Study of an Ultra Large Container Ship Under
Pure Vertical Bending Moment

Joana Patrícia Sousa Franco

Thesis to obtain the Master of Science Degree in
Naval Architecture and Ocean Engineering

Supervisor: Prof. José Manuel Antunes Mendes Gordo

Examination Committee
Chairperson: Prof. Carlos António Pancada Guedes Soares
Member of Committee: Prof. Yordan Ivanov Garbatov
Supervisor: Prof. José Manuel Antunes Mendes Gordo

December 2020
“All you need is Faith, Trust, and a little bit of Pixie Dust.”

“Tudo o que precisas é de Fé, Confiança e um pouco de Pó de Fada.”

- Walt Disney
Acknowledgements

First and foremost, I would like to express my gratitude to Professor Carlos Guedes Soares, as coordinator of the Master of Science Degree in Naval Architecture and Ocean Engineering, for the opportunity to take this course.

I must thank Professor José Manuel Gordo for his guidance, comprehension, insight and, especially, for his knowledge. His devoted presence and interest to discuss and develop the thesis topic has made this dissertation possible. I also want to thank to Mesut Tecgoz for time spend explaining me critical tools to this dissertation.

I would like to thank my closest friends outside of the university, Ana Rita Santos, Catarina Gabriel, Rita Marques, Ridhi Nautamlal, César Correia, Rita Evangelista, Paulo Caseiro, Mariana Machado and Inês Reis for their encouragement and motivation throughout the dissertation and course. I must also thank my course colleagues that I pretend keep for life, Ana Costa, Hugo Loureiro, Gabriel Marreiros, Eduard Lotovskiy, João Escabelado, Jorge Cabral, Andrea Lasheras and Ricardo Nunes for their companionship and assistance for this dissertation.

A very special and deep gratitude to my parents, Paula and Filipe Franco, for all their support and patience: mom, dad, you can finally sigh with relief, I think this is it. To my cousin Matilde Sousa that lives with me: after everything you have been through and continue to do, you are a warrior and like a younger sister to me, I love you. Finally, thanks to the newest members of the family: Zazu and Duquesa: although you make a lot of noise, you are very cute, intelligent, and loving, I adore you two.
Os navios porta-contentores possuem grandes aberturas no convés e, em consequência, apresentam um comportamento na resistência à flexão e torção diferente quando comparado a outro tipo de navios. Para perceber o comportamento deste tipo de navios à flexão vertical, é analisado neste trabalho um modelo de uma secção transversal real de um navio porta-contentores de 10.000 TEUs à flexão vertical pura nas condições de alquebramento e contra-alquebramento. Para tal, foi criado um modelo em elementos finitos usando o software ANSYS e a linguagem APDL disponível no mesmo. Os valores obtidos através da análise do modelo de elementos finitos no ANSYS foram analisados e comparados com a teoria linear elástica da viga-navio. Finalmente, os valores obtidos pela análise do modelo de elementos finitos foram comparados com os obtidos pela teoria da flexão de vigas demonstrando que o comportamento à flexão deste tipo de navios é pouco afetado pela instabilidade elasto-plástica local devido à muito baixa esbeltez dos painéis sob maior esforço, em particular da braçola da escotilha. São estudadas as consequências de diferentes níveis de imperfeições iniciais na resistência à flexão deste porta-contentores.

A aprovação do projeto estrutural pelas entidades regulatórias e em particular pelas normas da International Association of Classification Society deve ser feita não nas condições de construção mas deduzia a margem para corrosão, pelo que foi analisado o modelo de elementos finitos após deduzida a margem de corrosão comparando os resultados com os do navio nas condições geométricas de construção. Esta dissertação propõe ainda condições fronteira a aplicar a modelos de pequena dimensão longitudinal que permitem identificar efeitos secundários associados à flexão pura da viga navio e que não podem ser detetados por outras condições fronteiras normalmente utilizadas na análise dos elementos finitos deste tipo de estrutura.
Abstract

Container ships have large deck openings. As a result, they have a different structural behaviour under bending and torsion of the hull girder resistance when compared to other types of vessels. This work studies a model of a real-world midship cross-section of a 10 000 TEUs capacity container ship, when subjected to a pure vertical bending moment in sagging and hogging conditions. A finite element model was created using the ANSYS software and the APDL language to study the structural behaviour of the container ship under vertical bending moment. The finite element model's results were analysed and compared with the linear elastic bending theory of the hull-girder. Through the prismatic beam theory, demonstrating that container ships' bending behaviour is slightly affected by the local elasto-plastic instability. Such conclusion results from the very low slenderness of the panels under higher stresses, particularly the hatch coaming. The consequences of different levels of initial imperfections on the bending strength of container ships are also studied.

The approval of the structural design by the regulatory authorities and, in particular, by the International Association of Classification Society's standards must be complied, not under construction conditions, but by deducting the margin for corrosion. Hence, two finite element models were analysed, in as-built condition and after removing the corrosion margin. A comparison between the two scenarios was made considering geometric construction conditions versus the same conditions but deducting the corrosion margin. Additionally, this master thesis proposes new bending boundary conditions applicable to models of smaller length. These are useful in identifying the secondary effects of pure bending of the ship that cannot be detected with other usual boundary conditions in the finite element analysis of container-ships’ structures.

Keywords

Ultra large container ship; Vertical bending moment; Finite element analysis; Ultimate strength
# Table of Contents

Table of Contents .................................................................................................................. ix
List of Figures ......................................................................................................................... xi
List of Tables .......................................................................................................................... xiii
List of Acronyms .................................................................................................................... 14
List of Symbols ....................................................................................................................... xv

1  Introduction ....................................................................................................................... 1
   1.1  Overview ....................................................................................................................... 2
   1.2  Motivation and Contents ............................................................................................. 2
   1.3  Objectives .................................................................................................................... 3
   1.4  Document Structure .................................................................................................... 4

2  Literature Review .............................................................................................................. 6

3  Theorical Concepts ........................................................................................................... 13
   3.1  Motion and Sectional Load Components .................................................................... 14
      3.1.1  Vertical Bending Moment .................................................................................. 15
      3.1.2  Design Vertical Bending Moments .................................................................... 16
   3.2  Static Analysis - Beam Theory ................................................................................. 17
   3.3  Finite Element Model ................................................................................................. 20

4  Description of the ULCS ................................................................................................... 23

5  Finite Element Model ....................................................................................................... 28
   5.1  Geometry of the Structural Model ............................................................................. 29
   5.2  Mesh Creation .............................................................................................................. 30
   5.3  Initial Imperfections .................................................................................................... 32
   5.4  Boundary Conditions .................................................................................................. 35
   5.5  Static Analyses in FEM ............................................................................................. 38
   5.6  Corrosion Effects ........................................................................................................ 42

6  Results Analysis ............................................................................................................... 44
   6.1  Design Vertical Bending Moment ............................................................................... 45
   6.2  Beam Theory .............................................................................................................. 46
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.3</td>
<td>FEM Geometry</td>
<td>46</td>
</tr>
<tr>
<td>6.4</td>
<td>Vertical Bending Moment on FEM</td>
<td>47</td>
</tr>
<tr>
<td>6.5</td>
<td>Influence of Initial Imperfections</td>
<td>56</td>
</tr>
<tr>
<td>6.6</td>
<td>Corrosion Margin</td>
<td>61</td>
</tr>
<tr>
<td>7</td>
<td>Conclusions</td>
<td>64</td>
</tr>
<tr>
<td>8.1</td>
<td>Achievements</td>
<td>65</td>
</tr>
<tr>
<td>8.2</td>
<td>Further Work</td>
<td>66</td>
</tr>
</tbody>
</table>
List of Figures

