Fatigue Analysis of a Tension Leg Platform: Fatigue Life Improvement

Adrian Hita Espejo

Master Thesis

presented in partial fulfillment

of the requirements for the double degree:

“Advanced Master in Naval Architecture” conferred by University of Liege

"Master of Sciences in Applied Mechanics, specialization in Hydrodynamics, Energetics and Propulsion” conferred by Ecole Centrale de Nantes

developed at West Pomeranian University of Technology, Szczecin

in the framework of the

Supervisor: Dr. Maciej Taczala, West Pomeranian University of Technology, Szczecin

Reviewer: Prof. Patrick Kaeding, University of Rostock

Szczecin, February 2016

ABSTRACT

A fatigue life analysis of a Tension Leg Platform (TLP) was performed; concretely it was verified how the fatigue life (F.L) of a TLP is affected by means of structural modifications in critical fatigue areas. The main aim is quantify the effect of such modifications to guarantee a good fatigue design without over-dimension certain structural elements. For the study, a standard design of TLP is used to obtain realistic results, exposed to waters of the North Sea.

A Stochastic fatigue analysis using linear wave theory was considered. A Potential Flow Solver combined with Morison Formulation coupled with a Finite Element solver, is used to determine the pressure distribution generated by the waves on the structure and mooring line loads. The loads on tendon elements are neglected in this study. After determined, loads are transferred to the Global model and the fatigue study is performed to identify the critical fatigue areas. For the structural assessment, a quasi-static analysis is used since dynamic effects are not to be so relevant in this particular case of study.

Once critical areas are identified, a local analysis is developed considering the hydrodynamic loads from the global model and the boundary conditions are applied in terms of displacements. From the local model, the hotspots are identified and the fatigue life is determined.

Since known the loads and boundary conditions to be applied in the local model, structural modifications of the identified critical areas are proposed and the effect of such modifications in the F.L is quantified. In addition, the effect of surface treatments in the F.L is also verified. The treatments considered for the study are Grinding, TIG dressing and Hammer peening.

It is expected that when increasing the thicknesses of certain structural elements, the F.L will tend to increase proportionally to such increments. Nevertheless, the aim is to quantify in an accurate way the cyclic life improvement to provide a guideline to guarantee a good fatigue design without over-dimensioning, valid for TLP’s to be installed in the North Sea

CONTENTS

List of Figures

List of tables

Declaration of Authorship

I declare that this thesis and the work presented in it are my own and have been generated by me as the result of my own original research.

Where I have consulted the published work of others, this is always clearly attributed.

Where I have quoted from the work of others, the source is always given. With the exception of such quotations, this thesis is entirely my own work.

I have acknowledged all main sources of help.

Where the thesis is based on work done by myself jointly with others, I have made clear exactly what was done by others and what I have contributed myself.

This thesis contains no material that has been submitted previously, in whole or in part, for the award of any other academic degree or diploma.

I cede copyright of the thesis in favour of the West Pomeranian University of Technology, Szczecin

Date: *Signature*

1 INTRODUCTION

The development of the offshore field, its improvement and the global oil-gas production have been going hand in hand. In order to have an overview about the offshore field, is necessary to understand how the market had been behaved the last years.

Fossil fuels, including coal, oil and natural gas, are still the world's primary energy source. Formed from organic material over the course of millions of years, fossil fuels have fuelled U.S. and global economic development over the past century.

The Offshore Structures had played a vital role to fulfill the increasing demands of fuels and energy in today’s world. Offshore oil & gas production is forecast to grow from 39 million barrels oil equivalent (boe) per day in 2004 to 55 million boe by 2015. From providing around 34% of global oil production and 28% of global gas production in 2004, offshore oil & gas is expected to reach 39% and 34% respectively by 2015.

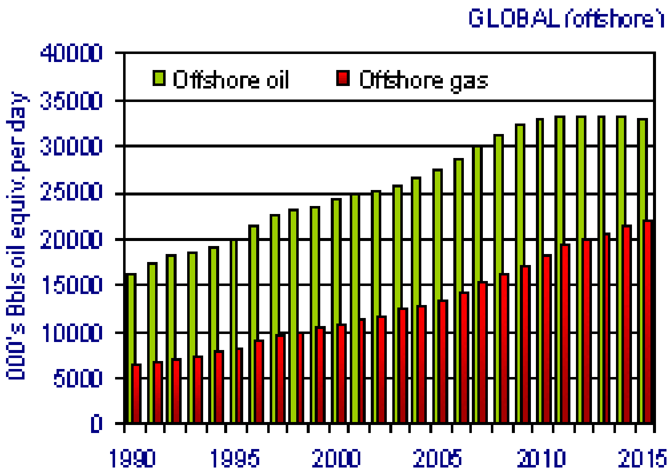

Figure 1-1 Offshore oil & gas production in 1000’sBpd between 1990 and 2015. Available from http://www.marinetalk.com/articles-marine-companies/art/Offshore-Oil-and-Gas-Industry-Projections-DOU006103030OT.html

Of course the production increments had not been free of charge. Although in the beginning the increment was associated to improvements in the drilling and productions methods, inevitably the increase in the production have supposed to explore new remote areas at ultra-Deep-water.

As consequence of this development of the industry, the new designs for floating offshore structures are required to be as robust as possible to face extreme environmental condition.

These structures will be largely exposed to significant cyclic stresses, generated principally by the waves. Loads will be applied thousands of cycles during the lifetime of the structure, having as consequence the fatigue. After many cycles the accumulated damage reduces the ability of the structural member to withstand loading. Global Fatigue analysis is one of the approach applied to quantify wave induced load effects. Based on this analysis the integrity and structural safety of the offshore platform can be ensure.

The load environmental exposure combined with a corrosive medium will affect the structure integrity and modify the estimations of the un-corroded condition, thus this issue should be consider during such type of analysis.

Due to the significant investment that such type f structures represent, the industry and owners interest is to guarantee longer in-service periods, and increase it if possible by recognized techniques or methodologies... Previous scientists had presented in the past succesfull methodologies that nowadays are considered by Class Societies as recommended practices for fatigue life improvement.

Between these different proposals, the most common are:

- Reduce the residual stress.
- Increase the thickness of the critical area.
- Avoid welding in areas where low ductility or fracture toughness steel is (e.g. K areas of wide flange members, corners of hollow steel sections).
- Provide adequate protection from the environment (e.g. provide measures to prevent galvanic action between dissimilar metals).
- Design details to reduce stress and strain concentrations.
- Improve surface conditions.
- Improvement of fatigue life by fabrication: Grinding, TIG dressing and Hammer peening.
- Use of high-performance alloys resistant to corrosion fatigue. This measure have the disadvantage that for most engineering applications this approach may not be practical, due to the availability and cost of these alloys.

1.1 Objective

The purpose of this master thesis is quantify the influence of certain structural modifications on the fatigue life (F.L) of an offshore platform, concretely in a TLP. In order to offer tangible results a design from an existing platform will be used to generate the structural global and local modelling.

Two main objectives are set:

- Quantify the effect of in the F.L of the following structural modifications:
    - Increment in the thickness of critical areas.
    - Improvement of fatigue life by fabrication: Grinding, Tungsten inert gas (TIG) dressing and Hammer peening.

The main reason to quantify these improvement is to offer concrete effect of this measures in the fatigue design, mainly related to the critical areas which are identified by the classification societies.

- Reduce the number of design iterations necessaries to achieve an optimal fatigue design

Once the effect of this measures will be quantified, the designer will have the possibility to applied alternative solutions for problems like:

- Critical points in the structure (hot spot), which exceed the limit usage factor.
- Shortfall of fatigue life in a critical area.

1.2 Scope of Work

Presented the objective of this master thesis, a methodology is proposed for the study of fatigue life improvement in TLP. The scope of the analysis methodology used, is mentioning below:

- Structural model developed by finite element method (FEM):
    - Global model:
        - Principal structural elements are modelled using 4- and 8 node shell elements. Longitudinal and transversal bulkheads and hull shell are modelled in such way.
        - Secondary structural elements are modelled with 2-node beam elements. Stiffeners are modelled in this way.
    - Local model:
        - All the structural elements are modelled using 4 and 8 node shell elements.
- The Hydrodynamic analysis is performed in the frequency domain, where:

Modelling of Hydrodynamic loads is divided in Inertial and drag load calculations. In case of inertial load modelling for large bodies:

- Inertial forces are determined using Potential to calculate first order radiation and diffraction effects on large volume structures.3D panel method to evaluate velocity potentials and hydrodynamic coefficients with direct numerical integration to determine the pressure distribution on the hull surface.
- First order velocity potential is used to determine the linear wave loads.

In the case of the drag force modelling for small bodies:

- Drag forces are determined using Morison formulation solved with the wave kinematics determined with the diffractive potential.

Modelling of large bodies with considerable drag effect.

- Under the presence of large bodies with non-negligible drag forces, inertial loads are calculated by Potential flow Theory and separately Morison formulation is used to determine drag forces on such bodies considering the characteristic length and wave kinematics from the diffractive potential. I.e. superposition of both effects is applied to determine the loads on such bodies to correct forces, damping and added mass matrices.

- Quasi-static analysis of the structural response for the global and the local models.

Structural response due to dynamic loading is analysed by a quasi-static method since the frequencies of the load, wave's frequencies in this case of study, are far enough from the eigenfrequencies of the structure.

- Stochastic linear fatigue analysis based on s-n data

In this analysis linear load effects and responses are assumed. The total fatigue damage (Miner sum) is computed by summing up the contributions from each sea state taking into account their probability of occurrence, wave spreading and the corresponding S-N curve.

- Hot spot calculation base on refine finite element mesh.

The size of the mesh used in the hot spot area correspond to t by t meters, being t equal to the plate thickness.

- Structural modifications of the TLP quantified:
    - Increment in the thickness of the critical area.
    - Improvement of fatigue life by fabrication: Grinding, Tungsten inert gas (TIG) dressing and Hammer peening.

2 OFFSHORE STRUCTURES: PLATFORMS

2.1 Classification Attending to Mobility Criteria

In this chapter a classification of the offshore platform attending to their mobility, as well as a brief description about the structure of a TLP, types, purpose and operational areas is proportionated.

There are different types of offshore platforms, a main classification could be establish attending to the mobility. Being divided between: fixed and floating platforms.

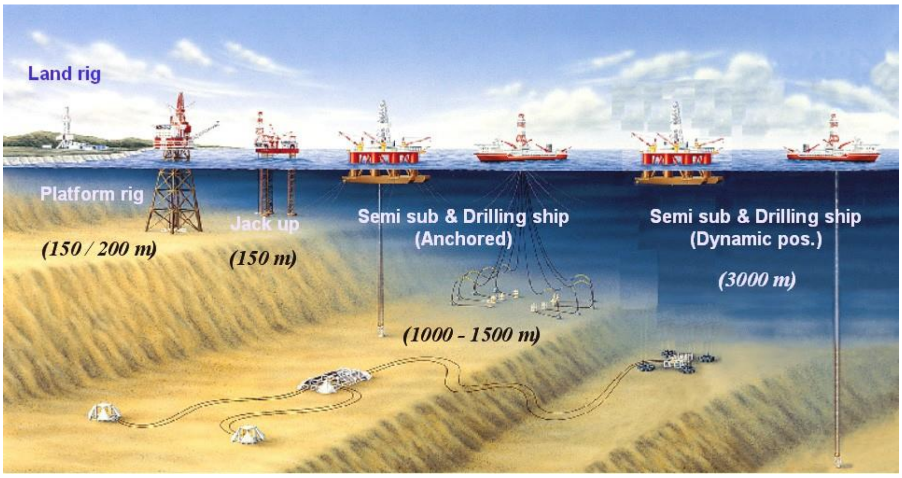

Figure 2-1 Different types of offshore rigs. Available from http://rigbook.blogspot.fr/2012/08/rig-type.html

The expenses associated with fixed production platforms at depth waters are no longer within a feasible range making a floating production platform design a far more economical choice. Certainly, the usage of floating platforms have advantages and disadvantages respect fixed platforms:

- Advantages
  - The cost associated with jacket type platforms
  - The floating structure offers earlier oil production.
  - The floater offers a lower risk solution when the nature of the field is uncertain.
  - The floater may provide oil storage
  - Well testing phase can be extended and overlapped with the production phase, in order to obtain a better idea of the long term production rates without committing to the large risk of a field specific platform.

- Drawbacks
  - Movement of the platform due to the sea state, larger waves, so that drilling or production operations may have to be suspended during storms.

2.2 Tension Leg Platform

A TLP is defined as a buoyant unit connected to a fixed foundation (or piles) by pre-tensioned tendons which acts as mooring system. The tendons are normally parallel, near vertical elements, acting in tension, which usually restrain the motions of the TLP in heave, roll and pitch. The platform is usually compliant in surge, sway and yaw. Figure 2-2 shows an example of a tension leg platform.

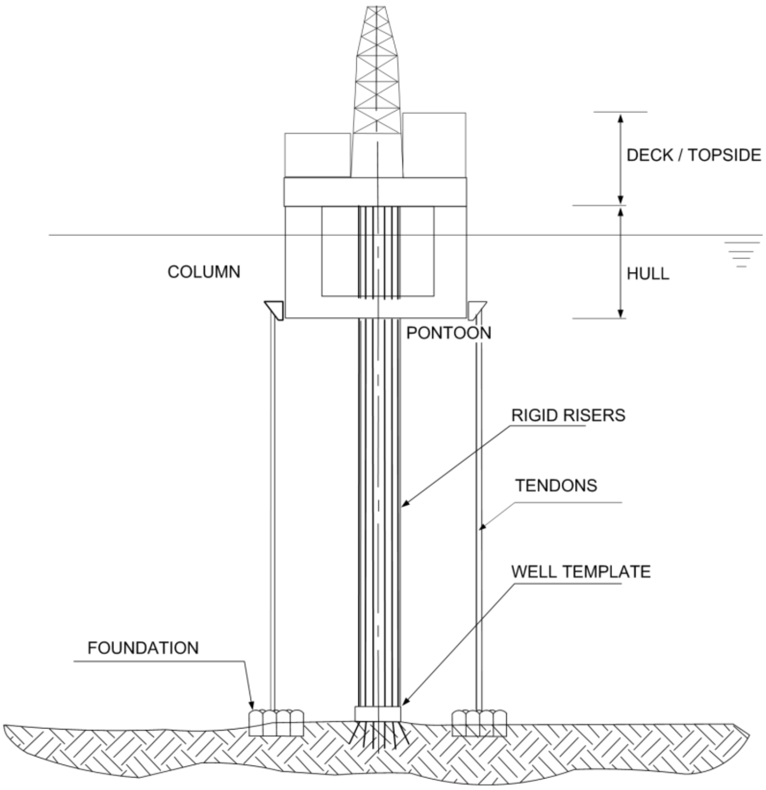

Figure 2-2 Example of a tension leg platform Available from Offshore standard. DNVGL-OS-C105

The mooring system is composed by a set of tendons, which are providing the link between the platform and the foundation (build with tubular sections, solid rods etc.). Normally, due the lengths of the tendon elements, intermediate connections of couplings along them are required.

A TLP is usually applied for drilling, extraction and production and export of hydrocarbons. Storage may also be a possible TLP function; finally, we can mention that this kind of structures are worldwide located, in which can be mentioned the North Sea, Gulf of Mexico, Southeast Asia and West Africa as main locations.

2.2.1 Types of TLP

Currently, there are three different types of TLPs: full-size TLPs, mini TLPs and wellhead TLPs. Corresponding the TLP analysed to the first type.

- Full-size TLP: also known as conventional usually composed by four columns connecting the pontoons with the deck. Normally the mooring system is connected to the column´s bases

Figure 2-3 Example of Full-size TLP. Available from http://www.shipspotting.com/ships/ship.php?imo=8768725

- Mini TLP: In this case, the platform also has four column but the distance between columns have been reduced. As a consequence the columns ar not located at the pontoon edges. The legs, were the mooring system is connected, are situated farther than the columns respect to the centre of gravity.

Figure 2-4 Mini TLP. Available from
http://www.modec.com/fps/tlp/about/index.html

Other possibility is to have just one center column. This model offer some advantages such as: small size, moderate cost, simple construction and installation.

Figure 2-5 SeaStar mini-TLP. Available from
http://www.offshore-mag.com/articles/print/volume-56/issue-6/news/general-interest/tlp-technology-seastar-minimal-platform-for-small-deepwater-reserves.html

This type of TLP are more for smaller discoveries. While Full-size TLPs, which have been built to support drilling rigs or completion units ,are appropriate for larger discoveries.

3 TENSION LEG PLATFORM ANALYSED

The case being analysed consist in a TLP with a lower hull composed by pontoons, where a square disposition creating a ring is adopted. The connection of the upper hull is achieved by 4 columns situated in the corners of the ring. Regarding the upper hull, it is composed by four decks: bottom, cellar, tween and main decks. In addition the mooring system results with a combination of four tendons in each corner of the platform. The main dimension of the platform are reported in Table 3.1

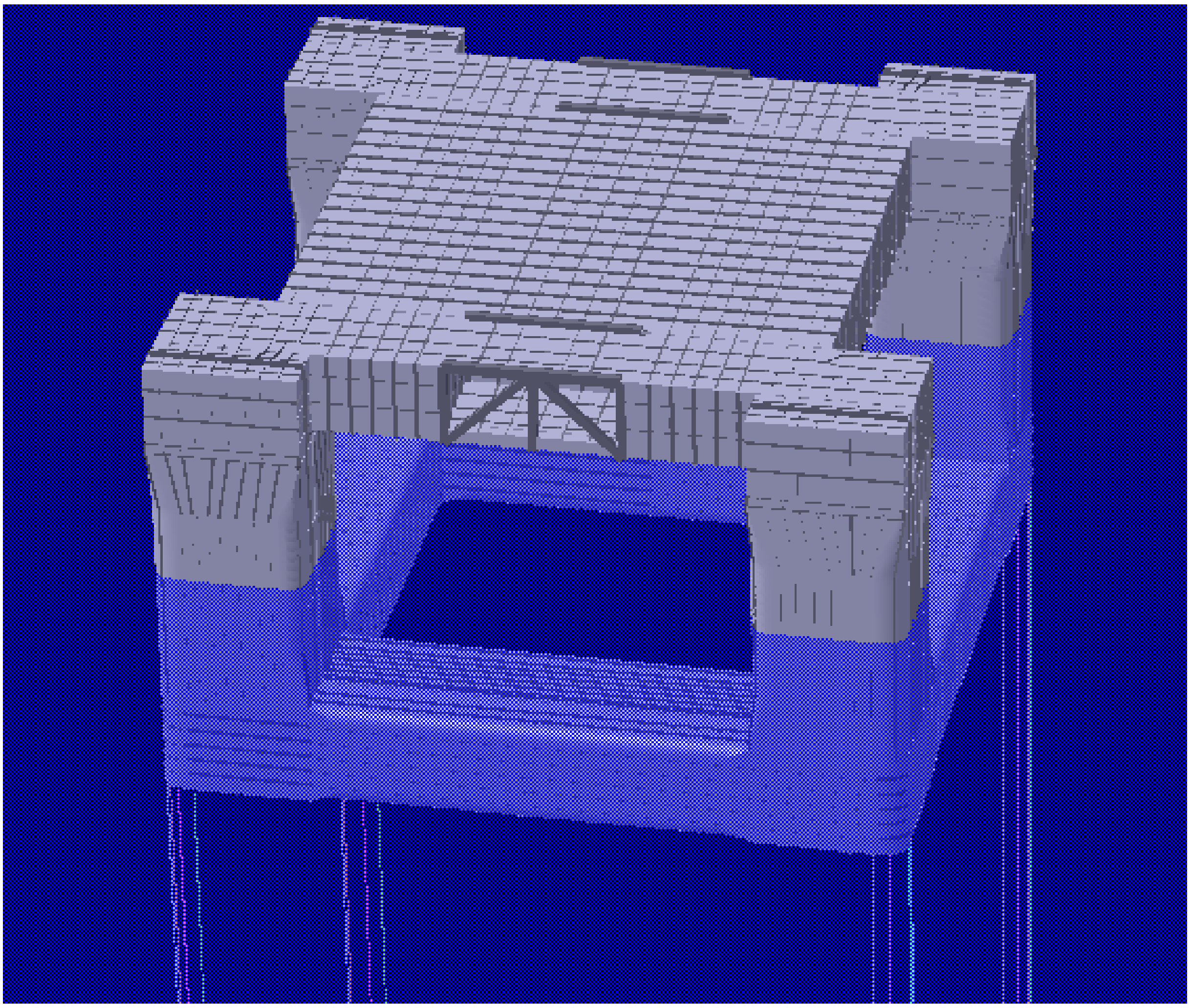

Figure 3-1 Hydrodynamic analysis: Global model

The structure is based in the drawings of an existing platform. And the structural model will be developed according its dimensions.

Since the case analysed pretend to be a general example, the North Sea localization was selected as operation area, which a water depth of 327.5 m. This location is selected since it is linked to the hardest environmental conditions a platform of such type can face in the all over the world.

Table 3-1 Main Dimensional Parameter for the TLP Analysed

Length overall	85 m
Beam	85 m
Depth	54 m
Draught	27.5 m
Column elevation	43.38 m
Column breadth	17.5 m
Main deck elevation	54 m
Bottom deck elevation	44.5 m
Pontoon elevation	12 m
Pontoon breadth	17.5 m

Note: All the vertical dimensions are given respect the base line, which is situated in the lowest point of the pontoons.

4 OFFSHORE STANDARS FOR TLP

In this charter a general review of the DNV GL standards with concert the structural fatigue analysis and environmental loads of TLP are presented. The rules which make reference to the fatigue analysis of TLP are listed below.

- DNV RP C205: Environmental Conditions and Environmental Loads
- DNV-RP-F205: Global Performance Analysis of Deep-water Floating Structures
- DNV-OS-C101:Design of Offshore Steel Structures, General (Load and Resistance Factor Design method)
- DNVGL-OS-C105: Structural design of TLPs - LRFD method
- DNVGL-RP-C203: Fatigue design of offshore steel structures
- DNV-OS-C106: Structural Design of Deep Draught Floating Units (LRFD Method)

Such standards offer especial attention to the fatigue limit state, fatigue analysis method and fatigue cracking failure modes considered.

4.1 Fatigue Limit State

The fatigue limit state (FLS) is the one which is generated by cumulative damage due to repeated loads. The aim of fatigue design is to ensure that the structure has an adequate fatigue life.

Know the F.L is necessary for efficient inspection programmes during fabrication and the operational life of the structure. This feature will be cover in next chapter, where the methodology to be followed in order to localize the fatigue critical areas will be explained.

When a fatigue analysis is performed the first question that the designer should ask to himself is: How much will be the design life of the structure? Or in other words, how many years is the structure supposed to operate?

The standards consulted state that there are two possible methods to perform accurate fatigue analysis:

- Analysis based fatigue tests (S-N curves)
- Analysis based on fracture mechanics.

Considering the standard: DNV OS C101, the fatigue analysis will be carried out based on S-N data, determined by fatigue tests of the considered welded details, and the linear damage hypothesis.

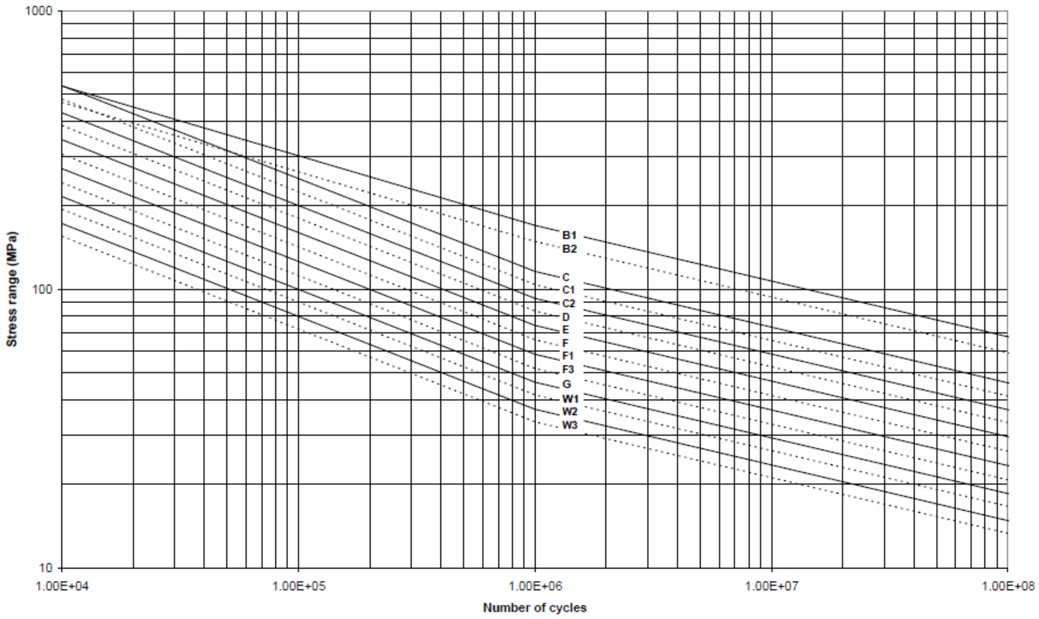

Figure 4-1 S-N curves in seawater with cathodic protection. Available from Offshore standard. DNVGL-RP-0005

S- N curves represent a stress range (S) versus number of cycles to failure (N) based on fatigue tests, which were derived by fatigue testing of small specimens in test laboratories. These curves are based on the mean-minus-two-standard-deviation curves for relevant experimental data. The S-N curves are thus associated with a 97.7% probability of survival.

