FCTUC DEPARTAMENTO DE ENGENHARIA CIVIL
FACULDADE DE CIÊNCIAS E TECNOLOGIA
UNIVERSIDADE DE COIMBRA

Steel framed structures for subsea applications: structural criteria and analysis methodologies

Dissertação apresentada para a obtenção do grau de Mestre em Engenharia Civil na Especialidade de Mecânica Estrutural

Autor

Francisco da Silva Arede

Orientadores

Prof. Doutor Luís Alberto Proença Simões da Silva

Prof. Doutor Per Thomas Moe

Coimbra, Julho, 2015

ACKNOWLEDGEMENTS

Firstly I would like to thank both my supervisors Prof. Dr. Luís Alberto Proença Simões da Silva and Prof. Dr. Per Thomas Moe, for their time, feedback and supervision of my work.

I show my gratitude to FMC Technologies Kongsberg, specially the Structural Well Integrity department, headed by Anders Mogensen, for providing me with this opportunity.

I thank Torulf Hagner, for his expertise, patience during our discussions, and suggestions.

Lastly, I also thank my family, father, mother and brother, who were always present.

RESUMO

Na indústria energética *offshore*, vários tipos de componentes estruturais são necessários. Estes componentes são normalmente produzidos *onshore*, pelo que é necessário fazer o seu transporte para o local de produção, *offshore*. Esta tese, ir-se-á focar nos requisitos estruturais e metodologias de análise apropriadas para o dimensionamento de estruturas metálicas que passam por esta fase de transporte, sendo posteriormente usadas em campos subaquáticos de extracção de hidrocarbonetos, fase de utilização. Para tal, vai-se proceder à comparação dos critérios estruturais de três normas: DNV 2.7-3, respeitante ao transporte de *Portable Offshore Units*; ISO 13628-7 referente à fase de *completion* e *workover riser equipment*; e Eurocódigo 3, destinado à generalidade das estruturas metálicas usadas na construção civil.

De modo a proceder à comparação entre os diferentes critérios estruturais e metodologias de análise, vai-se recorrer a modelos numéricos de elementos finitos, permitindo obter a carga limite de acordo com cada norma. Este estudo também irá incidir sobre vários métodos de modelação estrutural usando *software* de elementos finitos, considerando diferentes tipos de elementos, curvas de material e os tipos de análise.

Três ligações em T constituídas por duas barras de secção circular oca, CHS, vão ser analisadas. Cada ligação tem uma classe de secção transversal diferente: classe 1, 3 e 4, de acordo com a classificação do Eurocódigo 3.

Após uma comparação detalhada dos resultados obtidos, observações e comparações vão ser realizadas relativamente aos critérios estruturais e às metodologias de modelação e análise. Orientações e directrizes para futuras análises vão ser fornecidas, visando a realização de análises mais eficazes para as estruturas em estudo.

ABSTRACT

In the offshore energy production business several types of units are required. As these units are manufactured onshore, they require transportation to the exploration site. Afterwards they are deployed and the production stage shall proceed. This thesis will focus on the structural criteria and analysis methodologies feasible of being used when designing steel framed structures which undergo these stages and are intended to be used in subsea oil and gas production fields. The structural criteria which will be compared are provided by: DNV 2.7-3, related to the transportation stage of Portable Offshore Units; ISO 13628-7, which concerns the production stage of completion/ workover riser equipment; and Eurocode 3, intended for steel structures as a whole.

In order to retrieve results, to perform the structural criteria comparison, finite element analysis will be used. This thesis also focuses on different modelling techniques available in finite element software, such as element type, stress strain relationship and analysis type.

Resembling part of a steel framed structure, three T shaped connections will be analysed, covering three classes according to EC 3 categorization: class 1, 3 and 4. The T connection is performed by two circular hollow section beams.

After a detailed comparison of results, observations shall be made concerning both the structural criteria provided by each standard and the finite element modelling techniques used. As a conclusion, guidelines will be provided for future analyses, aiming towards a more efficient analysis strategy for such structures.

CONTENTS

1. INTRODUCTION 1
1.1. Thesis background 1
1.1.1. The offshore industry 2
1.1.2. Subsea production systems 3
1.1.3. Operational stages and applicable standards for C/WO and PO units 12
1.1.4. Standards and Recommended Practices 13
1.1.5. Structural analysis through the Finite Element Method 14
1.2. Purpose 17
1.3. Organization of the thesis 17
2. OVERVIEW OF THE MAIN STANDARDS 19
2.1. DNV 2.7-3 19
2.1.1. PO Unit types 20
2.1.2. Operational classes 21
2.1.3. Design loads 21
2.1.4. Design criterion 23
2.2. ISO 13628-7 24
2.2.1. Design loads 24
2.2.2. Design criteria 25
2.2.3. Elastic analysis 25
2.2.4. Perfectly elastic plastic and plastic analysis 26
2.3. EC 3 27
2.4. Analysis methodology – FEA 29
2.4.1. DNV 2.7-3 29
2.4.2. ISO 13628-7 30

3. NUMERICAL MODELLING AND VALIDATION 31
3.1. Connection geometry 31
3.2. FEA parameters studied 33
3.2.1. Material models 33
3.2.2. Element types 36
3.2.3. Analysis types 37
3.2.4. Other modelling parameters 38
3.2.5. Finite Element models 38
3.3. Validation of the FE models 39
4. PARAMETRIC STUDY 42
4.1. DNV 2.7-3 criterion 42
4.1.1. Beam models 42
4.1.2. Shell models 44
4.1.3. Solid models 47
4.2. ISO 13628-7 criteria 48
4.2.1. Beam models 49
4.2.2. Shell models 49
4.2.3. Solid models 51
4.3. EC 3 criteria – hand calculation 52
4.3.1. Member capacity check 52
4.3.2. Connection capacity check 54
4.3.3. Chord face failure 55
4.3.4. Punching shear failure 56
5. COMPARATIVE ASSESSMENT 58
5.1. Comparison between modeling techniques 58
5.1.1. Bending and rotation capacities 58
5.1.2. 1st and 2nd order analysis 59
5.1.3. CPU cost 60
5.1.4. Other modelling parameters 61

5.2. Comparison between standards criteria 61
5.2.1. Comparison between possible DNV 2.7-3 analysis methodologies 61
5.2.2. Comparison between ISO 13628-7 analysis methodologies 63
5.2.3. Comparison between DNV 2.7-3, ISO 13628-7 and EC 3 64
5.2.4. Observations on analysis methodology according to DNV 2.7-3 65
6. CONCLUSIONS 67
6.1. Conclusions and comments 67
6.2. Further work 69
7. REFERENCES 70

1. INTRODUCTION

1.1. Thesis background

This thesis was written during an internship in FMC Technologies Kongsberg, in the Structural Well Integrity/Well Access Systems department. The Structural Well Integrity (SWI) department is responsible for performing analysis, through Finite Element Analysis (FEA), mainly on subsea structures used in oil and gas production systems.

Structural analysis, on steel framed structures, for subsea purposes is normally within SWI’s scope of work. These structural analyses are relatively complex, mainly due to different standards and different structural criteria that have to be satisfied, according to the different operational stages of such structures. Performing the design check on complex problems requires also complex tools, such as Finite Element software.

Besides knowing which structural criteria to apply, it is also of interest to better understand and compare the different analysis methodologies used when assessing the structural capacity of such structures. If the existing codes and standards do not provide a clear set of recommendations, lacking clarity, the analysis can lead to results that do not satisfy the safety factors required.

In order to make this problem more evident, this thesis will focus on the possible reasonable interpretations, according to the criteria provided by the standards used for analysis of steel framed structures with subsea purposes. With the objective of retrieving results for a comparison between the different interpretations, a T shaped connection will be studied through FEA. Since different modelling techniques are possible (different element types, material curves and analysis types) another comparative study will be made, concerning the influence that these different modelling techniques have in the structural capacity assessment.

1.1.1. The offshore industry

Large amounts of energy are consumed every day in modern society, making the energy industry one of the most important, as we are so dependent of it. Since society is evolving, and so is technology, energy demand is constantly increasing and new ways of producing and harvesting it are being developed and enhanced, such as the offshore energy industry. In this thesis, offshore production refers to the energy sector located beyond the shoreline.

The first offshore energy production system was introduced in 1890s. It was related with the oil and gas industry and consisted on a drilling field in shallow waters resorting to wooden piers connected to shore [1]. Since then, this activity has increased, both in quantity and in diversity. Nowadays there are several different branches in the offshore industry, such as oil and gas, wind, tidal, and wave energy harvesting. As depicted in Figure 1.1, there is an increase in the offshore oil and gas production when compared to onshore. When comparing shallow water with deep water production, it is noticed an increasing trend of oil and gas production in deeper waters, since reserves there are larger. Moving the production systems to deeper waters will lead to new challenges, such as high pressure and temperature (HPHT), followed by more complex engineering solutions.

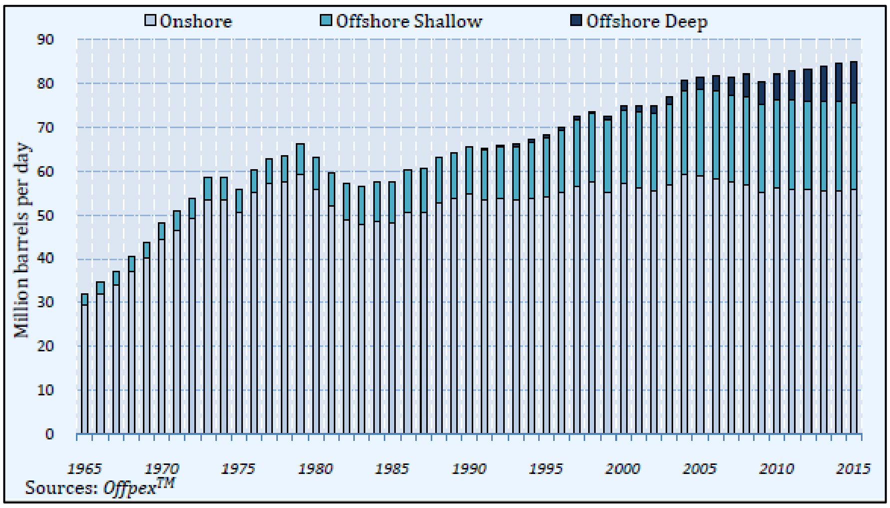


Figure 1.1 - Worldwide oil production (Infield Systems, 2012)

Due to the scale of this industry, the number of existent structural standards and recommended practice is extensive. Some of the relevant institutions that created these guidance documents containing provisions for offshore structures are: International Organization for Standardization (ISO); Det Norske Veritas (DNV); American Petroleum

Institute (API), amongst others. Therefore, there are several standards and recommended practices suitable to be applied to the same component or structure, throughout its lifecycle and operational stages. Having this in mind, it is relevant to perform a comparative study on the design implications on each of these, mainly due to possible differences in requirements, structural criteria, safety factors and analysis techniques.

1.1.2. Subsea production systems

To better understand some of the background context of this thesis, a brief overview of the subsea systems and equipment will be presented hereafter.

Subsea oil and gas is a branch of the offshore energy production field and has proven to be a profitable way for companies to extract hydrocarbon resources. Subsea systems, in some cases, are a more cost-efficient alternative to fixed platforms. The conjunction of a floating production vessel with a system of underwater equipment has proven to be a cost-efficient solution for the development of oil and gas fields. These are versatile solutions, allowing to be expanded if required. Subsea production systems are quite complex and come in many varieties, depending on the needs being addressed. Due to the complexity of these systems, a simple explanation of a subsea production system is a difficult task, but it can be said that the difference between these and a traditional offshore platform system is that, in the first one, due to engineering need (large depths, and adverse weather conditions) or due to economic reasons, the production system, or at least part of it, is bound to the seabed.

A subsea field can be categorized according to the water depth:

- Shallow-water, if the water depth is less than 200 m;
- Deepwater for water depths comprised between 200 m and 1500 m;
- Ultra-deepwater if the water depth is greater than 1500 m.

When planning a new subsea production system an economical study shall be performed, in order to assess which is more profitable, a tie-back or a stand-alone system:

- A tie-back system shall be chosen if there is any pre-existent subsea production field nearby. If the conditions are appropriate, connecting the new to the old system might lead to a reduction of the expenditure, since both systems may share the same platform infrastructures.
- A stand-alone system, as the name suggests is a fully independent production system, disregarding its surroundings.

During the design of a production field layout, there are some alternatives to consider regarding the disposition of the wells:

- Satellite systems: Normally used for small dimension production systems, where the reserves are smaller, requiring few wells. These consist in individual wells, normally connected to the main facilities through tie-back.
- Clustered systems: When the reserves are larger and more wells are required. Usually a clustered system is a more interesting option from an economical perspective. These consist in a group of the wells close to each other, thus being able to share some equipment, such as the manifold, and saving on other apparatus, mainly flow lines. In a clustered system it may be chosen to use a template, which is an appliance that mainly offers protection against impact on the wells. This equipment will also be able to assemble and provide housing to other equipment.

The most relevant equipment used in a subsea production system are:

- Subsea well and wellhead: The well is the hole that has to be drilled in order to reach the natural reserve. To prevent the well from collapse, due to the high pressures, a subsea wellhead is installed which can also be fitted with monitoring devices, Figure 1.2. The wellhead, as the name suggests, is located at the top of the well. Since the oil and gas reserves can be located far from the seabed, additional components are required to provide structural resistance, throughout the well's length. These components are called casings and will connect the wellhead to the reserves.

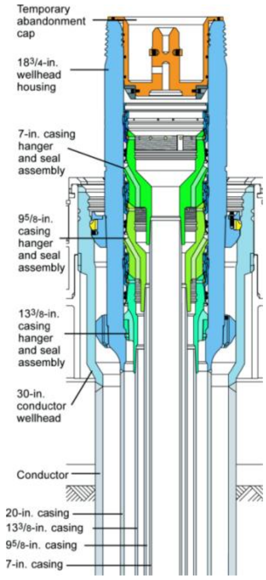


Figure 1.2 – Generic wellhead system (Petrowiki, 2015)

- Subsea production tree: It is an assembly of valves, pipes and fittings, which is placed on top of the wellhead, controlling and monitoring the flow of fluids, in and out of the well. There are two types of production trees or Christmas trees: vertical and horizontal, Figure 1.3. The main difference between these two is related to the valves' orientation.

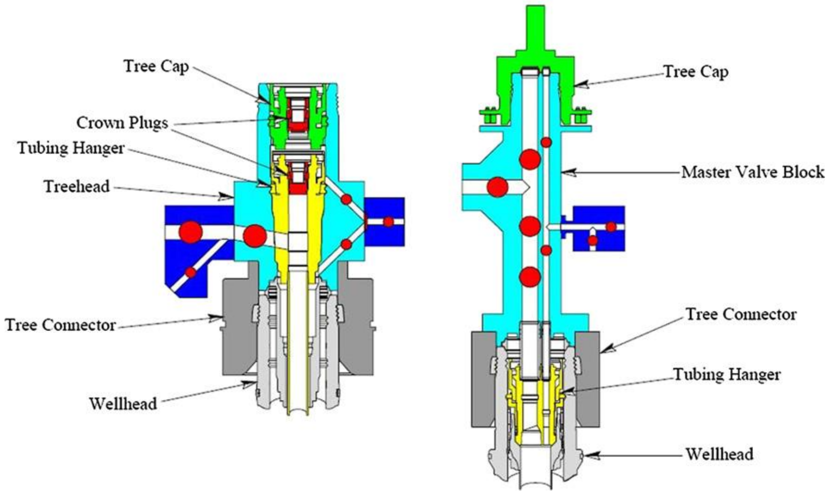


Figure 1.3 - Horizontal tree (on the left) and vertical tree (on the right) (Bai and Bai, 2010)

- Subsea manifold: This equipment is used to simplify the amount of flow lines required, increasing the efficiency of the subsea field. It consists of an arrangement of pipes and valves that collects and distributes the production fluids from the wells, Figure 1.4.

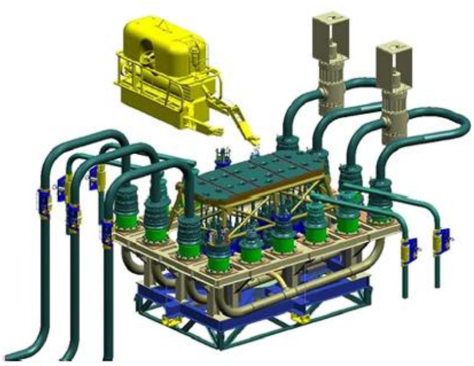


Figure 1.4 – Manifold (Bai and Bai, 2010)

- Template: Large framework structure that precisely spaces and protects the wells, thus simplifying the drilling and completion stages, Figure 1.5. A template is normally used when multiple wells are required, reducing the expenditure since the required equipment between wells is shared. Besides this advantage, it also serves as a support structure that can provide housing for other subsea equipment such as manifolds and flow lines.

Figure 1.5 – Template (FMCTI, 2015)

- Umbilical systems: Consist of an arrangement of pipes within an armoured shell that provides electrical power and/or control fluids (chemical or hydraulic) to the subsea equipment, Figure 1.6.

Figure 1.6 – Umbilical (Bai and Bai, 2010)

- Subsea pipelines: The main purpose of pipelines is to transport the production fluids to onshore or offshore facilities.

- Riser system: The riser system is a flow line that connects the well to the topside facilities. It consists of an assembly of small components. There are different types of risers, for each operation mode.

- Topside Structure: The topside structure comprises all the floating platforms and onshore facilities where the control panels, operators, storage tanks and supplies are located.

There is more equipment used in subsea fields that are not summarized in this work. Only the essential equipment was referred. Part of a subsea production field is presented in Figure 1.7.

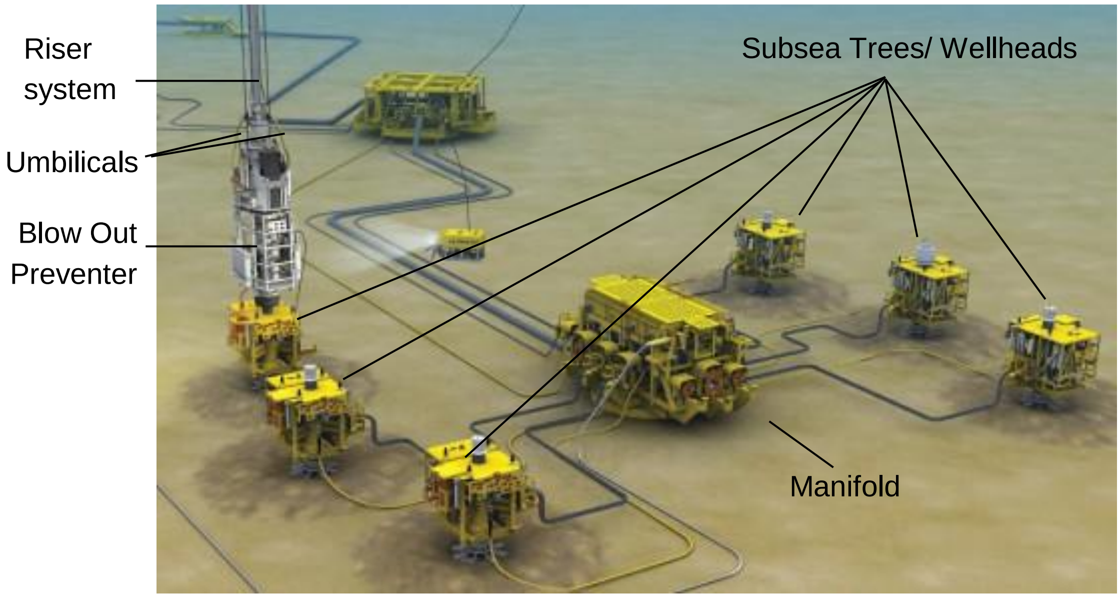


Figure 1.7 - Part of a subsea production system (adapted from Subsea World News, 2015)

During the lifetime of a subsea production field, there are several stages to consider, from the first survey, to the well abandonment. Firstly, geologists and geophysicists define the apparent extension of the hydrocarbon reservoir. After performing the proper analysis, if creating a well seems viable, further studies are performed, including prospect evaluation, where the geometry and volume of the reservoir is determined in a more detailed manner. After this step the layout of the subsea production starts to be designed, so it fits best to the reservoirs geometry. The next step is related with manufacturing the equipment, testing and implementing it in the production site. Afterwards, the field is able to start producing oil and gas. After a considerable amount of fluid being extracted from the reservoir, the production rate will start to decrease, due to the decrease of pressure in the reservoir. When this happens, the well undergoes a recovery step, where the pressure is increased and the well is stimulated. Throughout the well's life cycle, maintenance procedures are also required. The final step will be the abandonment of the well, either when it is depleted or when it stops being economically viable. For some of the mentioned steps, there are sub-steps, including: drilling, when the seabed is perforated; completion, when setting up the production field; and workover, when performing well maintenance.