Figure 1.0.1 - Capacity of CS in seaborn trade from 1980 to 2019 (in million dwt) (taken from [2]) ................................................................. 3
Figure 1.0.2 - Container ship evolution (taken from [5]) ................................................................. 3
Figure 3.1 – Definition of: (a) motion components; (b) sectional load components (taken from [34]) ................................................................................. 14
Figure 3.2 – Wave crest for: (a) Hoggıng Condition; (b) Sagging Condition (adapted from [35]) 15
Figure 3.3 -Hull girder bending for: (a) Hoggıng Condition; (b) Sagging Condition (adapted from [35]) ................................................................. 15
Figure 3.4 – Moment-curvature relationship for rectangular cross-section and stress distribution .............................................................................. 18
Figure 3.5 - A 2-D domain partial mesh composed by 4 elements and 9 nodes ........................................ 20
Figure 3.6 - FEM elements: (a) 1-D elements; (b) 2-D elements; 3-D elements ................................................. 21
Figure 3.7 - Mesh size: (a) fine mesh; (b) gross mesh.................................................................................. 21
Figure 3.8 – Poorly dimensioned (left side) and well dimensioned (right side) mesh ............................... 21
Figure 4.1 – Transverse midship section of the ULCS (all dimensions in mm) (taken by [41] [32]) ................................................................. 25
Figure 4.2 - Transverse midship section of the ULCS (all dimensions in mm) (taken by [41]) ... 27
Figure 5.1 - Object ship FE model with and without the transverse frames...........................................29
Figure 5.2 – Initial FE model with (right side) and without transverse frames (left side) ................. 30
Figure 5.3 - Free (a) and mapped (b) meshes (adapted from [43]) ................................................................. 31
Figure 5.4 - SHELL93 geometry (taken from [43]) .................................................................................. 31
Figure 5.5 – Mesh of the ULCS ........................................................................................................... 32
Figure 5.6 - Initial deflection example (adapted from [44]) ................................................................. 33
Figure 5.7 - Representation of the three first buckling modes (adapted from [45]) ........................................ 33
Figure 5.8 - Initial deflection of independent stiffeners due to angular rotation (adapted from [46]) ................................................................. 35
Figure 5.9 - Example of an unstrained and strained bending curve of a beam (adapted from [47]) ........ 35
Figure 5.10 – Boundary conditions at fore and aft section for master nodes method ................. 36
Figure 5.11 – Boundary Conditions on fore and aft sections ................................................................ 37
Figure 5.12 – BC implemented (a); BC by master nodes (b) ................................................................. 38
Figure 5.13 - Newton-Raphson’s method (taken from [50]) ................................................................. 39
Figure 5.14 - Newton-Raphson’s method problem (adapted from [51]) ..................................................... 40
Figure 5.15 - The Arc Length Method (taken from [53]) ......................................................................... 40
Figure 5.16 - (a) Snap–Through, (b) Snap–Back, (c) Through and Snap–Back instabilities (taken from [53]) ............................................................................. 41
Figure 5.17 - Typical convergence graph ............................................................................................... 41
Figure 5.18 – Static analysis using Newton and arc-length methods (ANSYS code) ......................... 42
Figure 5.19 – Corrosion addition in mm (taken from [55]) ..................................................................... 43
Figure 6.1 – Progressive collapse behaviour for sagging and hogging conditions .......................... 48
Figure 6.2 – Convergence solution for hogging bending moment ......................................................... 50
Figure 6.3 – Convergence solution for sagging bending moment Error! Bookmark not defined.
Figure 6.4 – Deformation of FEM when applied hogging moment .................................................. 51
Figure 6.5 – x component of stress for hogging condition .......................................................... 51
Figure 6.6 – x component of elastic strain for hogging condition .............................................. 52
Figure 6.7 – x component of plastic strain for hogging condition ............................................. 52
Figure 6.8 – z component of plastic strain for hogging condition ............................................. 53
Figure 6.9 - Deformation of FEM when applied sagging moment ............................................. 53
Figure 6.10 – x component of stress for sagging condition ........................................................ 53
Figure 6.11 – z component of stress for sagging condition ........................................................ 54
Figure 6.12 – x component of elastic strain for sagging condition ............................................. 54
Figure 6.13 – x component of plastic strain for sagging condition ............................................. 55
Figure 6.14 – z component of plastic strain for sagging condition ............................................. 55
Figure 6.15 – x component of stress using a higher scaler factor ............................................. 55
Figure 6.16 – Moment-curvature curve for different levels of initial imperfections .................. 56
Figure 6.17 – Convergence solution for Hogging in severe level of initial imperfections .......... 58
Figure 6.18 – x component stress for hogging condition (0 Imperfections) .............................. 58
Figure 6.19 – x component stress for hogging condition (average level of initial imperfections) 58
Figure 6.20 – x component stress for hogging condition (severe level of initial imperfections) .. 59
Figure 6.21 – x component of elastic strain for hogging condition (no initial imperfections) ...... 59
Figure 6.22 – x component of elastic strain for hogging condition (average level of initial imperfections) ............................................................................................................ 60
Figure 6.23 – x component of elastic strain for hogging condition (severe level of initial imperfections) ............................................................................................................ 60
Figure 6.24 – z component of plastic strain for hogging condition (average level of initial imperfections) ............................................................................................................ 60
Figure 6.25 – z component of plastic strain for hogging condition (severe level of initial imperfections) ............................................................................................................ 61
Figure 6.26 – Moment-curvature curve before and after removing the corrosion margin .......... 61
List of Tables

Table 3.1 – Coefficient depending on the type of vessel ................................................................. 17
Table 4.1 - Main dimensions of the ULCS ..................................................................................... 24
Table 4.2 - Plates and stiffeners’ thicknesses of the ULCS ............................................................. 25
Table 4.3 - Material characteristics of the ULCS .......................................................................... 26
Table 5.1 – Boundary Conditions on fore and aft sections ............................................................ 37
Table 6.1 – Characteristics of the ULCS .......................................................................................... 45
Table 6.2 – Design still water and wave induced bending moment ................................................ 45
Table 6.3 – Section properties ......................................................................................................... 46
Table 6.4 – Yielding and plastic moments, and maximum curvature for the ULCS ......................... 46
Table 6.5 – FEM section properties ................................................................................................. 46
Table 6.6 – Relative error of FEM section properties ...................................................................... 47
Table 6.7 – Beam theory and FEM results in elastic regime in hogging condition ......................... 48
Table 6.8 - Beam theory and FEM results in elastic regime in sagging condition ......................... 48
Table 6.9 – Plastic moment by beam theory and maximum plastic moment achieved for hogging and sagging conditions .......................................................................................... 49
Table 6.10 – Hogging results before and after removing the corrosion margin .............................. 62
Table 6.11 – Sagging results before and after removing the corrosion margin .............................. 62
Table 6.12 – Initial yielding moment without corrosion addition vs IACS ....................................... 62
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Full Form</th>
</tr>
</thead>
<tbody>
<tr>
<td>DNV</td>
<td>Det Norske Veritas</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Model</td>
</tr>
<tr>
<td>IACS</td>
<td>International Association of Classification Societies</td>
</tr>
<tr>
<td>IMO</td>
<td>International Maritime Organization</td>
</tr>
<tr>
<td>TEU</td>
<td>Twenty-Foot Equivalent Unit</td>
</tr>
<tr>
<td>ULCS</td>
<td>Ultra Large Container Ship</td>
</tr>
</tbody>
</table>
## List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_S$</td>
<td>Still water bending moment along the ship</td>
</tr>
<tr>
<td>$M_W$</td>
<td>Wave induced bending moment along the ship</td>
</tr>
<tr>
<td>$k_{sm}$</td>
<td>Distribution factors for still water moment along the ship</td>
</tr>
<tr>
<td>$k_{wm}$</td>
<td>Distribution factors for wave induced moment along the ship</td>
</tr>
<tr>
<td>$M_{SO}$</td>
<td>Design still water bending moment amidships</td>
</tr>
<tr>
<td>$M_{WO}$</td>
<td>Design wave induced bending moment amidship</td>
</tr>
<tr>
<td>$C_W$</td>
<td>Wave load coefficient</td>
</tr>
<tr>
<td>$C_B$</td>
<td>Coefficient block</td>
</tr>
<tr>
<td>$v$</td>
<td>Design speed</td>
</tr>
<tr>
<td>$M$</td>
<td>Bending moment in elastic regime</td>
</tr>
<tr>
<td>$M_y$</td>
<td>Initial yielding moment</td>
</tr>
<tr>
<td>$M_P$</td>
<td>Plastic moment</td>
</tr>
<tr>
<td>$E$</td>
<td>Young Modulus</td>
</tr>
<tr>
<td>$I$</td>
<td>Moment of Inertia about the neutral axis</td>
</tr>
<tr>
<td>$I_{zz}$</td>
<td>Second moment of area</td>
</tr>
<tr>
<td>$d\nu$</td>
<td>Slope of the deflection curve</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Curvature radius</td>
</tr>
<tr>
<td>$k$</td>
<td>Curvature</td>
</tr>
<tr>
<td>$k_{Max}$</td>
<td>Maximum curvature</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$h_{NA}$</td>
<td>Neutral Axis Height</td>
</tr>
<tr>
<td>$Z_D$</td>
<td>Section Modulus about the deck</td>
</tr>
<tr>
<td>$Z_K$</td>
<td>Section Modulus about the keel</td>
</tr>
<tr>
<td>$\sigma_y$</td>
<td>Yield stress</td>
</tr>
<tr>
<td>$w_0$</td>
<td>Maximum amplitude of initial deflection</td>
</tr>
<tr>
<td>$a$</td>
<td>Distance between transverse frames</td>
</tr>
<tr>
<td>$b$</td>
<td>Distance between longitudinal stiffeners</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Slenderness parameter</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Chapter 1 provides a brief overview of the dissertation that has been developed. Before exposing the overview and establishing the objectives of the thesis, a summary of the dissertation, motivations and contents are brought up. At the end of the chapter, the thesis’ structure is briefly provided.
1.1 Overview

Container vessels are continuously increasing their capacity and hence their size. Consequently, container ships and especially ultra large container ships (ULCS) acquire low torsional and bending resistance due to the size of their deck openings, which are very large when compared with other conventional ships. Thus, it is important to realise some studies about this type of ships when they are subject to vertical bending moment related to sagging and hogging conditions.