Further details regarding S.N curves can be find in chapter 9 and in the reference [18].

4.1.1 Fatigue analysis – FEM

According to the standard DNVGL-RP-0005, when performing finite element analysis for design of plated structures it is often found more convenient to extract hot spot stress from the analysis instead of nominal stress. This demand will be satisfied performing a local FEM model, which use a fine mesh with size t x t.

Based in the effective notch stress method, the following fatigue cracking failure modes will be considered in the analysis:

- Fatigue crack growth from the weld toe into the base material.

In welded structures fatigue cracking from weld toes into the base material is a frequent failure mode. The fatigue crack is initiated at small defects or undercuts at the weld toe where the stress is highest due to the weld notch geometry. A large amount of the content in this RP is made with the purpose of achieving a reliable design with respect to this failure mode.

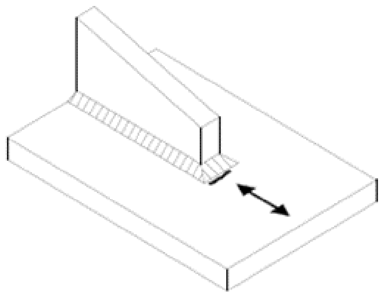

Figure 4-2 Fatigue crack growth from the weld toe into the base material. Available from Offshore standard. DNVGL-RP-0005

- Fatigue crack growth from the weld root into the section under the weld.

Fatigue crack growth from the weld root into the section under the weld is observed during service life of structures in laboratory fatigue testing. The maximum number of cycles linked to this failure mode is likely to fatigue cracking cycles from the weld toe in an “as-welded” condition.

During the analysis, the fatigue life will be calculated based on the S-N fatigue approach under the assumption of linear cumulative damage (Palmgren-Miner rule).

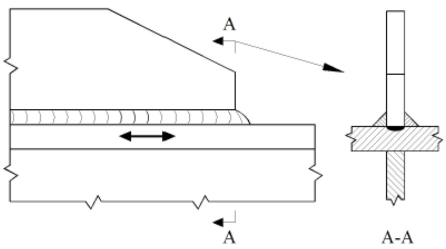

Figure 4-3 Fatigue crack growth from the weld root into the section under the weld. Available from Offshore standard. DNVGL-RP-0005

4.1.2 Usage Factor

The usage factor is defined as the design life - life in service - divided by the calculated fatigue life.

$$Usage\ factor = \frac{Design\ life}{Fatigue\ life} \tag{1}$$

The reason to introduce the concept of usage factor is related to the results presentation. Base in the rules, the design fatigue life for structural components should be based on the specified service life of the structure. If a service life is not specified, 20 years should be used.

By selecting 20 years as the design life, the fatigue life of the different component should be at minimum 20 years to guarantee the safety of the structure will be safe again fatigue during the life in service. I.e. a condition for usage factor is imposed, for which it should be lower than 1.

$$Usage\ factor = \frac{20}{20} = 1 \tag{2}$$

Unavoidably, when the usage factor is above 1, the structure will start to present cracks in the most components.

The standards expose that:

- To ensure that the structure will fulfil the intended function, a fatigue assessment shall be carried out for each individual member which is subjected to fatigue loading. Where appropriate, the fatigue assessment shall be supported by a detailed fatigue analysis. It shall be noted that any element or member of the structure, every welded joint and attachment or other form of stress concentration is potentially a source of fatigue cracking and should be individually considered.

This recommendations will determine the way in which the methodology of this master thesis will be developed.

5 ANALYSIS METHODOLOGY

In this chapter an explanation of the work flow and methodology followed for the study to be performed is provided. A description of the software's used is presented, covering the steps required for the calculations, the generation of the FE models, and the local fatigue analysis of critical areas

Detailed explanation about the FE modelling, hydrodynamic or fatigue calculations required will be discussed in following chapters.

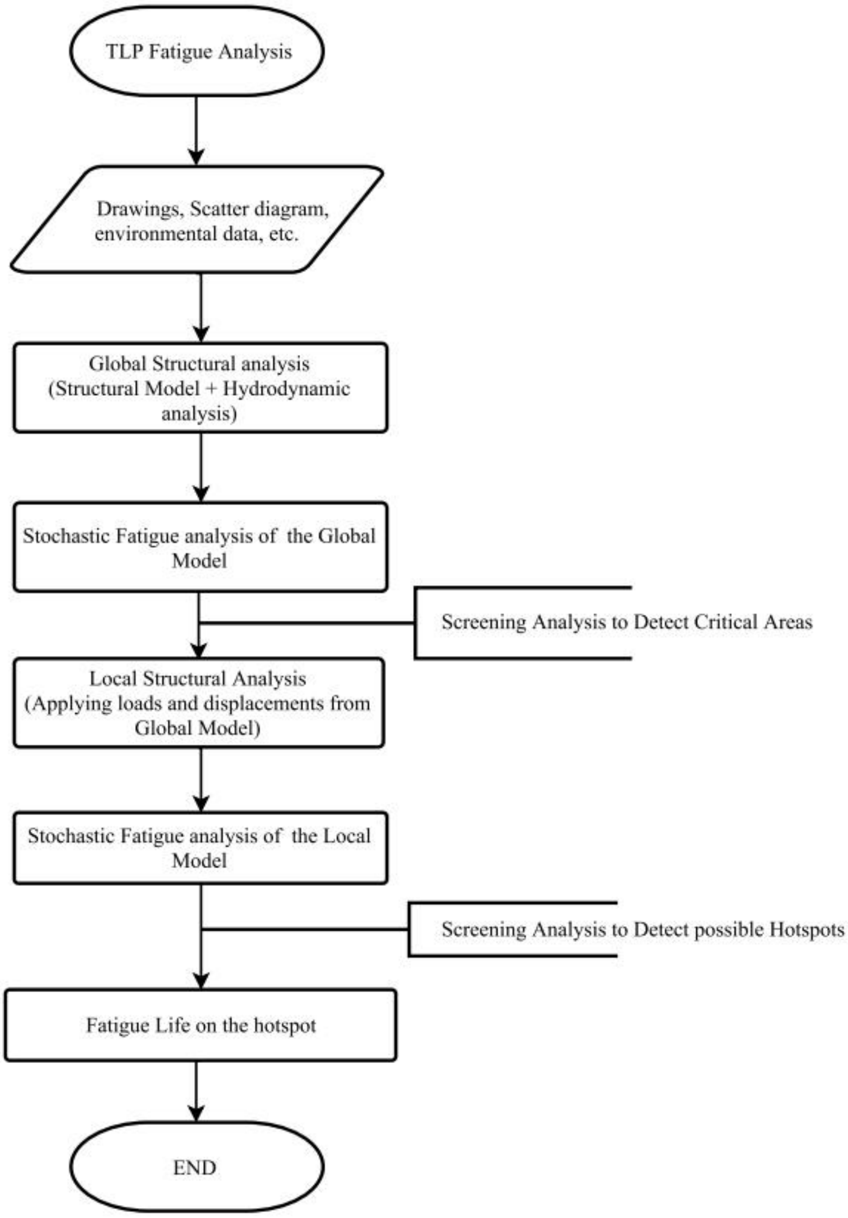

Figure 5-1. General flow chart

The analysis methodology followed to perform the fatigue analysis of the TLP is presented in the figure 5-1

The fatigue analysis of the TLP will start from the development of the corresponding FE Global structural model. In this model all the relevant structural elements, which contributing to the global stiffness of the structure must be included. Once the global model is finished the next step is a Hydrodynamic analysis. The objective of this analysis is to obtain the pressure distribution acting on the wet surfaces generated by the waves.

When loads are determined, a load transfer from to the FE model is performed to solve the global structural analysis (done by means of a static, quasi-static or dynamic method). Based on the output obtained from the structural analysis of the global model, a stochastic fatigue analysis is performed to determine the usage factor distribution. This is done to determine the critical areas, procedure known as screening analysis.

Located the critical areas, a local model of the critical areas have to be developed. This local model will included a more detailed structural model to describe with accuracy the irregular stress distribution in the area. This means that simplifications as usage of beam or trust elements are to be avoided so shell and solid elements would me the dominant modelling technique. At this point, loads and displacements are transferred from the global model to the local model and so, the new structural model is solved by means of a static, quasi-static or dynamic method as before.

Once again a stochastic fatigue analysis is performed and a more detailed usage factor distribution is determines to locate the possible hotspots. To ensure that the fatigue life obtained is accurate enough an effective notch stress method is applied in the hotspots, the explanation of the effective notch stress method can be found in chapter number 9.

With this last step fatigue analysis will be accomplished. Considering the methodology described several structural modifications of the critical areas will be applied, being the local structural model modified and therefore the fatigue life in the hotspots. Based on the variation of the fatigue life in the hotspots, the effect of the structural modification applied will be quantified.

5.1 Software features and limitations

The software program used to perform the sets of analysis was: Sesam- Strength assessment of offshore structures. Sesam is an integrated and comprehensive software suite for hydrodynamic and structural analysis. It forms a complete strength assessment system for engineering of ships, offshore structures and risers based on the finite element methodology. SESAM package involve different programs, which go from the modelling in FE to the post processing of the results (fatigue, animations, etc.), in next Figure the SESAM package overview is showed:

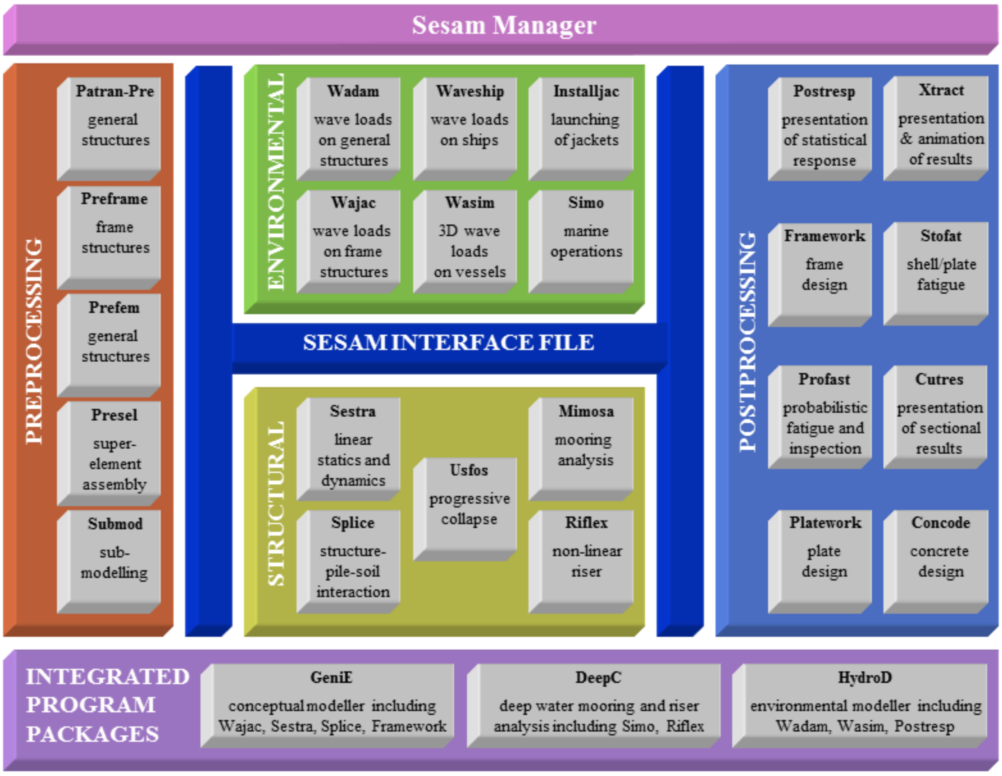

Figure 5-2 The Sesam overview. Available from

Sesam user manual

During the development of this master thesis, 7 program of the SESAM package will be used:

- Sesam Manager
- Xtract
- Genie
- Submod
- HydroD:Wadam
- Sestra
- Stofat

On the other hand, the main tasks, which will be carried out, correspond basically to:

- Modelling.
- Sub-Modelling
- Hydrodynamic analysis.
- Quasi-static analysis of the structural response.
- Stochastic fatigue analysis.

A brief description of the 7 programs will be given. Especial attention will be paid to the finality and limitations of the last five.

5.1.1 Sesam Manager

Sesam manager is a program for the management of Sesam package programs. You may use it as your single entry point to Sesam.

In Sesam Manager you establish your analysis task as a job containing execution of several Sesam programs (applications) organised as activities in a workflow. The workflow may be of any length and complexity involving any and all of the Sesam programs. Sesam Manager takes care of the data flow between the Sesam programs.

In short, the main function of this program will be connect the different programs which will be used in the different analysis.

5.1.1. Xtract: Analysis Post Processing Tool

Xtract is the program of Sesam package used for the results visualisation. It offers general-purpose features for selecting, further processing, displaying, tabulating and animating results from static and dynamic structural analysis as well as results from various types of hydrodynamic analysis.

Its intuitive and high-performance 3D graphics enables easy and efficient interactive rotation, zooming and panning of the model while viewing and animating the results. The software is helpful to visualize all relevant points belonging to an integrated hydrodynamic analysis like e.g. the finite element model (e.g. several super elements), the water pressure, the displacements (structure and water) and the stresses.

Xtract will be used to visualize the result of the fatigue screening analysis, detection of critical areas and possible hotspots. Also the match between local and global model, where the coincidence of Hydro pressure, displacements and stresses is required, will be checked.

5.1.2 Genie – Modelling

For the structural modelling and analysis of TLP under consideration a standard finite element modelling software, Genie, has been selected.

Genie is a software tool for design and analysis of fixed and floating offshore and maritime structures, where all analyses are based on the Finite Element Methodology. The software supports work phases from initial concept studies to mature design and re-analysis:

- Intuitive user interface and strong features for 3D visualisation of model and results.
- Combined plate and beam modelling (curved as well as planar).

In addition the features of the software included:

- Finite element mesh generation
- Partial mesh generation
- Local mesh control
- Finite element analysis

This software will be used to model the TLP structure, global and local model. To develop the models different types of elements are available. The type of elements which will be used are shell and beam elements, however there are different types of beam and shell elements.

The elements that finally will be used are:

- SCTS:6-nodes triangular thick shell
- SCQS: 8-nodes quadrilateral thick shell
- BTSS: 3- node beam elements
- BEAS:2-node beam elements (Since not 3 –node beams could be used for the Morison model)

All the structural element included in the different models are described in next chapter, were a division of the structure is carried out.

5.1.2.1 Limitations of the Software

The modelling of solid elements is not allowed by the software. Therefore the properties of the structure across the thickness of stiffeners or plates could not be studied.

5.1.3 Submod - Sub-modelling

This software allow to implement the sub-modelling technique, where a part of a model could be re-analysed to produce more accurate results locally. To take advantage of the sub-modelling technique is not necessary to make any provisions prior to the global analysis. It is an option which is available after having performed a global analysis where is evident that the results are not detailed enough in certain areas.

Submod will be used to transfer the displacement of the global model to the local model by applying boundary conditions with prescribed displacements. The procedure is based on an interpolation of the results extracted from the global model and to apply these onto the sub-model. The finite element mesh of the sub-model is normally fine so as to produce more accurate results within the sub-model region.

5.1.3.1 Limitations of the Software

The software have limitations regarding the type of elements used in the modelling, also the nodes situated in the boundaries are summited to some restrictions presented as follows:

- Coupling nodes must be defined as nodes with prescribed degrees of freedom (DOF) in the sub-model first level elements before starting Submod. Default setting is that only nodes where all degrees of freedom are prescribed will be taken as coupling nodes.
- For ’element node coupling’ all available shell elements, i.e. 3, 4, 6 and 8 node elements, and all available solid elements, i.e. 4, 6, 8, 10, 15 and 20 node elements are implemented. The 2- and 3-node beam elements are also implemented, but the displacement may only be found (interpolated) along the neutral axis of the beam elements.
- The coupling nodes of the sub-model must have 3 or 6 DOF. Coupling nodes with 3 DOF may match solid elements or shell elements in the global model.

- The transfer of forces from e.g. a beam global model to a shell / solid sub-model are not implemented in Submod.

5.1.4 Hydrod: Wadam - Hydrodynamic Analysis

HydroD is an interactive application for computation of hydrostatics and stability, wave loads and motion response for ships and offshore structures (hydrodynamic). The wave loads are computed by a sub module included in the HydroD package named as Wadam.

Wadam is a general analysis program for calculation of wave-structure interaction for fixed and floating structures of arbitrary shape, e.g. semi-submersible platforms, tension-leg platforms, gravity-base structures and ship hulls.

The analysis capabilities in Wadam comprise:

- Calculation of hydrostatic data and inertia properties
- Calculation of global responses including:
    - First and second order wave exciting forces and moments
    - Hydrodynamic added mass and damping
    - First and second order rigid body motions
    - Sectional forces and moments
    - Steady drift forces and moments
    - Wave drift damping coefficients
    - Internal tank pressures
- Calculation of selected global responses of a multi-body system
- Automatic load transfer to a finite element model for subsequent structural analysis including:
    - Inertia loads
    - Line loads for structural beam element analysis
    - Pressure loads for structural shell/solid element analysis
    - Pressure loads up to the free surface

From the previous capabilities, remark the importance of the added mass and damping, since they will influence the movement of the platform and therefore the interaction of the wave loads with the structure. For this reason the Morison model will be included in the analysis.

The main objective is to assure that the load transfer will be correctly done, for this reason is also important to take into account the correct movement of the platform. In order to have precise calculations.

The loads will be calculated by using:

- First order 3D potential theory for large volume structures
- Morison's equation for slender structures

This sub module uses Morison's equation and first and second order 3D potential theory for the wave load calculations. Analysis can be performed in frequency domain or in time domain. In this case of study the frequency domain will be used for the analysis.

5.1.4.1 Limitations of the Software

The main limitations of the software is related to the size of the hydro model analysis. Despite the size of the structural model used in the detailed load calculation is virtually unlimited, the rest of the models have limitations. The size dependent limitations are listed below:

- Panel model:
    - Maximum number of panels (for the basic part of the model) = 15000
    - Maximum number of free surface panels (for the basic part of the model) = 3000
- Morison model:
    - Maximum number of nodes in the Morison model = 5000
    - Maximum number of elements in the Morison model = 5000

On the other hand there are additional limitations regarding the input parameters:

- Maximum number of wave frequencies = 60
- Maximum number of wave headings = 36

In the current analysis just 23 wave frequencies and 8 wave headings will be used, so no problems should appears. Respect to the panel and Morison models, both of them are under the limitations described above, more detail could be found in chapters 7 and 9.

5.1.5 Sestra - Quasi-static Analysis

Sestra is the program for linear static and dynamic structural analysis within the SESAM program system. It uses a displacement based finite element method.

The software is computing the local element matrices and load vectors, assembling them into global matrices and load vectors. The global matrices are used by algebraic numerical algorithms to do the requested static, dynamic or linearized buckling analysis.

In the present master thesis the analysis to perform is a dynamic analysis. However a simplification will be carried out, moving from a dynamic analysis to a quasi-static analysis.

Structural response due to dynamic loading may be analysed by a quasi-static method, i.e. a static analysis in Sestra with ‘quasi-static’ loads. ‘Quasi-static’ loads may be either complex loads corresponding to a frequency domain analysis, which correspond to the wave loads in this analysis.

This simplification have the main advantage to reduce the CPU time consuming, but is limited by the relation between frequency or time-variation of the load and the eigenfrequency of the structure. The quasi-static analysis is often applied when the frequencies of the load far enough from the eigenfrequencies of the structure. The justification of neglecting dynamic effects could be found in chapter 8 of this document, where eigenfrequencies and the relation with Load frequencies will be studied in more detail.

5.1.5.1 Limitations of the Software

The main limitation of Sestra is regarding to the finite elements use in the mesh. This limitations correspond to the dimension of the triangular and square elements. The elements that do not pass this criteria will not be analysed, creating errors in the solution.

The criteria used for refusing elements are as follows:

- For triangles:

$$100 * A - L^2 < 0 \tag{3}$$

Where

A: area of the triangle, in m^2

L: length of longest edge, in m

- For rectangles:

$$25 * C - D < 0 \tag{4}$$

Where

C: shortest diagonal, in m

D: longest diagonal, in m

Other important limitations are:

- Sestra is based on linear theory assuming small displacements and linear materials.
- Size of the model: Depending of computer capacity and accuracy.
- Meshing:
  - Fine and regular mesh is needed in areas where stress gradients are high. Areas with small stress gradients can be modelled with a coarse mesh.

5.1.6 Stofat - Stochastic Fatigue Analysis

Stofat is an interactive postprocessor performing stochastic fatigue calculation of welded shell and plate structures. The fatigue calculations are based on responses given as stress transfer functions. The stresses are generated by hydrodynamic pressure loads acting on the model. These loads are applied for a number of wave directions and for a range of wave frequencies covering the necessary sea states. The loads are applied to a finite element model of the structure whereupon the finite element calculation produces results as stresses in the elements. Stofat uses these results to calculate fatigue damages at given points in the structural model.

The assessment is made by an SN-curve based fatigue approach accumulating partial damages weighted over sea states and wave directions. The program delivers usage factors representing the amount of fatigue damage that the structure has suffered during the specific period. Stofat may also account for the effect of static stresses from still water load cases in the fatigue assessment (Considered in this case of study).

The program will be used firstly to perform a screening analysis of the global model to localize the critical areas. Function of the results obtained in the first analysis, a second screening, in the local model, to identify possible hotspot and finally a calculation of the fatigue life in the hotspots of interest.

5.1.6.1 Limitations of the Software

The main limitation of the software is related to the model size. Stofat saves principal stresses and part damage results for all sea states at all stress points of the elements included in the run. For large models this may sum up to quit a big number of data to be saved.

The size limit is a function of the number of wave directions, wave frequencies, sea states and elements included in a Stofat run. Analysis results are stored in 8 data base directories and provides a maximum possible utilization of the data base capacity for a Stofat run.

The number of data saved for an element in directory 3 and 4:

$$nsave3 = npnt * (1 + 2 * nwdir * nfreq) + 21 \tag{5}$$

$$nsave4 = npnt * (3 + 2 * nwdir * nfreq) + 21 \tag{6}$$

The number of data saved for an element in each of the directories 5 to 10:

$$nsave5 - 10 = npnt * nsea + 15 \tag{7}$$

Where

npnt = number of stress points of the element

nsea = number of sea states(=nwdir*nscpnt, in case of same scatter diagram for all wave directions)

nwdir = number of wave directions

nscpnt = number of points of scatter diagram

nfreq = number of wave frequencies

nsave3 = Number data saved for each element in directory 3

nsave4 = Number data saved for each element in directory 4

nsave5-10 = Number data saved for each element in each of the directories 5 to 10

For this reason the screening diagram of the global model will be perform just for one quarter. There is not issue since the structure is symmetrical respect the axis X and Y.

6 STRUCTURAL MODELLING

The objective of the structural modelling, is to create a standard TLP structure as close as possible to the reality. Two different models will be developed: Global model and a combination of sub and local model. While in the global model just the element which contribute to the global stiffness, in the sub model all the secondary stiffeners will be included. In addition, in the critical areas, which correspond to the local mode, higher grade of detail such as: Brackets, flange terminations for main girders and stiffeners connection will be considered.

In this case, the structural model is equal to the mass model of the hydrodynamic analysis. The FE used for the modelling of the structure, are shell and beam elements

Along this chapter the strategy followed to represent each component of the structure and boundary conditions will be detailed. Also the assumptions considered, to complete the omitted data due to confidentiality reasons, will be exposed. Finally the meshing of the different model will be presented indicating the point where special attention should be paid.

6.1 Assumptions

Before start to explain the modelling process, is important to numerate the assumptions which will be followed in order to simplify the modelling process and also approximated the models to the reality.
Two main assumptions are applied to the structural and mass model developed in this master thesis

6.2 Structural Assumption

Structures components not contributing to the global stiffness, are not included in the analysis. However the mass of these elements will be included in the model.

This decision correspond to the necessity of reduce the time consumption employed for the development of the FE model and the performance of the different calculations required.

Due to this assumptions some part of the structure will not be modelled. These part are numerated in the next list:

- Spider deck.
- Superstructure situated above the main deck.
- Derrick.

6.3 Mass Assumption

Difference between the buoyancy force, given by the design draft, and the displacement offer by the global structural model shall be equal to zero.

From the design draught of the TLP and based in the drawings, the total displacement of the platform could be calculated. For confidentiality reasons, no list of weight corresponding the blocks and the equipment's installed in the platform were proportionated. However in the hydrodynamic calculation is necessary to have an equilibrium between buoyancy force and displacement of the platform. Hence the difference of weight between the global structural model and the real structure need to be compensated.

By following the present chapter, the final weight of the global model will raise 13196.112 t. This weight do not reach the buoyancy imposed by the design draft. Therefore is necessary to increase the weight of the global model.