Drilling:

In the drilling stage the seabed is perforated until the reservoir is reached. It is required a semi-submersible rig, a marine riser, with a drilling riser, and a Blow Out Preventer (BOP). The marine riser is the assembly of joints that connects the semi-submersible rig to the wellhead. The drilling riser is located inside the marine riser and is used to reach the reservoir through drilling. The BOP, as the name suggests, is a device required to take control of the well and prevent blowouts, if needed. In case of happening, a blowout can lead to serious environment and economic damages, as well as human losses. The BOP is designed to cope with the high pressure installed on the reservoir. During the drilling stage, the wellhead and the casings are also fitted. While the installation of the wellhead consists in fitting it onto the wellbore, the casing installation is more complex. As the well gets deeper, several casings have to be installed to ensure that the well does not burst. To prevent this, a gap between the rock and the casing is left. This gap is filled with cement that is pumped through the riser, creating a resistant tight connection. This process is repeated several times as the diameter of the casings being installed reduces, see Figure 1.8.

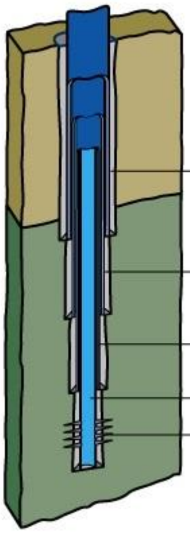


Figure 1.8 - Casing on a well (adapted from Rigzone, 2015)

Completion and workover:

Completion and workover operations take place at two different stages and can be performed with the same equipment. The completion stage consists in fitting and installing the equipment required for the extraction of hydrocarbons. The workover stage consists in any kind of invasive intervention in the well. As already referred, there are two possible production trees that can be used: vertical (VXT) and horizontal (HXT), which will be installed during the completion step. The completion procedure depends on which type of tree is used. If using a horizontal tree, then the tree is installed first, followed by the production casing and the tubing hanger (component that holds the production casing), resorting to a BOP or a Lower Riser Package (LRP) plus an Emergency Disconnect Package (EDP), Figure 1.9. The use of an EDP and a LRP is an alternative to the BOP that will also aim towards preventing blowouts. If a vertical tree is used, then this process is reversed. Firstly the casing hanger is installed followed by the production casing, also resorting to a BOP or LRP plus EDP. After that, the vertical Christmas tree is installed, Figure 1.10.

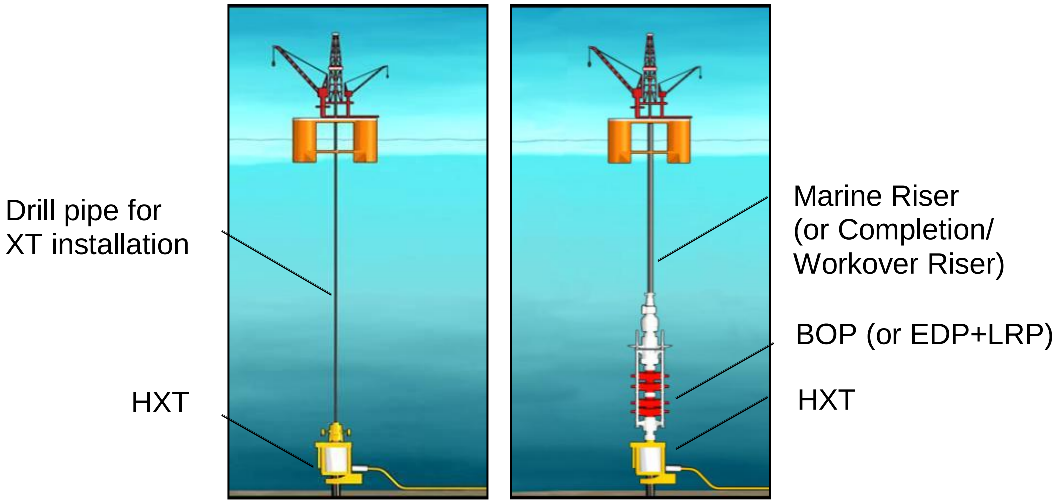


Figure 1.9 - Completion steps for a horizontal tree (adapted from FMCTI, 2015)

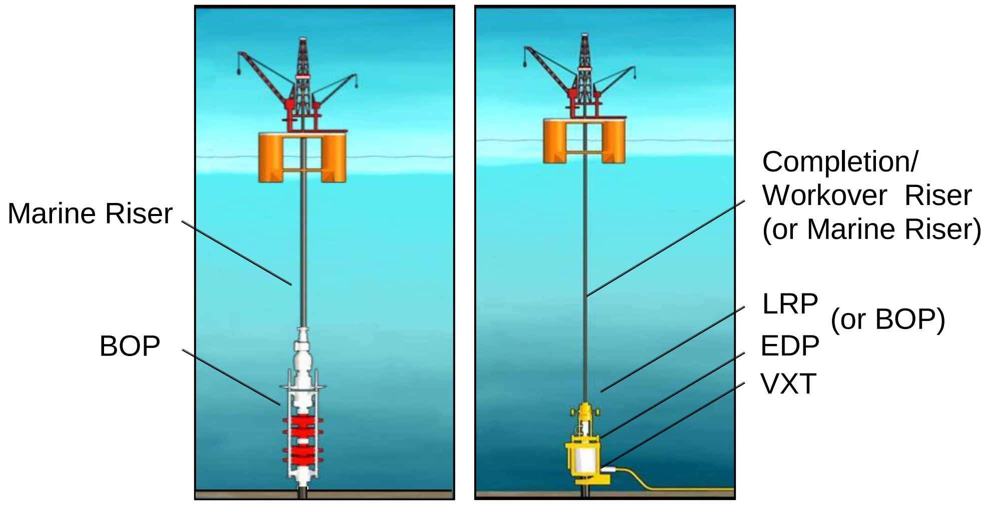


Figure 1.10 - Completion steps for a vertical tree (adapted from FMCTI, 2015)

As presented in the figures above, a Completion/Workover (C/WO) riser can be used to perform this step. This riser comprises several components, which can be seen in Figure 1.11.

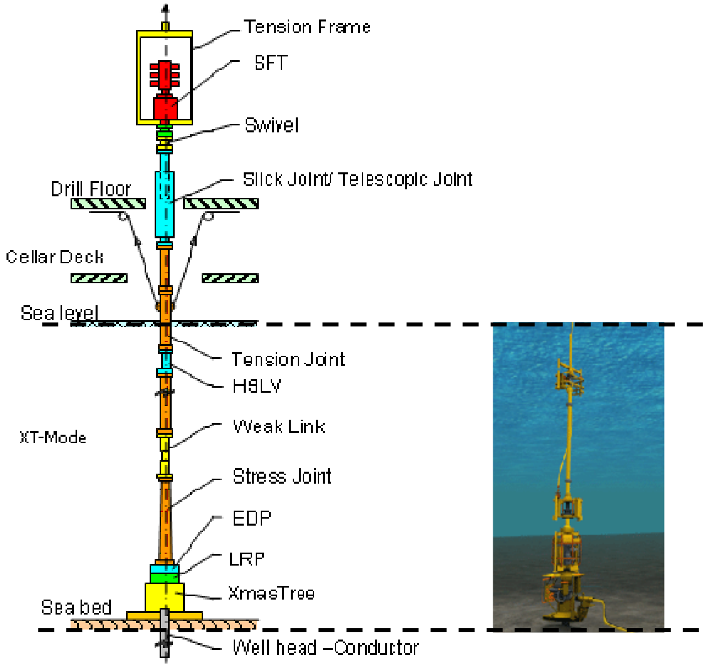


Figure 1.11 - Illustration of the components of a C/WO riser (adapted from FMCTI, 2015)

The EDP and LRP, Figure 1.12, are two steel framed structures located in the lowermost part of the C/WO riser. Their main purpose is also related with environmental hazard prevention. These components are attached together, and have common goals:

- The Emergency Disconnect Package main purpose is to separate the subsea tree from the riser in case of emergency, due to adverse weather conditions and/or vessel drift off. As a result of this separation feature, it is necessary a retainer valve, which is a mechanism that will cut the hydraulic flow prior to the disconnection of the EDP, preventing hydrocarbon leakage to the environment.
- The Lower Riser Package main purpose is to create a barrier between the well bore and the outer environment. This component also has the ability to shut down the well, acting similarly to a plug.

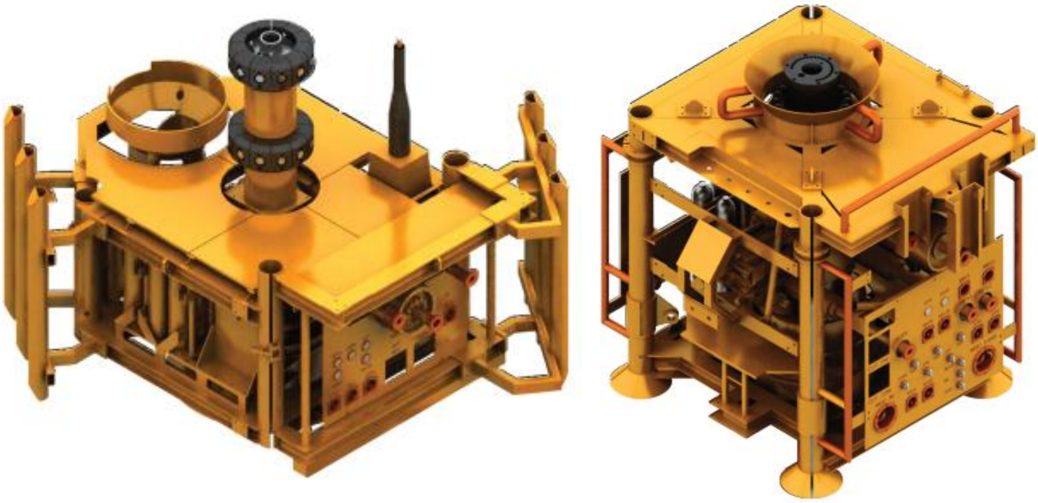


Figure 1.12 - EDP (on the left) and LRP (on the right) (FMCTI, 2015).

1.1.3. Operational stages and applicable standards for C/WO and PO units

Packages like the EDP and LRP are complex assemblies, consisting of interconnected components with high precision devices, and when transported from the manufacturing site (onshore) to the production site (offshore), and eventually down to the seabed (subsea), it is crucial to maintain the integrity of the structure and the safety of the personnel involved in these operations. To ensure this, a structural exoskeleton is required. These components undergo several transportation stages, such as lifting, sea fastening and subsea lifting. Lifting includes lifting from onshore to ship/floating vessel and from ship/floating vessel to semi-submersible rig/platform. Sea fastening consists in securing the cargo on deck during sea transportation. Subsea lifting comprehends lowering the device onto or from the seabed.

According to DNV 2.7-3, Portable Offshore Units (PO Units) are packages intended for offshore transportation that can also be designed for subsea lifting. Both the EDP and the LRP have to be checked according to DNV 2.7-3, regarding the transportation stage. Afterwards there is also an operation stage, which consists in using these units in the subsea production systems. For this step, ISO 13628-7 is applied, providing structural requirements for C/WO operations, where the LRP and EDP are included. The different operational stages and the applicable standards are summarized in Figure 1.13.

One of the main goals in this thesis is to perform a comparative study between standards, their structural requirements and analysis methods for this specific family of structural assemblies, categorized both as PO Units and as C/WO units.

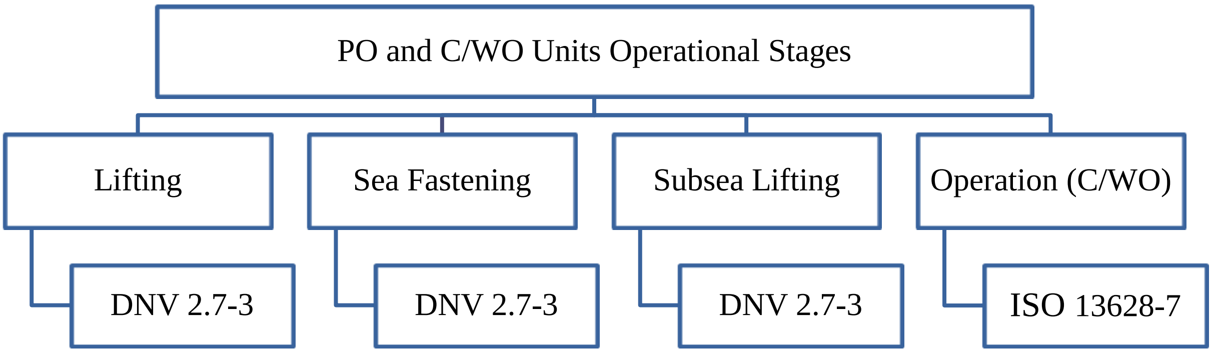


Figure 1.13 - Operational stages for PO and C/WO units

1.1.4. Standards and Recommended Practices

A study resorting to three standards will be made in order to assess the different structural criteria and requirements. The standards to be compared are DNV 2.7-3, ISO 13628-7 and EC 3:

- DNV 2.7-3 is a standard specific to PO Units, and covers requirements regarding design, manufacturing, testing and certification.
- ISO 13628-7 is a standard related with subsea operation of the oil and gas industry, covering completion and workover riser systems.
- Eurocode 3 is a standard related to steel construction, therefore does not provide specific requirements for offshore structures. It will be used in this study in order to have input on hand calculation procedures.

Due to almost non-existent studies regarding this matter, a comprehensive analysis shall be performed. As explained, the standards being compared apply to different operational stages, making a comparison between load formulae not possible. Hence, only the material resistance criteria will be compared, which is also more interesting because the main challenge in structural analysis lays in the interpretation of the possible analysis methodologies. In order to perform this analysis a numerical model must be created so results can be retrieved.

1.1.5. Structural analysis through the Finite Element Method

In order to perform this comparative study, FEA will be performed to determine the structural capacity of the components in analysis. The numerical models will be solved using ABAQUS CAE v6.13-3 software.

The Finite Element Method (FEM) has been subject of many studies and discussions due to its potential. It is considered to be one of the most important developments in the mathematical field, which was highlighted with the growing capabilities of computational technology, allowing to solve more complex problems. It is extensively used in different areas of interest, from the engineering field to the biomedical field. The FEM started to gain relevance around the 1960s [2]. Since then, many authors developed research on this subject. Two of the main authors are Zienkiewicz (Zienkiewicz et al, 2005a), (Zienkiewicz et al, 2005b), (Zienkiewicz and Taylor, 2005) with a vast amount of publications about FEM, and Bathe (Bathe and Wilson, 1976), (Bathe, 1982), (Chapelle and Bathe, 2011), (Bathe, 2014) with an extensive explanation on finite element procedures.

FEA is based on numerical simulations and it is a very powerful engineering tool that enables the study of a component's behaviour under specific loading. Although powerful, it only provides approximate results. Therefore, the results obtained through FEA, shall be validated with either hand calculation or test results, assuring the reliability of the numerical model.

FE software are complex tools, prompting the user with a vast array of options on how to model a component. One major concern on FEA is how to interpret results and acknowledge them as accurate or not, since they rely largely in the modelling stage. This proves the point that validating the model is an important step, since a wrongful assumption in the modelling stage can lead to inaccurate results.

Since the outcome relies on the modelling stage and that one same component can be modelled in different ways, another goal of this thesis will be to compare the outcomes of different modelling techniques, through a parametric study, in which the following parameters will be studied: element type, material stress strain relationship and analysis type (Figure 1.14).

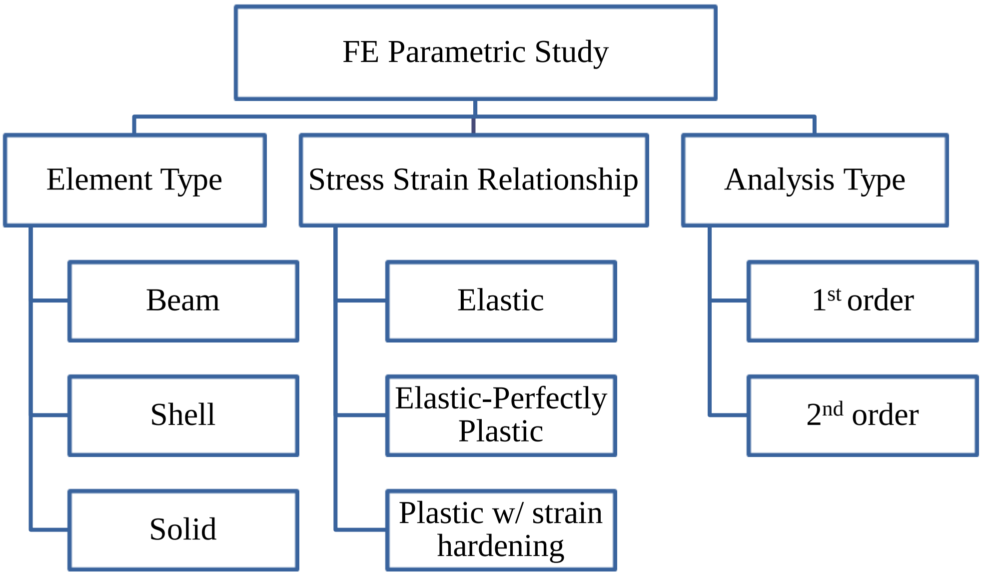


Figure 1.14 - FE parametric study

As the element type will be one of the parameters to be changed, the beam, shell and solid element types' fundamentals will be covered:

- The beam element type can only represent objects where one of the dimensions is considerably greater that the other two, resembling a bar. It is modelled in the FE software as the center line of the beam. With beam elements, plane sections normal to the beam center line will remain plane and normal to it throughout the analysis. It is the lightest element, meaning that it is the one that requires least CPU cost when computing the solution, because it leads to the least amount of mathematical equations. It also has some drawbacks, such as not being able to address local effects, where local buckling is included.
- The shell element type will more accurately represent elements which have one dimension much smaller than the other two. It is adequate for planar elements such as slabs, walls and thin-walled elements. This element is not as fast to compute as beam elements, but it is faster than solid elements.
- The solid element type is appropriate to be used in elements where none of the directions can be neglected in the analysis. It will more accurately represent block like objects or hollow elements that due to its thickness cannot be classified as thin walled. It is considered to be the element that will lead to the highest duration when performing analysis.

Other parameter that will also be studied is the stress strain relationship. The steel material curve can be divided into 3 regions: elastic, plastic, and strain hardening. There are several possible approaches when modelling the stress strain relationship. The ones studied in this thesis will be those that better fit the criteria provided by the studied standards:

- Elastic: a fully elastic behaviour;
- Perfectly elastic plastic: an elastic behaviour with a plastic plateau at a certain yield stress;
- Plastic with strain hardening: a material curve that tries to reproduce the real one.