Therefore, it was created a finite element model (FEM) of the midship section of a real ultra large container ship of 10,000 TEUs (Twenty-Foot Equivalent Unit), using ANSYS software. The 3D model is composed by plates, longitudinal stiffeners and frames. To simplify the model design, the bulb flat stiffeners were modelled by equivalent thicknesses. The mesh applied in this structural analysis is made by quadratic shell elements holding eight nodes, considering 2D plates. Boundary conditions different from the usual were carefully designed, and the initial imperfections were introduced. A comparison about different levels of initial imperfections was also presented. Finally, the FE model without the corrosion addition was created to verify if it fulfils with the International Association of Classification Society standards for structural strength.

The theoretical and analytical figures were analysed, and some conclusions were drawn.

1.2 Motivation and Contents

In the last decades, a host of political, economic, and technological changes clearly accelerated the trend toward globalization. Consequently, an increasing number of countries abandoned their old policies and opted for free trade, expanding the transport of people and goods around the world. At that time, land transport was carried out by navigable roads and rivers, with coastal ships used to transport heavy goods by sea since canals had not yet been built. This improved transportation infrastructure where canals were the first technology to allow bulk raw materials to be easily transported across the world, (cheaper and faster than before) [1].

The rise in commerce is currently driving the expansion of the global market, underscoring the need for an efficient transport system. Hence, the industry is being shaped by a cost reduction strategy, which minimizes costs per TEU (see Figure 1.0.1) [2].

Additionally, as shown in Figure 1.0.2, the increase in vessel capacity has led to an increase in vessel width leading to an increase in torsional and wave bending which can cause big damages or even break the vessel like the MOL comfort container ship did in 2013 that suffered a crack amidships in bad weather while hogging. These types of events also occur in tankers as it happened in 1952 where SS Fort Mercer and SS Pendleton vessels broken into two in a gale [3].
So, it is crucial to understand and know the limits of these type of ships to minimize the number of accidents related to these cases. Hereupon, to study the behaviour of this type of vessels classification society norms establish procedures for ship structural analysis to be performed for the midship section. It is where the maximum bending stresses occurs [4].

To perceive the strength characteristics of these ships when submitted to these efforts, the ideal option would be to make a real scale model of the midship section. This way, it would be possible to apply the efforts and see the damages, (if it is the case), generated in the structure. However, such manufacture would be quite expensive and for that reason scientific research choose not to do so. Thus, engineers apply finite element method to analyse the ship hull structure. [6].

1.3 Objectives

The propose of this thesis is to study the behaviour of an ultra large container ship of 10 000 TEUs when submitted to pure bending moment (sagging and hogging moments) using the finite element method. For this purpose, the theoretical approach of the prismatic beam theory which considers the hull girder as a simple beam is estimated. The finite element model with initial imperfections and boundary
conditions applied is created and the results of the moment-curvature curve concerning the elastic regime are analysed and compared with the scores from beam theory. Additionally, the model structure without imperfections and for two different levels of initial imperfections (average and severe levels) are compared. The FE analysis for the structure model under construction conditions and after deducting the margin for corrosion are compared. For the last condition, is verified if the model structure still complies with the IACS norms.

To summarize, the main objectives are:

- Verify if the FE model for elastic regime is in accordance with the Prismatic Beam Theory;
- Verify if the FE results fulfils with the IACS standards;
- Compare and analyze the moment-curvature curves obtained by FE method for different levels of initial imperfections and with no imperfections;
- Compare and analyze the moment-curvature curves obtained by FE analysis with and without corrosion addition;
- Draw conclusions.

1.4 Document Structure

This dissertation is divided into seven chapters.

This chapter (Chapter 1) introduces the subject studied in this dissertation. After briefly summarizing the research work conducted under the scope of this dissertation, is presented an overview of the subject and the motivations underlying this research work. Finally, a list of the main objectives and description of the document's structure is provided.

In Chapter 2, related literature about some existing methods and previous works developed on this research’s topics of interest are revised. This helps understanding the interest in this subject and the objectives of the studies conducted.

Chapter 3 is an overview of the theoretical concepts relevant to better understand the subject. The rudimentary concepts about motion and section load components, such as vertical bending moments are explained. Finally, a detailed explanation about beam theory and finite element model analysis are presented.

Chapter 4 starts with a description of ultra large container ships in general. Then, the main dimensions of the ULCS of 10 000 TEUs under study is provided. The characteristics of the elements which form the structure are described, while some drawings of the midship section are provided.

Chapter 5 first explains the geometry of the structural ship in study in ANSYS software, detailing all the steps done to accomplish the final structure. After that, a description about how the mesh is created is presented. Finally, the boundary conditions are explained, including the static analysis mentioning the arc-length and Newton methods.
Chapter 6 shows the final main results obtained by subjecting the model to sagging and hogging moments. Quantitative results were obtained from the ANSYS software, which allowed us to calculate scores to validate the model. The Chapter closes with a detailed analysis and discussion about the results obtained in the ANSYS software, where the structural model with no imperfections and with two different levels of initial imperfections are considered. The FEM analysis with respect to the moment-curvature curves for the structural model with and without corrosion addition is realized. Some illustrations of the structural model behaviour are provided along this chapter.

The dissertation closes with Chapter 7, where a review and discussion of the main conclusions of this dissertation are made. The future work regarding this subject is discussed.
Chapter 2

Literature Review

This chapter makes a historical summary about the subject under study and exposes some of the studies done so far related to this dissertation where shows the level of development reached by now. It provides an overview of current knowledge and allows to identify relevant theories, methods, and gaps in the existing research. It also includes substantive findings as well as theoretical and methodological contributions to this topic.
It was in 1871 that Edward James Reed showed his concern about neglected calculations divisions in ship building science. Although the methods ensuring ease and control of ship movement had been studied in an elaborate way, few naval architecture writers had pursued the subject to its legitimate and necessary development by that moment. Although, modern events at the time, such as the introduction of steel in shipbuilding, showed weaknesses in many ships, pointing to the need for further research in this direction. Therefore, this has triggered a few consequential studies about this issue [7].

One of these studies stated that the bending of a vessel can be simplified to a cylindrical beam exposed to bending [8]. This had greatly simplified calculations concerning to the ability of a vessel to support weather rough conditions at sea.

For the first time, in 1969, a paper from Society of Naval Architects and Marine described the application of the finite element method to ship structures [9]. It started with the development of a package of computer programs, starting with basic information about the ship and the loading conditions, passing through the structural analysis of the entire vessel, followed by local analyses to obtain a detailed stress distribution. An account of the generation of data, both geometrical and structural, that leded to the complete definition of the ship was achieved.

In 1973, a practical method of overall finite element analysis of a ship structure based on the modern theory of a beam was proposed [10]. It analysed the shear deformation of a ship structure based on the finite element formulation and concluded that computer programs for overall ship structural analysis would be very practical with respect to computer time and labour from the design point of view. In short, it was concluded that combined use of the present computer program at the basic design stage and the conventional general-purpose software at the detail design stage become ship structural analysis effective and powerful. Then quickly, Royal Institution of Naval Architects developed a design system capable of application to large complex ship structures. In a nutshell, the computer-based system consisted of a general model of automatic analysis of finite elements linked to an optimization routine. The key to the document was the use of the finite element method in design and the efficient optimization of major design problems [11].

Finally, regular shaped elements were used in the calculation of stress intensity factors of elastic fracture mechanics. Examples of the plane eight noded isoparametric element still included the constant strain and rigid body motion modes. Stress intensity factors were then calculated for several plane strain and three-dimensional problems. Still the structural behaviour of open ships was not matched with the analytical theory typically used. Considerable inconsistencies could be found between theory and reality. So, by closer interpretation of results from finite element calculations it was possible to explain the snaking phenomenon of open vessels where bending stresses of considerable magnitude occurred in the deck structure. Hither, the ship was finally treated as a thin-walled prismatic beam and the results presented a huge and enormous success [12].