The exactly quantity of weight, in which is necessary to increment the global model mass is given by next Table:

Table 6-1 Displacement discussion

Buoyancy given by draught 27.5 m	76375.62 t
Initial weight of global model	13312.085 t
Difference to compensate	76375.62 -13312.085 = 63063.535 t

To this difference should be discount the pre-tension offer by the mooring system. Mention that the TLP have an excess of buoyancy which is compensated by the mooring, in order to be always in tension and maintain his position during the operation. In chapter dedicated to the hydrodynamic analysis the mooring system and how the pre-tension is calculated will be described in detail.

Finally, taking into account the pre-tension of the mooring, the final weight of the global to have hydrostatic equilibrium shall be:

Table 6-2 Final displacement of the global model

Difference to compensate	63063.535 t
Pre-tension of the mooring	16858.736 t
Weight to add into the global model	63063.535-16858.736 = 46204.799 t
Final displacement of the global model	59516.884t

In order to obtain this final displacement of the global model one of the features of genie will be used: Scale Mass densities.

When making a model it is quite common not to include all details. In order to overcome this limitation is possible to scale structural masses to a given value. When the masses all scale the new weight will be distributed along all the structure selected. In this case the complete model will be selected since no weight or exactly position of any equipment or block is known.

The target mass will be introduced and a scaling factor will be showed. In the current analysis the scaling factor will be:

$$Scaling\ factor = \frac{Target\ mass}{Original\ mass} = \frac{59516.884}{13312.085} = 4.471 \tag{8}$$

6.4 Global Model

Finite element analysis is required to obtain accurate stress distribution in the TLP structure. There are several levels of finite elements models used in the various phases of design. In this case the model which is describe correspond to the top level: The global structural model.

The global structural model is a coarse mesh of the unit, which represent the actual global stiffness of the platform comprising sub-assemblies like pontoons, columns and decks. The FE used for the modelling of the structure are:

- Shell elements for principal structural elements

The principal structural elements as the hull shell, longitudinal and transversal bulkheads, column shell frames, double bottom, and deck are modelled with •8-noded shell elements or improved 4-noded shell elements with additional internal degrees of freedom will be used for the FE modelling of the structure. 6-node triangular elements will be also used in areas where a mesh of 8-noded shell elements otherwise is difficult to fit.

- Beam elements for secondary structural elements.

Secondary structural elements as bars, and stiffeners form the hull, columns and decks structure are modelled with 3-noded beam elements

In order to represent the global stiffness, it is important include:

- The longitudinal stiffness of the pontoons.
- The stiffness of the vertical columns.
- The stiffness of the main bulkheads as well as the shear and bending stiffness of the upper hull.

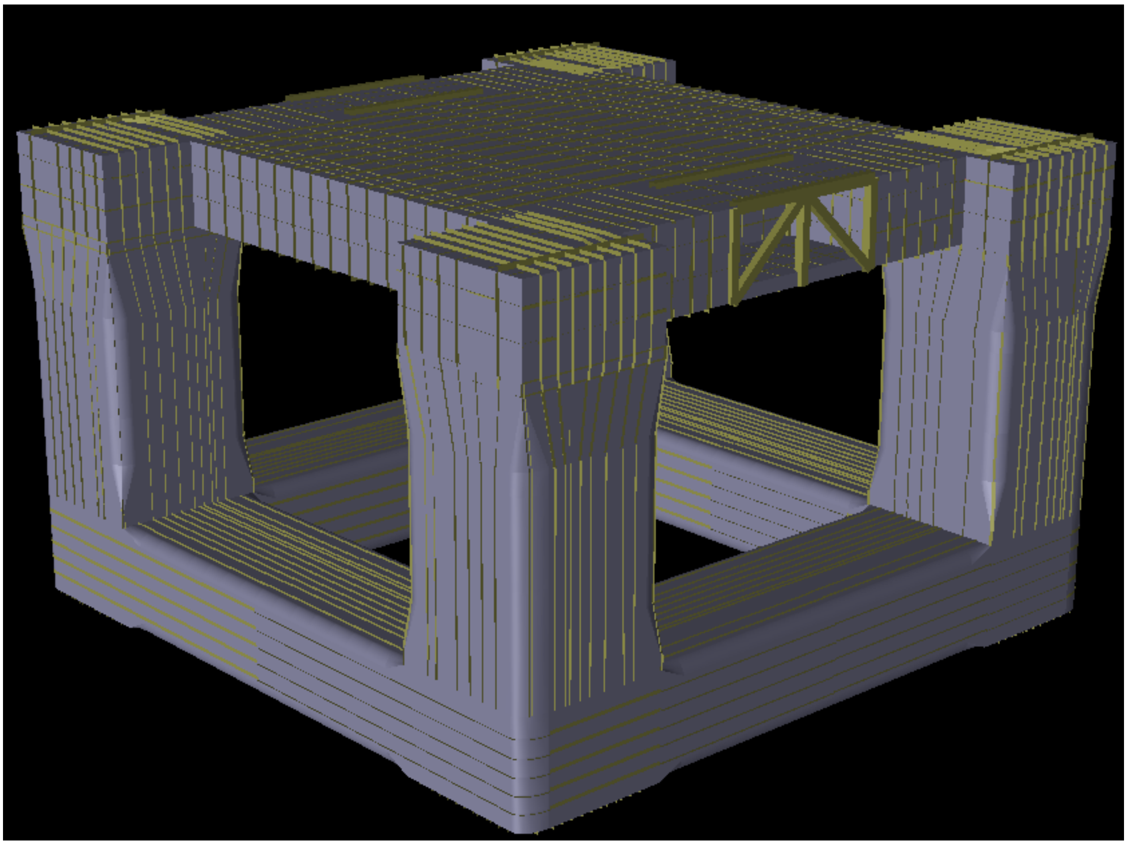

Figure 6-1Global structural model in Genie

Structure not contributing to the global strength of the unit will be disregarded. Nevertheless the mass of disregarded elements shall be included in the model. Smaller openings such as doors, man holes, pipe penetrations are not modelled. Large openings are modelled, as the stiffness of the area is reduced.

6.4.1 Pontoons

The structure in longitudinal direction will be included in the model. This implied that longitudinal bulkheads, girders and pontoon shell will be represented in the model. Local reinforcements like minor reinforcement will be omitted.

The transverse bulkheads and frames in the pontoon are modelled in order to achieve a representative geometric stiffness between the pontoon's vertical shell sides and the longitudinal bulkhead. If this stiffness is omitted, the pontoon may deflect incorrectly depending on the distribution of the ballast in the pontoon and the length of the free span between the columns.

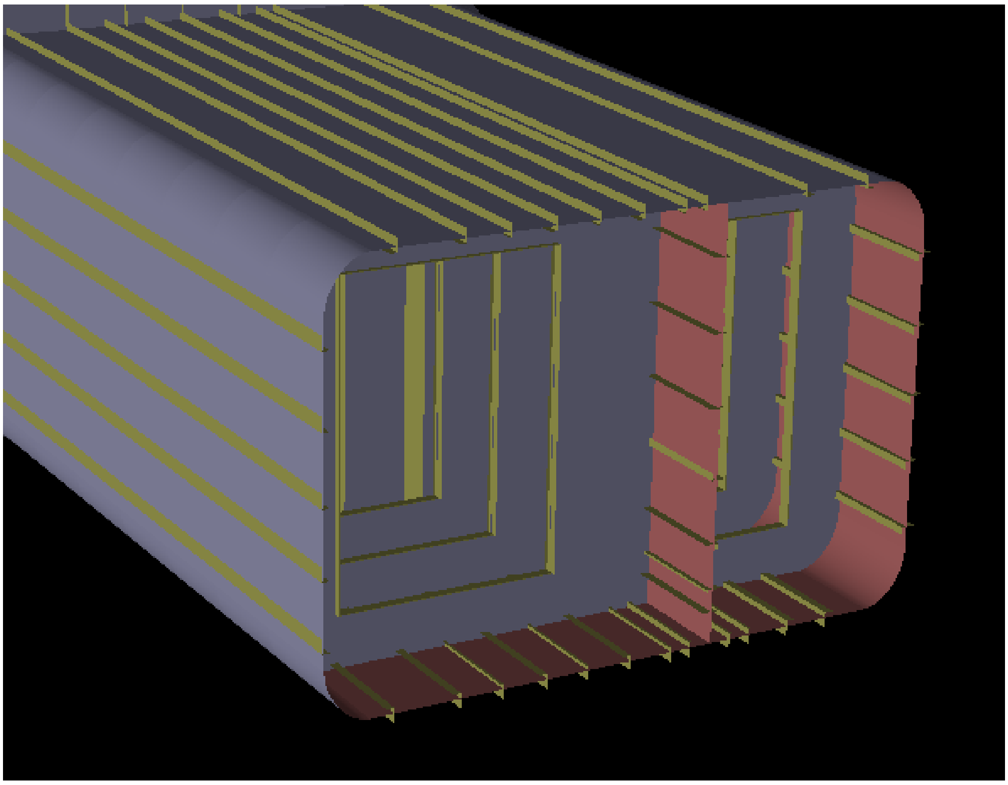

Figure 6-21Global structural model: Pontoon

Local reinforcements such as longitudinal girders covering over short distances, frames has been neglected as their contribution to the global stiffness of the structure is minimal.

6.4.2 Columns

The vertical elements of the structure are important for the global stiffness. Hence, transverse and longitudinal bulkheads and column shell will be properly included in the analysis model.

In addition the column´s decks will be included, it is not strictly necessary but stress peaks from the mass model will be reduced.

Since the trunk will attract stress from the global response, is also included in the model.

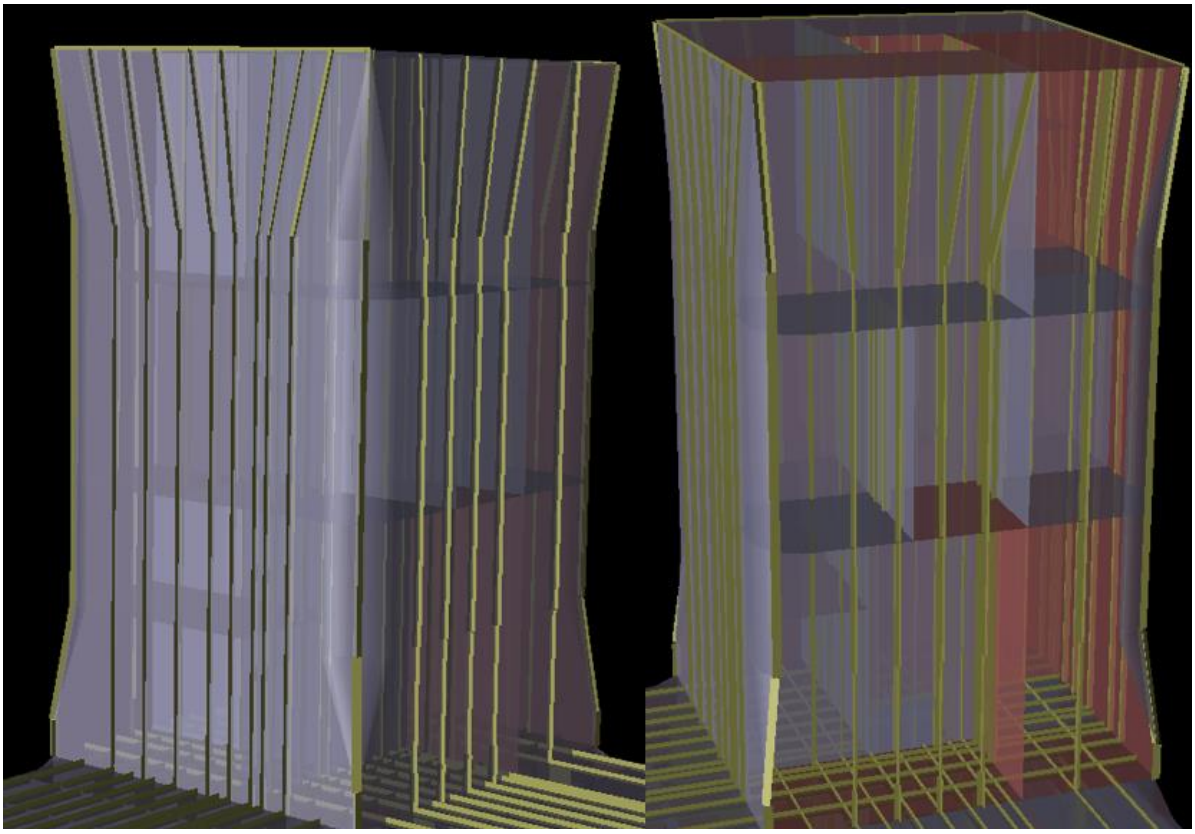

Figure 6-3Global structural model: Column

6.4.3 Upper Hull: Decks

The upper hull is composed by four decks, but also by the connection areas with the columns. The bulkheads and main girders connecting the columns with lowest deck will be modelled. Regarding to the decks, four of them will be modelled: Bottom, cellar, tween and main deck.

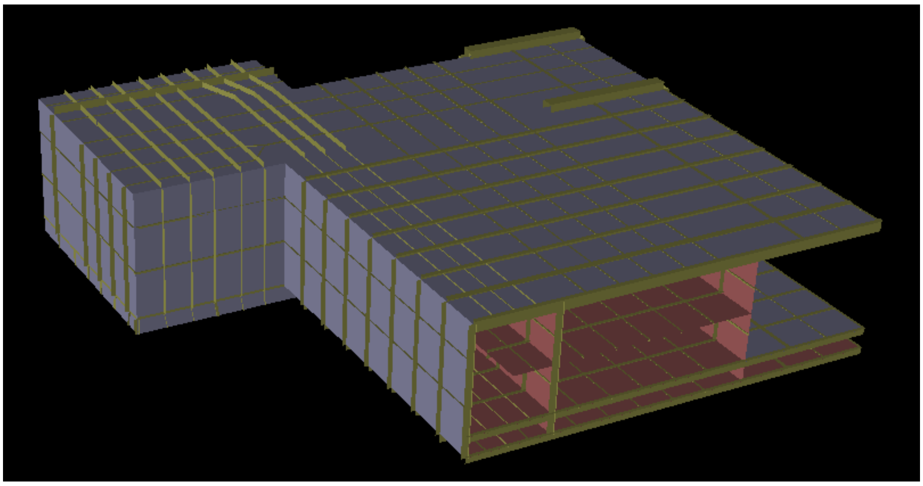

Figure 6-4 Global structural model: Upper hull

The bulkheads running across the transversal and longitudinal direction will act as girder-webs, while the decks will act as flanges. The decks represents the shear stiffness of the upper hull even though the thickness of the deck may be small. Local details such as brackets, buckling stiffeners, smaller cut-outs such as doors etc. will be neglected in the global model.

6.4.4 *Model Assembly*

Especial attention to the connection between upper hull/columns, columns/pontoons should be paid. Base in the offshore standard DNV-OS-C106, the typical fatigue sensitive areas for the platform analysed are:

- Column to deck connections
- Column to pontoon connections

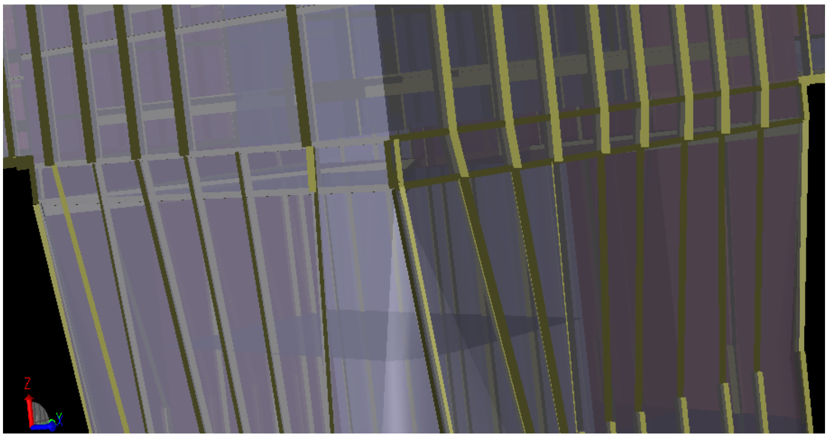

Figure 6-5 Global structural model: Column to upper hull connection

If these areas are not modelled correctly the stress distribution will not be accurate. Since the fatigue life calculation is based on the stresses suffered by the structure during the operational life, the results obtained will be wrong. For this reason these areas will be modelled accurately by included bulkheads, decks, frames, etc. as it showed in Figure 6-5.

6.4.5 *Stiffeners Modelling: Lumping*

The stiffeners of the global model will be modelled without any offset. If the stiffeners are modelled with offset additional bending stress coming from the stiffeners misaligned will be introduce in the plate due to water pressure. Therefore the calculated bending stress will not be correct, due to the coarse mesh and the larger stiffeners which will be used in the global model.

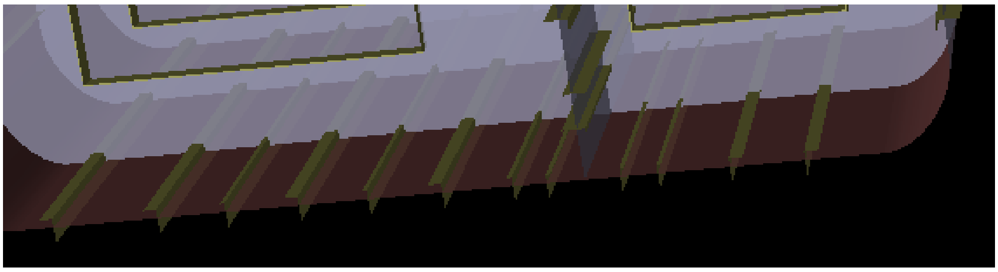

Figure 6-6 Global structural model: stiffener details

For this reason no offset will be considered in the stiffeners for the global model as it showed in Figure 6-6.

With the intention of reduce the complexity of the global model, time consuming in the calculations and also because of meshing reasons, the stiffeners will be lumped as show in Figure 6-7.

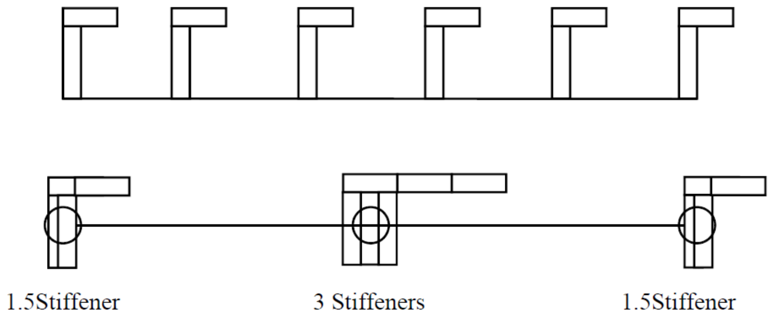

Figure 6-7 Lumping procedure. Available from
Offshore standard DNVGL-RP-0001

Should be notice that the lumping of stiffeners need to be limited, if too much stiffeners are group together local membrane stresses will appear, generating inaccurate fatigue results.

6.4.6 Boundary Conditions

According to the recommended practices given by DNV GL: All 6 DOF in the analysis model need to be defined in order to avoid singularity in the global stiffness matrix, fixed boundary conditions may be used for a statically determined set of boundary conditions while spring stiffness is more appropriate for a statically undetermined set of boundary conditions (Figure 4-20), in order to avoid stress concentration around support points.

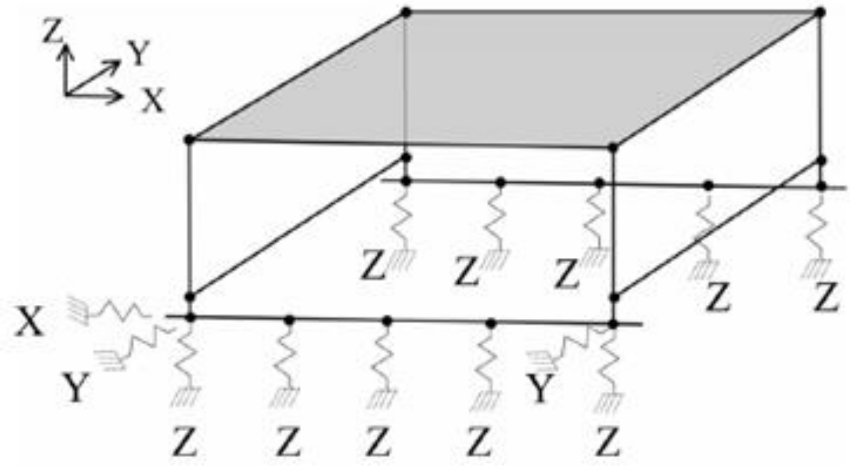

Figure 6-8 Statically undetermined boundary conditions with springs. Available from Offshore standard DNV-RP-C103

In the current analysis, a statically undetermined set of boundary conditions with spring stiffness will be applied. The total vertical stiffness is calculated according to the water plane area by following formula number 9. The corresponding area for one column is calculated and multiplied by 4 to obtain the total water plane area of the structure assuming 0 trim loading condition at operating draft. The result obtained is presented in Table 11.

$$k = \rho * g * A_w \tag{9}$$

Where:

ρ: density of water

g: acceleration of gravity

A_w: total water plane area of the structure

Table 6-3 Spring stiffness

Aw= total water plane area of the structure	1207.864 m^2
g= gravity	9.81 m/s2
ρ = density of water	1025 kg/ m3
K= total vertical stiffness	12145378.51 N/m

The spring stiffness will be applied at significant number of distributed region to minimize the stress concentration in the pontoon shells and to minimize the unbalance of the force, spring's distribution showed in Figure 6-9 is adopted. Moreover these springs have been applied at "strong" points in the model (for e.g. intersection of the bulkheads etc.) so as to limit the effect of unphysical support reactions.

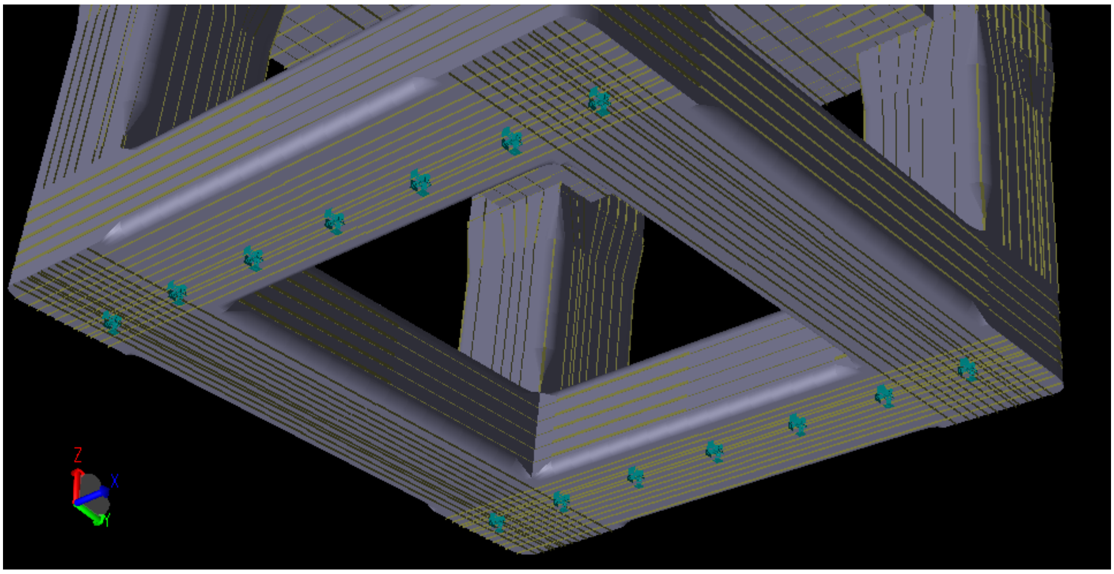

Figure 6-9 Detail of the boundary conditions

Base in the standards given by DNVGL-RP-0001 horizontal supporting, in transverse and longitudinal direction, is represented by springs equal to 10% the total vertical spring stiffness. The horizontal stiffness is distributed along all the springs in order to reduce the false stress concentration areas around the supports. In addition, the points where the supports will be situated are far from the areas of interest of this study.

6.4.7 Meshing

When the meshing of a model will start, the elements size should be selected depending on the geometry of the structure to model. A typical maximum size of the elements used for a global model of a TLP is approximately 2 by 2 meters. However, the size is often smaller due to shift in plate thickness and internal structure such as bulkheads, frames, geometric details etc. Another aspect to be considered is to model the elements as rectangular as possible, and with a length to breadth ratio less than 4:1.

In the case the modelling start in the columns, where is recommend to begin with the meshing. The mesh size select for the global model is equal to 1.25 by 1.25 meters. This size, correspond to the minimum spacing between stiffeners in the columns.