As for the analysis type, a 1^{st} or 2^{nd} order analysis can be chosen. By performing a 1^{st} order analysis the stiffness matrix and geometry will remain constant throughout the analysis. When performing a 2^{nd} order analysis, the problem's matrix will be updated during the analysis, therefore the geometrical nonlinearity of the system will be considered. The 2^{nd} order analysis will represent more accurately the reality, accounting for local effects such as global and local buckling, but it also leads to an increase of analysis time.

Considering the advantages and setbacks of the presented parameters, a study will be performed, accounting for these different modelling techniques. In order to get a broader perspective of the results and to provide a more accurate comparison, three connection geometries will be studied. The major difference between them is related to the wall thickness, which was assigned in order to achieve cross sections that qualify as class 1, 3 and 4 according to Eurocode 3 categorization. It was chosen not to perform analysis on a class 2 section model, because the results should be similar to the class 1, and local effects, should be more evident in slenderer sections.

Another problematic that will also be assessed is the interpretation of the structural criteria provided by each of the mentioned standards. Some requirements provided by these standards are unclear and prone to personal interpretation. As an example DNV 2.7-3, does not state which stress strain relationship shall be adopted, nor if FEA shall be performed through a 1^{st} or 2^{nd} order analysis. Also concerning DNV 2.7-3, it only provides a structural criterion based on a stress limit. Since no material curve is advised, if one uses a perfectly elastic plastic material curve with yield plateau inferior to the stress limit provided, then this stress limit will never be reached. This leads to believe that this standard could provide more accurate guidelines. The aforementioned issue could be surpassed by imposing a local strain limit, for example. Other standards provide strain limits, such as ISO 13628-7 and EC 3 Part 1-6. Using DNV 2.7-3 criterion with the endorsement of a strain criterion will be studied further on.

1.2. Purpose

Summing up what has been described, this thesis will focus on:

- A comparative analysis between ISO 13628-7, DNV 2.7-3 and EC 3 structural requirements and criteria for the design of steel structures;
- Analysis and discussion on the different modelling techniques used when performing FEA;
- Highlighting the current problematic when performing analysis for these units, mainly due to the lack of guidelines on FEA.

This thesis will be supported by results obtained through FEA, on a generic T shaped connection, resembling a geometric detail of a PO Unit. The outcome of this thesis will provide better insight on the most efficient modelling techniques while performing structural analysis of such structures, having always as target optimizing the analysis time without compromising safety.

1.3. Organization of the thesis

The content of this project is divided into 6 chapters:

Chapter 1 – INTRODUCTION – presents the background of the thesis, where the problematic and the scope of work are explained. Also, the analysis methodology is introduced.

Chapter 2 – OVERVIEW OF THE MAIN STANDARDS – looks in detail to the provisions presented in DNV 2.7-3 and ISO 13628-7. Different analysis methodologies according to these standards are explained here. Also, EC 3 is introduced.

Chapter 3 – NUMERICAL MODELLING AND VALIDATION – describes the finite element models used, their geometry and the different modelling techniques chosen to simulate a simplified steel framed structure in ABAQUS CAE. The validation of the FE models is also accomplished in this chapter.

Chapter 4 – PARAMETRIC STUDY – presents the results and the post processed data regarding the FEA study.

Chapter 5 – COMPARATIVE ASSESSMENT– contains a comparison of the results, drawing conclusions on the different analysis methodologies used.

Chapter 6 – CONCLUSIONS – sums up the most relevant conclusions retrieved through this study, also providing guidelines for future studies on this subject.

2. OVERVIEW OF THE MAIN STANDARDS

This study is dedicated to three main standards. In order to better understand why these standards have been chosen, a schematic relation is presented, in Figure 2.1, providing better insight on their range of applicability.

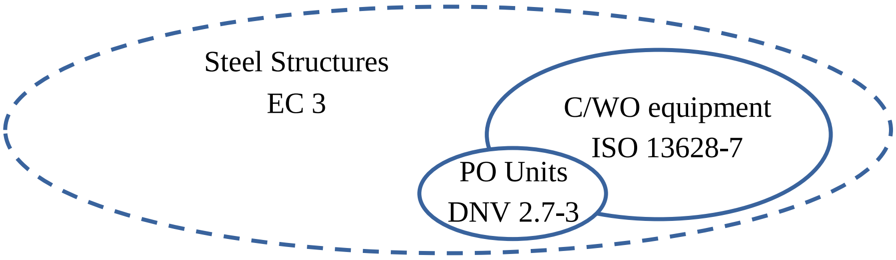


Figure 2.1 – Standards applicability

The studies conducted in order to assess the differences between ISO 13628-7, DNV 2.7-3 and EC 3 are almost non-existent. Thus a proper literature revision is challenging. This is rather understandable due to vast quantity of existent standards, but also because DNV 2.7-3 is a recent standard, being revised in 2011.

2.1. DNV 2.7-3

DNV GL is an international certification body addressing mainly maritime, oil and gas, and energy industries. DNV 2.7-3 is standard for certification that covers the requirements of all PO Units, other than portable offshore containers. A PO Unit is a package intended for offshore transportation and lifting. To ensure safety, this standard requires that PO Units:

- Shall be lifted individually by one crane;
- Are normally not designed to be lifted by a sling set including spreader bar;
- Can only be stacked if designed for such purpose;
- Shall be handled according to IMO's "Code of safe practice for supply vessels" or according to other special made transport procedure.

2.1.1. PO Unit types

In this standard there is a sub-classification according to the unit type, Figure 2.2.

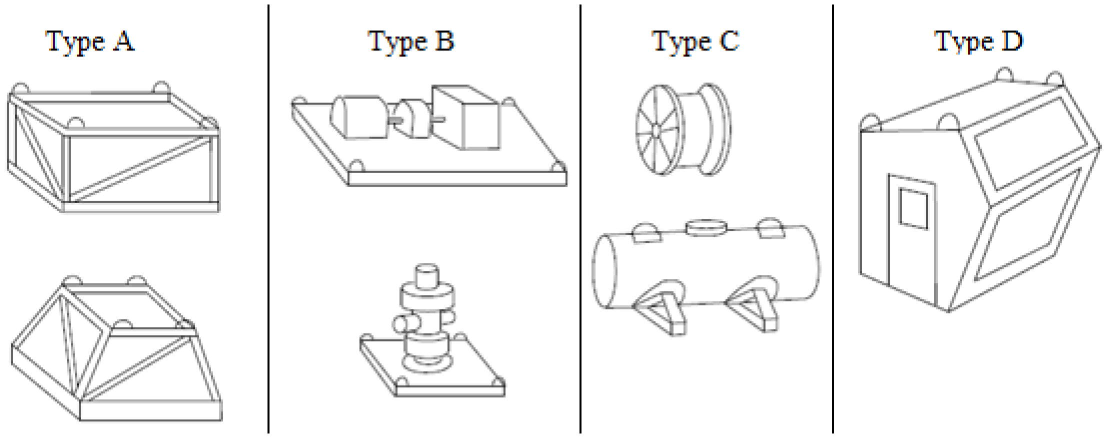


Figure 2.2 - PO unit types (adapted from DNV 2.7-3, 2011)

Type A: PO units with primary structure frame, including skids arranged with crash frame;

Type B: These units might have installations with the same functions as the ones mentioned for type A but without a primary structure frame;

Type C: PO Units that have self-supporting systems but that lack a dedicated skid or frame;

Type D: Mainly boxes or units of stress skin design where, in order to making them suitable for transportation, attachments are required.

Type E: Units that do not fall in either of the mentioned types, and are not PO containers.

These unit types might be used in subsea operations, but this standard only covers the transport and lifting certification. This standard also has other restriction: the units must weigh less than 100 tonnes.

2.1.2. Operational classes

Since the design of PO Units depends on their purpose (if intended for subsea use or not) and the weather conditions that have to withstand (mainly wave height and frequency), a division by class is required. The division provided by DNV 2.7-3 is based on Mass Gross Weight (MGW), unit type and risk evaluation.

According to the mentioned parameters, a PO Unit shall be categorized as R30, R45 or R60. In addition, a PO Unit shall also be noted as "subsea" if intended for subsea use and "SE" if designed for single lifting events. These classes will affect the maximum wave height that a vessel can be exposed to, when lifting these units.

2.1.3. Design loads

Lifting loads:

If the lifting is performed through slings, the lifting design load shall be calculated based on F, the greater of F_{Air} and F_{Sub}:

$$\mathrm{F_{Air}} = \mathrm{DF} \times \mathrm{MGW} \times \mathrm{g} \tag{1}$$

$$\mathrm{F_{Sub}} = 2.5 \times \mathrm{MGW} \times \mathrm{g} \tag{2}$$

The Design Factor (DF) is calculated according to the operational class and the MGW, Table 2.1.

Table 2.1 -DNV's design factor

Operational Class	MGW<50 tonnes	MGW>50 tonnes
R60	$1.4 + 0.8 \times \overline{50 \quad MGW}$	2.2
R45	$1.4 + 0.6 \times \overline{50 \quad MGW}$	2
R30	$1.4 + 0.4 \times \overline{50 \quad MGW}$	1.8

If the lifting is performed by a fork lift truck instead of slings, the design load shall be:

$$F_F = 1.65 \times MGW \times g \tag{3}$$

Impact loads:
The following values shall be used for horizontal impact on corners and bottom rails:

-R60&R45:

$$F_{HI} = 0.08 \times test\ load \tag{4}$$

- R30:

$$F_{HI} = 0.05 \times test\ load \tag{5}$$

For end or side structure and upper rails the load is:

$$F_{HIR} = 0.6 \times F_{HI} \tag{6}$$

The test load is given by Table 2.2:

Table 2.2 - Test load

MGW	Test Load
≤25	$\mathrm{Min}\ \ \mathrm{F}\,;2.5 \times \mathrm{MGW} \times \mathrm{g}$
25-50	$1 - 0.01 \times\ \ \mathrm{MGW} - 25\ \ \ \times \mathrm{F}$
>50	$0.75 \times \mathrm{F}$

For vertical impact, the load to consider is:

$$F_{VI} = 0.08 \times F \tag{7}$$

Where F, the design load for lifting, is the greater of F_{Air} and F_{Sub}.

Acceleration loads:
The horizontal loads, due to vessel motion during sea transportation stage, are given by:

$$F_H = MGW \times g \tag{8}$$

The vertical loads are given by:

$$F_{Vmax} = 1.3 \times MGW \times g \tag{9}$$

$$F_{Vmin} = 0.7 \times MGW \times g \tag{10}$$

DNV also states that these loads shall be combined in any direction in order to cover the worst case scenario. As the horizontal forces have variable directions, it is advised to create a set of eight possible directions, each direction being incremented by 45º, covering an entire span of 360º. Due to symmetry, the number of load cases could be reduced, but many times the center of gravity of PO Units is not centered. Hence, analysing all eight horizontal directions is preferable. All loads should be combined resulting in sixteen cases, eight cases from the maximum vertical load and other eight for the minimum vertical load.

Wind loads:

During the transportation process the wind must also be accounted for, with an equivalent pressure of 1kN/m^2.

2.1.4. Design criterion

DNV's acceptance criterion imposes a limit on the maximum allowable stress. According to DNV 2.7-3, design loads shall not produce von Mises stresses higher than 85% of the yield stress. These statements can be a bit ambiguous since DNV 2.7-3 does not provide guidelines on how to perform this check. Choosing hand calculation over FEA can lead to different results, because the first one does not capture local peak stresses, due to hot stop stress concentrations. By hot spot stress concentrations, it is referred to the peak stresses that tend to be present in geometrical details, mainly on geometrical transitions, which are captured in FE models. Also, regarding FEA, the analysis methodology is not clear on how to use this 1 0.85 safety factor. Three possibilities come to discussion: to consider a stress limit of 85% of the yield stress on the FE model, which seems very conservative, particularly if stress hot spots develop; to affect the yield stress of material curve by the same factor; or to apply the safety factor afterwards, on the obtained capacity.

Regarding buckling effects, it is stated that all elements that are subject to compression must be checked against buckling, where the maximum utilization factor is 0.85.

2.2. ISO 13628-7

The International Organization for Standardization (ISO) is a worldwide federation of national standards bodies, composed by 163 member countries, with more than 19500 international standards published to date. The ISO 13628 is a group of standards that provide requirements for the design and operations of subsea production systems. As for the part 7 of this group, ISO 13628-7, it contains provisions and recommendations for subsea C/WO riser systems running from a floating vessel.

2.2.1. Design loads

The ISO 13628-7 part does not provide explicit formulae for load calculation, but it contains some guidelines. Therefore, the loads applied to C/WO units are assessed through global riser analysis simulating a riser as presented in Figure 1.11. Quoting ISO 13628-7, for environmental loads effects:

- For permanent operational conditions the most probable extreme combined load effect for a 100-year return period shall apply;
- For temporary operational conditions the most probable extreme combined load effect for the following return period values shall apply:
    - A 100-year return period if duration is in excess of 6 months;
    - A 10-year return period for the actual seasonal environmental condition if duration is over 3 days but less than 6 months;
    - Specified extreme load condition for duration less than 3 days.

This standard also provided some insight regarding which loads to consider:

Environmental loads:

- Surface waves – including wave height, spectral peak period, spectral shape and directionality;
- Current – currents velocity, profile and direction shall be selected using recognized statistical methods;
- Water depth and tide;
- Temperature – including the maximum, average, and minimum seasonal air and sea temperatures at the site;
- Vessel offset and motions - static offset, wave frequency motions, low-frequency motions, and set-down and draught variations;
- Hydrodynamic loads.

Accidental loads:

- Accidental loads may be defined according to system review and risk analysis.

2.2.2. Design criteria

Regarding the component design criteria, the following relevant failure modes shall be checked:

- Excessive yielding;
- Buckling;
- Fatigue;
- Brittle fracture;
- Excessive deflections;
- Leak-tightness;
- Corrosion and wear;
- Sudden disengagement;
- Mechanical function.

Annex D of this standard contains requirements and acceptance criteria to determine plastic collapse, or ultimate load capacity, of the covered components. The methods used for such purpose consist of either experimental testing or calculations. This last one implies the use of numerical methods such as FE models, which is preferable to experimental testing. Within FEA, some methodologies are provided, including:

- Elastic analysis;
- Limit analysis based on perfectly elastic plastic material model and small deformation theory;
- Plastic collapse analysis based on material strain hardening and large deformation theory.

2.2.3. Elastic analysis

In this method, an elastic analysis is performed through FEA and the design check is performed through stress verification. In order to perform this check, stresses shall be linearized and divided into primary and secondary stresses. After this division is performed, stresses shall be checked, distinctly for the main structure and for bolts, according Figure 2.3.

Linearized equivalent von Mises stress value	Allowable stress	
	General	Bolts (root area)
General membrane primary stress, P_m [a, e]	$(\sigma_{eq})_{Pm} \leqslant \frac{2}{3} \times C_f \times \sigma_y$	$(\sigma_{eq})_{Pm} \leqslant \frac{2}{3} \times C_f \times \sigma_y$
Local primary membrane stress, P_l [b]	$(\sigma_{eq})_{Pl} \leqslant C_f \times \sigma_y$	NA
Primary membrane (general or local) plus primary bending stress ($P_m + P_{bs}$) or ($P_l + P_{bs}$)	$(\sigma_{eq})_{P} \leqslant C_f \times \sigma_y$	$(\sigma_{eq})_{P} \leqslant C_f \times \sigma_y$
Primary (P) plus secondary (Q_s) membrane + bending stress	NA	$(\sigma_{eq})_{Pm+Qms+Pbs} \leqslant 0{,}75 \times C_f \times \sigma_y$
Primary-plus-secondary stress range [c] ($P_m + P_{bs} + Q_s$) or ($P_l + P_{bs} + Q_s$)	$(\Delta\sigma_{eq})_{P+Qs} \leqslant 2{,}0 \times \sigma_y$	$(\Delta\sigma_{eq})_{P+Qs} \leqslant \sigma_y$ [d]
Primary average shear, P_{sh}	$(\tau_{sh})_{Pm} \leqslant 0{,}4 \times C_f \times \sigma_y$	$(\tau_{sh})_{Pm} \leqslant 0{,}4 \times C_f \times \sigma_y$
Average bearing stress, P_{br} [f, g]	$(\sigma_{br})_{P+Qs} \leqslant \sigma_y$	NA

[a] General membrane primary stresses for pipes connected to the component are ensured by the pipe wall criteria, hence need not be controlled by this method.

[b] The distance over which the local primary membrane stress, P_l, exceeds 0,75 times yield strength should not extend in the axial direction more than $\sqrt{r \times t_2}$.

[c] The component limit of primary-plus-secondary stress range has been placed to ensure shakedown to elastic conditions after a few repetitions of the maximum operating load range. In the determination of the maximum primary-plus-secondary stress range, it may be necessary to consider the superposition of cycles of various origins that produce a total range greater than the range for any of the individual cycles.

[d] Secondary stresses after bolting-up include stresses produced by preload and thermal expansion.

[e] The bolt stress is primary during the bolting-up condition.

[f] In the case where the distance to a free edge is greater than the distance over which the bearing load is applied, the bearing stress may be increased by a factor of 1,5.

[g] When bearing loads are applied on parts having free edges, such as a protruding edge, the possibility of a shear failure shall be considered.

Figure 2.3 - ISO 13628-7 elastic analysis method (ISO 13628-7, 2006).

This division of stresses tends to become more difficult with the increase of the structure's detail. Since the structures targeted by this study are rather complex, this process is not commonly used. Besides, ISO 13628-7 recommends choosing either the perfectly elastic plastic or plastic analysis methods over this one, as they are more accurate.

2.2.4. Perfectly elastic plastic and plastic analysis

The criteria provided for perfectly elastic plastic analysis with small deformation theory and plastic analysis considering material strain hardening and large deformation theory are the same. What differs between these two methods is the modelling stage, namely the material stress strain relationship and the analysis type. Instead of the stress check imposed by the elastic method, these methods require strain checks, comprising both a local and a global strain check. According to ISO 13628-7, the limit load shall be determined as the minimum value of the following:

- Local failure criterion - load which will cause von Mises plastic strain to exceed:

$$\varepsilon_{peq} \leq min\ \ 0.1; 0.5 \times\ \ 1 - \frac{\sigma_y}{\sigma_u} \qquad (11)$$

- Global failure criterion - load which will cause overall structural collapse: 2% max principal structural strain.

After determining the capacity through FEA, the ultimate strength limit is given by:

$$S_d \leq F_d \times R_{uc} = \frac{2}{3} \times C_f \times R_{uc} \qquad (12)$$

Where F_d is the design factor, R_{uc} is the ultimate capacity and C_f is the design condition factor. ISO 13628-7 also provides design condition factors (partial factors) to be applied, depending on the load, see Figure 2.4.

Load condition (case)	C_f	Failure mode calculation basis
Assembly (bolting-up or make-up) and disassembly (break-out)	1,35	Based on actual design values at assembly/disassembly temperature
Mill/FAT hydrostatic pressure test	1,35	Based on actual design values at test temperature, fluid (hydrostatic)
Normal operation	1,00	Based on corroded wall thickness at design metal temperature
Extreme operation	1,20	Based on corroded wall thickness at design metal temperature
System (in-service) pressure test	1,20	Based on corroded wall thickness at test temperature
Temporary operation	1,20	Based on corroded wall thickness at actual metal temperature
Accidental (survival)	1,50	Based on corroded wall thickness at actual metal temperature

Figure 2.4 - ISO design condition factors (ISO 13628-7, 2006).

2.3. EC 3

The Eurocode program consists in a family of codes developed by the Comité Européen de Normalisation, intended for the elimination of technical problems, through a normalization of specifications regarding the civil engineering field. Eurocode 3 in particular, applies to engineering steel design, comprising several standards. The ones that this study will focus on are: EC 3 Part 1-1, providing general rules for buildings; EC 3 Part 1-6, containing provisions regarding the strength and stability of shell structures; and EC 3 Part 1-8, concerning the design of joints.