Then, it was implemented a family of new element models in discrete structural analysis. These models consisted of finite number of small rigid bodies connected with springs distributed over the contact area of two neighbouring bodies. In general, the size of stiffness matrices of these elements was smaller than those of conventional finite elements which brought to a reduction of computing time [13].
Later, Jensen et al. concluded that for container ships the nonlinear theory predicted significant contributions from the quadratic terms to the peak values of the bending moments, while for the tanker, the effects of the nonlinearities were small. For the container ships the effect of the nonlinearities was the increase of the sagging bending moment and at the same time the decrease of the hogging bending moment. The numerical results also showed when flexibility or springing was considered, the variance of the wave induced midship bending moment increases due to nonlinear effects. Therefore, seemed that was important to consider the nonlinear effects to predict spring in container ships, whereas for the tanker the nonlinear terms were negligible influence showing that linear theory should be sufficiently accurate [14].

Steel structures are typically constructed by welding, and the structural members are consequently followed by initial imperfections due to welding such as welding residual stresses and distortions. That brought to a review about the equations for the design of ship plate. The effect of several parameters was quantified showing that a design method should account explicitly for plate slenderness, residual stresses, initial distortions, and boundary conditions [15].

Tetsuya Yao et al. proposed a new simple method to simulate the progressive collapse behaviour of a ship’s hull girder subjected to vertical bending moment. In this method, the cross section of a hull girder was divided into minor elements composed of a stiffened and associated plating. This new method slowly increases the curvature which was applied to the hull girder assuming that the plane cross section remains plane. At each incremental step, the tangential flexural rigidity of the cross section was evaluated using the tangential slope of the average stress-strain curves of the elements as well as the incremental bending moment due to the curvature increment. It was found that the full plastic bending moment cannot be achieved due to buckling of the deck plates under sagging condition and that of the bottom and inner bottom plates under hogging condition. It was also found that the maximum vertical bending moment carried by the cross section under sagging bending moment condition is 20% lower than that under hogging bending moment condition for the ship in analysis [16].

In 1992 a new bending theory of thin-walled girders was calculated by introducing the concept of stiffness and mass parameters as average quantities. The problem was reduced to the girder cross-section plane where the stiffness and mass properties were determined from equivalence of girder deformation energy and inertia work making this modern beam theory more accurate than the classical one. Hence, the finite element analysis was applied, using orthotropic strip elements that are necessary for the ship structures [17]. One year later, B.W.Golley et al. introduced a new finite strip-element method for the dynamic analysis of orthotropic plate structures [18].

A method based on a simplified approach was presented to estimate the ultimate longitudinal strength of a hull girder. The work was developed to provide a computationally inexpensive procedure to assess the ultimate longitudinal strength with adequate accuracy and predicted the behaviour of the hull girder under predominant longitudinal bending with quite accurate. It was found that the ultimate bending moment in the upright position is greater than the moment at small angles of heel, no matter if the ship is in hogging or sagging. The deck stiffeners made in bars did not had much flexural-torsional rigidity and the calculated tripping stress were lower than the flexural buckling stress of the stiffener with
associated plate. That led to a very high reduction of the ultimate bending moment in sagging when compared with the moment in hogging, (where the deck is in tension) [19].

One year later, the ultimate collapse of the midship section of tankers and container ships under combined vertical and horizontal bending moments were determined by Gordo and Guedes Soares [12] by an approximate method where were able to construct the moment-curvature relation for the hull girder. It was concluded that the exponent of the interaction formula for containerships is different for sagging and hogging. However, the disperse of the results relative to the interaction formula is much higher on tankers when the bending moment varies from pure sagging to pure hogging.

From that time on, several studies were applied in shipbuilding construction, such as icebreaking ship structures’ analyses, modelled corrugated shell, etc [20]. Up to 2001, static analysis of stiffened shells was carried out using an eight-noded isoperimetric element for the shell and a three-noded curved beam element for the stiffener. So, an improvement over the degenerated shell concept was made by a modified technique to analyse the shell. The stiffness matrix of the curved beam element was generated irrespective of its position and orientation within the shell element where, then, the stiffness matrix of the stiffener was transferred to all the nodes of the shell element [21].

In 2004 an experiment of four points bending test on a box girder subjected to pure bending moment was considered where initial loading cycles which allows the relief of residual stresses was performed. This experience was chosen to be representative of the conditions that can occur near the midship region of conventional ships. It was concluded that the transverse distribution of the strain at the middle of the box girder shows a loss of efficiency of the stiffened panels under tension in regions distant from the main vertical plating and at high applied moment. Also, when initial loading cycles were applied producing a large residual stresses relief, was obtained in the panels under tension some loading memory on the structure influencing the moment curvature relationship. The geometry of the buckled structure compared well the distribution of the residual plastic strain after collapse and unloading [22].

Some concerns about the construction of ULCS began to be raised. The main concern is related to a possible increase in the global hull girder loads as consequence of the increased hull flexibility as well as the possible decreases of fatigue life. The vertical bending moment (VBM) response of an ULCS was analysed numerically and compared with old experiments, having a good agreement with the experimental results in general. For the flexible response of the ship, a nonlinear time domain method based on strip theory method was used where the experimental VBM had a mean sagging value partly due to the steady speed of the vessel in still water and due to the dynamic effects of waves. The VBM peaks showed less asymmetry in moderate seas probably due to the long parallel middle body of the vessel. In high seas the distribution of the peaks was clearly asymmetric, and the asymmetry depended more on the speed of the vessel than on the significant wave height. The numerical VBM also had a mean sagging value, however less than the experimental value, probably because they consider only for the dynamic effect due to waves [23].

In 2017 a paper focused on the ultimate strength characteristics of a typical ULCS of 10,000 TEU implemented a hull girder under pure vertical bending moment, pure horizontal bending moment and pure torsion and finally, ultimate strength interaction relationships between two load components each
and between three load components. The writer concluded that ultimate strength decreases as the magnitude of initial deformation increases and decreases very quickly when the magnitude approaches or exceeds the corresponding plate thickness. Lastly, the combined loads could result in lower ultimate strength for ship hulls than under pure bending or pure torsion actions [24].

A year later, a very recent paper investigated the ultimate longitudinal strength characteristics of an ultra large container ship through experimental and numerical analysis, and mainly concluded that progressive collapse behaviour of ULCS are similar with middle container ships, but the proportional elastic limit and ultimate strength are very different from each other which are almost multiplied with the cargo capacity, which can provide a reference for the ship structure design of different type of container ships. The model test results showed that the ship cross-section stress along the horizontal neutral axis has a good symmetry, which conforms to the distribution law of normal stress on the beam section under pure bending moment, indicating beam bending theory could be applied to hull girder. In this study was used a similar scaled model for a typical 10,000 TEU container ship under hogging bending moment and numerical analysis for three type of container ships representing middle and ultra large container ships respectively was carried out to investigate the ultimate strength characteristics of ULCS [25].

It is important to identify the main factors that affects the ultimate strength of 3D structures under predominant bending moment. In 2009 an experimental result of the collapse of three box girders made of high tensile steel subjected to pure bending moment was produced. It was concluded that the efficiency of this type of steel was quite good when applied on the box girders subjected to bending moments. The global efficiency was higher than the expected under compression due the material. Also, it was concluded that for actual design practice one may obtain a global efficiency of this type of steel taking as basis the normal mild steel structure [26].

The finite element analysis is the most advanced tool for ultimate strength analysis. The efforts to obtain realistic information on the ultimate capacity of structures are focused on the development and application of effective procedures for modelling of elasto-plastic material behaviour, large displacements, rotations, etc. Although, it requires complex theoretical and numerical methodologies as well as experimental verification and usually requires extremely time-consuming calculations. Various adaptive FEA procedures, combined with mesh refinement techniques, have been created improving the predictive capability for linear and non-linear assessments [27]. As an alternative method to FEM, other methods as the Mesh-Free Method have been used where the advantageous over FEM in some cases, such as moving boundary problem, crack growth with arbitrary and complicated paths, and phase transformation problems are used. However, it is more time-consuming than FEM [28].

The global ultimate strength of a structure is frequently considered in the initial phases of design which makes the local ultimate strength an important part of structural design and analysis. For that reason, the ultimate strength is a critical and fundamental assessment in the design of a ship. In 2015 three different hold models of bulk carrier hull were made in FEM where was difficult to obtain the ultimate strength of a full-scale hull girder by using a static finite element solver due to the huge number of elements with negative stiffness after buckling. However, when using time dependent dynamic analysis, it was easy to obtain the ultimate strength for the three hold models. It was concluded that the effect of
initial imperfections on the hull girder ultimate strength was more significant under the sagging condition than under the hogging condition. Also, the effect of imperfection on ultimate strength in compression is larger than in tension. Hence, the reduction of ultimate strength due to imperfections is larger in sagging condition than in hogging condition [29].