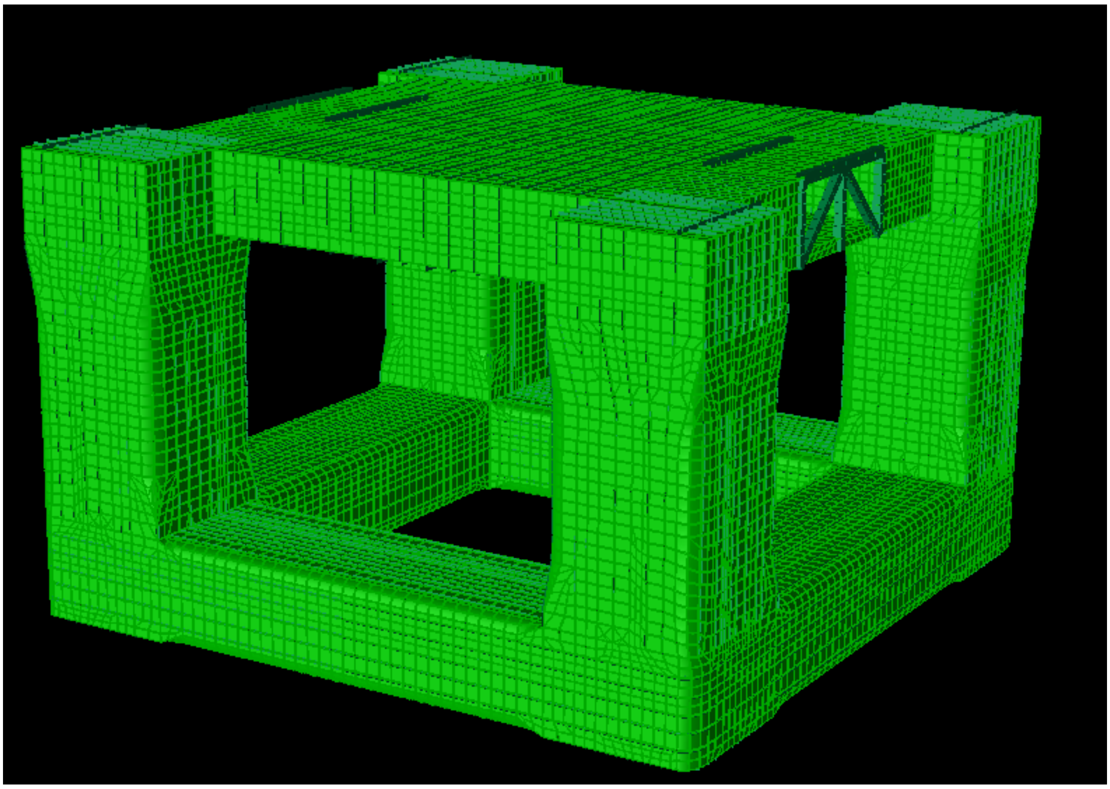

Figure 6-10 Global model: Overview of the mesh

It is an extended practice in the meshing of FE models, use the minimum spacing between stiffeners as basic size of the mesh. Also is important to mention that where stiffeners are lumped, the element edges will be as straight as possible in order not to produce spurious hot spots.

Finally the appearance of the mesh can be observe in Figure 20.The main objective of generate a regular mesh, which avoid stress concentration, was achieved. In Figure 6-11, 6-12 and 6-13 could be observe the mesh in the pontoons, decks, columns and connection areas.

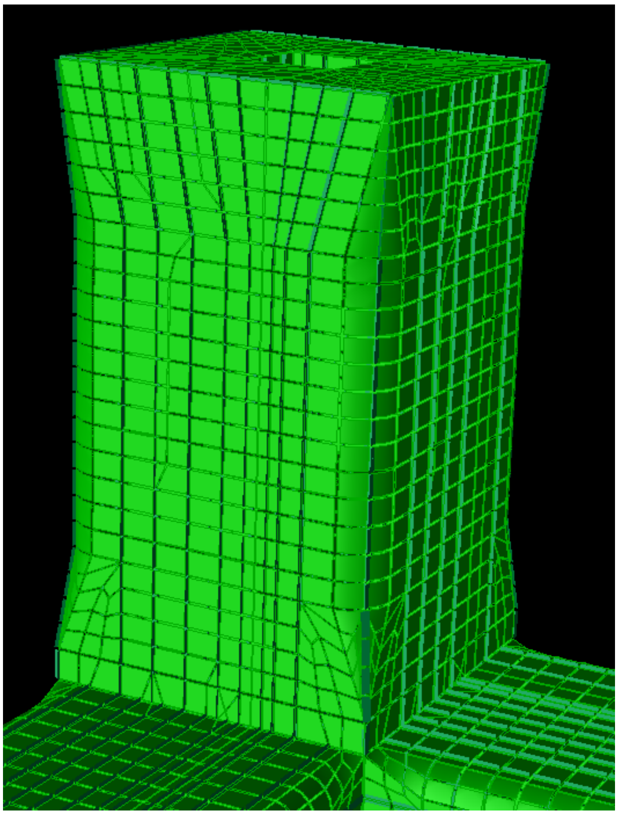

Figure 6-11 Global model: Column meshing

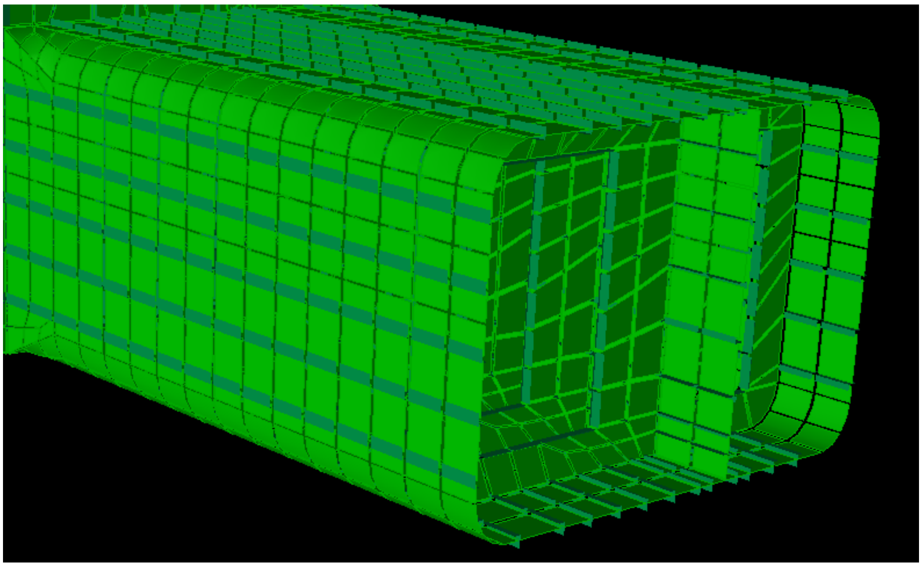

Figure 6-12 Global model: Pontoon meshing

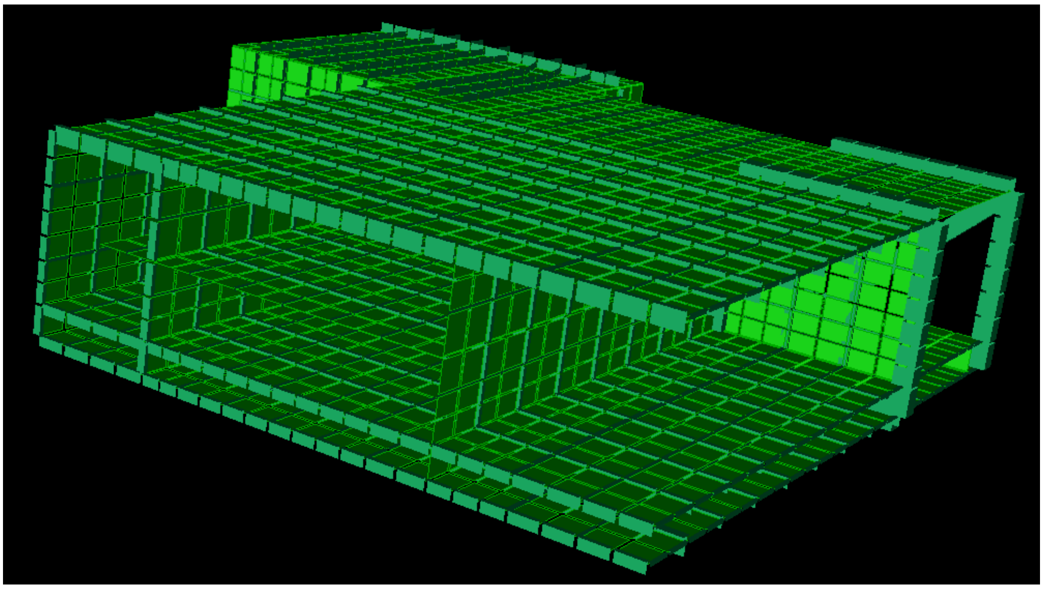

Figure 6-13Global model: Upper hull meshing

6.5 Sub Model

As it was mention in previous chapters, a sub-modelling technique will be used to arrive till local analysis. Therefore is necessary to generate a FE sub model which will allow the use of the sub-modelling technique.

The sub- modelling technique allows a part of the global model to be re-analysed to produce more accurate results locally without changing or re-running the original global model. By creating a separate model, typically with a more detailed structural description of a specific area, the responses from the global structural model can be transferred to the boundaries of the local model by means of complex prescribed displacements. In this way the local detail or model does not need to be an integrated part of the global model.

There are three main reason to apply this technique: the first is related to the need for reducing the size of the analysis model (time consuming). The second reason respond to limitation of the software regarding the virtual memory, which reduce the size of the FE to be analysed at the same time. The last reason is given by the standards which specified that for complex connections, where the dynamic stresses are found to be most severe, a sub-modelling technique should be used for calculation of the notch stress.

Base in the fatigue screening analysis of the global model, some critical fatigue areas were founded. Since the time available to perform this master thesis was limited, just one of these areas founded will be analysed in detail by creating a separate model (sub-model). This area correspond to the connection between column and pontoons (Figure 6-14).

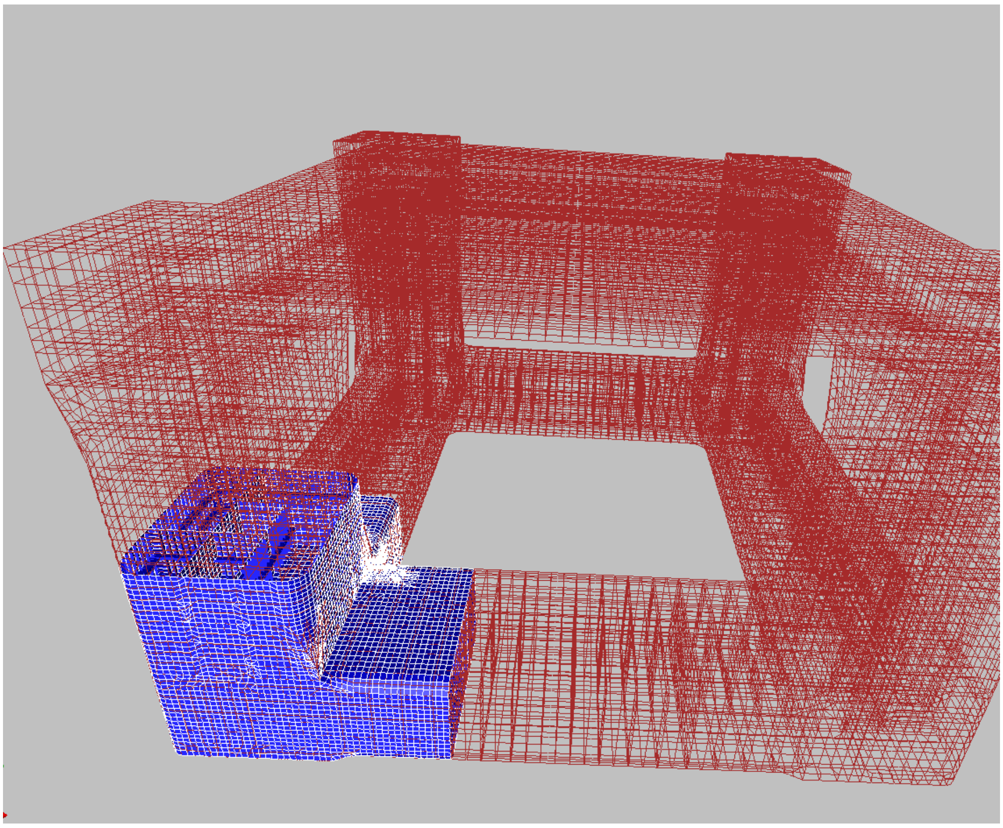

Figure 6-14 Location of the sub model in the global model

Although the sub-model technique offers good flexibility, there are some precautions which need to be taken into account in order to ensure reasonable transfer of displacement from global model to the local model.

The extension of the model will be determinant, it shall be sufficiently large to ensure that the calculated results are not significantly affected by assumptions made for boundary conditions and application of loads.

The rules of the sub-modelling technique specified that the boundaries of the sub-model should not be midway between two frames or decks if the mesh size of the parent model is such that the displacements in this area cannot be accurately determined. The extension of the sub-model analysed could be observe in Figure 6-14, being the boundary conditions situated in decks or bulkheads.

Related to the geometry to be included, the grade of detail will be increase in the sub-model. The lumping developed in the global model will be unpick and secondary stiffeners will be included in the modelling. This changes could be observed in the image 6-15 and also in the image 6-16 were a closer view of a frame with the secondary stiffeners is presented.

In the same way that for the global model, the FE used for the modelling of the structure are:

- Shell elements for principal structural elements

The principal structural elements as the hull shell, longitudinal and transversal bulkheads, column shell frames and double bottom, are modelled with •8-noded shell elements or improved 4-noded shell elements with additional internal degrees of freedom will be used for the FE modelling of the structure. 6-node triangular elements will be also used in areas where a mesh of 8-noded shell elements otherwise is difficult to fit.

- Beam elements for secondary structural elements.

Secondary structural elements as bars, and stiffeners form the hull, columns and decks structure are modelled with 3-noded beam elements

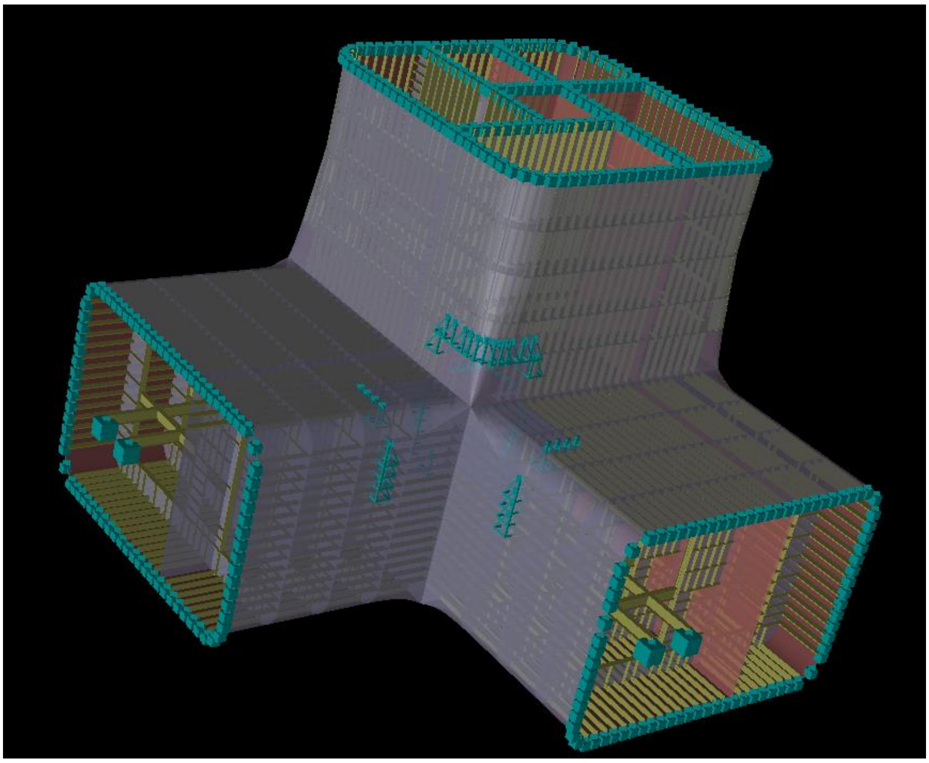

Figure 6-15 Overview of the sub model structure

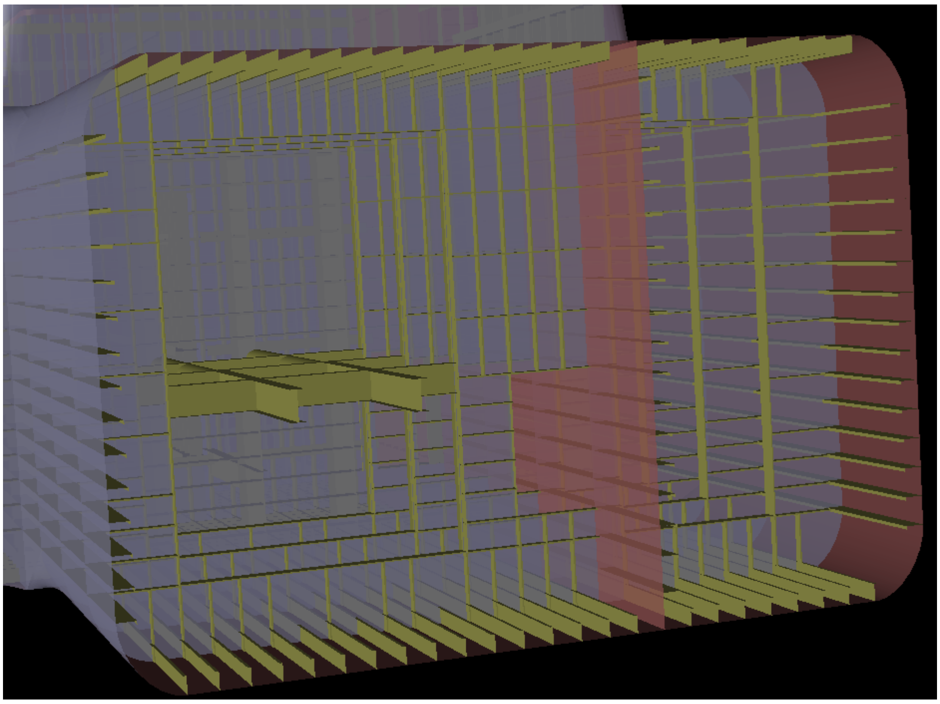

Figure 6-16 Sub model: stiffener details in the pontoon

6.5.1 Meshing

As in the global model, the size of the mesh is depending on minimum stiffeners spacing. The size selected correspond to the minimum spacing between stiffeners in the pontoons, also being influence by the plate thickness and internal structure such as bulkheads, frames, geometric details etc. Taking into account all the considerations the size selected was equal to 0.625 by 0.625 meters.

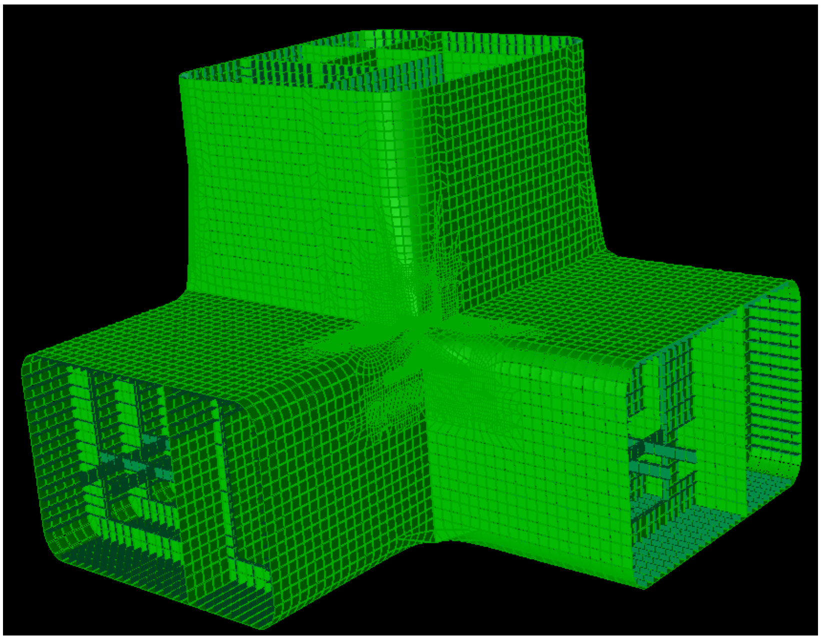

Figure 6-17 Overview of the sub model mesh

Mesh size is modified when required to avoid stress concentration or irregular mesh in critical areas. Figure 6-17 and 6-18 could shows the mesh built.

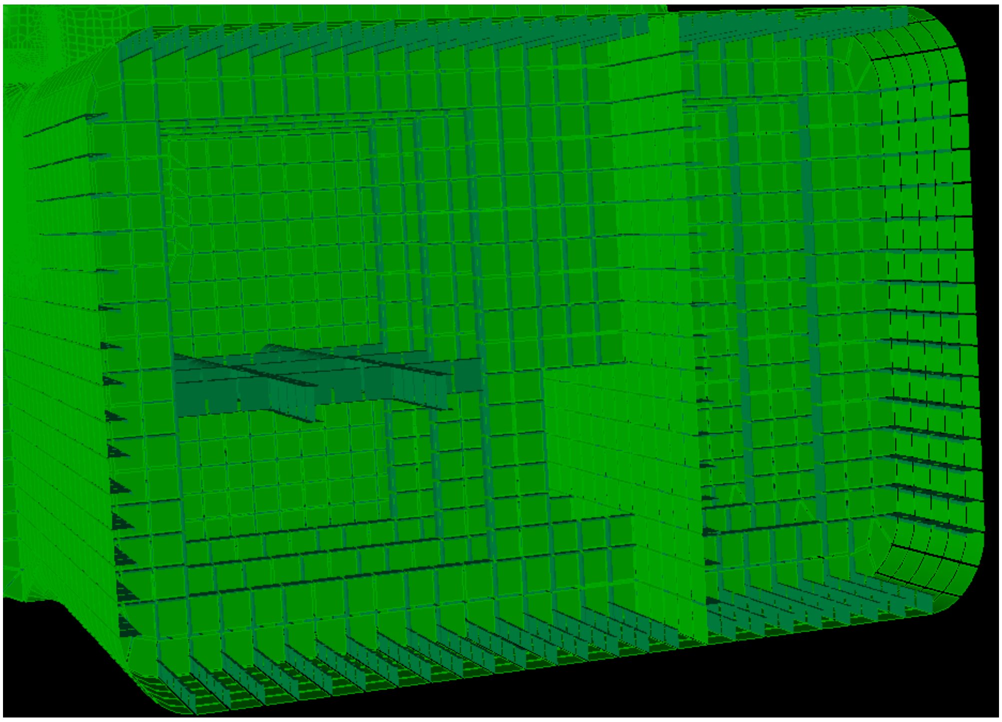

Figure 6-18 Sub model: Pontoon meshing

6.6 Local Model

The local or Stress concentration model is the one which has higher grade of detail. The aim of this model is to allow the calculation of the stress at the weld toe due to the presence of the attachment, denoted hot spot stress. This will be accomplished by the use of local fine mesh with size t x t, being t the plate thickness.

The FE used for the modelling of the structure are:

- Shell elements for all the structural elements

The principal structural elements, secondary structural elements and structural detail are modelled with •8-noded shell elements or improved 4-noded shell elements with additional internal degrees of freedom will be used for the FE modelling of the structure. 6-node triangular elements will be also used in areas where a mesh of 8-noded shell elements otherwise is difficult to fit.

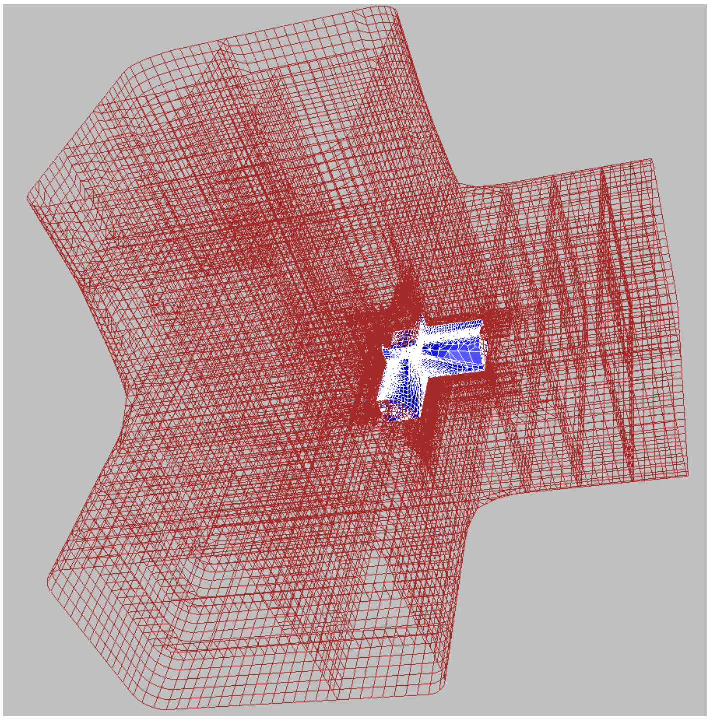

Figure 6-19 Location of the critical area in the sub model

The details considered in the local model are:

- Bracket and flange terminations for main girder systems.
- Stiffener connections.