EC 3 Part 1-1 and Part 1-8 are used when calculating the capacity of structures through hand calculation. A detailed overview of these standards will not be made since they are extensive and have large amounts of information that are not related with the subject in study. To provide an easier comprehension of the current document, the formulae and provisions used are shown in the Chapter 4.3. Although, in this thesis, EC 3 is mainly used to perform hand calculation, EC 3 Part 1-6 provides guidance regarding FEA that is worth to be mentioned. According to this standard, depending on the limit state, one or multiple of the following analysis should be used:

- Linear elastic shell analysis (LA): Considers the use of an elastic material curve and the small deflection theory. It should only be used when it is assumed that throughout the analysis the geometry remains undeformed;
- Linear elastic bifurcation analysis (LBA): Used when the requirements of the previous analysis are met. It will provide the elastic critical buckling resistance, obtaining the lowest eigenvalue;
- Geometrically nonlinear elastic analysis (GNA): includes the influence of geometrical 2^{nd} order effects during the analysis;
- Materially nonlinear analysis (MNA): Provides the plastic limit load assuming a small deformation theory;
- Geometrically and materially nonlinear analysis (GMNA): Combines the two previous analyses, GNA and MNA. Provides the plastic limit load assuming an updated geometry of the structure throughout the analysis.
- Geometrically nonlinear elastic analysis with imperfections included (GNIA): Similar to GNA, but used for structures where imperfections must be accounted for;
- Geometrically and materially nonlinear analysis with imperfections included (GMNIA): Similar to GMNA, but used for structures where imperfections must be accounted for;

Some of the presented analysis, GNA, GMNA, GNIA and GMNIA, might also be used to perform buckling load evaluation. In that case, the eigenvalues of the system should be checked to ensure that these analyses do not fail to detect a possible bifurcation in the load path. These different analyses are used, by themselves or combined, in order to check different limit sates:

- Plastic limit (LS1): The plastic limit state is related to a failure mode due to yielding of the material. In this case the limit load derives from the plastic collapse load;
- Cyclic plasticity (LS2): Used when a structure experiences several loading and unloading cycles at the same point, leading to local cracking;
- Buckling (LS3): Considered when a structure loses its stability, leading to sudden large displacements;
- Fatigue (LS4): Concerns structures than experience several cyclic loads with different stress levels, causing a fatigue crack.

2.4. Analysis methodology – FEA

Possible methodologies used to determine the maximum capacity through FEA, according both to DNV 2.7-3 and ISO 13628-7, are hereby presented.

2.4.1. DNV 2.7-3

When DNV 2.7-3 states that the maximum allowable stress is 85%, it does not provide any guideline on which analysis type to use, nor which stress strain relationship shall be used. The analysis methodologies that one may use to retrieve the structural capacity, through FEA, according to this standard include:

- Material curve: Elastic; Criteria: First fibre to reach 85% x σ_y;
- Material curve: Perfectly elastic plastic with yield plateau at 85% x σ_y; Criteria: Limit analysis - load for which the model stops converging or at a certain strain limit (not specified by DNV 2.7-3);
- Material curve: Perfectly elastic plastic with yield plateau at σ_y; Criteria: Limit analysis - load for which the model stops converging or at a certain strain limit (not specified by DNV 2.7-3). After determining the load capacity, it shall be multiplied by 85%.

For the elastic analysis, first methodology, the von Mises stress shall be checked at a node level.

Another challenge that analysts face when performing FEA through DNV 2.7-3, is how to know when the maximum capacity is reached. Normally standards will provide a strain limit, but since DNV 2.7-3 does not provide any, this shall also be studied in this thesis. An article based on a real scale test (Healy and Zettlemoyer, 1993) provides a rotation limit that will be used when determining the maximum bending capacity of the models. But this limit is geometry dependent, therefore shall not be used for all structures in general. Additionally, for the elastic plastic analyses, second and third methodologies, the following methods will compared in order to assess the structural capacity:

- Consider a strain limit (EC 3 Part 1-6 strain limit);
- Consider the maximum capacity recorded in a 2^{nd} order analysis.

2.4.2. ISO 13628-7

When assessing the structural capacity according to this standard, the perfectly elastic plastic and plastic methodologies are commonly used. For these methods, the modelling techniques recommended by ISO 13628-7 are, respectively:

- Material curve: Perfectly elastic plastic with yield plateau at σ_y; Analysis type: 1^{st} order;
- Material curve: Plastic with strain hardening; Analysis type: 2^{nd} order.

For both alternatives, the same criteria are used, probing PEEQ (equivalent plastic strain, which stands for strains as von Mises stands for stresses). Regarding the local criteria, PEEQ shall be probed in the integration point. As for the global criteria, the average Logarithmic Strain (LE) or PEEQ shall be probed across a path. It has to be mentioned that the global criteria is prone to engineering judgment, since there are potentially many paths that can be defined and not all lead to a loss of stability of the structure.

3. NUMERICAL MODELLING AND VALIDATION

In order to perform a comparison between the presented standards and the different modelling techniques, FEA is performed. An existing study (Healy and Zettlemoyer, 1993), the same that provided the rotation limit previously mentioned, gathered several previous real scale test results, providing a database of experimental CHS T-joint tests. This study serves as a base case for the presented comparison, allowing validating the FE models against real test data.

The chosen geometry, Figure 3.1, consists of a generic T shaped connection between two circular hollow section members, resembling a frame joint.

3.1. Connection geometry

The connection used in this validation of results was experimentally tested by Stol, 1985. The relevant characteristics of this connection, labelled TNO 10, are as follows:

Geometric and material properties:

- Connection angle: 90 degrees;
- Chord section: 168.3 x 5.90 mm (outer diameter x thickness);
- Chord length: 840 mm;
- Chord steel yield stress: 309 MPa (estimated);
- Brace section: 114.6 x 5.95 mm;
- Brace length: 590 mm;
- Brace steel yield stress: 312 MPa (estimated) – used value was 309 MPa.

Capacities:

- Yield moment capacity of the brace: 16.34 kNm;
- Ultimate capacity recorded in test: 15.80 kNm.

As presented in Figure 3.1, the supports are pinned in both sides, with the axial direction of the chord restrained only on the left support. The load consists of a bending moment applied in the uppermost part of the brace.

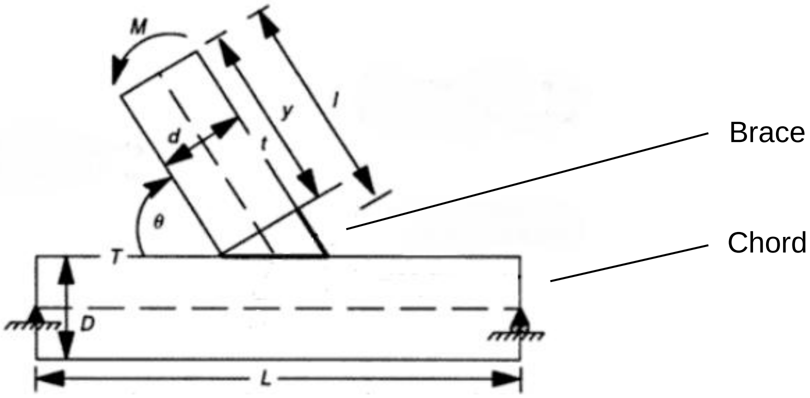


Figure 3.1 - Representation of the modelled geometry (adapted from Healy and Zettlemoyer, 1993)

This connection has some particularities, such as its capacity is not well determined by the load-displacement curve, since it does not exhibit a well-defined peak. To prevent excessive yielding its capacity has to be determined imposing a deformation limit. The deformation limit proposed by the author derives from a formula developed by Yura, 1980, which is representative for typical offshore structures:

$$\theta = \frac{80F_y}{E} \tag{13}$$

In addition to the presented experimental connection, two more geometries were studied in order to cover class 3 and class 4 sections (EC 3 categorization). The parameter modified to define these two other geometries was the wall thickness of both chord and brace, maintaining all other parameters, therefore only the new cross sections are presented:

Class 3 geometry properties:

- Chord section: 168.3 x 3.6 mm
- Brace section: 114.6 x 2.4 mm

Class 4 geometry properties:

- Chord section: 168.3 x 2.8 mm
- Brace section: 114.6 x 1.9 mm

Class 3 and class 4 models were only modelled with shell elements.

3.2. FEA parameters studied

In this thesis, several FE models were created using ABAQUS CAE. Hereby the modelling options chosen are discussed. The options that were not mentioned were taken according to ABAQUS default settings.

Sketch and part type:

Firstly the model was sketched. Afterwards the part types chosen were deformable wire (for beam models), deformable shell and deformable solid.

Assigning sections:

For the beam models a pipe section was created. For the shell models a section was specified using 5 Integration Points (IP) across thickness. As for the solid models, the section was automatically specified in the sketching stage.

Load Steps:

In all models a static general step was created in order to apply the load.

Boundary conditions:

In the shell and solid models, 3 constraints – couplings – were created to apply both the boundary conditions and the load. Structural distribution couplings were used. For beam, shell and solid models, in both supports, displacement/rotation mechanical boundary conditions were applied. To the support on the left (see Figure 3.1) all displacements and the chord’s torsion were restrained. As for the support on the right, all displacements except the chord’s axial displacement were restrained, as well as chord’s torsion. The bending moment applied had an intensity of 25 kNm.

3.2.1. Material models

In this chapter the stress strain relationships used are presented. They are defined according to the FEA methodologies explained in Chapter 2.4.

The material curves that had elastic behaviour, were modelled using a Young’s Modulus of 200 GPa and a Poisson coefficient of 0.3. The material curves used in these models were:

Elastic:

A fully elastic stress strain relationship (Figure 3.2).

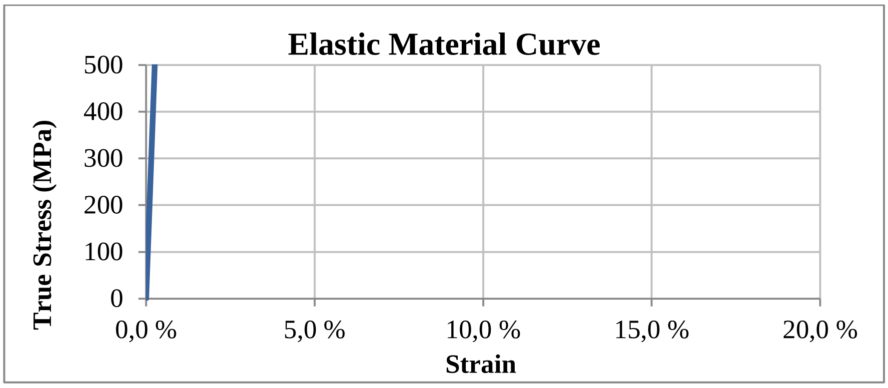


Figure 3.2 - Elastic material curve

Perfectly elastic plastic:

A material curve that is elastic up to a yielding stress of 309 MPa, becoming then perfectly plastic, Figure 3.3.

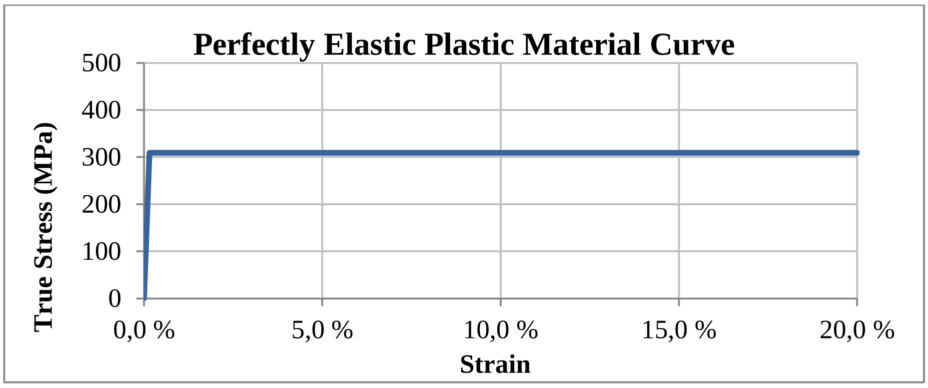


Figure 3.3 - Perfectly elastic plastic material curve

Perfectly elastic plastic with cut-off at 85%:

A stress strain relationship similar to the previous one, but with a cut off at 85% of the 309 MPa yield strength, having a yielding strength of 262.65 MPa, Figure 3.4.

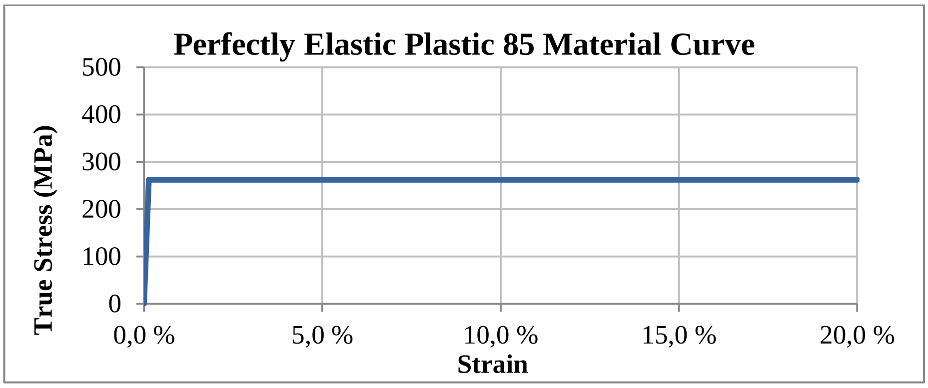


Figure 3.4 - Perfectly elastic plastic with cut off at 85% of yield strength

ASME VIII Division 2 material curve:

A more realistic stress strain relationship, which features strain hardening, Figure 3.5. When modelling this curve a ferritic steel type was considered with a design temperature of 20 °C.

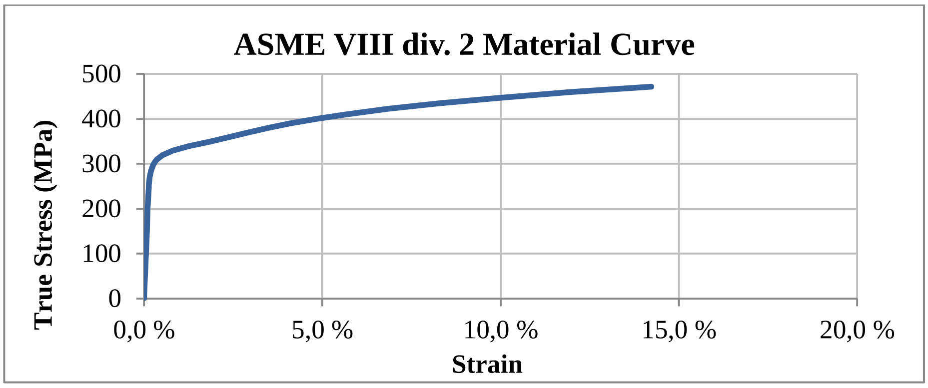


Figure 3.5 - ASME VIII div. 2 material curve

DNV-RP-C208 material curve:

Another stress strain relationship that also features strain hardening, but specific for FE nonlinear analysis, Figure 3.6.

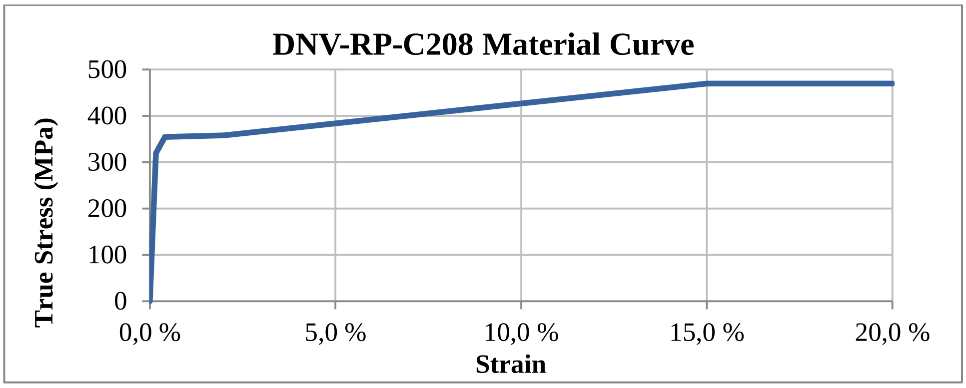


Figure 3.6 - DNV-RP-C208 material curve

This stress strain relationship was only used in class 1 shell models, since ASME VIII div.2 curve provides closer results to the real scale tests.

3.2.2. Element types

Three element types were chosen, beam, shell and solid, Figure 3.7. The mesh commands assigned to each models were as follows:

- Beam - B31: 2-node linear beam in space;
- Shell - S4R: 4-node doubly curved thin or thick shell, reduced integration, hourglass control, finite membrane strains;
- Solid - C3D8R: 8-node linear brick, reduced integration, hourglass control.

Additionally, another model with solid quadratic elements was created, Figure 3.8, to get more accurate results. It is composed by two different solid elements: C3D20 solid elements - a 20-node quadratic brick - applied in the connection region, while, in the remaining regions, C3D8R elements were considered. Also, in the connection region the mesh was finer and in the remaining regions it was coarser.

In all models, except this last one and those used to perform mesh convergence study, the mesh size used was 5 mm. In the solid models three elements were adopted across the thickness, to ensure that local effects could be properly evaluated.

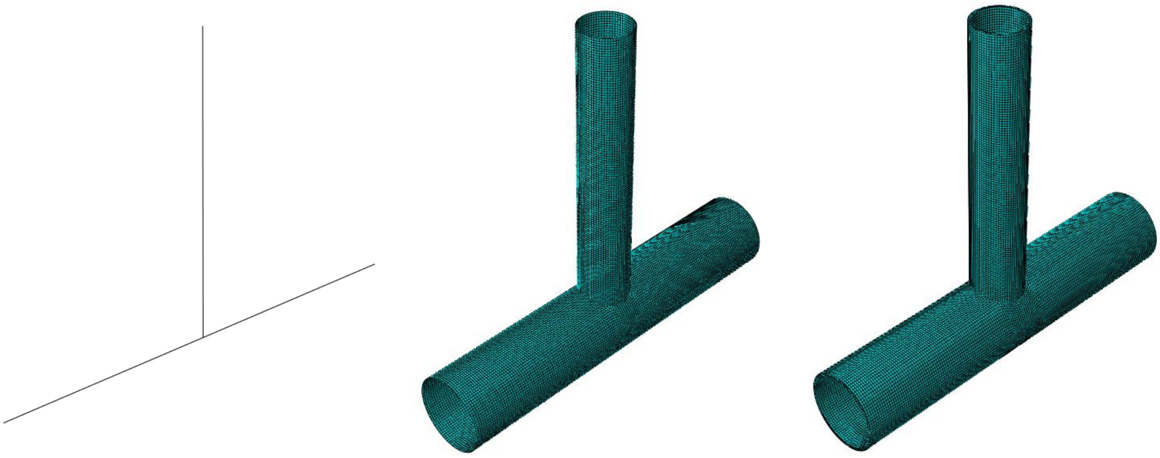


Figure 3.7 - Beam, shell and solid models (from left to right)

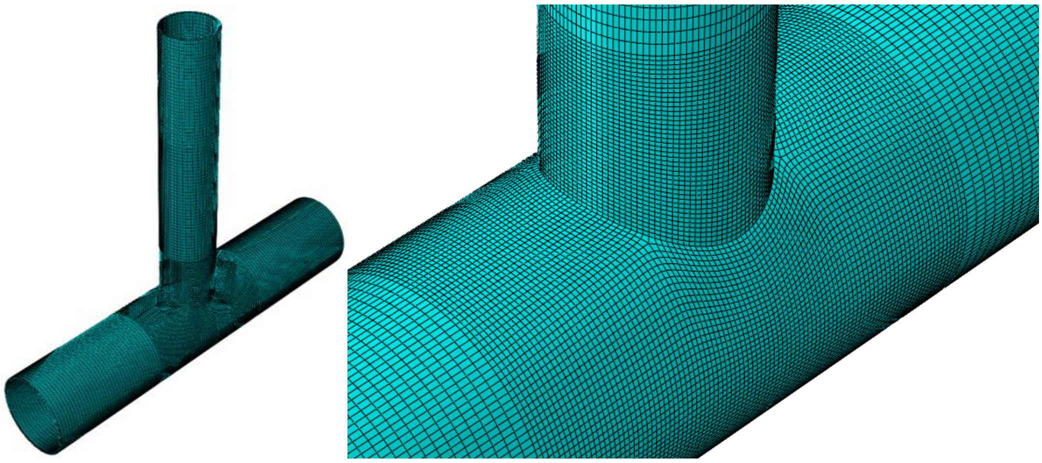


Figure 3.8 – “Solid Quad ASME NL” model, whole model (on the left) and close up (on the right)

3.2.3. Analysis types

These models were created considering both a 1st order, or small displacement analysis, and a 2nd order, or large displacement analysis. This is achieved by switching on NLGeom (nonlinear geometry analysis) in the appropriate step.