It is very important to predict the ultimate strength of a ship's hull girder in damage scenarios when subjected to asymmetrical bending. When a vessel suffers a damage, the ship strength is reduced which leads to modifications in the ship structural cross-section. Due to asymmetrical bending moment, the neutral axis translates and rotates. Therefore, a study about bending moment-curvature relationship of a container ship in intact and damage conditions, subjected to asymmetrical bending loading was analysed using finite element model and the progressive collapse analysis as defined by the common structure rules (where the hull girder cross-section is divided into several structural components composed by stiffened plates, assuming that the cross-section remains plane as it is progressively loaded and deformed). To improve the common structural rules, an iterative method for global bending moment equilibrium was produced into the solution of common structure rules to account for the effect of the neutral axis rotations along with the neutral axis translations. The study concluded that when the ship hull structure without damages is subjected to an asymmetrical bending moment the finite element analysis and the common structure rules approaches presented a similar bending moment - curvature relationship. Also, the method used to update the progressive collapse analysis defined by common structure rules showed a very similar result with the one of finite element analysis in predicting the neutral axis rotations. It also concluded that, for the container ship structure, the neutral axis rotation has a larger influence on the ultimate strength in the hogging condition than the one in sagging in the case of intact hull and the side damage has a significant influence on the ultimate sagging bending moment [30].

One of the most fundamental strength of ships is the ultimate longitudinal bending strength of a hull girder. Container ships are subjected to large buoyancy forces in the midship, and cargoes are loaded over the length. Therefore, container ships are usually in the hogging condition in still water. Also, the upward local loads due to bottom sea pressure produce convex deflection in the double bottom. Consequently, longitudinal and transverse compressive stresses are generated in the outer bottom plating. These stresses may have a substantial influence on the ultimate longitudinal bending strength of container ships in the hogging condition. So, in 2020, a paper regarding the ultimate hull girder strength of container ships subjected to combined hogging moment and bottom local loads is studied and analysed using nonlinear finite element method for two container ships: one of 8,000TEU and other of 14,000TEU. It was concluded that the ultimate hogging strength of container ships has been achieved immediately after the buckling collapse of the outer bottom for both ships. This resulted from the increased longitudinal force in the outer bottom plate, (following by a reduction of its buckling strength) and due to the low efficiency of the inner bottom which is on the tension side of the local bending of the double bottom. It was also concluded that the longitudinal curvature of the double bottom due to upward local loads causes the tensile stress in the flange of bottom stiffener which have a significant influence on the ultimate strength of the bottom stiffened panel [31].
An extending Smith’s method for pure bending collapse analysis of a ship’s hull girder was developed where the double bottom is idealized as a plane grillage and the rest of the cross section as a prismatic beam. In the Smith’s method, the cross section of the ship is modelled as a combination of stiffened panel elements, and the average stresses strain relationships under uniaxial tension/compression which considers the effect of buckling and yielding are applied to those elements. The method performed elastoplastic large deflection analysis by nonlinear finite element method to obtain the average stress-average strain relationships. It was concluded, among others, that the modelling area of the plane grillage affects the ultimate strength and the post-collapse behaviour. It also concluded that the extended Smith’s method with the extended grillage, gives the approximation of the progressive collapse behaviour and ultimate strength of the hull girders under combined the hogging moment and the water pressure with a reasonable accuracy [32].
Chapter 3

Theorical Concepts

This chapter is essential to understand the following dissertation. It provides a range of subjects, crucial to comprehend the ship behaviour in relation to the weather conditions. It starts by illustrating the motion and sectional load components. Then, it makes a deep and detailed description about the hull girder response according to classification society rules and beam theory. Lastly, a detailed explanation about finite element is brought up.
3.1 Motion and Sectional Load Components

To understand the ship’s behaviour in some conditions, it is crucial to take a brief explanation about motion and sectional load components and how it can influence the structure of the ship. Ship motion is divided into six components with six different degrees of freedom. A vessel can experience these motions while standing still or sailing due to wave crossing, tide change, wind, sea currents and so on.

Ship motions are divided into two different types: rotational motions (oscillatory rotations), and translational motions (oscillatory translations). Motion components are divided in longitudinal, transverse and vertical axes which are known as roll, pitch, and yaw respectively (see Figure 3.1a). The linear vertical motion, linear transverse motion and linear longitudinal motion are the translational motions known as heave, sway, and surge respectively (see Figure 3.1a). Note these movements do not weaken the vessel since it keeps up with the movement all along the ship [33].

Conversely, ship sectional load components do not have the same movement along the vessel: as evidence by Figure 3.1b, part of the vessel is static, and the other is not. The six components of internal sectional loads include three forces: axial force (Fx), horizontal shear force (HSF) and vertical shear force (VSF), and three moments: horizontal bending moment (HBM), vertical bending moment (VBM) and torsional moment (TM) [33].

![Figure 3.1](image)

(a) ![Figure 3.1](image) (b)

Figure 3.1 – Definition of: (a) motion components; (b) sectional load components (taken from [34])

In general, for tankers and container ships which have large open decks, the study of the vertical bending moments: sagging bending moment and hogging bending moment are very important since vessels are constantly subject to vertical loads when sailing in head waves.

As explained in Chapter 2, accidents with container ships and tankers have been reported essentially due to bending phenomena. And that is why is important to make a detailed explanation about vessel’s behaviour when submitted to vertical bending moments. This leads to the next sub-chapter which explains in detail the VBM and its design according to classification society rules.
3.1.1 Vertical Bending Moment

Vertical bending moment is divided in two different cases: hogging bending moment and sagging bending moment.

**Hogging bending moment** occurs when the ship’s buoyancy amidships exceeds the weight due to loading or when the wave crest is amidships as shown in the Figure 3.3a (wave induced bending). In this condition the overall weight is greater near the bow and stern, with buoyancy being larger near amidships. In this situation the keel is in compression and the deck is in tension. In other words, hogging causes the centre to bend upward such as demonstrate in the Figure 3.4a [35].

Unlike hogging bending moment, **sagging bending moment** occurs when the weight amidships exceeds the buoyancy or when two wave’s crests pass through bow and stern as shown in Figure 3.3b. Therefore, sagging occurs when a wave which has the same length as the ship pass at the bow and stern. This causes the middle of the ship to bend down, putting the keel in tension and the deck in compression (see Figure 3.4b) [35].

![Figure 3.2 - Wave crest for: (a) Hogging Condition; (b) Sagging Condition (adapted from [35])](image)

![Figure 3.3 - Hull girder bending for: (a) Hogging Condition; (b) Sagging Condition (adapted from [35])](image)

Therefore, bending is caused by two distinguished phenomena: still water bending, (static bending) and wave induced bending, (dynamic bending). The first one is caused due to unsymmetrical cargo loading over forward and aft of the ship, whereas the second case is caused due to unsymmetrical hydrodynamic wave loading on both extremes of the vessel. Also, mass acceleration forces contribute to dynamic bending moments [35].

In conclusion, these events may happen during loading and unload cargo. That cargo is something that is controllable due to knowledge of engineers and some rules that must be followed by operators. On the contrary, the design of the ultimate strength for hogging and sagging conditions is a huge concern since depends on the level of bend what can cause the hull crack or snap [36]. Due to the properties of the structure of container ships which have large bulbous bow and a salient stern, the beats induced by waves in the hull make the ship responses less strong to sagging conditions.
3.1.2 Design Vertical Bending Moments

The total design vertical moment $M_T$, according to classification society rules [37] is the sum of two components: still water bending moment along the ship, $(M_s)$ and wave induced bending moment along the ship $(M_w)$ which are, respectively:

$$M_s = k_{sm} M_{SO}$$  \hspace{1cm} (3.1)

$$M_w = k_{wm} M_{WO}$$  \hspace{1cm} (3.2)

The distribution factors for still water moment $(k_{sw})$ and for wave induced moment along the ship $(k_{wm})$ are provided respectively by Equations 3.3, 3.4, where $L_{pp}$ is the length of the ship between perpendiculars and A.P. and F.P. means aft and forward peak correspondingly.

$$k_{sm} = \begin{cases} 1 & \text{within } 0.4L_{pp} \text{ amidships} \\ 0.15 & \text{at } 0.1L_{pp} \text{ from A.P. or F.P.} \\ 0.0 & \text{at A.P. and F.P.} \end{cases}$$  \hspace{1cm} (3.3)

$$k_{wm} = \begin{cases} 1 & \text{between } 0.4L_{pp} \text{ and } 0.65L_{pp} \text{ from A.P.} \\ 0.0 & \text{at A.P. and F.P.} \end{cases}$$  \hspace{1cm} (3.4)

Whereas the design still water bending moment amidships, $(M_{SO})$ and the design wave induced bending moment amidship, $(M_{WO})$ for sagging and hogging, respectively, are given by the Equations 3.5, 3.6, below:

$$M_{SO} = \begin{cases} -0.065 C_W L_{pp}^2 B (C_B + 0.7), [kN.m] & \text{in Sagging} \\ C_W L_{pp}^2 (0.1225 - 0.015C_B), [kN.m] & \text{in Hogging} \end{cases}$$  \hspace{1cm} (3.5)

$$M_{WO} = \begin{cases} -0.11\alpha C_W L_{pp}^2 B (C_B + 0.7), & \text{in Sagging} \\ 0.19\alpha C_W L_{pp}^2 B C_B, & \text{in Hogging} \end{cases}$$  \hspace{1cm} (3.6)