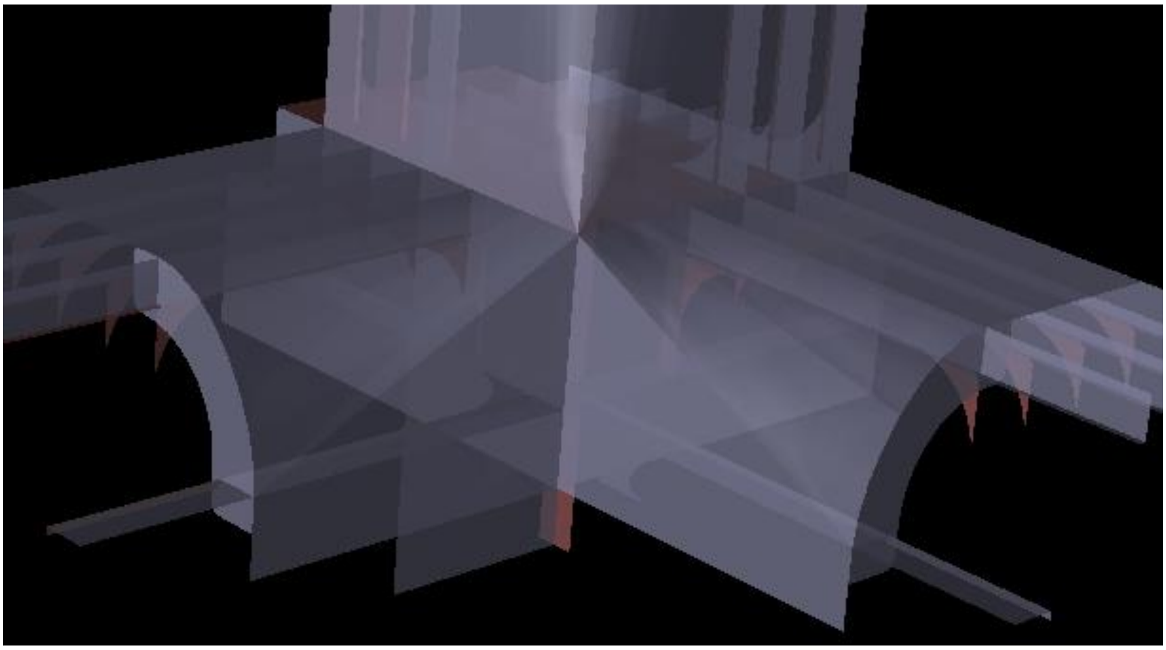

Figure 6-20 Structural detail included: Brackets

To develop the local model, there are two possibilities:

- Create a new sub-model with boundary conditions
- Refine the mesh in the existing sub-model, decreasing the mesh size in the critical areas in a progressive way, till arrive to a mesh size of t x t where the hotspots are supposed to be.

The option selected was the second, for this reason a progressive increment in the mesh density can be observe, Figure 6-21.

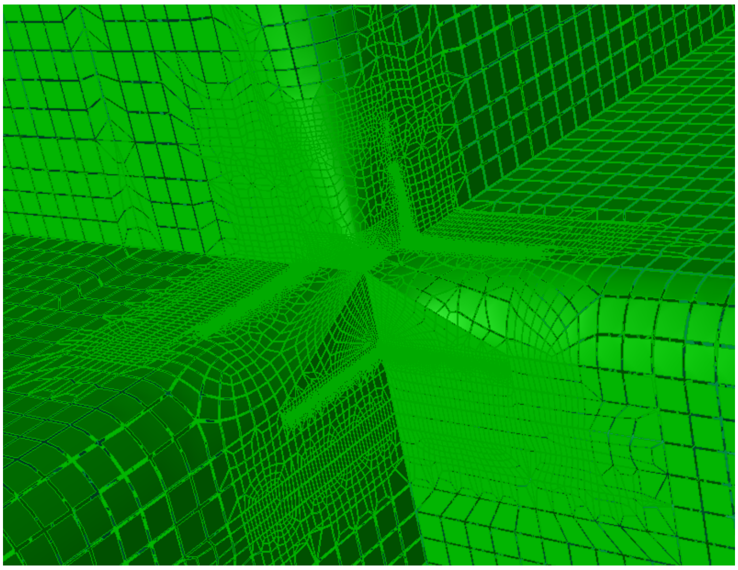

Figure 6-21 Overview of the mesh in the critical area

6.6.1 From Beam to Shell Elements

The local model will be created by using shell elements or alternatively with solid elements. Since the software used is not able to generate solid elements, shell elements will be used.

Stiffeners and other elements modelled before as beam elements should now be modelled as shell to determine the irregular stress distribution around them. I.e. in the transition area between the global models to the local model, where stiffeners were beam elements were located now shell elements will substitute them.

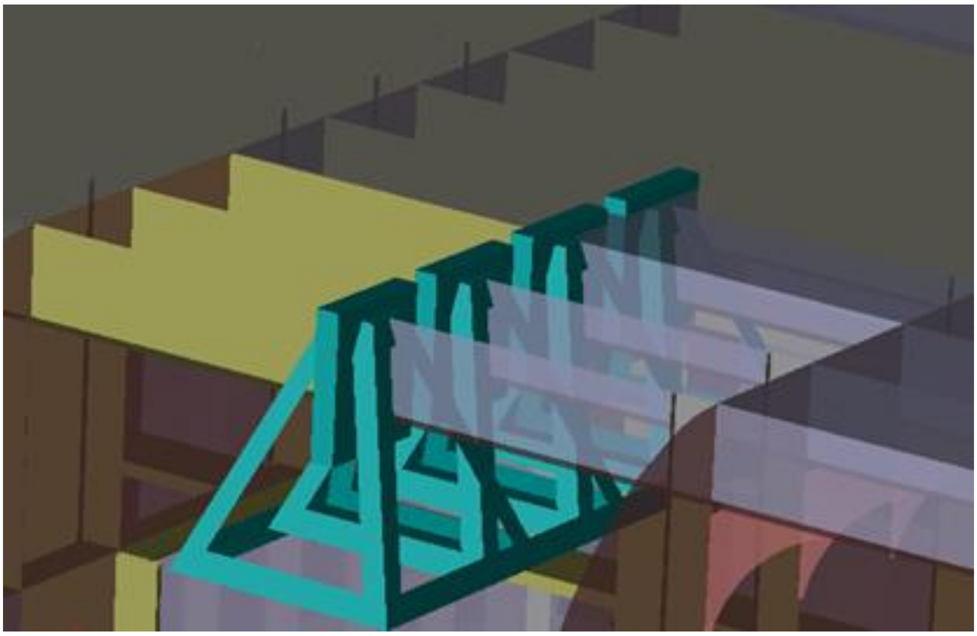

Figure 6-22 Transition area between beam and shell element

The stiffeners close the critical area of the local model, will be divided between frames or bulkhead. In the division point a support rigid link will be situated, defining a region of the structure which will act as a rigid body. This region will join the stiffener represented by beam elements, with the part represented by shell elements (part of the local model). Prcedures can be observed in figures 6-22 and 6-23.

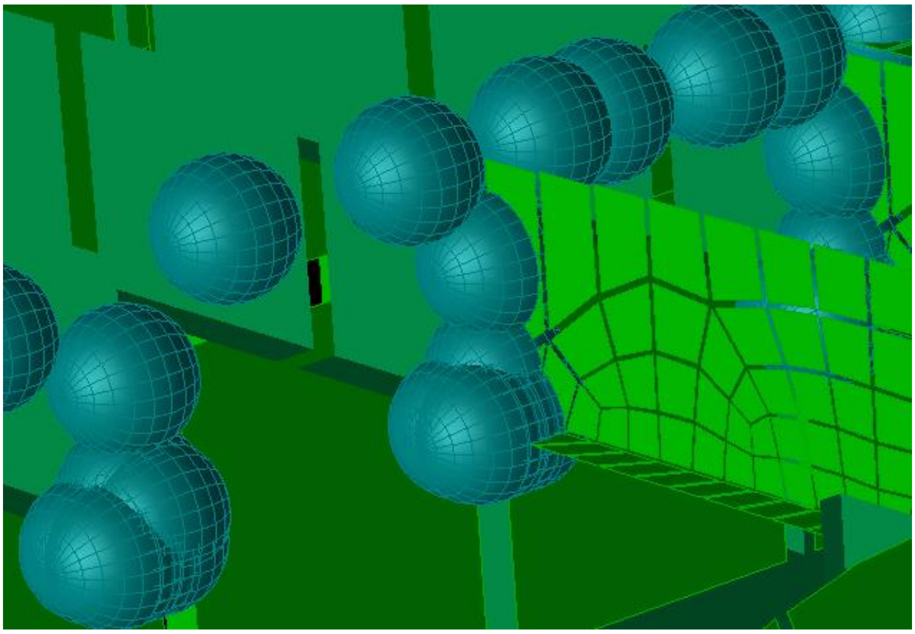

Figure 6-23 Transition between Beam and shell elements: mesh view

6.6.2 Meshing

The size of the mesh which will be used in the local model, will be equal to t x t, being t the thickness of the plate situated in the critical area.

Finally the appearance of the mesh could be observe in Figure 6-24. The areas where the density of the mesh is more elevated, correspond to the critical areas of the local model and points where the probability to have hotspots is higher.

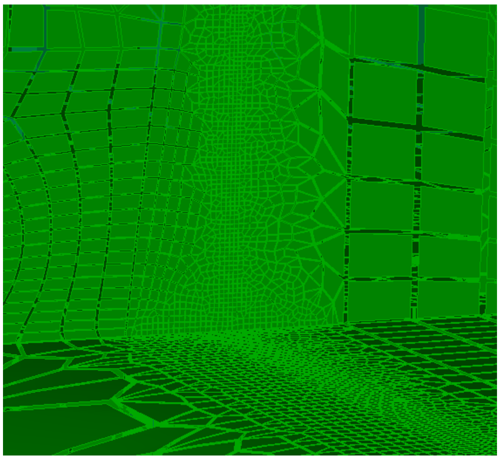

Figure 6-24 Close view of the mesh in the hotspot

6.7 Relevant Notes

Due to confidentiality reasons some of the structural details were not proportionated. Thus simplifications for structural areas are considered, especially for sharp angles in corners between pontoons and column.

It is known that this assumption will generate a reduction in the fatigue life. This reduction attend to the influence of structural discontinuities due to the geometry of the detail.

Figure 6-25 Area not considered in the fatigue analysis

For this reason, contribution of the corners will not be taken into consideration in the fatigue analysis. Being the analysis of this areas focus in the union between cast steel and steel elements

7 HYDRODYNAMIC ANALYSIS

In this chapter an explanation of the Hydrodynamic analysis and methodology followed is provided. In addition, it is present the theory behind the analysis.

A calculation of the environmental loads acting in the TLP structure will be carried. The calculation of the loading will be limited to the wave forces.

The analysis will be performed in the frequency domain, concretely a wave frequency dynamic analysis will be performance by using linear wave theory in order to determine first-order platform motions and wave loading. A sufficient number of wave approach headings will be selected for analyses. Potential flow theory in combination with Morison equation will be used to determine the wave loading.

Two types of calculations will be performed:

- Hydrostatic calculations.
- Load calculations, in which the pressure distribution acting on the hull panels is solved by direct numerical integration. The loads are transferred to the structural model for subsequent quasi-static structural analysis performed by SESTRA.

To perform the hydrodynamic analysis a hydro model was created in HYDRO-D software, Figure 7-1.

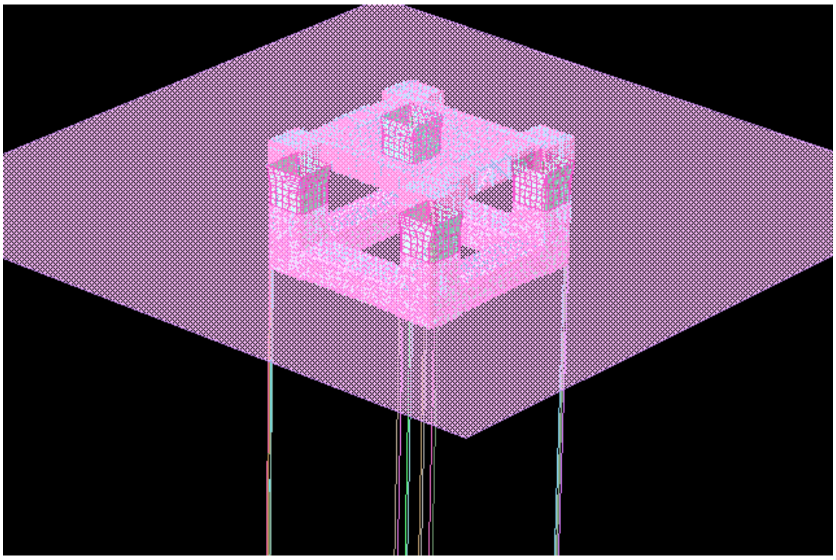

Figure 7-1 Overview of the hydro model in Wadam

From a practical point of view is interesting to know which are the main forces involved in the current analysis. After determined, is possible to select the method needed to include this forces in the analysis.

The basic equation for a floating structure is given in next formula:

$$[M]\ddot{X} = F_{WS} + F_{PTO} + F_{gravity} + F_{moorings} + F_{others} \tag{10}$$

Where the floating structure, TLP in this case, will be affected by different forces. From previous formula, some of the components could be neglected since do not affect the current analysis. The effect which could be neglected directly are:

- F_{PTO}, since no power take off device is considered.
- F_{other}, these effect take into account:
  - Wave drift forces: 2nd order constant force due to response of system in waves
  - Nonlinear wave forces
  - Drag forces due to current and wind

  The analysis will be carried out by using linear wave theory. Also wind and current loads are not considered in the scope of the analysis. Thus these effect are not considered

After these simplification the previous formula result in:

$$[M]\ddot{X} = F_{WS} + F_{gravity} + F_{moorings} \tag{11}$$

From this formula is extracted, that the main forces which affect the analysis are:

- F_{WS}, wave structure forces, related to the interaction between waves and the structure.
- $F_{moorings}$, mooring forces. Will be taking into account by the TLP mooring system included in the analysis.
- $F_{gravity}$, take into account the effect of the gravity. This gravity force will be combined with the buoyancy force included in the wave structure forces, giving as result the hydrostatic force.

The main aim is the study of the wave structure forces with included the most important effects which will taking into account in the TLP structure. These effect are:

- Hydrostatic forces: Forces induced by the displacement of the platform from the mean position, are obtained by calculating the difference between buoyancy force and gravity force. In case of small displacement could be modelled as a restoring force represented as:

$$F_{hydrostatic} = -[K_H]X \tag{12}$$

Where
K_H: hydrostatic stiffness matrix
X: displacement

- Excitation forces: Excitation force is the sum of the incident wave force and the diffraction wave force.

$$F_{ex} = F_I + F_D$$

  - Froude-Krylov forces or incident wave force:
    Force on the body introduced by the unsteady pressure field generated by the undisturbed wave field and the surface.

  - Diffraction forces:
    Consequence of the unsteady pressure field generated by the diffracted wave, as result of the interaction between the body and the incident waves.

- Radiation forces: Product of the unsteady pressure field associated with the radiated wave field, which is generated by the unsteady motion of the platform.

The radiation force could be summarized as follows:

$$F_R = -[A]\ddot{X} - [C]\dot{X} \tag{13}$$

Where
A: three dimensional hydrodynamic coefficient of added mass
C: three dimensional hydrodynamic coefficient of damping

- Drag forces: Forces generated as consequence of flow detached from the body.

Finally, the equation of motion for this particular analysis will be:

$$[M]\ddot{X} = F_{hydrostatic} + F_{ex} + F_{radiation} + F_{drag} + F_{moorings} \qquad (14)$$

$$[M+A]\ddot{X} + [C]\dot{X} = -[K_H]X + F_I + F_D + F_{drag} + F_{moorings} \qquad (14)$$

7.1 Forces Involved in the Hydrodynamic Analysis

Previously, the different forces which could affect the TLP structured were numerated. However not always these forces are included in the analysis.

From the equation 15, if the mooring force is neglected (The loads on tendon elements are neglected in this study):

$$[M+A]\ddot{X} + [C]\dot{X} = -[K_H]X + F_I + F_D + F_{drag} \qquad (15)$$

In order to determine which force will or not being taken into account in the analysis, the Keulegan–Carpenter number (KC) will be used, evaluating the pontoons and columns of the TLP.

KC is one dimensionless number, which base in a relation between de main dimension of the body and the wave length of the waves affecting the structure, describe the relative importance of the drag forces over inertia forces for bodies that oscillate in a fluid at rest. For small Keulegan–Carpenter number inertia dominates, while for large numbers the (turbulence) drag forces are important.

$$KC = \frac{2\pi A}{L} \qquad (16)$$

Where:
A: wave amplitude
L: characterize dimension of the body

Depending of the KC value, the body analysed could be classified between Slender or large bodies:

- Slender bodies KC (>10)
    - Flow separation
    - Incident flow is not perturbed much.
    - Drag forces are dominating
    - Diffraction/radiation forces are negligible

- Large bodies KC (<2)
    - Flow is attached to the body
    - Large perturbation of the incident flow
    - Diffraction/radiation forces are most important
    - Effect of drag forces important only at resonance

As it was mention before no analysis of the mooring will be performed, so just the pontoons and columns are analysed.

The design wave parameter have been selected based on one of most extreme climates i.e. in the North Sea with 100 year return period found for the current operating locations of mobile offshore units as given in Table 8 below.

Table 7-1 Design wave parameter

Significant wave height (m)	13.6
Spectral peak period a (s)	16

With all the previous data and known that the characterize dimension for columns and pontoons is equal to 17.5, the resulting KC number is:

$$KC = \frac{2\pi A}{L} = \frac{2\pi * 6.8}{17.5} = 2.441 \qquad (17)$$

Theoretically and Based on the results obtained, which is situated in the zone between large and slender bodies, a Morison model should be added to the panel model, in order to predict the drag forces, and correct the added mass and damping coefficients. However KC number is closer to the large bodies effects were the inertial forces will be the most important and the drag forces will be important just in resonance (The resonance of the TLP will be far of the frequencies analysed as it will be demonstrated in the following sections). So it is clear that the main purposed of the Morison model the correction of the added mass and damping matrixes.

DNVGL standards offer another way to check the influence of the relevance of drag and inertia forces. The evaluation is based in two dimensionless numbers:

- $\pi L/\lambda$
- H/L

Where:
L= characterize dimension of the body
H= wave height
λ= wave length

As could be observe the value of the wave length is necessary to perform the evaluation. Since the analysis is base in linear wave theory, is possible to obtain the wave length for the columns and pontoons by using the dispersion relation in deep waters.

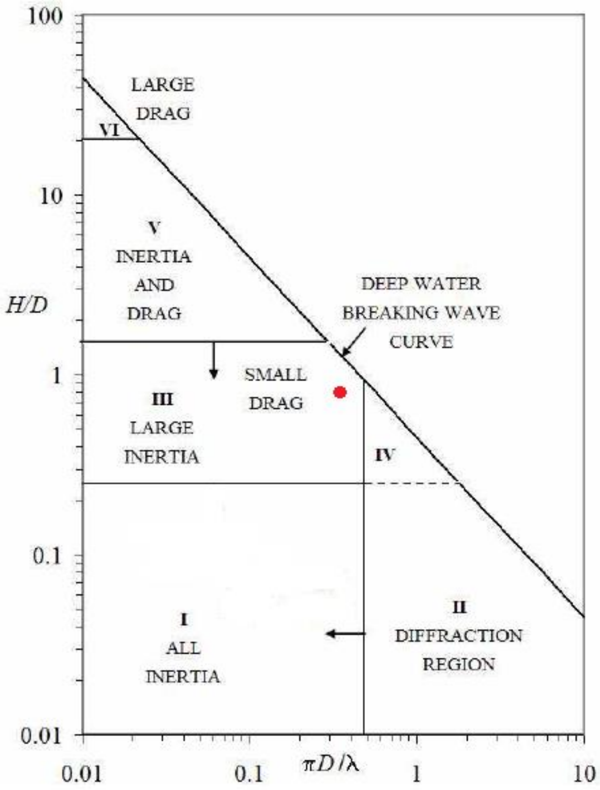

Figure 7-2 Different wave force regimes. Available from Offshore standard DNV-RP-C205

$$w^2 = gk\,tanh(kd) \tag{18}$$

$$w = \frac{2\pi}{T} \tag{19}$$

$$k = \frac{2\pi}{\lambda} \tag{20}$$

Where:

d = water depth

T= wave period

Giving as results:

Table 7-2 Dimensionless parameters

Water depth	327.5 m
characterize dimension	17.5 m
wave length	399.63 m
$\pi L/\lambda$	0.13
H/L	0.777

By situated the values obtained in the graphic of DNVGL, red point in the Figure 7-2, is possible to observe that the main forces correspond to the inertia being the drag force small. Being agree with the conclusions obtained from the KC number.

In short, for the development of the hydrodynamic analysis will be necessaries two model:

- Panel model
- Morison model

Being the Morison model included mainly due to motion purposes.

7.2 Analysis Set Up

In this section, the steps followed in the hydrodynamic analysis and the input used will be explained. Must be mentioned that the data used in the analysis correspond to the environmental condition of the North Sea.

In the analysis a load coming from the waves is transferred to the structural model. The hydro model is defined as floating.

7.2.1 Translation of Models and Location of the Platform

The models used in the analysis are: Structural, Morison and Panel models. The structural model was describe in chapter 6. Being the Morison and Panel model described in the sections 7.3 and 7.4.

The coordinate system used for the hydrodynamic analysis has been taken the same as the global structural modelling except for the Z axis (Notice that the draught of the TLP is 27.5 m). The hydro model is translated in the vertical direction –27.5 m to ensure that mean sea level is around z= 0 as per recommendation of the software manuals.

Regarding the location of the platform in the analysis, it was assumed to be uniform 327.5 m which is in coherence with the range of water depth expected in the central North Sea region and typical water depth for operation of TLP in that region.

7.2.2 Sea State

Short term stationary irregular sea state.is characterized by statistics, including the wave height, period, and wave spectrum. It is common to assume that the sea surface is stationary for a duration of 20 minutes to 3 to 6 hours, in this case the sea state was defined for the duration of 3 hour according to the recommended practices in the offshore design codes.

Design wave was selected based on most extreme environmental conditions of North Sea with 100 year return period. Significant Wave height Hs=13.6m and Peak Period (Tp) =16s.

Respect to the wave spectrum the rules assign a particular wave spectrum to the analysis of column stabilised units: The Pierson-Moskowitz wave spectrum (PM). This spectrum is given by:

$$S_{PM}(\omega) = \frac{5}{16} * H_S^2 \omega_P^4 * \omega^{-5} \exp\left(\frac{-5}{4}\left(\frac{\omega}{\omega_P}\right)^{-4}\right) \tag{21}$$

Where

ω_p: angular spectral peak frequency

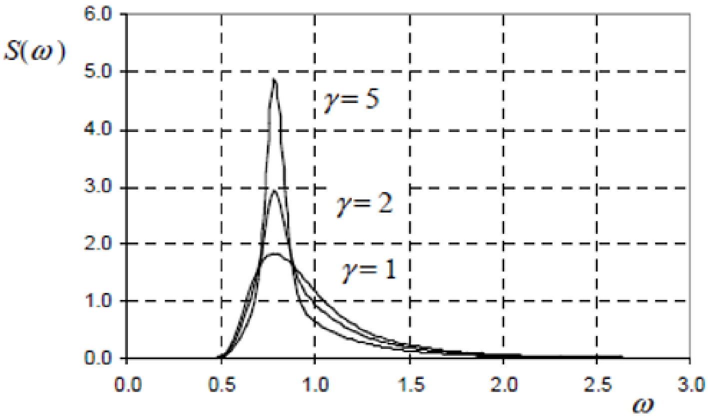

Figure 7-3 Pierson-Moskowitz wave spectrum γ=1. Available from Offshore standard DNV-RP-C205

For $\gamma = 1$ the JONSWAP spectrum reduces to the Pierson-Moskowitz spectrum.

7.2.2.1 Direction Set

For wind generated sea it is often a good approximation to assume that wind and waves are inline. The main direction may be set equal to the prevailing wind direction if directional wave data are not available. In this analysis main heading and therefore the prevailing wind direction is assumed to be 60 degree with respect to longitudinal axis.

A sufficient number of wave approach headings shall be selected for analyses (e.g. with basis in global configuration, number of columns, riser configuration etc.). For the Long term wave statistics, required to perform the stochastic fatigue analysis, the 360º range is typically divided into eight 45º, twelve 30ºor sixteen 22.5ºdirectional sectors.

Since the case studied correspond to a standard design of TLP and mainly due to software limitations, the directions selected are: First direction with 0 degree and last direction 315 degree, as it is showed in Table number 13.

Table 7-3 Wave headings

No. of Direction	Direction(deg)
1	0

No. of Direction	Direction(deg)
2	45
3	90
4	135
5	180
6	225
7	270
8	315

7.2.2.2 Wave Spreading Function

The distribution of wave directions within a short-term sea state has a relevant influence on the fatigue loading of offshore structures. It is common practice in the industry to simulate wave loads according to different sea states in the frequency domain by linear superposition of the particle kinematics of many linear waves having different amplitudes, periods, and phase angles. However, this approach assumes that the wave crests are infinitively long, so that all waves propagate only into one direction.