3.2.4. Other modelling parameters

Additionally, other modelling parameter was studied: moment controlled model (impose a bending moment) versus rotation controlled (establishing a rotation).

3.2.5. Finite Element models

After combining all the parameters presented before, element type, material curve and analysis type, several models were created. Table 3.1 presents a list with the most relevant FE models used. This list of twenty seven models presents the designation and the differences between them.

Table 3.1 - Model description

Model name	Model Description			
	Element type	Material Curve	Analysis Type	Class
Beam Elastic	Beam	Perfectly Elastic	1st order	1
Beam ElastPlast 85		Perfectly Elastic Plastic; σ_y=262.65 MPa		
Beam ElastPlast		Perfectly Elastic Plastic; σ_y =309 MPa		
Beam ASME		ASME VIII div.2		
Beam Elastic NL		Perfectly Elastic	2nd order	
Beam ElastPlast 85 NL		Perfectly Elastic Plastic; σ_y =262.65 MPa		
Beam ElastPlast NL		Perfectly Elastic Plastic; σ_y =309 MPa		
Beam ASME NL		ASME VIII div.2		
Shell Elastic	Shell	Perfectly Elastic	1st order	1, 3 and 4
Shell ElastPlast 85		Perfectly Elastic Plastic; σ_y =262.65 MPa		
Shell ElastPlast		Perfectly Elastic Plastic; σ_y =309 MPa		
Shell ASME		ASME VIII div.2		
Shell DNV		DNV RP C208		
Shell Elastic NL		Perfectly Elastic	2nd order	
Shell ElastPlast 85 NL		Perfectly Elastic Plastic; σ_y =262.65 MPa		
Shell ElastPlast NL		Perfectly Elastic Plastic; σ_y =309 MPa		
Shell ASME NL		ASME VIII div.2		
Shell DNV NL		DNV RP C208		
Solid Elastic	Solid	Perfectly Elastic	1st order	1
Solid ElastPlast 85		Perfectly Elastic Plastic; σ_y =262.65 MPa		
Solid ElastPlast		Perfectly Elastic Plastic; σ_y =309 MPa		
Solid ASME		ASME VIII div.2		
Solid Elastic NL		Perfectly Elastic	2nd order	
Solid ElastPlast 85 NL		Perfectly Elastic Plastic; σ_y =262.65 MPa		
Solid ElastPlast NL		Perfectly Elastic Plastic; σ_y =309 MPa		
Solid ASME NL		ASME VIII div.2		
Solid Quad ASME NL		ASME VIII div.2		

3.3. Validation of the FE models

Validating the FE models is required to understand if the results provided by these are reliable. This step assures that the conclusions are drawn based on accurate FE models. In the following chart, see Figure 3.9, the load-displacement curves from the real scale test and also from beam, shell and solid "ASME NL" models are presented. Only ASME VIII div.2 material curve models are shown because this is the curve that better replicates steel properties. Also, these are 2nd order analysis models, since their results are more realistic than the 1st order models.

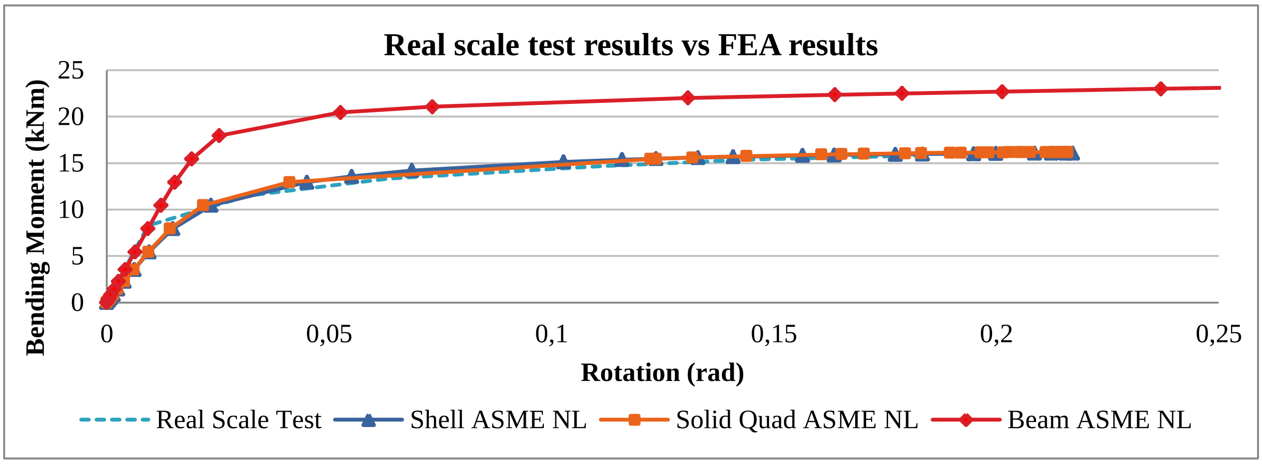


Figure 3.9 - Real scale test results vs FEA results, bending moment-rotation chart

Two points were chosen to perform the validation of the results: the deformation limit point provided; and the failure or maximum capacity point, see Table 3.2.

Table 3.2 - Comparison between FEA results and real scale test results

Bending Capacity (kNm)	Real Scale Test	Shell ASME NL	Diff.	Solid Quad ASME NL	Diff.
Deformation Limit	14.80	15.47	4.5 %	15.47	4.5 %
Failure	15.80	16.08	1.7 %	16.18	2.4 %
		Average	3.1 %	Average	3.4 %

As presented in the previous table, the difference between shell and solid models results and the experimental data is reduced. An average difference of 3.1% for the shell model and 3.4% for the solid model is found. This confirms that the FE model curves fall close to the experimental data curve. These small differences might occur due to some uncertainties and assumptions made in the modelling stage, as well as imperfect boundary conditions in the real scale testing. Since there is a good alignment between the FE analysis results and the real scale test results the FE models are considered as valid.

The "Beam ASME NL" model was shown in the previous chart without the intent of performing a validation of the results, but to show the difference between beam models with shell and solid models. As it will be discussed further on, beam models cannot properly assess the connection’s capacity. But, curiously, out of the three models presented, it is the one that falls closer to the real scale test data in the 0 - 7.5 kNm range. This might be related to the initial stiffness and rotation capacity.

Another important step in FEA, which also aims towards assuring reliable results, is the mesh convergence study. The goal of this study is to understand if the mesh has influence in the results and, if so, it should be changed until the results are consistent. Figure 3.10 presents the results retrieved from five different mesh sized models. The model labelled "Shell ASME Mesh 5" is the control model, since it was assigned the same mesh commands used in the majority of the remaining models. The four other models in the chart consist in two models with a coarser mesh and other two with a finer mesh. In the models' label, the number that appears after "Mesh" represents the mesh size, for example in the model "Shell ASME Mesh 5" a 5 mm mesh size was assigned.

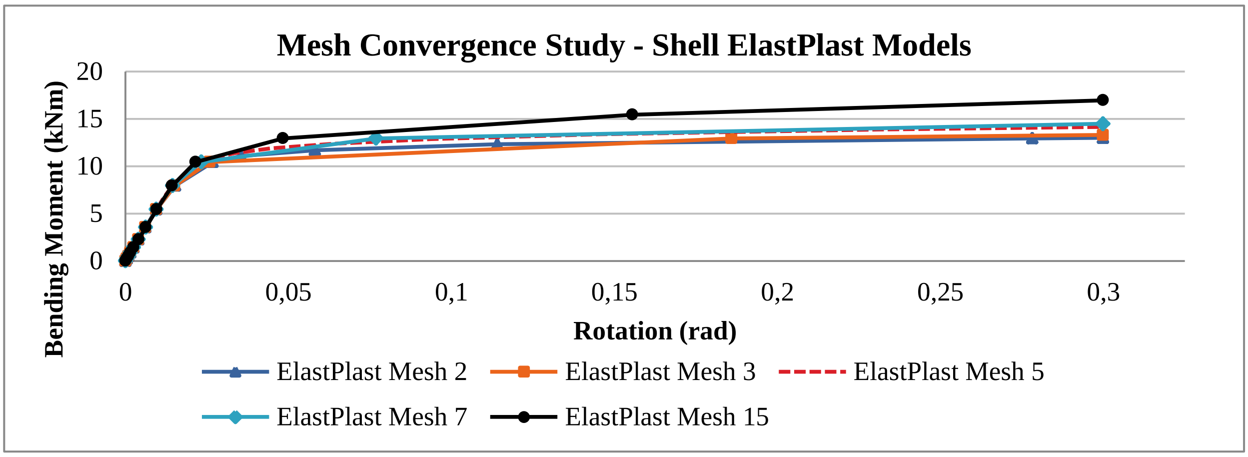


Figure 3.10 - Mesh convergence study, bending moment-rotation chart

The chart above shows that the "control" model results are identical to the finer mesh models, with slight differences, which are considered as acceptable.

4. PARAMETRIC STUDY

In this chapter the results retrieved from the FE models will be presented, and the ultimate capacity according to each standard will be determined. When going through this chapter, in order to be of better comprehension, it is suggested to resort to the models' names, Table 3.1, and to the adopted analysis methodologies according to each standard, Chapter 2.4.

4.1. DNV 2.7-3 criterion

The results for the maximum bending capacity obtained through DNV 2.7-3 criterion are presented in this chapter, see from Figure 4.1 to Figure 4.12, and from Table 4.1 to Table 4.3.

In combination with DNV 2.7-3 criterion a deformation limit was also used, see Chapter 3.1, which is presented in the following charts as vertical black dotted line.

4.1.1. Beam models

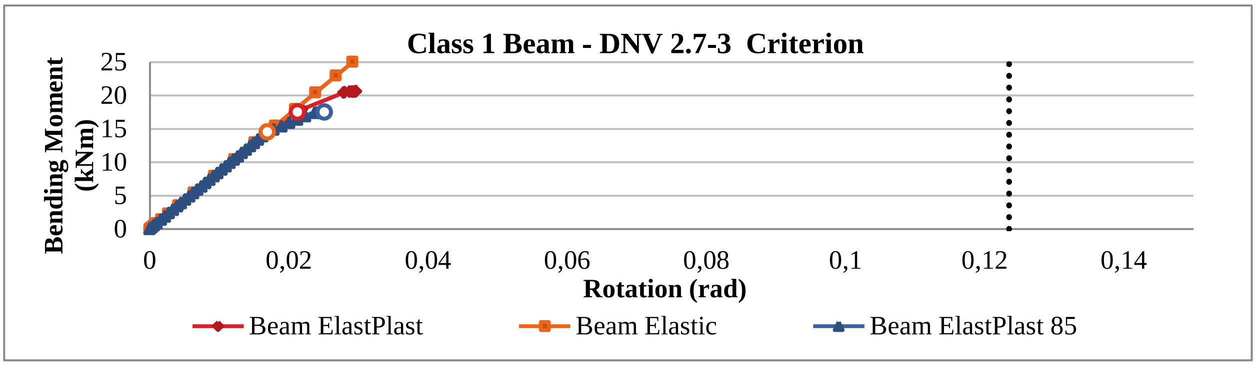


Figure 4.1 – DNV 2.7-3 criteria: "Class 1 Beam" models, bending moment-rotation chart

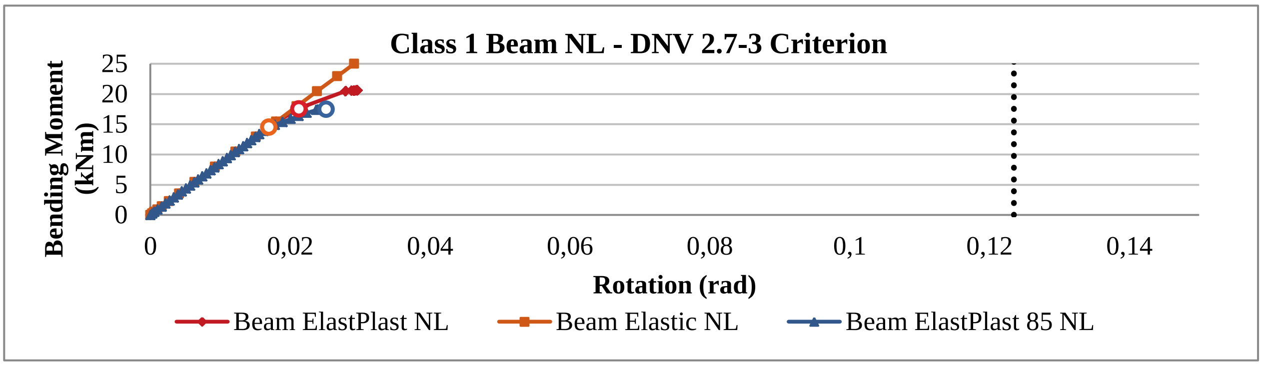


Figure 4.2 - DNV 2.7-3 criterion: “Class 1 Beam NL” models, bending moment-rotation chart

Table 4.1- DNV 2.7-3 criterion for beam models

DNV 2.7-3 Criterion	
Model	Bending Capacity (kNm)
Beam Elastic	14.53
Beam ElastPlast	17.51
Beam ElastPlast85	17.48
Beam Elastic NL	14.53
Beam ElastPlast NL	17.51
Beam ElastPlast 85 NL	17.48

When observing the presented charts, it is noticed that beam elements do not reach the rotation limit. Since the largest strains occur in that region, and beam elements cannot predict local failures, then the rotation limit is not reached. It is also noticed that there are no major differences between the capacities obtained through the different analysis types, in this case.

4.1.2. Shell models

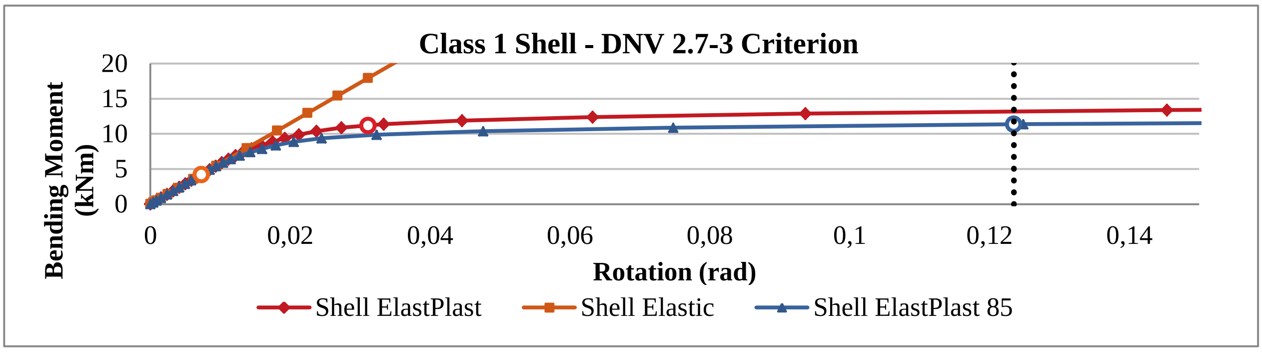


Figure 4.3 - DNV 2.7-3 criterion: "Class 1 Shell" models, bending moment-rotation chart

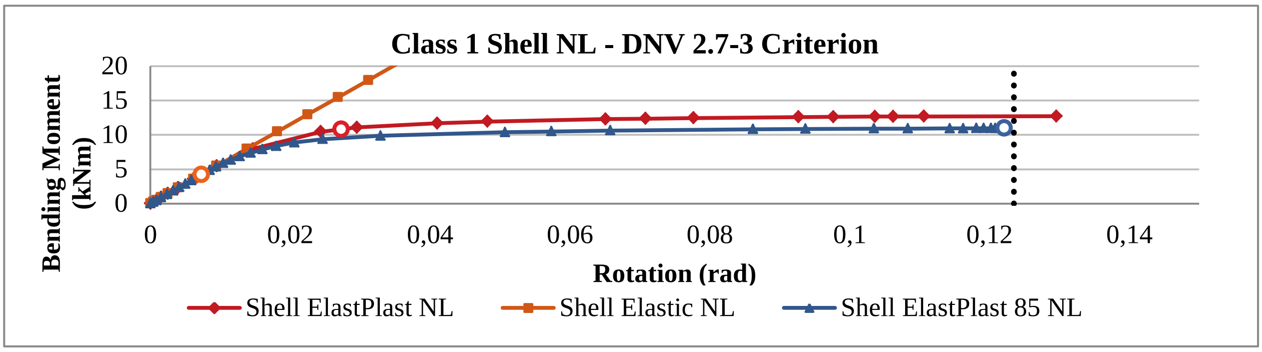


Figure 4.4 - DNV 2.7-3 criterion: "Class 1 Shell NL" models, bending moment-rotation chart

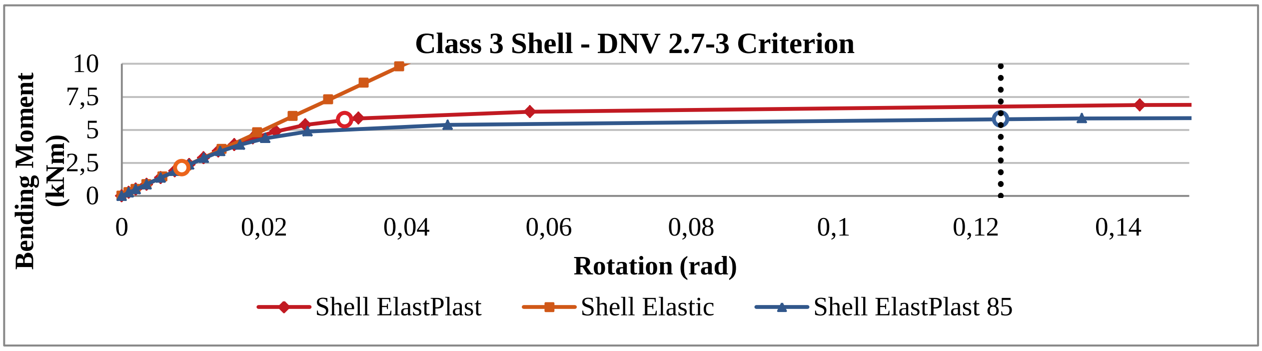


Figure 4.5 - DNV 2.7-3 criterion: "Class 3 Shell" models, bending moment-rotation chart