Where $\alpha = 1$ for seagoing condition and $\alpha = 0.5$ for harbour and sheltered water conditions (enclosed fjords, lakes and reavers). The wave load coefficient $(C_W)$ depends on the length of the ship and is given by Equation 3.7.

$$C_W = \begin{cases} 0.0792 L_{pp}, & L_{pp} \leq 100 \\ 10.75 - \left(\frac{300 - L_{pp}}{100}\right)^{\frac{3}{2}}, & 100 < L_{pp} < 300 \\ 10.75, & 300 \leq L_{pp} \leq 350 \\ 10.75 - \left(\frac{L_{pp} - 350}{150}\right)^{\frac{3}{2}}, & L_{pp} > 350 \end{cases}$$  \hspace{1cm} (3.7)
And the coefficient block, $C_B$ is given by Katsoulis method:

$$C_B = 0.8217 f L_{PP}^{0.42} B^{-0.3072} T^{-0.6135} v^{0.1721} \tag{3.8}$$

In which $f$ is in function of the type of the ship:

<table>
<thead>
<tr>
<th>Table 3.1 – Coefficient depending on the type of vessel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ro/Ro Reefers</td>
</tr>
<tr>
<td>---------------</td>
</tr>
<tr>
<td>0.97</td>
</tr>
</tbody>
</table>

And $v$ is the typical design speed of the vessel.

### 3.2 Static Analysis - Beam Theory

In static analysis the loads are static or gradually applied at a slow speed. It also assumes that loads do not change direction during the analysis and inertial and damping loading forces due to impact or dynamic loading are neglected.

To get the bending moment-curvature relationship graph, two distinguished parameters are established: linear and non-linear static analysis response for elastic and inelastic deformation, correspondently. When a force is removed, and the structure returns to its original shape, it means that the molecular space did not change, and it is in the elastic regime, (linear response). Otherwise, when a force is removed and the structure does not return to its original shape, it means that the molecular spaces already change, and the structure is now in the elastic/plastic regime (non-linear response), (see Figure 3.4). The straight line represents the linear response, and the curve line represents the non-linear response.

A linear static analysis is applicable to structural problems where stresses remain in the linear elastic range. Herein the model's stiffness matrix is constant, and the solving process is easy compared to a nonlinear analysis on the same model. Non-linear effects (large deformations, elastic-plastic material, etc.) results in a stiffness matrix which is not constant during the load application. As a result, a different solving strategy is required, and modern analysis software is crucial to obtain solutions.
The Euler-Bernoulli beam model is a simplification of the linear elasticity principle that provides a way of measuring the deflection characteristics of the beam under a given load. It requires calculating the load-carrying and deflection characteristics of beams. Additional analysis tools have been developed such as finite element analysis (FEA), but the simplicity of beam theory makes it an important tool in structural analysis. In sum, it is a model for beams behave under axial forces and bending as well. This theory is crucial because it is used to facilitate the hull girder response analyses [38].

In prismatic beam theory, the ship is idealized as a hollow thin-wall box beam. It assumes that the ship has the same cross-section through its entire length and assumes that only the bending moments cause distortions. It is true that this analysis treats the hull girder as a simple prismatic beam, which is a drastic simplification, and if there are large hatches over an extended length, or if there is any major discontinuity in the ship’s cross section, this assumption is grossly in error. But, has many studies demonstrated, in terms of bending moment it makes a good approximation of reality, opposite of what happens in torsion moment which non-prismatic beam theory, (the cross section of beam varies over its length), must be assumed [39].

Linear analysis is an analysis in which the relationship between forces and deformations are linear. It is applicable to problems where the stresses remain in the linear elastic range of the material used and follows Hooke’s law (for small deformations of an object, the displacement of the deformation is directly proportional to the deforming force). Given the differential equation of the basic bending curve of a beam by the Equation 3.9:

\[
\frac{M}{EI} = \frac{d^2v}{dx^2}
\]  

(3.9)

\(M\) is the bending moment, \(E\) is the Young’s modulus, (stiffness of a solid material) and \(dv\) is the slope of the deflection curve along the length \(x\). For small angles of rotation that follows Hooke’s law, a
simplification of Equation 3.9 is acquired: \[ \frac{M}{EI} = \frac{d^2 v}{dx^2} = \frac{1}{\rho} = k \] where \( \rho \) is the curvature radius and \( k \) is the curvature. \( I \) is the moment of inertia about the neutral axis which is given by the Equation 3.10.

\[ I = I_{zz} - Ah_N^2 \tag{3.10} \]

It depends on the sum of the structural elements’ moment of inertia about \( zz \) axis (height), (2\(^{nd} \) moment of area, \( I_{zz} \)), the sum of the structural elements’ areas (\( A \)) and the distance of the neutral axis above the keel (\( h_{NA} \)). The total inertia moment about \( zz \) axis is the summation between the local second moment (\( i \)) and the second moment (\( ah^2 \)) for every element structure, where \( a \) is the area of each element and \( h \) is their vertical height of the centroid. Finally, the distance of the neutral axis above the keel is represented in Eq. 3.11.

\[ h_{NA} = \frac{\sum a_i h_i}{\sum a_i} \tag{3.11} \]

Is now possible to calculate the Bending Moments (sagging and hogging) given by the Equation 3.9, in elastic regime, whereas the final main results are shown in the Results Chapter. Also, the results of the longitudinal elements of the container ship (plates and stiffeners), are shown in detailed in Appendix A. The elastic regime ends and starts the inelastic/plastic regime in the called initial yielding moment (\( M_y \)) given by the Equation 3.12:

\[ M_y = \sigma_y Z \tag{3.12} \]

Where \( \sigma_y \) is the yield stress and \( Z \) is the section modulus (about the deck (\( Z_D \)) or about the keel (\( Z_K \))), given respectively by:

\[ Z_D = \frac{I}{h_D - h_{NA}} \tag{3.13} \]

\[ Z_K = \frac{I}{h_{NA}} \tag{3.14} \]

Nonlinear analysis is an analysis where there is a nonlinear interaction between the forces exerted and the displacements. As mentioned before, non-linear analysis is tough to calculate in an analytic way. Even so, is possible to calculate the plastic moment (\( M_p \)) and the maximum curvature (\( k_{max} \)).

The plastic moment is the moment at which all cross section has reached its yield stress. That is, the maximum bending moment that the section can resist. As the moment increases, the plastic zones increase in depth which results in two stress blocks: one zone yielding in tension and one in compression. Figure 3.4 (c) represents the stress distribution in beams stressed to this stage. The plastic
zones fill the whole cross section and are described as being 'fully plastic'. When the cross section of a member is fully plastic under a bending moment, any effort to increase this moment causes the member to act as if hinged at the neutral axis, called the fully plastic hinge.

The plastic moment is calculated with the following formula where \( z_i^p \) is the elements’ height from plastic neutral axis:

\[
M_p = \sigma_y \sum A_i z_i^p \tag{3.15}
\]

### 3.3 Finite Element Model

The finite element method (FEM) is the most common method used for solving problems of engineering and mathematical models. Typical problems such as structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential are often solved by FEM. It is a numerical method for solving differential equations of boundary value problems.

To solve it, it subdivides a large system into smaller parts: the finite elements (FE) which are obtained by a space discretisation, that brings to the construction of a mesh that has a finite number of points, called nodes. So, a mesh is a group of elements which are delineated by a group of nodes as demonstrated in Figure 3.5.

![Figure 3.5 - A 2-D domain partial mesh composed by 4 elements and 9 nodes](image)

There are 3 types of FE described in Figure 3.6. These are divided by one-dimension elements (called beam/pipe/spring/line, etc. element), two-dimensions elements (plane/plate/shell, etc. element) and three-dimensions elements (solid element). 2-D elements can be triangular or quadrilateral in shape with 3 or 4 nodes respectively. 3-D solid elements can be based on triangles and quadrilaterals which produces all types of prisms.
Figure 3.6 - FEM elements: (a) 1-D elements; (b) 2-D elements; 3-D elements

The goal is to create a mesh with quality (well-shaped elements), and without a huge number of elements to avoid calculations difficult. The mesh should also be fine (have small elements) in areas that are important for the subsequent calculations. Concluding, greater number of elements, finer the mesh and more precise the values and time in computation. Figure 3.7 follows an example of a mesh with a large number of elements (a) and only a few numbers of elements (b). Obviously, this thermology depends on the problem itself.

