSYS, but has to be adapted for classification purpose. An automated post processing of the buckling results is deemed feasible in MAPDL, but not undertaken.

6 References

[1] Recommended practice column stabilised units, DNV-RP-C103, Det Norske Veritas, 2012

[2] Rules for building and classing Mobile Offshore Drilling Units, American Bureau of Shipping February 2014

[3] Guide For Buckling And Ultimate Strength Assessment For Offshore Structures, American Bureau of Shipping, 2004