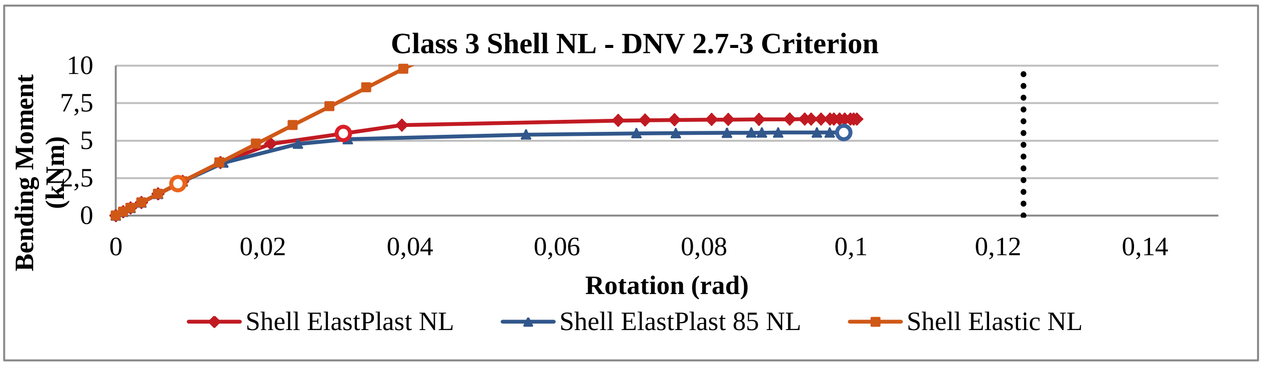


Figure 4.6 - DNV 2.7-3 criterion: “Class 3 Shell NL” models, bending moment-rotation chart

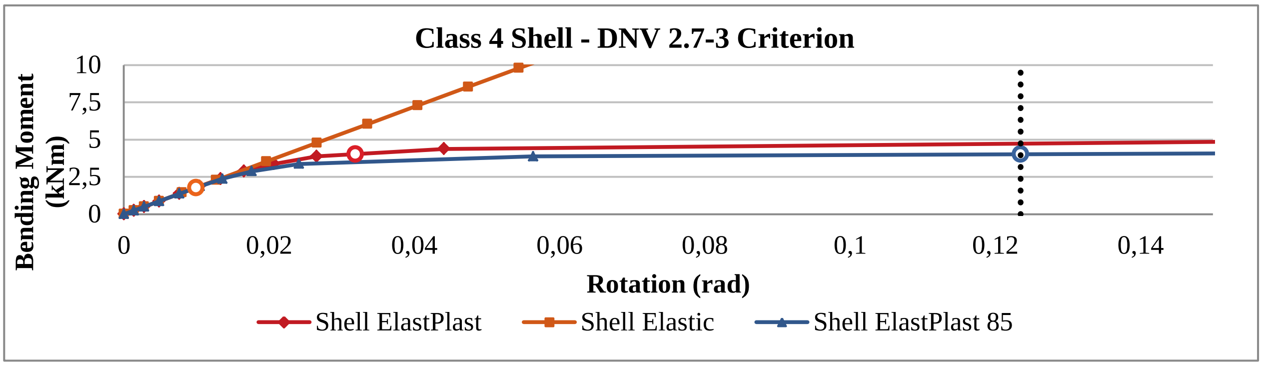


Figure 4.7 - DNV 2.7-3 criterion: “Class 4 Shell” models, bending moment-rotation chart

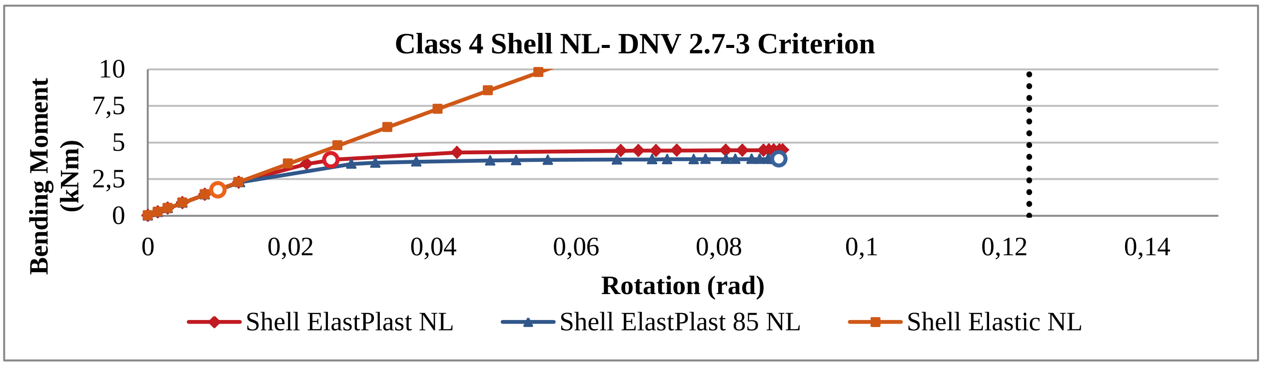


Figure 4.8 - DNV 2.7-3 criterion: “Class 4 Shell NL” models, bending moment-rotation chart

Table 4.2 - DNV 2.7-3 criterion for shell models

DNV 2.7-3 Criterion			
Model	Bending Capacity (kNm)		
	Class 1	Class 3	Class 4
Shell Elastic	4.21	2.12	1.78
Shell ElastPlast	11.19	5.75	4.03
Shell ElastPlast 85	11.36	5.81	4.02
Shell Elastic NL	4.21	2.12	1.76
Shell ElastPlast NL	10.80	5.47	3.80
Shell ElastPlast 85 NL	10.96	5.55	3.87

As expected, the models with smaller thicknesses achieved lower bending capacities. It is also noticed that reducing the wall's thickness will lead towards lower strains at the failure load, therefore class 3 and class 4 models with 2nd order analysis do not reach the rotation limit. Furthermore, the initial stiffness of slenderer models is considerably lower, since the bending moment-rotation curves of these have lower initial gradients. In Figure 4.9 is presented the deformation at the limit load of “Shell ElastPlast 85 NL” class 1, 3 and 4 models, were it is shown that slenderer section allow smaller displacements at failure load.

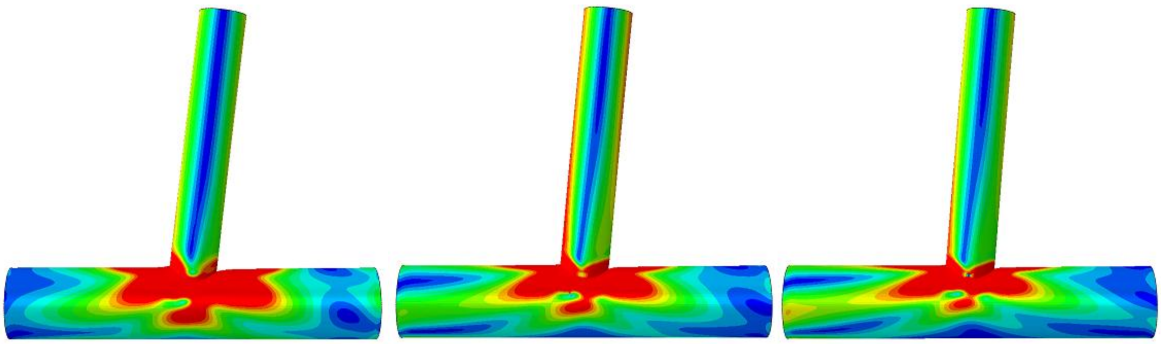


Figure 4.9 – Class 1, 3 and 4 “Shell ElastPlast 85 NL” models' deformation at limit load (from left to right)

When comparing the capacities obtained through the different analysis methodologies, defined according to DNV 2.7-3, it is noticed that the elastic methodology will provide substantially lower capacities. It is observed that the other two methodologies, using the 85% criterion in the material curve or in the limit load obtained, will lead to similar capacities, but the strains obtained by the first one are greater. This happens because using the 85% criterion in the material curve will lead to an earlier yield state.

The following figure presents the deformation of the structure at the failure load according to DNV 2.7-3 elastic analysis methodology criterion.

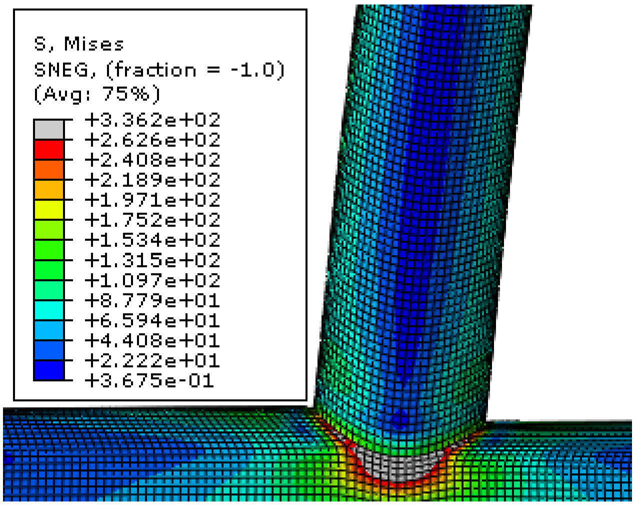


Figure 4.10 – “Class 1 Shell Elastic” DNV 2.7-3 criterion (units in MPa)

4.1.3. Solid models

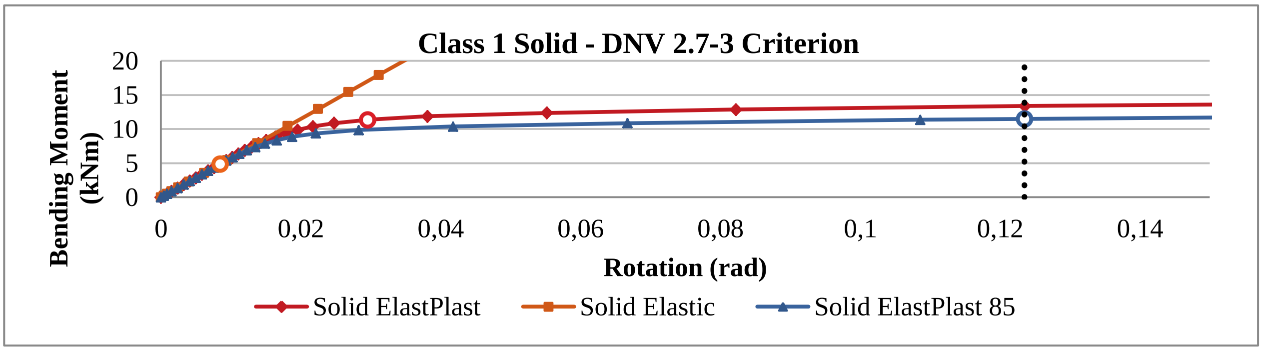


Figure 4.11 - DNV 2.7-3 criterion: “Class 1 Solid” models, bending moment-rotation chart

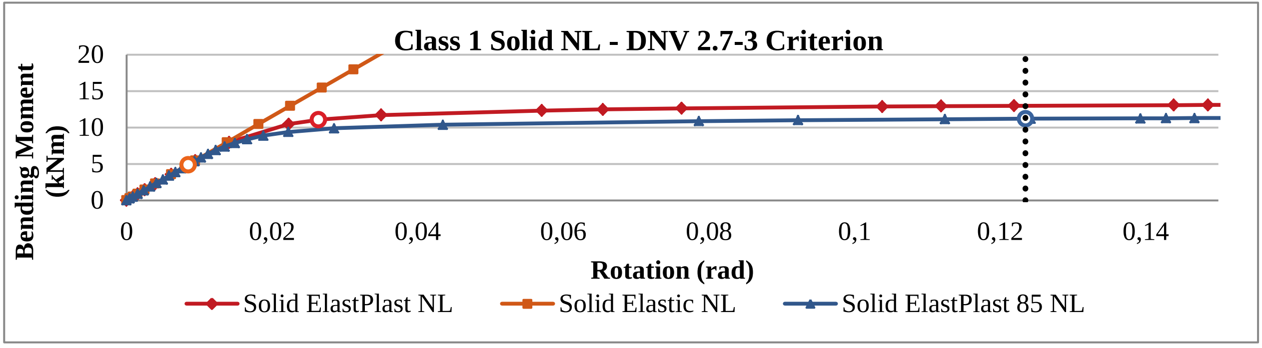


Figure 4.12 – DNV 2.7-3 criterion: “Class 1 Solid NL” models, bending moment-rotation chart

Table 4.3 - DNV 2.7-3 criterion for solid models

DNV 2.7-3 Criterion	
Model	Bending Capacity (kNm)
Solid Elastic	4.87
Solid ElastPlast	11.37
Solid ElastPlast85	11.49
Solid Elastic NL	4.87
Solid ElastPlast NL	11.04
Solid ElastPlast85 NL	11.18

It is observed that the results obtained both by shell models and solid models are quite different from the ones obtained by beam models. Unlike beam elements, shell and solid elements are able to assess the connection’s capacity, predicting local failure modes. In this case, the connection is the weakest component and, as expected, shell and solid models will provide lower capacities and higher strains. Since higher strains are reached, the rotation limit has to be used to prevent excessive yielding. When comparing the bending capacities obtained through solid and shell models, it appears that the results are close with each other.

4.2. ISO 13628-7 criteria

The results for the maximum bending capacity obtained through ISO 13628-7 criteria, both local and global, are presented in this chapter, from Figure 4.13 to Figure 4.18, Table 4.4 and Table 4.5.

4.2.1. Beam models

Since beam models do not reach the maximum strain limits provided by ISO 13628-7 their results are not presented. The reason for this is explained in Chapter 5.

4.2.2. Shell models

For shell models, only ISO's local criterion is shown. Shell models only have a single element across their thickness, therefore a failure path across the pipe's thickness could not be sketched. Hence, the global criterion was only determined for solid models.

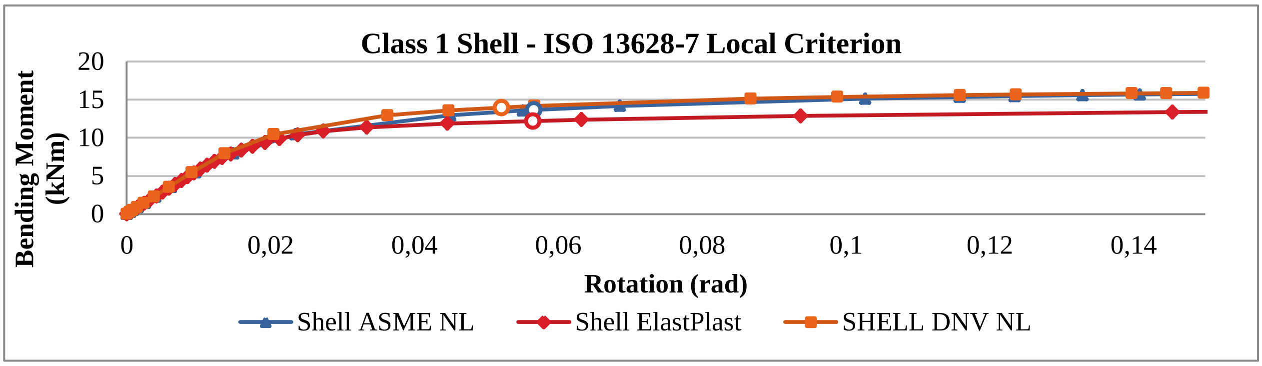


Figure 4.13 – ISO 13628-7 local criterion: "Class 1 Shell" models, bending moment-rotation chart

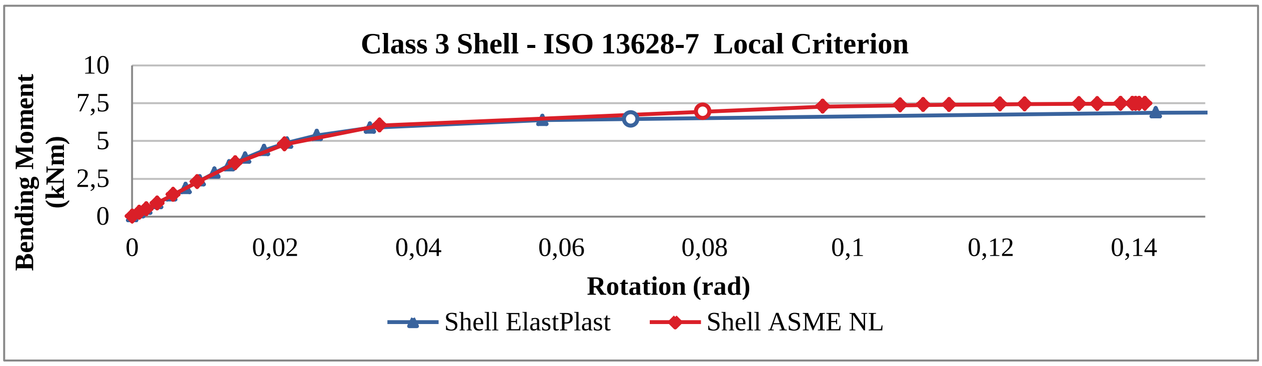


Figure 4.14 - ISO 13628-7 local criterion: "Class 3 Shell" models, bending moment-rotation chart

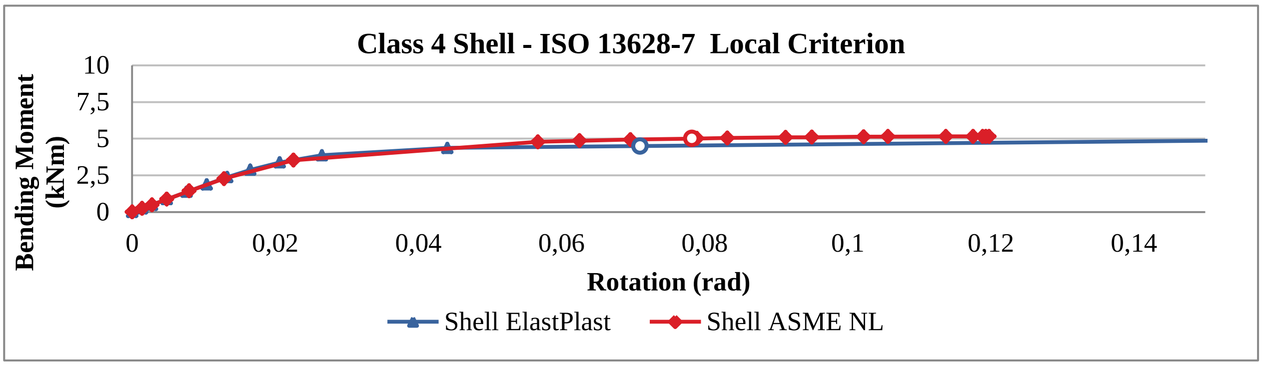


Figure 4.15 - ISO 13628-7 local criterion: “Class 4 shell” models, bending moment-rotation chart

Table 4.4 – ISO 13628-7 local criterion for shell models

ISO 13628-7 Local Criterion			
Model	Bending Capacity (kNm)		
	Class 1	Class 3	Class 4
Shell ElastPlast	12.19	6.45	4.50
Shell ASME NL	13.64	6.94	5.01
Shell DNV NL	13.96	-	-

In a similar fashion to the DNV 2.7-3 criterion, it can be stated that slenderer sections tend to achieve lower capacities and also have lower initial stiffness. In accordance to the material curves defined in Chapter 3, DNV-RP-C208 material curve was also used. This curve achieved higher bending capacities than ASME VIII div.2 material curve, which also features strain hardening.