Obviously, this leads to inaccurate predictions of fatigue loads when compared to real-life measurements, for this reason the spreading function is introduced. Better agreement between simulation and reality can be achieved if wave spreading is taken into account in the simulation of the sea states.

For this reason the spreading function is introduced in the simulations. This means the distribution of wave directions within a short-term sea state around the mean wave direction of the sea state. Such a sea state can be best described by a superposition of many linear waves having different wave heights, periods, phase angles, and directions. Fig. 7-4 illustrates the difference between a linear airy-wave and a short term sea state with wave spreading.

Base in the recommendation practice, DNV-RP-C103, a $\boldsymbol{\cos^4\alpha}$ wave spreading function was utilised. Being α the angle between direction of elementary wave trains and the main direction of the short-crested wave system.

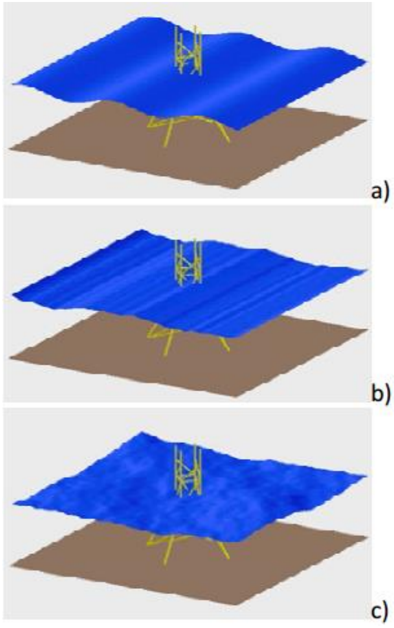

Figure 7-4 Illustration of the difference between infinitively long wave crests and more realistically spread wave directions. a) Airy wave; b) Irregular, unidirectional sea state; c) Irregular sea state with wave spreading. Available from [1]

7.2.2.3 Wave Periods

The various time periods for which the response was analysed were taken in the range of 4 seconds to 36 seconds with an interval of 1.5, which is related to the range of wave period in the North Sea (wave frequency analysis).

7.3 Morison Model

The Morison model is used to calculate hydrodynamic loads, viscous damping and added mass based on Morison's equation. Also it is used to include the effect of mooring, which restrict the movement of the platform and therefore affect the interaction with wave loads. In resume, the loads will be affected the use of Morison model have a direct relation with the structural model and the final fatigue life of the platform.

As it was mentioned before, the drag forces have small relevance in this analysis, however the effect of Morison models in the damping and added mass is considerable. The influence of the Morison model in the damping and added mass matrix, which are combined with the existing matrix coming from the potential theory, was one of the main reason to include this model in the analysis. The combination was obtained by using a dual-model (hydro model where a panel model and a Morison model represents the same part of a structure) in the columns and pontoons.

The basic idea with the dual model is that panel model with potential theory is used to include the forces related potential damping whereas the Morison model is used to include the viscous damping, added mass and drag forces. Also the damping and added mass matrix's conditioned the eigenvalues of the structure and rigid body motion of the platform. The eigenvalues are really important in order to justify the use of a quasi-static analysis (chapter 8).

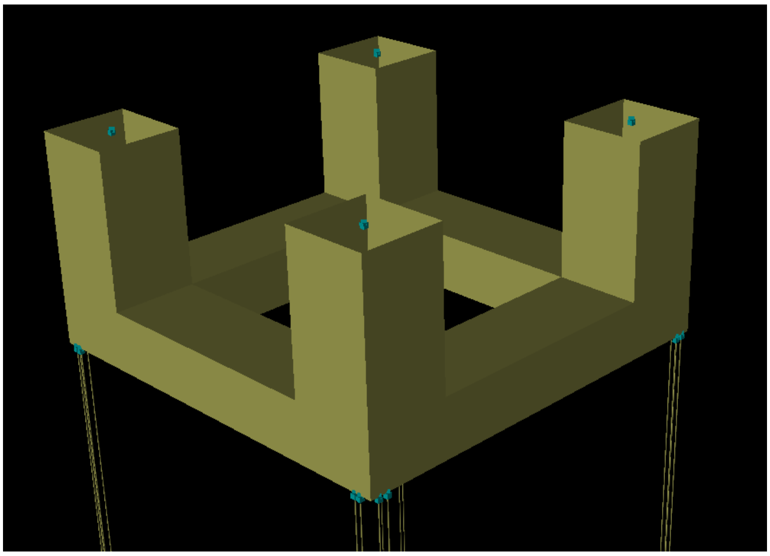

Figure 7-5 Morison model

In addition, the Morison model is used to include the restriction of movement coming from the mooring lines in the hydro model.

The Morison model is put together from a set of Morison elements. The Morison elements are actually defined by assigning hydrodynamic properties to beam elements. The type of Morison elements used for calculation of hydrostatic and hydrodynamic effects is:

- 2D Morison elements for calculation of hydrostatic and hydrodynamic loads on wet 2 node beam elements. Presenting the mesh the next aspect:

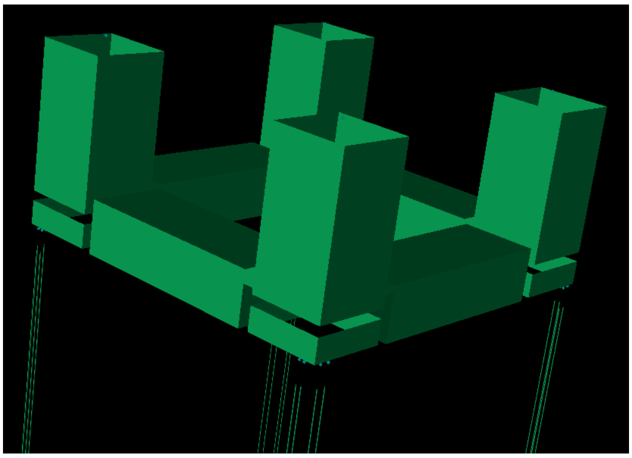

Figure 7-6 Mesh view of the Morison model

7.3.1 2D Morison Elements

2D Morison element are conveniently defined as a 2 node beam element in Genie and assigned section numbers to be matched by hydro property sections specified in HydroD.

The hydro property description for a 2D Morison element include added mass and viscous drag coefficients ,which were obtained from the DNV-RP-C205 appendix D, in the two directions perpendicular to the longitudinal element axis. The hydrodynamic coefficients are specified in a coordinate system local to each 2D Morison element.

These 2D Morison element correspond to:

- Columns
- Pontoons
- Tension leg anchor elements

It is important to remark that the hydrodynamic coefficients specified for a 2D Morison element apply to circular cross sections. For elements with non-circular cross-sections, which is the case of the columns and pontoons, the hydrodynamic coefficients in the ξ and ζ directions are directly related to an equivalent cross-sectional diameter. The equivalent diameter was calculated by Wadam as the circumscribing diameter as shown in Figure 2.10.

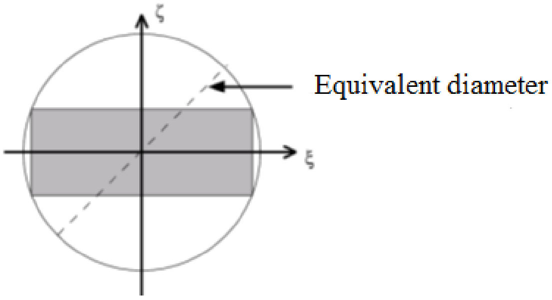

Figure 7-7 Equivalent diameter

Since the displacement of the columns and pontoons will be considered by the potential theory, panel model, the equivalent diameter of each 2D Morison element was divided by 10,000 to neglect the buoyancy of these elements. Apply this division to the diameter supposed that the drag coefficient should be also multiplied by 10,000. Finally the coefficients were:

Table 7-4 Added mass and drag coefficients

2D Morison element	Columns	Pontoons	Tension leg anchor elements
Coefficients			
Added mass Cay	0.52	1.51	1
Added mass Caz	0.52	1.51	1
Drag Cdy	22000	22000	7000
Drag Cdz	22000	10000	7000
Equivalent diameter (m)	0.00195 m	0.0008176 m	2.29 x e^-5 m

7.3.2 Mooring System: Tension Leg Anchor Elements

Reference to the mooring system was done in previous chapters, however not detail explanation have been proportionated yet. In this section all the data related to the mooring will be clarified.

It is important to remark that TLP´s usually have an excess of buoyancy which introduce a pre-tension in the mooring system reducing the possible movements. In the hydrodynamic analysis the mooring elements were introduced as TLP sections. The parameters that define a tension leg anchor elements are:

- Length of the tendons
- Tendon pre-tension
- Anchorage point location in the Platform and in the Seabed.
- Stiffness of the tendon

The water depth of site assessment is assumed to be uniform 327.5 m and the platform draft is equal to 27.5 m. The attachment point of the mooring to the structure are located at the bottom of the structure with length equal to 300 m.

In the structural modelling a pre-tension of the mooring was used to determine the final displacement of the global model. However no explanation for the calculation of this parameter was provided. In order to perform a realistic analysis, the data of two similar TLP´s operating in the North Sea are considered to estimate the pre-tension load required for our model.

Table 7-5 Similar TLP´s operating in the North Sea

TLP	Design displacement	Total Pre- tension
Hutton	63770 t	130 MN
Snorre	106520 t	250 MN

Considering the design displacement of our model and the data presented above, a gross estimation is made interpolating with the available data. Within this procedure, the value obtained for the pre-tension is:

Table 7-6 Pre-tension interpolation

TLP	Design displacement	Total Pre- tension	Total Pre-tension
Case of Study	76375.62 t	165.384 MN	16858.736 t

No offset was considered for any of the mooring lines. Concerning to the stiffness, the parameter is defined by the number of mooring lines, diameter, material and type of construction of the tendon selected.

For the analysis the number of mooring lines selected is equal to 16, where 4 lines are located at the bottom of each column. Finally the properties of the mooring lines are presented in Table 7-7.

Table 7-7 Properties of the mooring lines

Total Pre-tension	165.384 MN
Pre-tension in each tendon	10.336 MN
Pre-tension in each tendon in extreme climate conditions	41.346 MN
Diameter	0.235 m
Construction	Spiral strand
Security factor	1.2
Stiffness	49.615 MN/m

7.4 Panel Model

The panel model is used to calculate the hydrodynamic loads and responses from potential theory. It describe the entire wet surface of the structure and the parts which could be in contact with the water.

The model developed is composed by quadrilateral and triangular panels representing the wet surfaces of a body. By using two planes of symmetry, axis x and y, the computational effort is reduced.

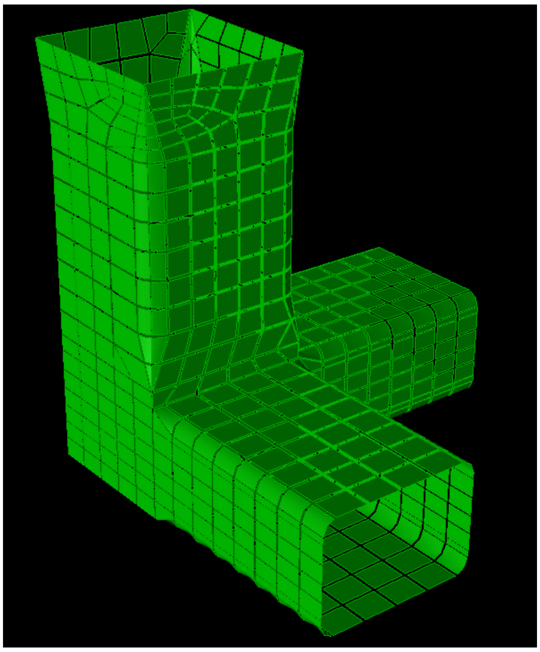

Figure 7-8 Mesh view of one quarter of the panel model

The mesh size correspond to the spacing between stiffeners in the global model, 2 by 2 meters.

8 SOLVING THE STRUCTURAL MODEL: QUASI STATIC ANALYSIS

In structures, Tree methods can be generally used to study the structures response under external loads and these are: Static Analysis, Quasi-Static Analysis and Dynamic Analysis. Each of them are strictly linked to the features of the loads acting on the body.

The dynamic structural response will depend on the ratio between the excitation frequency and the structural natural frequency, I.e. in case of a vessel, the structural response will be conditioned by the incoming wave excitation. Structural response due to dynamic loading may be analysed by a quasi-static method when the frequencies of the load, wave’s frequencies in this case of study, are far enough from the eigenfrequencies of the structure.

Base in this affirmation, it is necessary to determine the eigenvalues of the structure, for the six DOF, and compare them with the wave´s frequencies used. An eigenvalue analysis (also known as free vibration analysis) is carried out, the input parameters are structure, boundary conditions and masses. Being the mass matrix built up from the structural mass and the added mass, obtained from the hydrodynamic analysis.

The time periods for which the response was analysed were taken in the range of 4 seconds to 36 seconds with an interval of 1.5, which is related to the range of wave period in the North Sea (wave frequency analysis).

Once the eigenvalue analysis is carried out, the most critical periods due to their proximity to the wave frequencies values are presented in next Table.

Table 8-1 Eigenfrequencies of the structure

Degree of freedom - Movement	Period
Surge	80.67 s
sway	80.55 s
Heave	0.092 s
Rolling	0.065 s
Pitching	0.066 s
Yaw	53.03 s

Using this values and the wave frequencies, is possible to evaluate the introduction of any dynamic effects. The Dynamic effect in the structures could be represented by a Dynamic amplification factor (DAF). Being this effect relevant when DAF, formula 22, is higher than 1.1.

$$DAF = \frac{1}{\sqrt{(1-\Omega^2)^2 + (2\delta\Omega)^2}} \tag{22}$$

where:

δ: Damping

Ω: frequency ratio= Load frequency/ Natural frequency of the structure (eigenfrequency)

Representing this DAF again the frequency ratio in some graph, is possible to evaluate if the dynamic effects will affect the case of study.

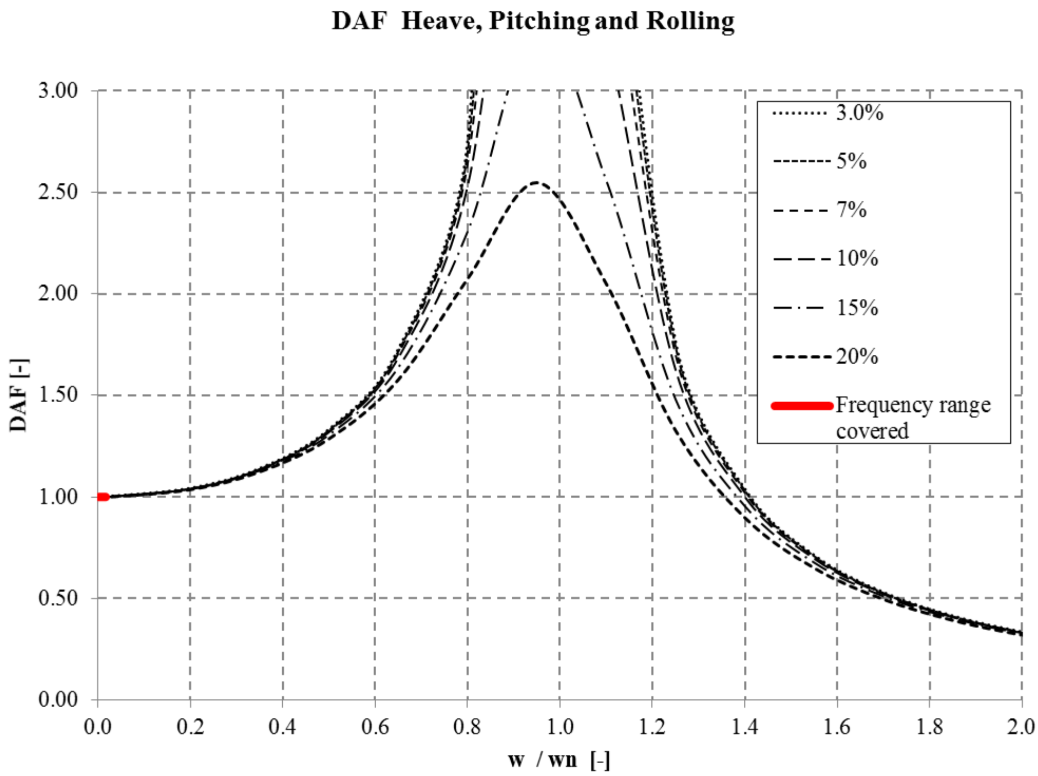

Figure 8-1DAF for the heave, pitching and rolling degrees of freedom (DOF)

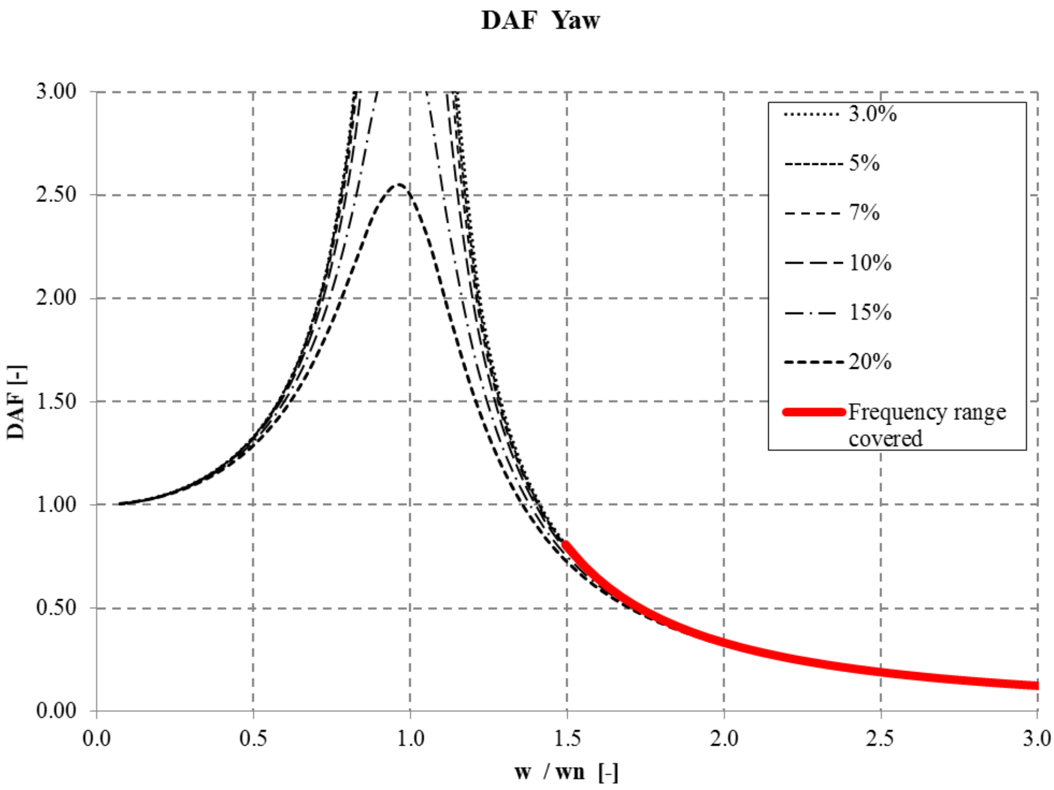

Figure 8-2 DAF for the yaw DOF

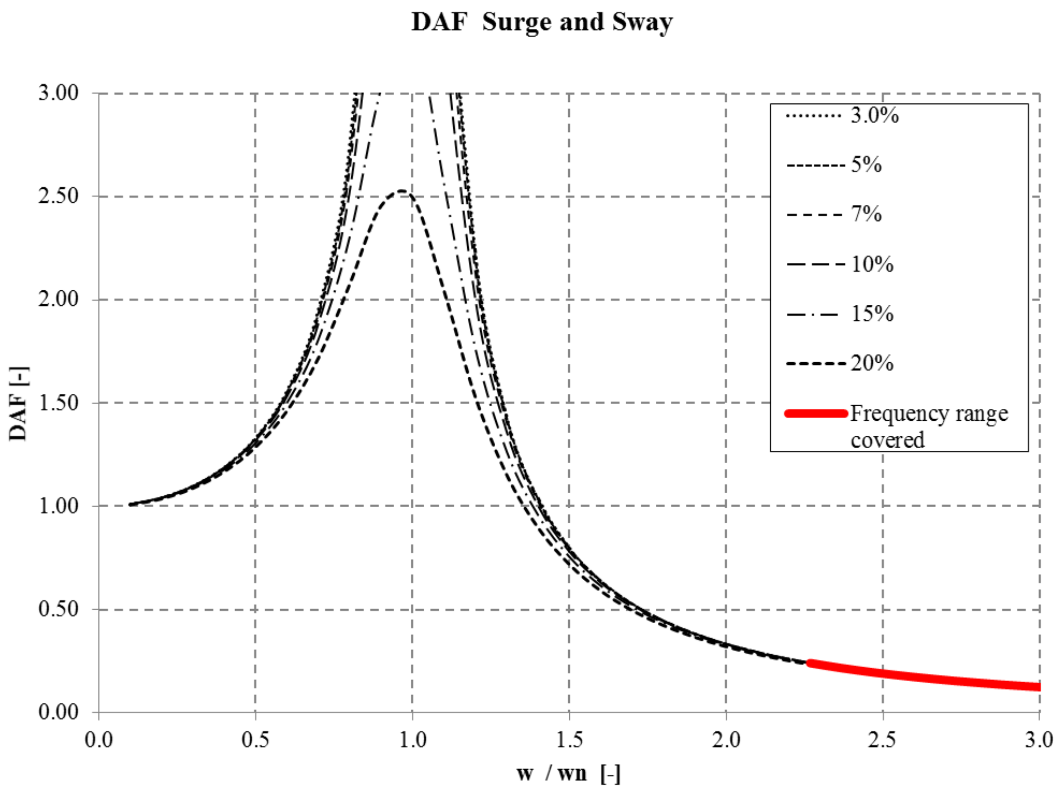

Figure 8-3 DAF for the surge and sway DOF

In all the cases the maximum value of DAF not exceed the value of 1, being under this value in the case of Surge, Sway and Yaw movements.

Consequently, is demonstrated that the use of a quasi-static analysis for the case of study is correct and what is more will offer conservative solutions. Being the dynamic response lower that the static one for frequency ratios over the square root of 2.

9 FATIGUE ANALYSIS

9.1 Stochastic Fatigue Analysis Based on S-N Data

The calculation of fatigue damage will be based on S-N data (RP-C203: Fatigue design of offshore steel structures given by DNV GL), determined by fatigue testing of the considered welded detail, under the following assumptions:

- Wave climate is represented by long term scatter diagrams (summation of short term conditions).Assumed to be represented as a Gaussian process.
- Rayleigh distribution applies for stresses within each short term condition.
- Cycle count is according to zero-up-crossing period, Tz, of short term stress response.
- Fatigue damage summation is according to Miner’s rule for linear cumulative damage.

In the analysis linear load effects and responses are assumed. The hydrodynamic loads and structural responses are calculated using 3D potential theory, Morison’s Equation and FE analysis, respectively.

Principal stresses used for calculation of fatigue in the critical areas are based on notch stress. The notch stress is either calculated using a local hot spot FE model. The total fatigue damage (Miner sum) is computed by summing up the contributions from each sea state taking into account their probability of occurrence, wave spreading, and the S-N curve (including effect of welds) and an appropriate stress concentration (given by the local FE model) for the critical area.

9.2 Analysis Set Up

Several fatigue analysis were performed in this document. A fatigue screening analysis of the global model sub model, and local model, where a hotspot fatigue calculation was also performed. These analysis have some input parameter in common which determine the results obtained. This section give a description of the environmental parameters relevant for fatigue assessment

9.2.1 Wave Loads

Reference should be done to the main loads which generate the fatigue in the structure, those induced by waves. In order to establish the characteristic response, the waves have to be described in detail.

Stochastic methods for fatigue analysis (FLS) are recognised as the best methods for simulating the irregular nature of wave loads. Motion characteristics are determined by stochastic methods by using North Atlantic environmental data.

The Pierson-Moskowitz wave spectrum is applied for evaluation of dynamic response in short term sea states. Marine growth, wind, ice, snow and current loading are omitted.

According to the standards, the operational history of the TLP, time spent at different locations should be documented and the unit's total history should be accounted for in the fatigue assessment. The operational history is typically described through site specific wave scatter diagram.
In this particular analysis, the basic description of the wave conditions take the form of a 2-dimensional scatter diagram (Hs, Tp/Tz diagram), Scatter Diagram for the North Atlantic, showing the relative frequency of various combinations of significant wave height and peak wave period (or zero-up-crossing period).

As it was mention in previous chapter, the effect of wave spreading was taken into account in the fatigue calculation with a spreading function of the form $\cos^4\alpha$, see Figure 9-2. Being α the angle between direction of elementary wave trains and the main direction of the short-crested wave system.