Figure 3.7 - Mesh size: (a) fine mesh; (b) gross mesh

Back to Figure 3.5, each of the four elements tries to calculate the respective stress' values for the respective four nodes. This means that all four elements attempt to calculate stress' values in the node 5 since it belongs to the four elements. Therefore, each element obtains a different value, based on its own shape functions which leads to different answers in node 5. Thus, the final value stress of node 5 is an average from all the answers each element sharing that node is producing. So, if there is a problem of big differences between nodal values in a model, such as happens on Figure 3.8 on the left, the best approach is to get the nodes closer to each other, this means, a finer mesh, as demonstrated on the right side of Figure 3.8 on the right.

Therefore, the characteristics of a FE are place in the element stiffness matrix. For a structural FE, it contains the geometric and material behaviour information and indicates the resistance of the element to deformation when subjected to loading. Such deformation may include torsional, bending and shear effects.

Figure 3.8 – Poorly dimensioned (left side) and well dimensioned (right side) mesh
In summary, the FEM formulation of a boundary value problem brings to a system of algebraic equations which approximates the unknown function over the domain. The simple equations that model these FE are subsequently assembled into a larger system of equations that models the whole problem. Finally, it uses variation methods to approximate a solution by minimizing an associated error function.
Chapter 4

Description of the ULCS

Chapter 4 starts by explaining about container ships in general and then, by making a detailed description and characterization of the ultra large container ship of 10,000 TEUs. It describes the main dimensions of the ship in conjunction with two figures of the midship section of the ultra large container ship. The elements of the structure and his materials are also described and characterized in this chapter.
Container ships are cargo ships that carry all their load in truck-size containers. This technique is called containerization and its capacity is measured in twenty-foot equivalent units (TEU). This type of vessels is divided into seven size categories: small feeder, feeder, feedermax, Panamax, Post-Panamax, New Panamax and Ultra-large. ULCS, short for Ultra Large Container Ship, is the common name for container ships with a container capacity of 10,000 TEU and above.

Containerships are well known for their big deck openings and large hatches meaning that there is only a small area for deck that can be used to contribute to the main hull girder strength of the vessel. In the present dissertation a section of the midship section of a 10,000 TEU container ship is design.

The main characteristics of the ULCS, dimensions of the transverse mid ship section such as plates, stiffeners distances and thicknesses has been brought [25] [24]. The main dimensions and the drawing of a typical transverse mid-ship section of a 10,000 TEU ultra large container ship are described and represented, respectively in Table 4.1 and Figure 4.1.

Table 4.1 - Main dimensions of the ULCS

<table>
<thead>
<tr>
<th>Description</th>
<th>Value [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length over all</td>
<td>337.00</td>
</tr>
<tr>
<td>Length between perpendiculars</td>
<td>320.00</td>
</tr>
<tr>
<td>Breadth Moulded.</td>
<td>48.20</td>
</tr>
<tr>
<td>Depth Moulded.</td>
<td>27.20</td>
</tr>
<tr>
<td>Design Draft</td>
<td>13.00</td>
</tr>
<tr>
<td>Scantling Draft</td>
<td>15.00</td>
</tr>
</tbody>
</table>

According to Figure 4.1 is possible to count the total number of longitudinal stiffeners for half of the transverse mid ship section. The hull of the ship is made up of welded plates and if these plates are not well stiffened, the bending moment on the structure can cause damage, and even failure. Therefore, there are a total of 102 longitudinal stiffeners: 91 bulbus flat plates (BP), 8 tee bars (TB) and 3 flat bars (FB).

The thicknesses of plates and stiffeners according to the different places of the mid ship section are described in Table 4.2. It also describes stiffeners’ types: flat bar (FB), bulbous flat plate (BP), tee bar (TB) and stiffeners’ steel materials’: AH and EH. Both materials are categorized as materials of higher strength, even so with different properties as shown on Table 4.3.

The material properties of the midship section such as the yield stress, the Young’s modulus, the Poisson’s ratio and the density are also shown on Table 4.3.
Figure 4.1 – Transverse midship section of the ULCS (all dimensions in mm) (taken by [40] [25])

Table 4.2 - Plates and stiffeners' thicknesses of the ULCS

<table>
<thead>
<tr>
<th>Description</th>
<th>Stiffeners</th>
<th>Plate Thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Above upper deck</td>
<td>900 × 70 EH FB</td>
<td>70 EH</td>
</tr>
<tr>
<td>Upper deck</td>
<td>760 × 58 EH FB</td>
<td>62 EH</td>
</tr>
<tr>
<td>2ND deck</td>
<td>260 × 10 AH BP</td>
<td>15 AH</td>
</tr>
<tr>
<td>4791 platform</td>
<td>320 × 12 AH BP</td>
<td>14 AH</td>
</tr>
<tr>
<td>Outer&amp;Inner bottom shell</td>
<td>200 × 12 AH BP</td>
<td>23&amp;16 AH</td>
</tr>
<tr>
<td>Outer&amp;Inner side shell</td>
<td>150 × 11 AH BP</td>
<td>16&amp;13 AH</td>
</tr>
<tr>
<td>Bilge side shell</td>
<td>350 × 11/150 × 15 AH TB</td>
<td>19 AH</td>
</tr>
</tbody>
</table>
Table 4.3 - Material characteristics of the ULCS

<table>
<thead>
<tr>
<th>Description</th>
<th>Material</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Stress</td>
<td>EH</td>
<td>390</td>
<td>MPa</td>
</tr>
<tr>
<td></td>
<td>AH</td>
<td>355</td>
<td></td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>EH</td>
<td>$2.06 \times 10^5$</td>
<td>MPa</td>
</tr>
<tr>
<td></td>
<td>AH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>EH</td>
<td>0.3</td>
<td>[-]</td>
</tr>
<tr>
<td></td>
<td>AH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>EH</td>
<td>$7.85 \times 10^{-9}$</td>
<td>$\frac{t}{mm^3}$</td>
</tr>
<tr>
<td></td>
<td>AH</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

For a better understanding, follow the following definitions of material’s properties:

**Yield stress**, $\sigma_y$, is the stress corresponding to the yield point at which the material begins to deform plastically. It represents the higher forces that can be applied without producing permanent deformation (elasticity deformation only). **Young’s modulus**, $E$ measures the stiffness of a solid material. It defines the relationship between stress and strain in a material in the linear elasticity regime of a uniaxial deformation. **Poisson’s ratio**, $\nu$ describes the expansion or contraction of a material in perpendicular directions to the direction of loading.

Notice that the yield stress for material EH is a slightly higher than material AH (see table 4.3). This means that upper deck and hatch coming, (material EH) bears a bigger amount of stress before it is permanently deformed. Besides yield stress, materials EH and AH have the same material properties.

In Figure 4.1 there are no stiffeners’ distances in decks and between upper and outer bottom. Therefore, in search of these information, another paper from a similar ship is considered and represented in Figure 4.2 [40]. This information is very important to create the model and give results’ credibility.

Comparing both figures, is possible to realize some disagreement between the two: while Figure 4.1 has two flat bars in the 12 564m, Figure 4.2 does not have any. Given the thickness of the plate, it has been concluded that there is no need for these two extra horizontal girders.

Initially, the object ship was design with two spacing between frames, (three number of frames) and for the entire transverse section. Although, due to time-consuming, only one spacing between frames with half of the transverse midship section was design. Therefore, the final object ship has a length, width and height of 3.044m, 24.100m and 29.200m respectively.
Figure 4.2 - Transverse midship section of the ULCS (all dimensions in mm) (taken by [40])
Chapter 5

Finite Element Model

Chapter 5 starts by explaining about the steps made for the construction of the half of the midship section of the ultra large container ship. Then, it describes the geometry of the object ship such as the mesh creation (including type, size and geometry). The initial imperfections in local plates and stiffeners are also defined. At last, the boundary conditions are described, and the static analysis is brought up in detail.
5.1 Geometry of the Structural Model

The model is constructed in ANSYS software which provides an easy parametric coding of the model that ensures speed and easiness to change parameters in case of need. To reduce the duration of each simulation, and at the same time, accomplish accurate results, one chose to study solely half of the midship section with one spacing between frames (3.044m). This way, one could run simulations with a refined mesh and obtain more reliable results. Therefore, the structural model according to the vessel in Figures 4.1 and 4.2, for only one spacing frame, has a height, width and length of 29.200m, 24.100m and 3.044m, respectively. It is worth to mention that to reduce the duration of each simulation, and at the same time, accomplish accurate results, one chose to study solely half of the midship section with one spacing between frames. This way, one could run simulations with a refined mesh and obtain more reliable results.

To create the structure in Ansys software, 365 keypoints are created for half of the transverse midship section. A followed path in keypoints organization is made to facilitate. Figure 5.1 shows the origin point of the structure, which is in the outer bottom shell at the intersection between the central longitudinal section and the left cross section of the finite element model, where x, y, and z indicates longitudinal, transverse, and vertical directions, respectively.