4.2.3. Solid models

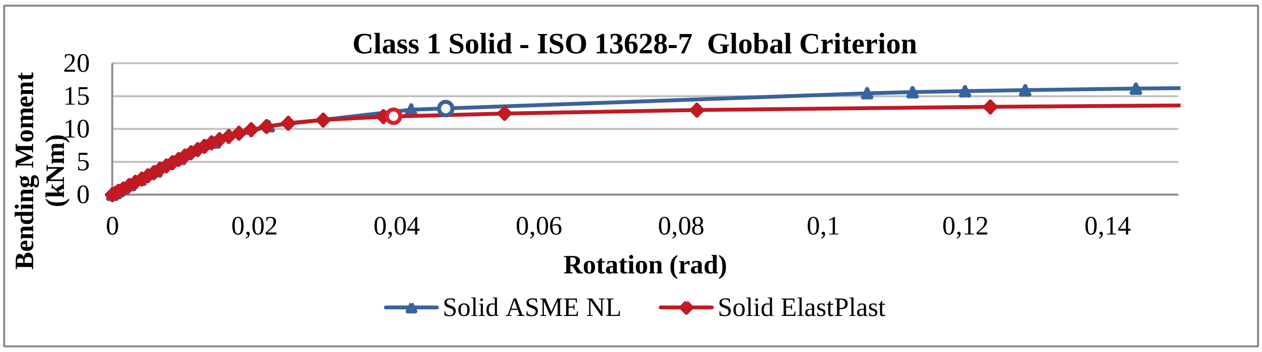


Figure 4.16 - ISO 13628-7 global criterion: "Class 1 Solid" models, bending moment-rotation chart

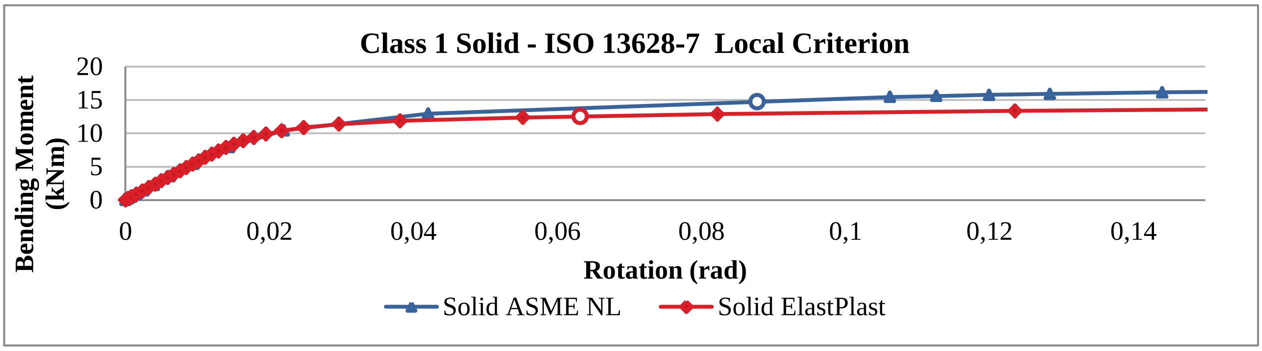


Figure 4.17 – ISO 13628-7 local criterion: "Class 1 Solid" models, bending moment-rotation chart

Table 4.5 - ISO 13628-7 local and global criteria for solid models

ISO 13628-7 Criteria		
Model	Bending Capacity (kNm)	
	Local Criterion	Global Criterion
Solid ElastPlast	12.52	11.92
Solid ASME NL	14.73	13.14

For both shell and solid models, it is perceived that the global criterion will govern failure, for this specific connection. The solid models tend to provide slightly higher capacities than the shell models. It is also seen that using the plastic with strain hardening material curve will provide higher capacities than the perfectly elastic plastic material curve, which is expected, since it allows higher stresses.

The following figure presents the deformation of the structure at the limit load according to the ISO 13628-7 elastic plastic analysis methodology. Both the local and global criteria are shown. The red line that appears in the figure on the right represents the path used to check the global criterion.

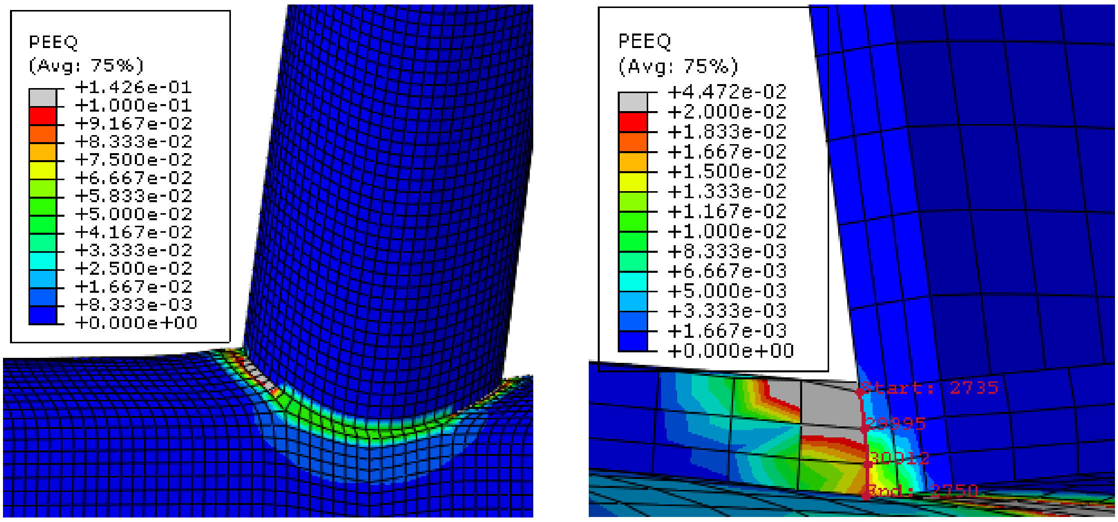


Figure 4.18 – ISO 13628-7 local (left) and global (right) criteria for "Class 1 Solid ElastPlast" model

4.3. EC 3 criteria – hand calculation

As mentioned before, in order to have some input on hand calculation procedures and address its differences compared with FEA, the design check was also made according to Eurocode 3. To check the T connection through EC 3, there are two stages that need to be attended: member capacity check and connection capacity check. The maximum bending capacity will derive from the lowest of these values. Hand calculation was only performed for class 1 cross sections, since the real scale test results did not provided experimental data for the class 3 and class 4 geometries.

4.3.1. Member capacity check

Before performing the member capacity check, the cross section class has to be determined. Eurocode 3 Part 1-1 provisions were used, Figure 4.19:

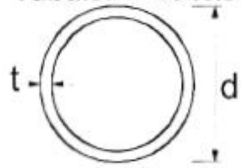
Tubular sections	
	
Class	Section in bending and/or compression
1	$d/t \leq 50\varepsilon^2$
2	$d/t \leq 70\varepsilon^2$
3	$d/t \leq 90\varepsilon^2$ **NOTE** For $d/t > 90\varepsilon^2$ see EN 1993-1-6.

Figure 4.19 – Class categorization for circular hollow sections (EN 1993-1-1, 2005).

Both cross sections are class 1 (Table 4.6), therefore the plastic material properties could be used when performing the member check.

Table 4.6 - Categorization according to class of the test models

		d_0/t_0	$50\varepsilon^2$	Class
D_0 (chord)	168.3	28.52	33	Class 1
T_0 (chord)	5.9			
d_0 (brace)	114.6	19.26		Class 1
t_0 (brace)	5.95			

The member check performed, only accounted for "pure" bending moment. According to EC 3, the plastic capacity is given by:

$$M_{c,Rd} = M_{pl,Rd} = \frac{W_{pl} \times f_y}{\gamma_{M0}} \quad (14)$$

With the input used in Table 4.7, the results obtained for the maximum bending capacity were 21.73 kNm for the brace and 48.10 kNm for the chord (see Figure 3.1 for difference between brace and chord). Since the bending capacity is governed by the weakest component, in respect to the member strength, the maximum capacity is 21.73 kNm. This result is quite accurate, as the real scale test data specified 21.90 kNm as the chord's plastic bending capacity.

Table 4.7 - Input for the calculation of the members' maximum bending capacity

f_y	309	MPa
W_{pl} (chord)	155673.64	mm^3
W_{pl} (brace)	70308.90	mm^3
γ_{M0}	1	-

4.3.2. Connection capacity check

To perform the connection capacity check, EC 3 Part 1-8 was used. Regarding the geometry being study, this standard provides different failure modes that have to be checked: chord face failure, chord side wall failure, chord shear failure, punching shear failure, brace failure and local buckling. If the connection being studied fits within certain geometric restrictions, which are also provided by this standard (Figure 4.20), only two failure modes need to be addressed: chord face failure and punching shear failure.

Diameter ratio		$0.2 \leq d_i/d_0 \leq 1,0$
Chords	tension	$10 \leq d_0/t_0 \leq 50$ (generally), but:
	compression	Class 1 or 2 and $10 \leq d_0/t_0 \leq 50$ (generally), but:
Braces	tension	$d_i/t_i \leq 50$
	compression	Class 1 or 2
Overlap		$25\% \leq \lambda_{ov} \leq \lambda_{ov,lim}$, see 7.1.2 (6)
Gap		$g \geq t_1 + t_2$

Figure 4.20 - Geometric restrictions for CHS T connections (EN 1993-1-8, 2005).

Since this connection fits within the restriction limits, see Table 4.8, only the two mentioned failure modes have to be verified.

Table 4.8 - Geometric restrictions check

		d_i/d_0		d_0/t_0	
D_i (chord)	156.5	0.93	Check	28.53	Check
D_0 (chord)	168.3				
T_0 (chord)	5.9				
d_i (brace)	102.7	0.90	Check	19.26	Check
d_0 (brace)	114.6				
t_0 (brace)	5.95				

4.3.3. Chord face failure

The chord face failure check is performed as follows, Figure 4.21:

$$M_{ip,1,Rd} = 4.85\frac{f_{y0}t_0^2 d_1}{sin\theta_1}\sqrt{\gamma}\beta k_p/\gamma_{M5}$$

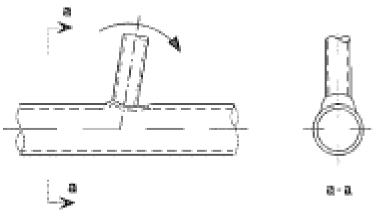


Figure 4.21 - Chord face failure design check (EN 1993-1-8, 2005).

Both β and γ are geometrical ratios. β is “the ratio of the mean diameter or width of the brace members, to that of the chord”. As for γ, it is “the ratio of the chord width or diameter to twice its wall thickness”. According to the input used in Table 4.9, the result obtained for the maximum bending capacity regarding this failure mode is 16.90 kNm.

Table 4.9 - Input for the calculation of the chord face failure mode

θ	90	º
d_1 (brace)	114.6	mm
t_0 (chord)	5.9	mm
f_{y0}	309	MPa
K_p	1	-
γ_{M5}	1	-
β	0.67	-
γ	14.26	-

4.3.4. Punching shear failure

The punching shear failure check is performed as follows, Figure 4.22:

$$M_{ip,1,Rd} = \frac{f_{y0} t_0 d_1^2}{\sqrt{3}} \frac{1 + 3sin\theta_1}{4sin^2\theta_1} / \gamma_{M5}$$

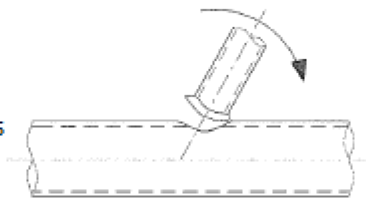


Figure 4.22 - Punching shear failure design check (EN 1993-1-8, 2005).

In order to perform this check, a geometric ratio has to be verified, see Table 4.10:

$$d_1 \leq d_0 - 2t_0 \tag{15}$$

Table 4.10 - Geometric ratio check

d_1	$d_0 - 2t_0$	
114.6	156.5	Check

As this geometric ratio is verified, the punching shear failure check formula is applicable. According to the input used in Table 4.11, the result obtained for the maximum bending capacity regarding this failure mode is 15.92 kNm.

Table 4.11 - Input for the calculation of the punching shear failure mode

θ	90	º
d_1 (brace)	114.6	mm
t_0 (chord)	5.9	mm
f_{y0}	309	MPa

Since the maximum bending capacity of the overall structure is governed by the weakest component, member or connection, the maximum bending load that this structure can withstand is 15.92 kNm, where punching shear drives as failure mode, see Table 4.12.

Table 4.12- Maximum bending moment for both member's and connection's capacity

		Bending Capacity (kNm)
Member Check	Chord Failure	48.10
	Brace Failure	21.73
Connection Check	Chord Face Failure	16.90
	Punching Shear Failure	15.92

Considering that the ultimate capacity recorded in the real scale test was 15.80 kNm, in a similar fashion as for the member check result, it is noticed that the formulae provided by EC 3 are very accurate, since the difference is 0.77%. It has to be noted that these results were obtained with partial factors equal to 1.0, thus they should not be directly compared with the results assessed according to DNV 2.7-3 or ISO 13628-7.

5. COMPARATIVE ASSESSMENT

5.1. Comparison between modeling techniques

5.1.1. Bending and rotation capacities

In Figure 4.1 and Figure 4.2 it is shown that beam models tend to stop converging with small rotations and with high bending capacities. Comparing them with the shell and solid models, Figure 5.1, it is noticed that the results are quite different. Shell and solid models stop converging at lower bending capacities but with higher rotations. When comparing with the real scale test data, the same conclusions are retrieved. Therefore it can be concluded that beam elements will only assess the members' capacity. If a structure has a connection and is modelled with beam elements, then its capacity will not be checked. Consequently, if the failure mode is related with the connection, like the present case, the use of beam elements is non-conservative. Also, beam models do not reach ISO 13628-7 strain limits, because it is in the connection region that higher strains occur.

Unlike beam elements, both shell and solid elements allow a proper check of the connection's capacity. The results are similar between these two element types, where the major difference is related to the rotation capacity. Solid elements tend to allow larger rotations before the collapse of the structure. This can be noticed in Figure 5.1.

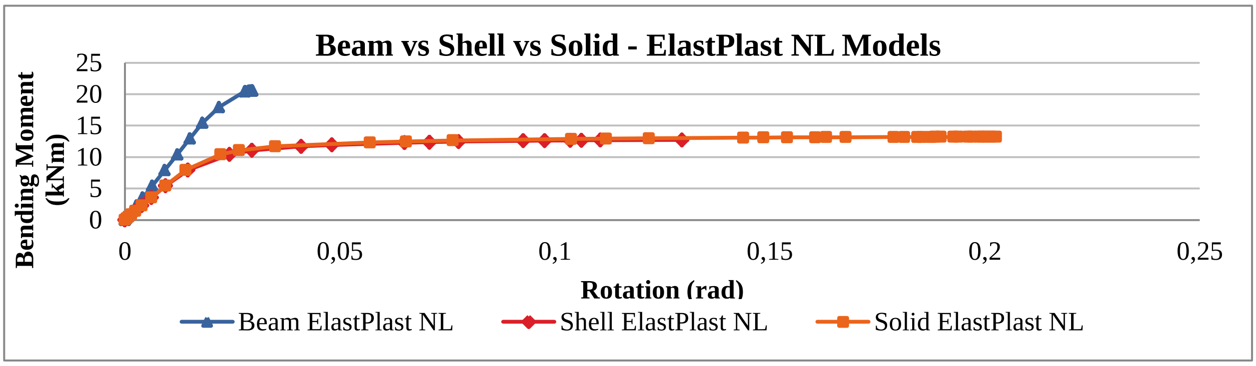


Figure 5.1 - Beam vs shell vs solid elastic plastic models, bending moment-rotation chart

5.1.2. 1st and 2nd order analysis

For beam models, a 1st or 2nd order analysis does not have major influence in the results. With regard to the other models, shell and solid, the results between a 1st and 2nd order analysis are distinct, mainly concerning the rotation capacity, with exception for elastic stress strain relationship models, see Figure 5.2.

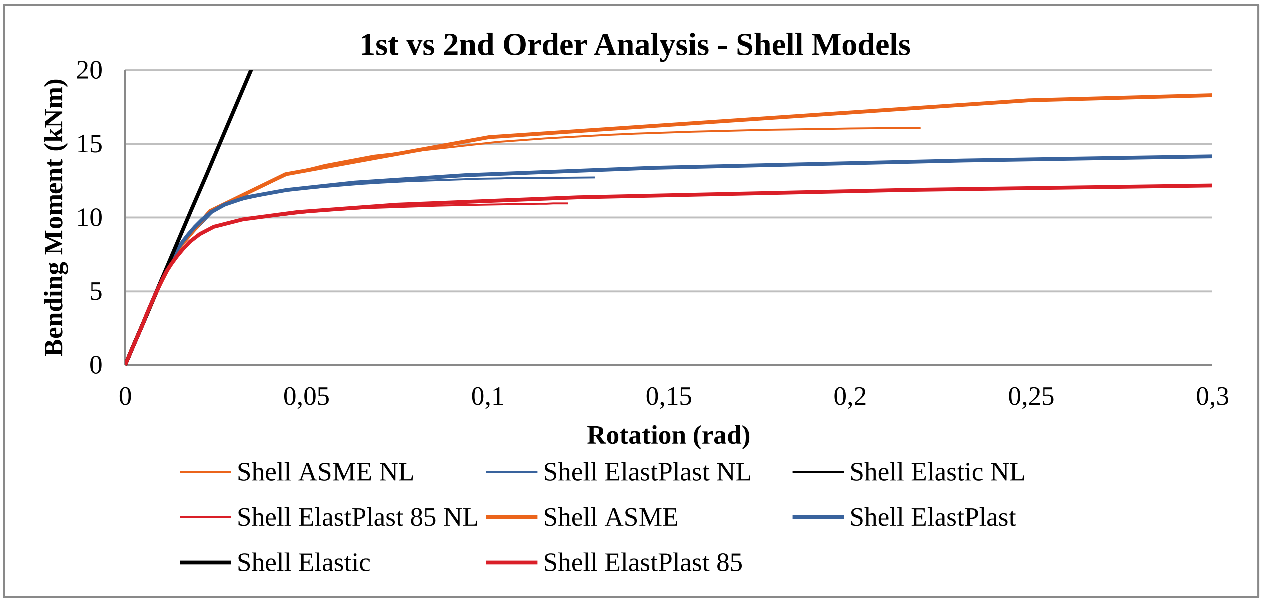


Figure 5.2 – 1st vs 2nd order analysis, bending rotation chart

When a 2nd order analysis is performed the stiffness matrix will be updated during the analysis stage, instead of remaining the same as in the initial stage. This will lead to an earlier collapse, which is more realistic. When the failure mode is related with the members' capacity, if a 1nd order analysis is performed, then the results obtained should be realistic. If the failure mode is related to local effects, like in the present case, then the results may not be realistic, leading in some cases to excessive yielding, see Figure 5.3. Performing a 2nd order analysis is always advised, although 1st order analysis results might be reliable if there are no instability issues and the failure mode is related with the members' capacity.

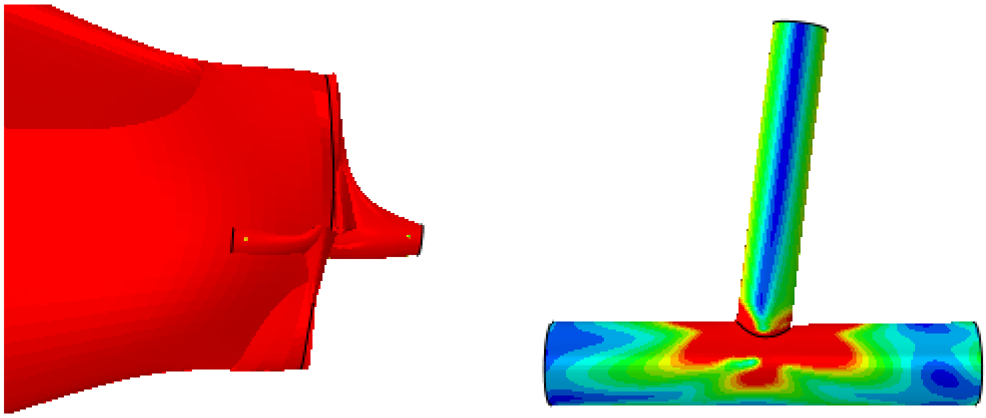


Figure 5.3 - Unrealistic “Class 1 Shell ElastPlast” model’s deformation (left) and realistic “Class 1 Shell ElastPlast NL” model’s deformation (right)

5.1.3. CPU cost

The resources that are required to compute the solutions of the presented models are different between each other, but a trend is noticed. The major features that will drive the duration of the analysis are the element type used and if a 1st or 2nd order analysis is adopted. Solid elements provide more information than shell or beam elements, and shell elements provide more information than beam elements. Naturally, increasing the complexity of the elements and their number, will lead to an increase of the calculation time, see Figure 5.4.