Table C-2 Scatter diagram for the North Atlantic

T_Z (s)	3.5	4.5	5.5	6.5	7.5	8.5	9.5	10.5	11.5	12.5	13.5	14.5	15.5	16.5	17.5	18.5	Sum
H_S (m)																	
0.5	1.3	133.7	865.6	1186.0	634.2	186.3	36.9	5.6	0.7	0.1	0.0	0.0	0.0	0.0	0.0	0.0	3050
1.5	0.0	29.3	986.0	4976.0	7738.0	5569.7	2375.7	703.5	160.7	30.5	5.1	0.8	0.1	0.0	0.0	0.0	22575
2.5	0.0	2.2	197.5	2158.8	6230.0	7449.5	4860.4	2066.0	644.5	160.2	33.7	6.3	1.1	0.2	0.0	0.0	23810
3.5	0.0	0.0	34.9	695.5	3226.5	5675.0	5099.1	2838.0	1114.1	337.7	84.3	18.2	3.5	0.6	0.1	0.0	19128
4.5	0.0	0.0	6.0	196.1	1354.3	3288.5	3857.5	2685.5	1275.2	455.1	130.9	31.9	6.9	1.3	0.2	0.0	13289
5.5	0.0	0.0	1.0	51.0	498.4	1602.9	2372.7	2008.3	1126.0	463.6	150.9	41.0	9.7	2.1	0.4	0.1	8328
6.5	0.0	0.0	0.2	12.6	167.0	690.3	1257.9	1268.6	825.9	386.8	140.8	42.2	10.9	2.5	0.5	0.1	4806
7.5	0.0	0.0	0.0	3.0	52.1	270.1	594.4	703.2	524.9	276.7	111.7	36.7	10.2	2.5	0.6	0.1	2586
8.5	0.0	0.0	0.0	0.7	15.4	97.9	255.9	350.6	296.9	174.6	77.6	27.7	8.4	2.2	0.5	0.1	1309
9.5	0.0	0.0	0.0	0.2	4.3	33.2	101.9	159.9	152.2	99.2	48.3	18.7	6.1	1.7	0.4	0.1	626
10.5	0.0	0.0	0.0	0.0	1.2	10.7	37.9	67.5	71.7	51.5	27.3	11.4	4.0	1.2	0.3	0.1	285
11.5	0.0	0.0	0.0	0.0	0.3	3.3	13.3	26.6	31.4	24.7	14.2	6.4	2.4	0.7	0.2	0.1	124
12.5	0.0	0.0	0.0	0.0	0.1	1.0	4.4	9.9	12.8	11.0	6.8	3.3	1.3	0.4	0.1	0.0	51
13.5	0.0	0.0	0.0	0.0	0.0	0.3	1.4	3.5	5.0	4.6	3.1	1.6	0.7	0.2	0.1	0.0	21
14.5	0.0	0.0	0.0	0.0	0.0	0.1	0.4	1.2	1.8	1.8	1.3	0.7	0.3	0.1	0.0	0.0	8
15.5	0.0	0.0	0.0	0.0	0.0	0.0	0.1	0.4	0.6	0.7	0.5	0.3	0.1	0.1	0.0	0.0	3
16.5	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.1	0.2	0.2	0.2	0.1	0.1	0.0	0.0	0.0	1
Sum	1	165	2091	9280	19922	24879	20870	12898	6245	2479	837	247	66	16	3	1	100000

The H_s and T_z values are class midpoints.

Figure 9-1 Scatter diagram for the North Atlantic. Available from

Offshore standard DNV-RP-C205

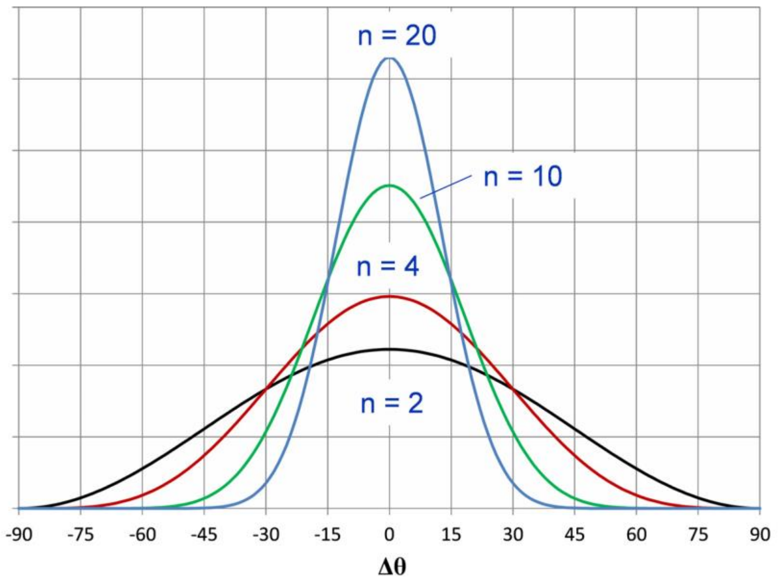

Figure 9-2 Wave spreading functions for different values of the cosine power n. Available from Offshore standard DNVGL-RP-0001

9.2.1.1 Wave Direction Probability

This defines the probability of occurrence for each main wave direction specified in the hydrodynamic analysis. The data is required in order to calculate the contribution of each main wave direction to the gross fatigue damage. Following the standard, equal probabilities from all directions will be taken.

Previously, a number of wave approach headings were selected for analyses. Being the 360º range is divided into eight 45º, twelve 30ºor sixteen 22.5ºdirectional sectors. Since the sum of probabilities for all directions must be 1.0 the wave direction probability for each heading is equal to 0.125.

9.3 Effective Notch Stress Method

The objective of this method is to obtain the notch stress in the hotspot which will be detected after carry out the different screening analysis. By using the notch stress the usage factor of this point will be calculated.

The stress due to the weld is included in the S-N curve and the hot spot stress is derived by extrapolation of the structural stress to the weld toe as indicated in Figure 9-3. It is observed that the stress used as basis for such an extrapolation should be outside that affected by the weld notch, but close enough to pick up the stress due to local geometry.

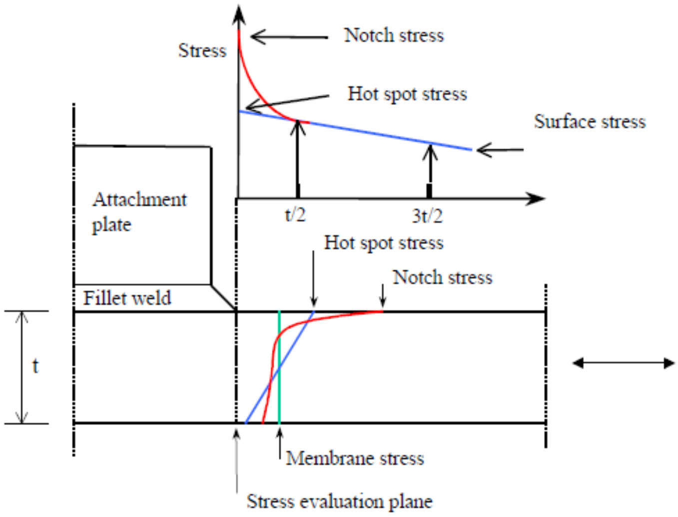

Figure 9-3 Schematic stress distribution at a hot spot. Available from Offshore strandardDNVGL-RP-0005

To take account of statistical nature and scatter of weld shape parameters, as well as of nonlinear material behaviour at the notch root, the real weld is replaced by an effective one.

The method is restricted to welded joints which are expected to fail from the weld toe or weld root. Other causes of fatigue failure, e.g. from surface roughness or embedded defects, are not covered. Also it is not applicable where considerable stress components parallel to the weld or parallel to the root gap exist.

The method is well suited for S-N classification of alternative geometries. Unless otherwise specified, flank angles of 30o for butt welds and 45° for fillet welds are suggested.
The method is limited to thicknesses t > 5 mm. For smaller wall thicknesses, the method has not been verified.

9.4 Improvement of Fatigue Life by Fabrication

In this section brief description of the possible measures to increase the F.L by fabrication, based on IIW Recommendations, are presented. Also the assumptions and limitations of this methods, which will restrict their applicability to this case of study, are numerated.

It should be noted that improvement of the weld toe will not improve the F.L if fatigue cracking from the root is the most likely failure mode. Therefore, the improvements presented in the following are for conditions where the root is not considered to be a critical initiation point.

9.4.1 Weld Toe Grinding

The primary aim of the grinding is to remove or reduce size of the weld toe flaws from which fatigue cracks propagate. At the same time, it aims to reduce the local stress concentration effect of the weld profile by smoothly blending the transition between the plate and the weld face.

Grinding is an efficient method for reliable fatigue life improvement after fabrication. Grinding also improves the reliability of inspection after fabrication and during service life. However, experience indicates that it may be a good design practice to exclude this factor at the design stage. The designer is advised to improve the details locally by other means, or to reduce the stress range through design and keep the possibility of fatigue life improvement as a reserve to allow for possible increase in fatigue loading during the design and fabrication process, see also DNV-OS-C101 Design of Steel Structures, section 6.

It is assumed that grinding is performed until all indications of defects are removed.

9.4.2 TIG Dressing

The aim of TIG dressing is to remove the weld toe flaws by re-melting the material at the weld toe. It also aims to reduce the local stress concentration effect of the local weld toe profile by providing a smooth transition between the plate and the weld face.

The geometrical changes of TIG-dressing can be quantified by defining the weld toe radius and weld toe angle (see Fig. 9-4). The main effect of TIG-dressing is the increase of the weld toe radius, which results in a lower stress concentration at the weld toe. The weld toe angle remains approximately the same for fillet welds, butt will be reduced for rather flat butt welds.

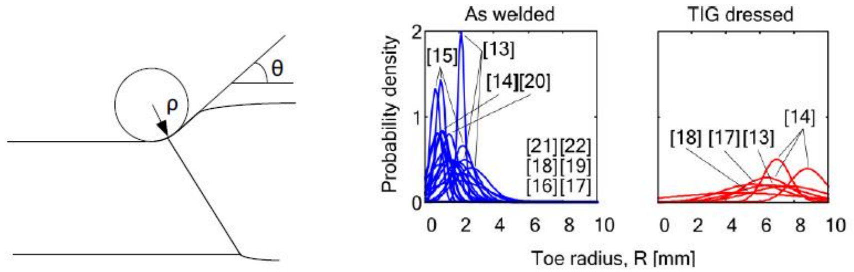

Figure 9-4 Definition of weld toe radius and weld toe angle (left), comparison of weld toe radii in as-welded and TIG-dressed conditions; the numbers between brackets refer to the sources of graphs in the original article (right). Available from [3].

9.4.3 Hammer Peening

In hammer peening method, compressive residual stresses are induced by repeatedly hammering the weld toe region with a blunt-nosed chisel. Compressive residual stresses are induced by mechanical plastic deformation of the weld toe region. Residual stresses then arise as a result of the constraint imposed by the surrounding elastic material.

This method is focus in improved the fatigue lives of welded joints that are most likely to fail from the weld toe. Another important limitation on the use of improvement techniques that rely on the presence of compressive residual stresses is that their fatigue lives are strongly dependent on applied mean stress. In particular, the benefit decreases as the maximum applied stress approaches tensile yield, disappearing altogether at stresses above yield. Thus, in general, the techniques are not suitable for structures operating at applied stress ratios of more than 0.5 or maximum applied stresses above around 80% yield. Note that the occasional application of high stresses, in tension or compression, can also be detrimental in terms of relaxing the compressive residual stress.

Figure 9-5 The view of the butt welds in as-welded condition (left side sample) and after application of UP (right side sample). Available in http://www.sintes.ca/Ultrasonic-Impact-Treatment.php

10 RESULTS

This section presents the results from the case studied. The section is divided into three subsections, dedicated to describing the methodology and results from the fatigue screening analysis, hotspot calculations and the effect of possible measures to increase the F.L by fabrication.

The results of the fatigue analysis will be presented in form of usage factor. Being the design life selected as 20 years.

10.1 Screening Analysis

Based on the effective notch stress method, several fatigue screening analysis were performed. In this section the description and results obtained of these analysis will be presented.

10.1.1 Screening Analysis of the Global Model

Based on the standard DNV-OS-C106, there are some typical fatigue critical areas for TLP´s which are:

- Hull and deck connections
- Collision ring area
- Hull and deck and stiffener connections at location of peak wave induced global bending moments
- Hull and mooring system interface
- Hard tank area
- Column and brace connections
- Riser frame/support and hull connections
- Openings in main hull
- Hard tank and truss connections
- Tubular joints
- Column to deck connections
- column to pontoon connections

The structure present a symmetry respect the axis X and Y. Also the wave direction probability is equal for all directions. Therefore the fatigue life for each quarter will be the same. For this reason and also due to the software limitations related to the model size (commented in the previous sections), just one quarter of the global model will be submitted to a fatigue screening analysis.

In a fatigue analysis based in the effective Notch Stress method, it is necessary to associate an S-N curve for each FE. The selection of one or another S–N curve is not so relevant in this point of the process, since the aim is to identify the areas where the fatigue life is lower. Looking for the maximum usage factors instead to the stress magnitude

In this screening analysis, the curve ”D” in Figure 4 -1 is selected considering structures exposed to seawater with cathodic protection, being applied for the entire model. This curve consider the effect of transverse butt welds, welded from both sides in flat position.

The results of this analysis are given in figure 10-1 and 10-2 were the usage factor is represented:

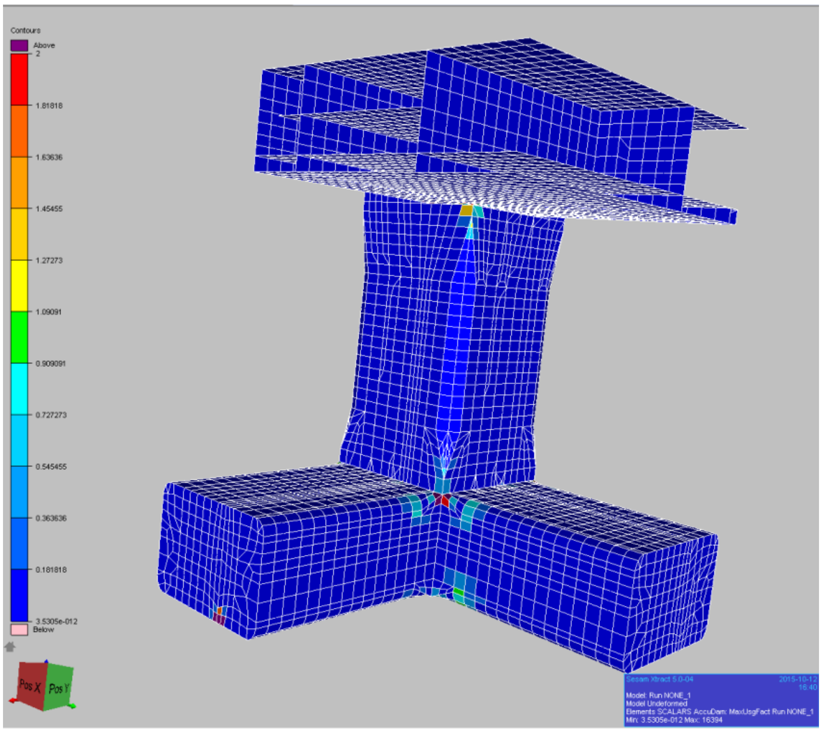

Figure 10-1 Global screening analysis: Usage factor

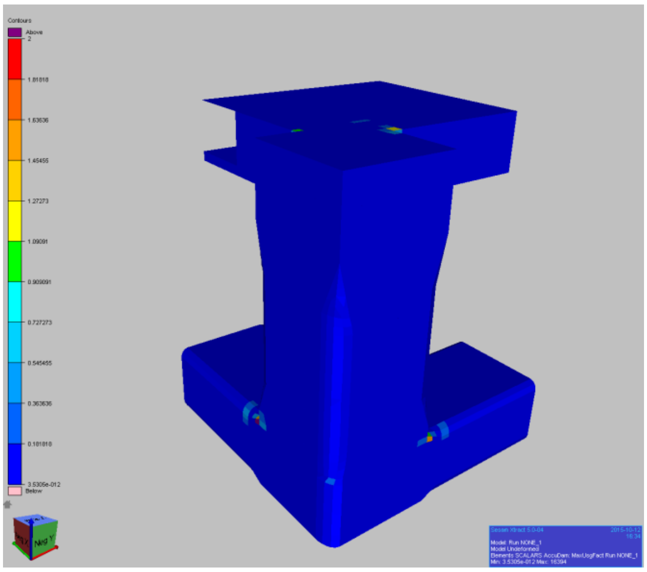

Figure 10-2 Global screening analysis: Usage factor

From Figures 10-1 and 10-2 is observed that the critical areas correspond to:

- hull and deck connections
- Column to deck connections
- Column to pontoon connections

This areas coincide with the ones mentioned by the standard, indicating that the results obtained are logical. All these areas should be verified nonetheless, this study will just consider the most critical area. Based in the results, this area correspond to:

- Column to pontoons connections

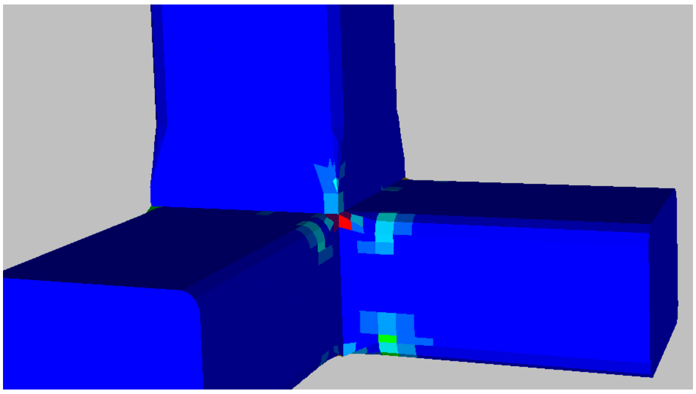

Figure 10-3 Location of the critical area under study

In order to apply the effective Notch Stress method, sub model and local model of this area are developed.

10.1.2 Screening Analysis of the Local Model

Subsequently to the screening procedure applied to the global model, an additional screening procedure is applied to a local model, where more structure details were considered. This procedure is aimed to identify the hot spots in the critical area delimited with the global model.

In the same way that for the global model, a screening analysis was applied to the local model. However the S-N curve associated to the different FE is selected carefully in this case, in function of the type of weld and material use for each plate. Hence the S-N curves in this analysis are DNV2010_CN-SEACP

The standards is recommended to use the C curve for cast nodes. This curve is assigned to the FE included in the cast steel node, which for part of the critical area. DNV2010_E-SEACP and DNV2010_F-SEACP for steel

This two curves, represented in Figure 4-1, correspond to welded joints with load carrying welds considering full penetration butt welded cruciform joint.

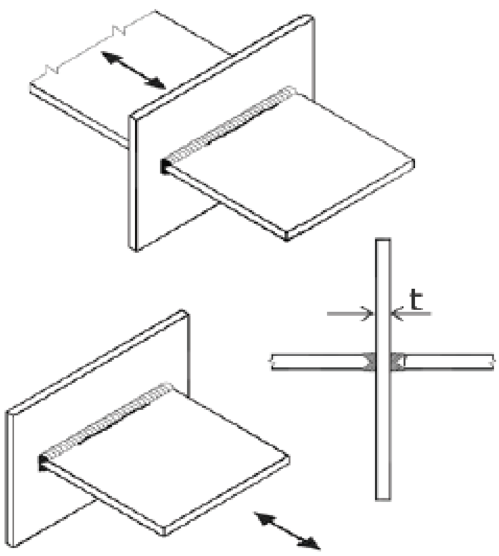

Figure 10-4 Structural detail considered in the case of study. Available from Offshore strandardDNVGL-RP-0005

This two curves are assigned to the steel plate in the area surrounding the cast steel node, which connect with the cast node. The selection of E or F curve depend on plate thickness, which will suffer modifications in order to quantify the effect of increment the thickness in the fatigue life.

The selection of these curves could derived in a conservative solution, in some areas where the weld joints will not be cruciform, thus being the solution in the safety side. More detailed explanation in the reference DNVGL-RP-0005.

The aim of this study is quantify the effect of some measures in the fatigue life of the structure. In this measures is included the increment in the thickness of the critical area. In order to achieve this aim, the thickness of all the critical area have been incremented millimetre by millimetre till being increased 9 millimetres respect the original thickness. This indicate the necessity of applied the local screening analysis for each increment in the thickness.

In the first local screening analysis, carried out for the original thickness, a hotspot is located (notice that the corner is not consider in the analysis, further explanation could be found in the section 6.7). This hotspot is situated in the vertical of the cruciform joint between cast node and the steel plate, for an elevation of 12.75 meters respect the base line, as represented in Figures 39, 40 and 41.

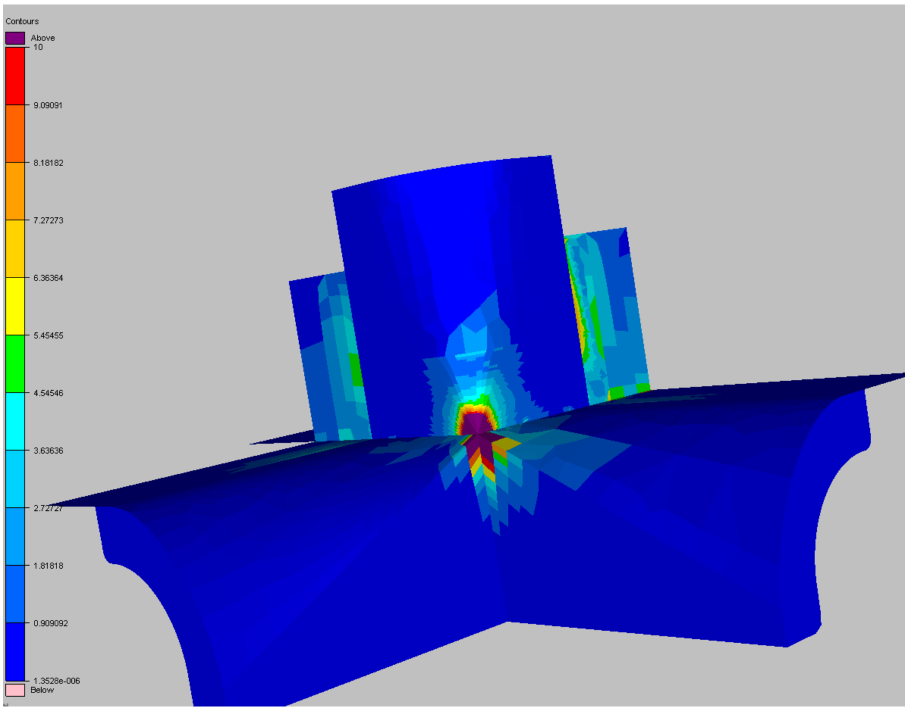

Figure 10-5 Screening analysis of the local model

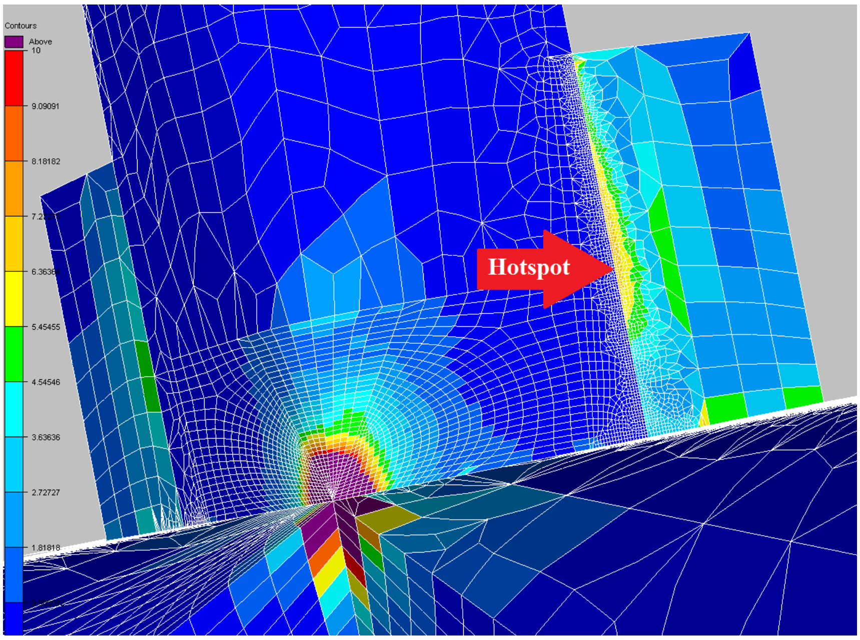

Figure 10-6 Hotspot location in the local model

In following analysis, the usage factor of the local model will suffer modifications as expected. This modification are motivated by the variation on the thickness and as consequence the new stress distribution.