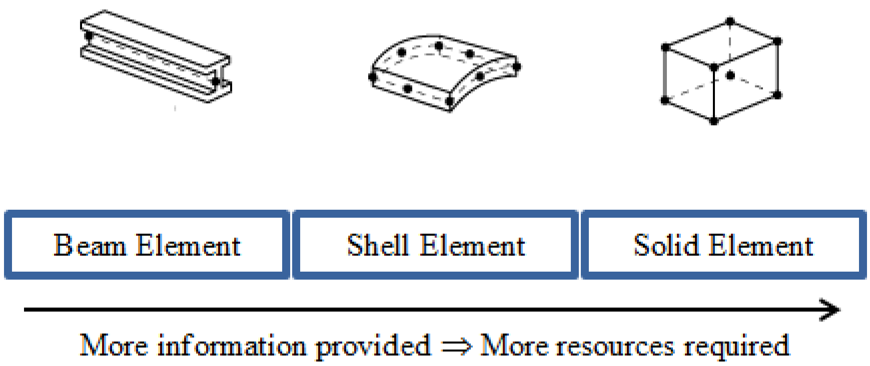


Figure 5.4 - Beam vs shell vs solid element types (adapted from ABAQUS User’s manual, 2013)

Table 5.1 presents the analysis times required by some of the models. Beam models clearly require the least time of all. As expected, solid models require the most amount of time to perform the calculations, in this case, taking over two times more than the shell models. The same can be said when performing nonlinear analysis. For this geometry, by switching NLGeom on, the analysis time will take at least two times more than the same model with NLGeom switched off. It can also be observed that the number of elements and the number of Degrees of Freedom (DOF) are correlated with the analysis duration.

Table 5.1 - Comparison between the processing time of different models

			CPU Time (s)		
Model	Number of Elements	DOF	ElastPlast 85 NL	ElastPlast NL	ASME NL
Beam	303	1824	212.6	10.6	6.4
Shell	25391	152976	1023.2	541.1	751.1
Solid	76740	308172	4834.3	4215.4	5768.9

The most accurate solid model was the one that also took the most amount of time. The "Solid Quad ASME NL" model took 88952 seconds of CPU time. This seems excessive for the extent of the current connection, showing that assigning quadratic elements must be carefully done.

5.1.4. Other modelling parameters

During the analysis stage other modelling techniques were experimented to understand their influence in the results. The more relevant was a comparison between rotation controlled models and moment controlled models. The results led to conclude that the outcome obtained through these two is very similar, regarding the maximum bending capacity obtained. But using rotation controlled models is advised, since in those models the rotation is limited, thus providing more control on the excessive yielding that occurs in models ran with 1st order analysis, see Figure 5.3.

5.2. Comparison between standards criteria

5.2.1. Comparison between possible DNV 2.7-3 analysis methodologies

Regarding the possible analysis methodologies described in chapter 2.4, it is without surprise that the elastic method is the most conservative of all. This method leads to the lowest maximum bending capacity. Clearly it is an overly conservative method, as it can be seen in Table 4.2 and Table 4.3, reaching less than half of the maximum bending capacities obtained through the other methods. It seems very conservative to consider that the maximum capacity of a structure is obtained based on the first fibre to reach 85% of yield stress. Besides this, when performing analysis on complex geometries, due to sharp corners, geometrical transitions or concentrated loads over a small area, stress hot spots will appear. These stress concentrations normally are not realistic and will largely influence this methodology.

Some conclusions can be drawn regarding the maximum bending capacity achieved by the other two methodologies, applying the 1 0.85 safety factor in the material curve or affecting the extracted capacity by the same factor. The maximum bending capacities achieved between these two methods are similar, see Table 5.2. The same cannot be stated for the rotation associated with this capacity. Applying the safety factor in the material curve will lead to an earlier yield state, resulting in high and unrealistic rotations. This leads to more than the double of the other method’s rotation, see Table 5.3. When applying the safety factor to the maximum bending capacity obtained, the rotations are closer to the real scale test data.

When comparing a 1st and a 2nd order analysis, the results are similar, with an averaged difference of 3.5%. This leads to believe that a 1st or 2nd order analysis have reduced influence for a material curve without strain hardening. This will only apply if a rotation or strain criterion is used to limit the maximum allowable displacements on a 1st order analysis, preventing excessive yielding.

Table 5.2 - Maximum bending capacity according to the different DNV 2.7-3 analysis methodologies

Bending Capacity (kNm)	Elastic Plastic 85	Elastic Plastic	Diff.	Elastic Plastic 85 NL	Elastic Plastic NL	Diff.
Class 1 Shell	11.36	11.19	1.5 %	10.96	10.80	1.5 %
Class 1 Solid	11.49	11.37	1.0 %	11.49	11.04	3.9 %
Class 3 Shell	5.81	5.75	1.1 %	5.55	5.47	1.3 %
Class 4 Shell	4.02	4.03	0.3 %	3.87	3.80	1.7 %
		Average	1.0 %		Average	2.1 %

Table 5.3 - Maximum rotation according to the different DNV 2.7-3 analysis methodologies

Bending Capacity (rad)	Elastic Plastic 85	Elastic Plastic	Diff.	Elastic Plastic 85 NL	Elastic Plastic NL	Diff.
Class 1 Shell	0.12	0.03	119.5 %	0.12	0.03	127.0 %
Class 1 Solid	0.12	0.03	122.7 %	0.12	0.03	129.6 %
Class 3 Shell	0.12	0.03	119.0 %	0.10	0.03	104.8 %
Class 4 Shell	0.12	0.03	117.9 %	0.09	0.03	110.1 %
		Average	119.8 %		Average	117.9 %

5.2.2. Comparison between ISO 13628-7 analysis methodologies

When comparing the ISO 13628-7 criteria a relevant difference between the element types can be noticed: the global strain check required by ISO 13628-7 could only be thoroughly performed in solid element models. As only solid models have more than one node across thickness, only in these a failure path could be sketched across the pipe's wall. Still concerning the global strain check, it was noticed that it is prone to engineering judgment. ISO 13628-7 states that this criterion is related with the overall structural instability, and, to prevent it, the principal structural strains should be checked. To perform this check, one has to know where this instability will occur or check several possible paths, considering the one that provides the lowest capacity value. For the current geometry it was not difficult to perform this check, even though the results could be different from analyst to analyst, but this task proves to be more arduous in more complex geometries, since the failure mechanism becomes less evident. Crosschecking this value against the load-displacement curve is helpful to work around this difficulty.

When comparing the two analysis methodologies studied through ISO 13628-7 criteria, perfectly elastic plastic material curve with 1st order analysis versus plastic with material strain hardening curve and 2nd order analysis, it is noticed that using the second alternative leads to higher bending capacities, 9.8% more, see Table 5.4. Thus it can be concluded that the perfectly elastic plastic methodology is a more conservative approach.

Table 5.4 - Maximum bending capacity according to the different ISO 13628-7 analysis methodologies

Bending Capacity (kNm)	Elastic Plastic NLGeom Off	ASME NLGeom On	Diff.
Class 1 Shell	12.19	13.64	11.2 %
Class 1 Solid	11.92	13.14	9.7 %
Class 3 Shell	6.45	6.94	7.4 %
Class 4 Shell	4.50	5.01	10.8 %
		Average	9.8.%

Regarding the two criteria proposed by the same standard, local and global, it is noticed that, for this specific geometry, the global strain criterion is the one that governs failure.

5.2.3. Comparison between DNV 2.7-3, ISO 13628-7 and EC 3

Although proven that hand calculation is a very accurate method for assessing the maximum bending capacity, it is easily understandable that sometimes this process is not viable, especially when it comes to more complex structures with several connections, different from each other, and even with unusual geometries. In conclusion, hand calculation provides reliable results, but performing it for all connections is a time consuming process.

In the following comparisons, only one DNV 2.7-3 methodology will be addressed: perfectly elastic plastic material curve with 1 0.85 safety factor being applied afterwards, to the maximum capacity obtained, since it appears to be the most accurate method out of the three assessed.

When comparing DNV 2.7-3 and ISO 13628-7, the first one provides more conservative results. When compared with the DNV 2.7-3 methodology, ISO's perfectly elastic plastic 1st order analysis achieved an average of 9% higher maximum bending capacities, Table 5.5. The methodology with ASME VIII div.2 material curve with 2nd order analysis achieved a bending capacity that is, in average, 18.5% higher than the DNV 2.7-3 methodology, Table 5.6.

Table 5.5 - DNV's perfectly elastic plastic analysis vs ISO's perfectly elastic plastic analysis

Bending Capacity (kNm)	Elastic Plastic NLGeom Off (DNV)	Elastic Plastic NLGeom Off (ISO)	Diff.
Class 1 Shell	11.19	12.19	8.6 %
Class 1 Solid	11.37	11.92	4.7 %
Class 3 Shell	5.75	6.45	11.5 %
Class 4 Shell	4.03	4.50	11.0 %
		Average	9.0 %

Table 5.6 - DNV's perfectly elastic plastic analysis vs ISO's plastic with strain hardening analysis

Bending Capacity (kNm)	Elastic Plastic NLGeom Off (DNV)	ASME NLGeom On (ISO)	Diff.
Class 1 Shell	11.19	13.55	19.1 %
Class 1 Solid	11.37	13.14	14.4 %
Class 3 Shell	5.75	6.94	18.9 %
Class 4 Shell	4.03	5.01	21.7 %
		Average	18.5 %

5.2.4. Observations on analysis methodology according to DNV 2.7-3

Following up on what was described in Chapter 2.4.1, the rotation limit used to achieve the maximum bending capacity according to DNV 2.7-3 was specifically adjusted for the studied geometry. Therefore, two alternatives to cover generic geometries are studied in this chapter. The first alternative considers the use of a strain limit based on EC 3 Part 1-6. The second one bypasses the use of a limit, but obliges a 2nd order analysis. The studies performed in this chapter were based on a perfectly elastic plastic material curve and, in all presented results, the maximum capacity obtained was multiplied by 0.85, so that the results were in accordance to DNV 2.7-3 criterion.

DNV 2.7-3 criterion with a strain limit:

EC 3 Part 1-6 is a standard intended for shell structures. It provides a strain limit, which fits FEA better than a rotation limit. The strain limit is given by the following formula:

$$\varepsilon_{mps} = n_{mps} \times \frac{f_{yd}}{E} \quad (16)$$

The recommended value for n_{mps} is 50, thus the strain limit is 7.725%. According to EC 3 Part 1-6, this strain limit applies to 2nd order analysis, but in the presented study, it is adopted for 1st order analyses. When this limit was considered together with the DNV 2.7-3 criterion, it was noticed that the results obtained were more conservative than ones obtained through previous analyses, where the rotation limit provided by Yura, 1980, was used. See Table 5.7.

Table 5.7 - Comparison between the results obtained through the rotation limit and EC 3 Part 1-6 strain limit

Bending Capacity (kNm)	Rotation limit	Strain limit	Diff.
Solid Class 1	11.37	10.46	8.3 %
Shell Class 1	11.19	10.16	9.6 %
Shell Class 3	5.75	5.48	4.8 %
Shell Class 4	4.03	3.82	5.3 %
	Average		7.0 %

DNV 2.7-3 criterion bypassing the strain limit:

The other methodology avoided using a strain limit. Since no deformation limit was used, a 2nd order analysis was required to prevent unrealistic large deformations. The maximum bending capacity is dictated by the maximum value recorded in test. The results obtained through this methodology were similar to the previous results obtained with the rotation limit and also a 2nd order analysis, as shown in Table 5.8.

Table 5.8 - Comparison between the results obtained through the rotation limit and a 2nd order analysis

Bending Capacity (kNm)	Rotation limit	2nd order analysis (without limit)	Diff.
Solid Class 1	11.04	11.22	1.6 %
Shell Class 1	10.80	10.81	0.1 %
Shell Class 3	5.47	5.47	0.0 %
Shell Class 4	3.80	3.80	0.0 %
	Average		0.4 %

After comparing the results obtained through both methods, it is noticed that the option of bypassing the strain limit and using a 2nd order analysis will provide more accurate results, whereas using EC3 Part 1-6 strain limit lead to more conservative results. These two alternatives may be valid to support future analysis.

6. CONCLUSIONS

6.1. Conclusions and comments

The presented study was driven with PO and C/WO Units in mind. These units go through several stages throughout their lifecycle, to which different standards apply (DNV 2.7-3 for PO Units and ISO 13628-7 for C/WO Units). The load formulae provided by those standards concern different stages, therefore only the material resistance criteria were compared.

This study aimed towards a comparison of the different structural capacities assessed in accordance to DNV 2.7-3, ISO 13628-7 and EC 3, with particular insight in the criteria provided by the first two. Another component of this study consisted in addressing the influence of the different FE modelling techniques on the FEA results and subsequent structural capacities.

It was explained how unclear some standards are regarding FEA, especially DNV 2.7-3. This thesis alerts for this problem, showing how different interpretations can lead to different results. Related to this, part of the complexity of FE tools, mainly during the modelling stage, and different modelling techniques were explained.

A rotation limit provided by the real scale test article was used to retrieve the maximum bending capacity according to DNV 2.7-3. That rotation limit consisted in a formula specific to the studied geometry. Observations were made regarding two broader alternatives to this limit: a 1st order analysis with EC 3 Part 1-6 strain limit and the limit load obtained by a 2nd order analysis without strain limit.

This document does not have broadness nor enough statistical value required to provide a definitive analysis methodology. Despite that, it can provide some guidelines and directions for further analysis and studies on this subject. In general, and for traditional steel structures:

Element types:

- Beam elements will greatly decrease the analysis duration, but these only assess the members' capacity, therefore their use has to be carefully thought of. There are some conditions where using beam elements might be acceptable: when the failure mode is related with the members' capacity; when hand calculation is performed for every connection; or when using a local sub-model with shell or solid elements in the connections' region.

- Using shell or solid elements to model generic steel profiles will lead approximately to similar results. The main differences are related to the analysis time, which is longer in solid models due to the greater amount of information provided. This extra information may be useful for some design checks, such as the ISO 13628-7 global strain check.

Analysis type:

- The use of a 1st or 2nd order analysis will lead to different results, mainly regarding the rotation capacity, which is greater in the 1st order analysis. Nonlinear analysis tends to achieve slightly lower structural capacities. None the less, within a small deformation range the results are similar. Using a 2nd order analysis is always advised, especially for materials with strain hardening. If a material without strain hardening is used, then a 2nd order analysis might be neglected, although a deformation limit should be imposed.

Material curve:

- The use of elastic material curves, in accordance to the criteria of the studied standards, will lead to overly conservative capacity results;
- Affecting the material curve with the 1 0.85 safety factor provided by DNV 2.7-3 is not the best alternative. It will lead to similar structural capacities as using a perfectly elastic plastic material curve and multiplying the capacity obtained by 0.85, but the rotation and displacements obtained by the first method are greater than what to expect from reality.

Standards criteria:

- ISO 13628-7 perfectly elastic plastic analysis methodology is more conservative than the plastic with strain hardening analysis methodology;
- DNV 2.7-3 structural criterion provides more conservative results than those obtained through ISO 13628-7. However, and according to ISO 13628-7, the structural capacity shall also be affected by a design condition factor. If this factor is applied, then, for normal operations, ISO 13628-7 leads to more conservative structural capacities;
- It was noticed that there is a major challenge assessing the capacity of a structure through FEA, where local effects and peak stresses are captured, in accordance with standards that may be intended for hand calculations procedures, which are not affected by peak stresses, such as DNV 2.7-3.

6.2. Further work

Further work can be developed in order to better understand and standardize an analysis methodology that is able to cover the different requirements coming from different standards. To achieve that it is suggested to:

- Expand the presented study, with different sections;
- Study different load conditions other than pure bending moment;
- Expand the presented comparison to more complex structures, capturing global effects.

7. REFERENCES

ABAQUS User's manual - Version 6.13 (2013). Dassault Systèmes, Simulia.

ASME Boiler and Pressure Vessel Code Section VIII, Division 2 (2011) "Rules for Construction of Pressure Vessels – Alternative Rules". The American Society of Mechanical Engineers.

Bai, Y., Bai, Q. (2010). "Subsea Engineering Handbook". 1th edition, ELSEVIER.

Bathe, K.J. (1982). "Finite Element Procedures in Engineering Analysis". Prentice Hall.

Bathe, K.J. (2014). "Finite Element Procedures". 2nd Edition, Watertown, MA: Klaus-Jürgen Bathe.

Bathe, K.J., Wilson, E.L. (1976). "Numerical Methods in Finite Element Analysis". Prentice Hall.

Chapelle, D., Bathe, K.J. (2011)."The Finite Element Analysis of Shells – Fundamentals" 2nd Edition, Springer.

DNV 2.7-3 (2011). "Portable Offshore Units". Det Norske Veritas.

DNV-RP-C208 (2013). "Determination of Structural Capacity by Non-linear FE analysis Methods". Det Norske Veritas.

European Committe for Standardization (2005). "EN 1993-1-1:2005 - Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings". CEN, Brussels.

European Committe for Standardization (2005). "EN 1993-1-8:2005 - Eurocode 3: Design of steel structures - Part 1-8: Design of joints". CEN, Brussels.

European Committe for Standardization (2007). "EN 1993-1-6:2007 - Eurocode 3: Design of steel structures - Part 1-6: Strength and Stability of Shell Structures". CEN, Brussels.

FMCTI (2015). Internal data. Kongsberg.

Healy, B.E. & Zettlemoyer, N. (1993). "In-Plane bending strength of circular tubular joints". Proceedings 5th International Symposium on Tubular Structures, Nottingham, UK. pp. 325-344.

Infield Systems (2012), "Offshore Outlook".

ISO 13628-7 (2006). "Petroleum and Natural Gas Industries - Design and operation of Subsea Production Systems - Part 7: Completion/Workover riser systems". International Organization for Standardization.

Petrowiki@(n.a.)."http://petrowiki.org/images/0/0e/Devol2_1102final_Page_359_Image_0001.png". [Viewed at 04/06/2015]

Rigzone@ (n.a.)." http://www.rigzone.com/training/insight.asp?i_id=326". [Viewed 04/06/2015]

Simões da Silva, L., Gervásio, H. (2007). “Manual de Dimensionamento de Estruturas Metálicas: Métodos Avançados”. 1th edition, CMM.

Subsea World News@ (2015).“http://subseaworldnews.com/wp-content/uploads/2015/03/GE-Lands-850-Mln-Equipment-Deal-for-OCTP-Ghana-480x294.jpg”. [Viewed 04/06/2015]

Zienkiewicz, O.C., Taylor, R.L. (2005). “The Finite Element Method for Solid and Structural Mechanics”. 6th edition, Butterworth-Heinemann.

Zienkiewicz, O.C., Taylor, R.L., Nithiarasu, P. (2005). “The Finite Element Method for Fluid Dynamics”. 6th edition, Butterworth-Heinemann.

Zienkiewicz, O.C., Taylor, R.L., Zhu, J.Z. (2005). “The Finite Element Method: Its Basis and Fundamentals”. 6th edition, Butterworth-Heinemann.

[1] (n.a.). “The History of Offshore Oil and Gas in the United States (Long Version)”. Staff Working Paper No. 22. National Commission on the BP Deepwater Horizon Oil Spill and Offshore Drilling.

[2] (n.a.). “Appendix O: The Origins of the Finite Element Method”. Aerospace Engineering Sciences, University of Colorado.