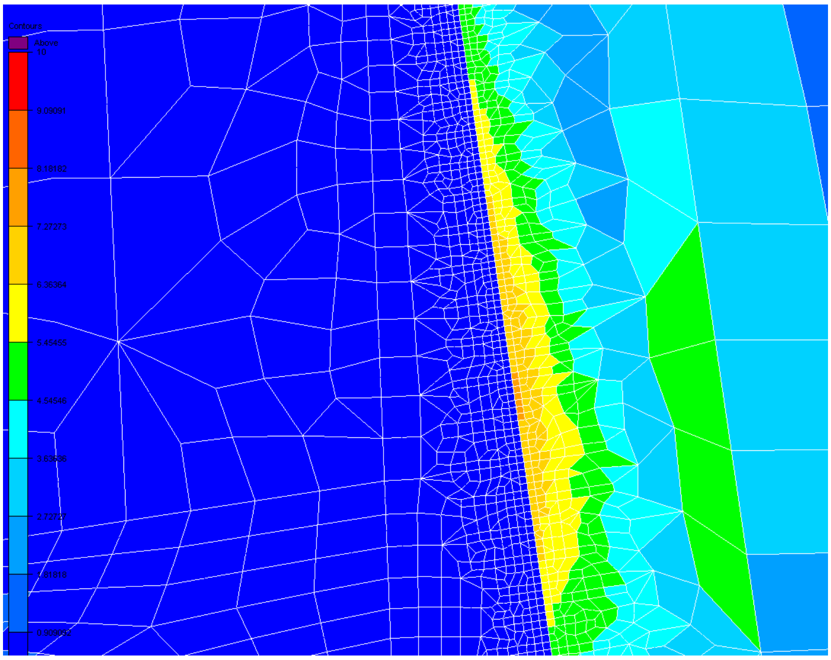

Figure 10-7 Detail of the hotspot

Once the hotspot is located, is time to move into the next step: The Hotspot fatigue analysis.

10.2 Hotspot Fatigue Analysis

In this section, the fatigue analysis of the hotspot and the results obtained will be presented. In the analysis, the notch effect due to the weld is included in the S-N curve and the hot spot stress is derived by extrapolation of the structural stress to the weld toe. The stress used as basis for such an extrapolation is outside that affected by the weld notch, but close enough to pick up the stress due to local geometry.

Since the element size used in the hot spot region is equal to t by t, being t the plate thickness, the stresses used for the extrapolation are evaluated as follows:

- The surface stress are evaluated at the corresponding mid-side points. Thus the stresses at the mid side nodes are used directly as stress at read out points 0.5 t and 1.5 t.

Stresses of these points are interpolated linearly to the hotspot and applied in the calculation of fatigue damage of the hotspot, see Figure 10-8. The stresses are interpolated component by component whereupon the principal stresses of the hotspot are calculated and applied in the fatigue calculation.

The hotspot stress σ_{hot} is expressed as:

$$\sigma_{hot} = \left(\sigma_{\frac{t}{2}} * L_{hot-\frac{3t}{2}} - \sigma_{\frac{3t}{2}} * L_{hot-\frac{t}{2}}\right) / L_{\frac{t}{2}-\frac{3t}{2}} \tag{23}$$

Where:
t: thickness of the plate, where the hotspot is situated
σ_{hot}: Stresses of hotspot
$\sigma_{t/2}$: Stresses of point t/2
σ_{hot}: Stresses of point 3t/2
L: distance in m

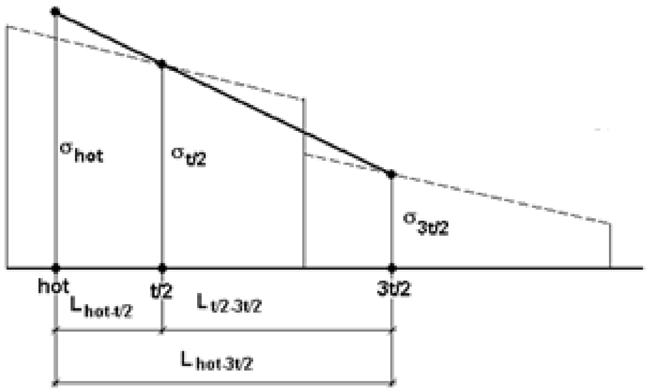

Figure 10-8 Interpolation of stresses to the hotspot. Available from Stofat user manual

The hotspot fatigue calculation has the same input parameter tat the previous screening analysis, which are described in the section 9.2.

Finally the notch stress is used to perform the analysis, being the results represented in form of usage factor in next Table and graph's.

Table 10-1 Usage factors in the hotspot for the different thickness

Thickness	Usage factor
Original	2.83
+ 1 mm	2.52
+2 mm	2.18
+ 3 mm	2.02
+ 4 mm	1.81
+ 5 mm	4.19
+ 6 mm	3.94
+ 7 mm	3.72
+ 8 mm	3.51
+ 9 mm	3.32

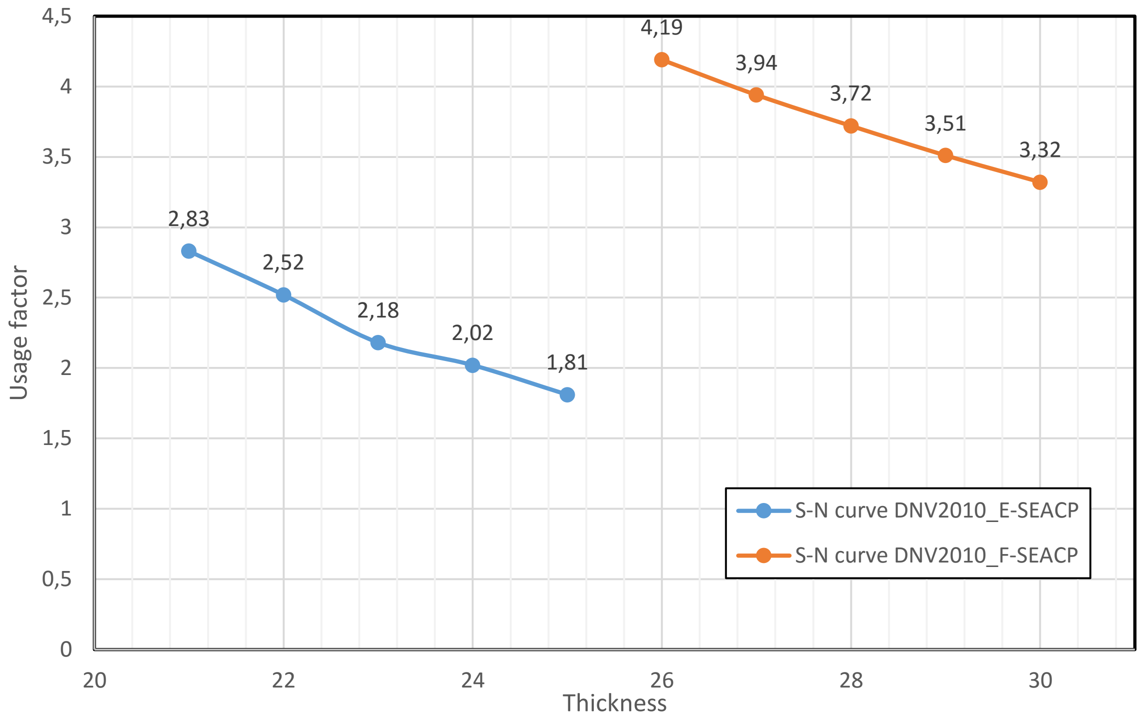

Figure 10-9 Evolution of the usage factor with the thickness´s increment

In addition, the increment in the fatigue life is presented for each modification of thickness:

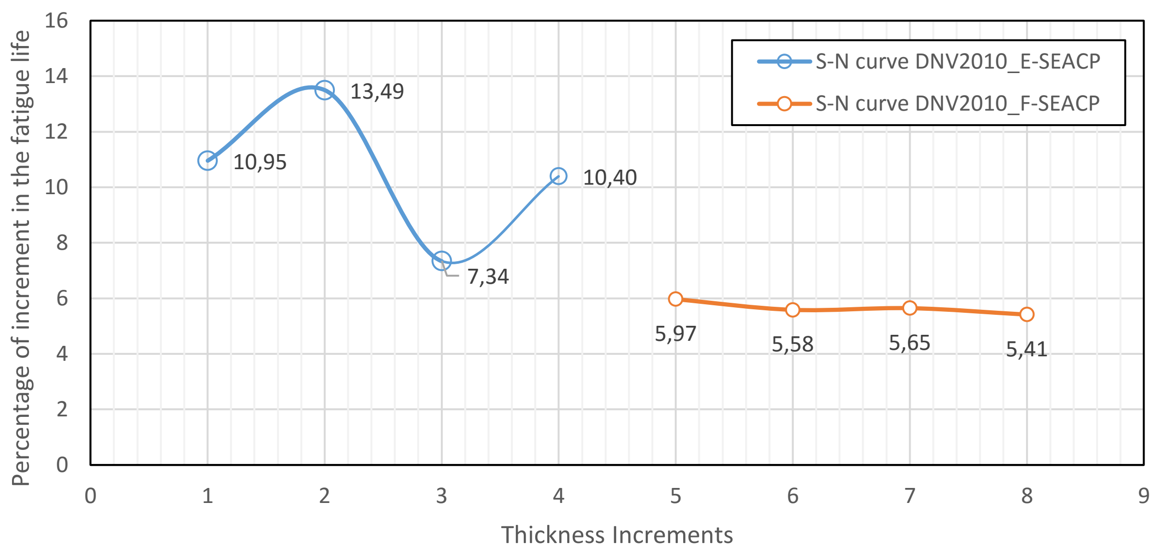

Figure 10-10 Increment in the fatigue life

As presented in the previous graph the F.L will be increased when the thickness of the plates raise. This increment decrease with the thickness, having an average of 10% in the first cases but being reduced to 5% on the last cases. This variation of the F.L improvement have its explanation of the thickness effect, which will influence the fatigue resistance.

The thickness effect is the phenomenon that the fatigue strength of a welded connection decreases when the thickness of load carrying plate increases. The thickness effect is observed in details where the fatigue crack initiation and propagation take place at the weld toe.

There are several reasons why an effect of plate thickness may appear in fatigue of welded joints. The three main reason are:

- Statistical, Örjasäter (1987) [5]

The probability of the initial defects refers to the dimension of the joint, so the larger volume structural component involves more imperfection and initial defects than thinner plate, which gives a weaker fatigue capacity for thicker plate.

- Residual stress

Because of more restrain in a thicker plate, the residual stress in large joint is higher than thinner one, which influences the fatigue life of welded structures.

- Geometric:

The radius of the weld toe does not depend on the wall thickness, but results in a relatively smaller radius for thicker joint components, Berge (1990) [6]. Therefore, for the thicker plate, the stress at weld toe is close to the surface stress, but for the thinner one, since its large stress gradient, the stress at weld toe drops dramatically from surface, which result a relatively small stress at the tip of the weld toe.

Gurney T.R. (1979) [7] demonstrated based on experimental evidence and fracture mechanics analyses that the effect of the plate thickness could be significant with the increased plate thickness of same geometry subjected to the same magnitude of stress.

In summary, the thickness will increment the F.L of the critical area but this increment will be limited by the counter-effect previously described.

10.3 Improvement on Fatigue Life in the Hotspot by Different Methods

In this section the effects of different methods to improve the F.L in the hotspot are presented. These methods were introduced in the section 9.4, being:

- Grinding
- TIG dressing
- Hammer peening

Different ways to represent the effect of these methods could be found in the bibliography, two possibilities are offered:

- Modified the use S-N curves with a more correct slope that represents the improved details. An example of such S-N curves is shown in Figure 44.

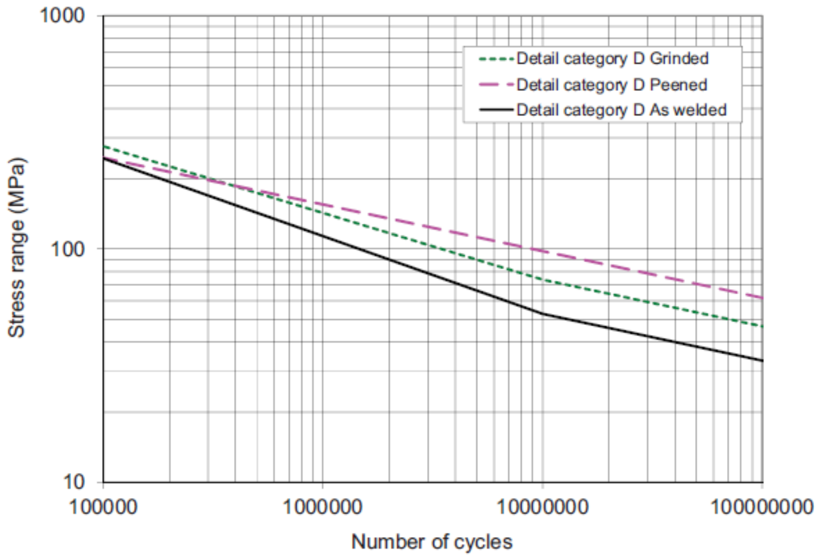

Figure 10-11 Example of S-N curves (D-curve) for a butt weld in as welded condition and improved by grinding or hammer peening. Available from Offshore strandardDNVGL-RP-0005

The S-N curves for improvement are in line with the recommendations from IIW for increased stress ranges at 2·106 cycles (increase in stress range by a factor 1.3 for grinding and a factor 1.5 for hammer peening).

- Applied an improvement factor base in Table 20, proportionated by the standard DNVGL-RP-0005.

Table 10-2 Improvement on fatigue life by different methods

Improvement method	Minimum sprecified yield strenght	Increase in fatigue life (factor on life)
Grinding	Less than 350 MPa	0.01 * yield strenght
	Higher than 350 MPa	3.5
TIG dressing	Less than 350 MPa	0.01 * yield strenght
	Higher than 350 MPa	3.5
Hammer peening	Less than 350 MPa	0.011 * yield strenght
	Higher than 350 MPa	4.0

Where the factor on fatigue life after improvement in this table is based on a typical long term stress range distribution that corresponds to wave environment for a service life of 20 years or more.

From these two possibilities to quantify the improvement, the second option is select. Knowing the material of the hotspot, NV/AH40 steel plate (yield strength equal to 390MPa), the final usage factors and fatigue life after applied the improvement methods are:

Table 10-3 Usage Factors for Hotspot.

HOTSPOT				
	As welded	Grinding	TIG dressing	Hammer peening
Thickness	Usage factor	Usage factor	Usage factor	Usage factor
21	2.83	1.42	0.81	0.71
22	2.52	1.26	0.72	0.63
23	2.18	1.09	0.62	0.55
24	2.02	1.01	0.58	0.51
25	1.81	0.91	0.52	0.45
26	4.19	2.10	1.20	1.05
27	3.94	1.97	1.13	0.99
28	3.72	1.86	1.06	0.93
29	3.51	1.76	1.00	0.88
30	3.32	1.66	0.95	0.83

Table 10-4 F.L for Hotspot.

HOTSPOT				
	As welded	Grinding	TIG dressing	Hammer peening
Thickness	F.L (years)	F.L (years)	F.L (years)	F.L (years)
21	7.07	14.13	24.73	28.27
22	7.94	27.78	27.78	31.75
23	9.17	32.11	32.11	36.70
24	9.90	34.65	34.65	39.60
25	11.05	38.67	38.67	44.20
26	4.77	16.71	16.71	19.09
27	5.08	17.77	17.77	20.30
28	5.38	18.82	18.82	21.51
29	5.70	19.94	19.94	22.79
30	6.02	21.08	21.08	24.10

Great increments are achieved, being the fatigue life of the hotspot extend even 4 times. However there are some recommendations, given by DNV GL, which offer a conservative vision of these improvement:

- Due to uncertainties regarding workmanship and quality assurance of the process, the Hammer peening method may not be recommendable for general use at the design stage.
- The maximum improvement factor from grinding only should then be limited to a factor 2 on fatigue life.
- Except for grinding method, the effect from different improvement methods as given are not consigned.
- The experience indicates that it may be a good design practice to exclude this factor at the design stage. The designer is advised to improve the details locally by other means, or to reduce the stress range through design and keep the possibility of fatigue life improvement as a reserve to allow for possible increase in fatigue loading during the design and fabrication process, see also DNV-OS-C101 Design of Steel Structures, section 6.

In short, this recommendations will lead to a conservative solution, where just the grinding method should be consider.

11 CONCLUSIONS

The purpose of this thesis is to study the F.L of a TLP when it is submitted to several structural modifications. Two main objectives are set: First, quantify the effect of these structural modifications in the F.L to guarantee a good fatigue design, without over-dimension certain structural elements. Then the second part focuses in proportionated to the design offices alternative solutions, based on the previous structural modifications, which could reduce the number of design iterations necessaries to achieve an optimal fatigue design.

1. Quantify the effect of the structural modifications in the F.L

The structural modifications studied are the thickness increment of the critical area and the improvement of F.L by fabrication. Assuming that the main source of loads coming from the waves and therefore performing the fatigue analysis in the wave frequency range these measures were evaluated.

For the first structural modification: Increase the thickness of the critical area. The results showed that:

- In the hotspot located, which correspond to transverse butt welds (welded from both sides), a minimum increment of 7.34% is experimented for thickness till 25 mm, being this minimum increment reduced to 5.41% for thickness greater than 25 mm.
- The F.L will suffer an improvement with the increment of thickness, however this will depend on the original thickness of the area and the classification of the structural detail. There are several reasons to why an effect of plate thickness may appear in the fatigue study of welded joints, being the three main: the residual stress, geometry and statistics.

In short, increasing the thickness a new stress distribution and therefore a new F.L will be achieved. The increment of thickness could be contemplated as a solution in cases were the TLP design is close to reach the design fatigue life, expecting increments of the 5 % range. This solution is limited to cases where the hotspot are located in structural categories equal or lower to the F category given by DNV GL. More detailed explanation in the reference DNVGL-RP-0005.

Respect the second measure to be quantified: Improvement of F.L by fabrication. The methods studied were: Grinding, TIG dressing and Hammer peening.

Meanwhile the case of study is evaluated considering the standards of DNV GL, the possible improvement will be limited to the grinding method. Being conservative the fatigue life will be just increase two times by this method. In addition, this increment will have an associated economical cost due to the additional treatment.

In resume, the use of these treatment could look like a fast solution to the design problems, but should be avoided except for punctual cases where the area of the hotspot could be easily submitted to grinding treatment.

2. Reduce the number of design iterations

This second objective is achieved as a consequence of the first. Once the effect of the thickness increment as well as the fabrication improvements techniques in the F.L are quantify for TLP structures, exposed to waters of the North Sea, it is possible that a design office could reduce the number of design iterations necessaries to achieve an optimal fatigue design, bases on the conclusions previously exposed.

The newness of this study is that offer to the designers of this type of structures, an alternative solutions of how to face possible problems as: critical points in the structure which exceed the limit usage factor and shortfall of fatigue life in a critical area. This information will be useful in matter that it will serve to reduce the engineering hours for the improvement of the design being studied.

12 FUTURE WORK

This thesis has done a fatigue analysis of a TLP, quantifying the effect of several structural modifications in the F.L. However, due to some objective reasons, there are still a lot of problems and tasks remain unsolved.

First, the results obtained in the fatigue analysis should be verified experimentally. The methodology used for this verification represent by itself a huge task to be face.

Regarding the structural modelling, in order to present a structure closer to the reality:

- Tendon system should be included, performing afterwards a fatigue analysis.
- Boundary conditions should be adjusted to reduce the possible reactions in the support´s areas.

To increase the F.L there are different proposals which can be found in previous research or in the different standards given by the classification societies. In this master thesis just few of them were quantified, nevertheless will be particularly interesting the evaluation of the following:

- Modification of the design: Increment the radius of the brackets situated into the critical areas.

Concerning the Hydrodynamic analysis, the study was performed in the frequency domain with linear wave theory approach, covering the range of the wave frequencies. However, there are two nonlinear behaviours of the TLP which should be studied: springing and ringing. These phenomena's occur when the Structural natural frequencies are several times higher than the dominant wave frequencies. Being defined Springing as a High frequency non-linear resonant response induced by cyclic (steady state) loads in low to moderate sea states and Ringing as the non-linear high frequency resonant response induced by transient loads from high, steep waves.

In order to evaluated the springing and ringing:

- High frequency analysis should be performed, which involve the performance of: dynamic analysis of the structural response for the global and the local models.
- Non-linear models to represent the wave forces should be used.

ACKNOWLEDGEMENTS

I would like to thank DNVGL to bring me the opportunity of develop this master thesis. My grateful thanks are also extended to various people; to Prof. Maciej Taczala to review this thesis; to Prof. Philippe Rigo and Zbigniew Sekulski who have been always supporting us in every possible way; to all the professors and lecturers who taught and encouraged me; to all staffs in ULG, ECN and URO involved in EMSHIP Erasmus programme.

I would like to delicate a special thanks to Mr. Maciej Jasinski, my mentor during the internship, for the constructive discussions and valuable insights in offshore works. His willingness to answer all my questions has been very much appreciated. I also want to show my gratitude to Mr. Tomasz Msciwujewski and Mr. Grzegorz Malinowski, from DNVGL, for their supports during my internship.

My gratitude for all classmates, who now I can call friends, for have been supporting me during the EMSHIP program.

Finally and most importantly, I would like to express my gratitude to my family and Marion, for the unconditional love that led me all the way to here. All will become meaningless without their supports.

This thesis was developed in the frame of the European Master Course in “Integrated Advanced Ship Design” named “EMSHIP” for “European Education in Advanced Ship Design”, Ref.: 159652-1-2009-1-BE-ERA MUNDUS-EMMC.

REFERENCES

[1] Prof. P. Schaumann, C. Böker, ForWind, 2007. Influence of Wave Spreading in Short-term Sea States on the Fatigue of Offshore Support Structures at the Example of the FINO1-Research Platform, (pp. 53-55).

[2] S.H.J. van Esa*, M.H. Kolsteina, R.J.M. Pijpersb, F.S.K. Bijlaarda, 2013. TIG-dressing of high strength steel butt welded connections – Part 1: weld toe geometry and local hardness.

[3] SEYUL SON, B.S, 2006. Design of ocean platforms against ringing response.

[4] Guanying Li, Yidong Wu, 2010. A study of the thickness effect in fatigue design using the hot spot stress method. Civil and Environmental Engineering. Chalmers University of technology.

[5] Örjasäter O., Dragen A., Haagensen P.J. Lian B. and Gunleiksrud A.1987. Effect of Plate Thickness on Fatigue Properties of Low Carbon-Micro-Alloyed Steel. Paper TS 6, Developments in Marine Technologu 3, Proc.3rd Int Conf on Steel in Marine Struct, SIMS'87, Delft, the Netherlands, 15-18 June, (pp. 315-335).

[6] Berge, S. 1990. The Plate Thickness in Fatigue Design - Again, Proc. Ninth Int. Conf. on Offshore Mechanics and Arctic Engineering. Houston. Texas, February 18-23, 1990, (pp185-190)

[7] Gurney T.R. 1979. The Influence of Thickness on the Fatigue Strength of Welded Joints. Boss'97, Imperial College, London.

[8] Gunther Clauss, Eike Lehmann, Carsten Ostergaard, 1992. Offshore Structures Volume I • Conceptual Design and Hydromechanics. Ed. Springer-Verlag.

[9] O. Faltinsen, 1993. Sea Loads on Ships and Offshore Structures. Part of Cambridge Ocean Technology Series.

[10] Pierre Ferrant. Lecture Notes on "Ship resistance under potential flow theory". Ecole Centrale de Nantes, 2014.

[11] Amirouche AMRANE. Lecture Notes on "Fatigue assessment of ship structures". Université de Liège, 2014.

[12] Det Norske Veritas, 2014. Offshore standard DNV-OS-C101: Design of Offshore Steel Structures, General (LRFD Method).

[13] Det Norske Veritas, 2014. Recommended practice DNV-RP-C102: Structural design of offshore ships.

[14] Det Norske Veritas, 2014. Recommended practice DNV-RP-C205: Environmental Conditions and Environmental Loads.

[15] Det Norske Veritas, 2014. Recommended practice DNV-RP-F205: Global performance analysis of deepwater floating structures.

[16] Det Norske Veritas, 2014. Classification notes No. 30.7: Fatigue Assessment of Ship Structures.

[17] Det Norske Veritas - Germanischer Lloyd (DNV GL) Group, 2014. Recommended practice DNVGL-RP-0001: Probabilistic methods for planning of inspection for fatigue cracks in offshore structures.

[18] Det Norske Veritas - Germanischer Lloyd (DNV GL) Group, 2014. Recommended practice DNVGL-RP-0005: Fatigue design of offshore steel structures.

[19] Det Norske Veritas, 2014. Offshore standard DNV-OS-C103: Structural Design of Column Stabilised Units (LRFD Method).

[20] Det Norske Veritas, 2014. Recommended practice DNV-RP-C103: Column-Stabilised Units.

[21] Det Norske Veritas - Germanischer Lloyd (DNV GL) Group, 2014. Offshore standard DNVGL-OS-C105: Structural design of TLPs - LRFD method.

[22] Det Norske Veritas, 2014. Recommended practice DNV-OS-C106: Structural Design of Deep Draught Floating Units (LRFD Method).

[23] Niraj Kumar Singh, 2013. Global Response Analysis for Semisubmersible Offshore Platform. Advanced Master in Naval Architecture” and "Master of Sciences in Applied Mechanics, specialization in Hydrodynamics, Energetics and Propulsion”. West Pomeranian University of Technology, Szczecin.

[24] Massachusetts Institute of Technology, 2011. Design of Ocean Systems, Moorings Dynamics.