To define the areas for half of midship section, new keypoints are created in a linear array in the x direction, in a ladder of 3.044m (distance between frames). After the first areas are created by selecting all the keypoints (which defines the areas in a continuous direction to avoid twisted areas), these areas are acting as a base to define the material thickness and mechanical properties.

Figure 5.1 - Object ship FE model with and without the transverse frames
So, the structural model with plates and longitudinal stiffeners is finished. Even so, the transverse frames are missing. To create them, additional areas are made inner the bottom, bilge and side structure. Areas of transverse frames must be delineated all over the lines. This issue has to do with Ansys software that does not allow to make areas perpendicular to each other that have coincident lines in the boundaries. Therefore, Figure 5.1 shows the midship section of the ULCS without and with transverse frames on the left and right side, respectively.

The initial structural model is shown in Figure 5.2 where is possible to see the total number of frames for the total transverse midship section. Is possible to visualise the different material colour on the hatch coaming.

![Figure 5.2 – Initial FE model with (right side) and without transverse frames (left side)](image)

### 5.2 Mesh Creation

Mesh creation is very important to make a finite element analysis. The type, size and geometry can influence the speed and the precision of the computational results. When the thickness of the vessel shell is small compared to other dimensions, (the ratio of their thickness between the radius of curvature is less than 1/20), the shell is called a membrane and the resulting stresses due to internal pressure are called membrane stresses [41]. These membrane stresses are tension or compression which act uniformly across the vessel wall.

This is important once looking at the structure of the ULCS is possible to realize that plate’s thicknesses are very small compared with other dimensions, meaning that structure has thin-walled structures. In bending simulation, shear strains should be negligible. But, due to triangular and quadrilateral elements
some inconsistencies create shear strains much different from zero showing a stiff behaviour known as shear locking [42]. Since triangular element forms a more stable structure than quadrilateral element, which is less rigid, quadrilateral element shows better behaviour about this subject [43]. With is and knowing that shell element can produce good results for solid models at lower time than tri-dimensional elements, and the structure has thin-walled plates, is chosen a 2D finite quadrangular element meshing.

Ansys can create free meshes and mapped meshes, see Figure 5.3. A free mesh has no restrictions on the organization of its elements or on type of geometry that can be meshed. The program makes almost all decisions regarding the mesh and may generate a slightly different mesh each time. Both triangular and quadrilateral elements can be used to free mesh an area. Free meshing is easy and robust, that is why ANSYS free meshes all models by default. In contrast to the free mesh, mapped mesh includes solely triangular or quadrilateral elements. It usually has a regular patter, with obvious rows of elements [44].

![Figure 5.3 - Free (a) and mapped (b) meshes (adapted from [44])](image)

Since the structure of the ship has different members orientated in different directions, is easier to choose by a free mesh.

**SHELL93** is a mesh suitable for analysing thin, to moderately thick shell structures and is an eighth-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. It is well-suited for linear, large rotation, large deflection, and large strain nonlinear applications. It is represented in Figure 5.4 and is the type of mesh chosen for this dissertation [44].

![Figure 5.4 - SHELL93 geometry (taken from [44])](image)

Figure 5.4 shows the geometry, node locations, and the element coordinate system for this element. The element is defined by shell section information and by eight nodes (I, J, K, L, M, N, O, P,) and can with quadrilateral geometry.
As already explained in previous chapter, as the number describing the mesh size increases, the particle size decreases. Thus, to get better accuracy nodes must be closer to each other. Divisions are automatically calculated (rounded upward to next integer) from line lengths. Therefore, for this dissertation problem, the default number of line divisions (elements) to be generated along the region boundary lines is a mesh size of 0.15m, see Figure 5.5.

![Figure 5.5 – Mesh of the ULCS](image)

5.3 Initial Imperfections

It is known that in shipbuilding the stiffened plates are often fabricated by fusion welding which can affect the ultimate strength of the hull girder members, and consequently, the vessel itself. These welds, such as the manufacturing, transporting, and handling processes can cause the so-called initial imperfections. Hence, weld imperfections are acquired during construction, meaning every new built ship will have initial imperfections.

Therefore, initial imperfections are inevitable and could have a significant effect on the collapse behaviour of the hull girder of the vessel. To achieve credible results, it is important that the structure is as close to reality as possible. Thus, at the beginning of the analysis it is necessary to model the initial imperfections in the structure which correspond to the application of geometric imperfections.

Initial geometric imperfections can be classified into two main categories: global imperfections, (also known as sway imperfections), and local imperfections, (known as well as bow imperfections). Initial global geometric imperfections are global profiles for the whole structural member along the length in any direction. On the other hand, initial local geometric imperfections can be found in the surfaces of metal structure members. They stand in the perpendicular directions of the structural member surfaces.
which is the most unfavourable direction for a certain load combination. In ship structures, only three types of initial deflections are used: the thin-horse mode for local plates between stiffeners, the column-type initial deflections for plate related stiffeners and the sideways initial deflections for independent stiffeners [25].

This dissertation implements the thin-horse mode of initial deflections of the local plates, and the sideways initial deflections of independent stiffeners due to angular rotation about panel stiffeners (see Figure 5.6 and 5.8, respectively). Therefore, below, is shown the initial imperfections in local plates:

![Initial deflection example](image)

**Figure 5.6 - Initial deflection example (adapted from [45])**

It is known that this initial deformation of a stiffened panel is equivalent to the first elastic buckling mode, (see Figure 5.7), which can provide less resistance against the ultimate strength [25] [40].

![Representation of the three first buckling modes](image)

**Figure 5.7 - Representation of the three first buckling modes (adapted from [46])**

Therefore, some researchers start to investigate the effects of deflection shape and the magnitude of the initial imperfection of welded steel plates. During studies, they conclude that Fourier series’ shapes (simple trigonometric functions of sines and cosines), are a good option to model the reality of the post-welding initial geometric imperfections on structures. It basically changes the original node position by imposing a perpendicular displacement of the structural member surfaces given by equation bellow [25]:
\[ w(x, y) = w_0 \sin \left( \frac{m \pi x}{a} \right) \sin \left( \frac{n \pi y}{b} \right) \] (5.1)

In the Equation 5.1, \( w_0 \) is the maximum amplitude of initial deflection of the local plates, \( m \) and \( n \) are the number of buckling half wave for \( x \) and \( y \) direction respectively, \( a \) and \( b \) are the distances between transverse frames and longitudinal stiffeners, respectively, and \( x \) and \( y \) are the coordinates' positions of the nodes.

The amplitude of initial deflection \( w_0 \) is defined as follows [25]:

\[
 w_0 = \begin{cases} 
 0.025 \beta^2 t, & \text{for slight level} \\
 0.1 \beta^2 t, & \text{for average level} \\
 0.3 \beta^2 t, & \text{for severe level} 
\end{cases} 
\] (5.2)

In which \( t \) is the plate thickness and \( \beta \) is the plate slenderness parameter which is defined by:

\[
 \beta = \frac{b}{t} \sqrt{\frac{\sigma_y}{E}} 
\] (5.3)

Where \( \sigma_y \) is the yielding stress of material and \( E \) is the Young’s modulus of material. In this dissertation the main initial deflection is for average level. This appears to be the appropriate choice once slight level is a very optimistic view and, by contrast, severe level is a pessimistic view.

For this model, initial imperfections are applied in the double bottom, upper deck and double side. It considers initial imperfections only for longitudinal stiffened elements of the structure.

Once there are different stiffeners’ distances and thicknesses along these plates, different volumes of \( w_0 \) are used for the slenderness parameters and initial deflections. Note that frames do not have initial imperfections. It is by formula observation, that for plates with low thicknesses and low yielding stress, the initial deformation gets higher. On the other hand, for plates with higher thicknesses and yielding stress, the initial deformation gets lower.

The longitudinal stiffeners suffer a rotation about the plates as shown in Figure 5.8.

To simulate the tripping mode of initial imperfections in longitudinal stiffeners, the following equation is provided:

\[
 v_S = v_{SO} \frac{z}{h_w} \sin \left( \frac{\pi x}{a} \right) 
\] (5.4)

Where the magnitude, \( v_{SO} = 0.001 m \) [25] and the \( h_w \) is the height of the stiffeners.
5.4 Boundary Conditions

A boundary value problem (BVP) is a differential equation with a set of additional constraints called boundary conditions (BC). The solution of a BVP is a result of the differential equation which also satisfies the BC [48].

To apply BC is crucial to comprehend the behaviour of the structure when bending is applied. Herewith, it means understanding what displacements a node takes before and after being subjected to sagging or hogging moment. Figure 5.9 illustrates a beam before bending. If only focused on a point/node with a random height, is possible to understand the displacement produced in longitudinal and vertical direction when a moment is applied (strained to curvature radius $R$).

![Figure 5.9 - Example of an unstrained and strained bending curve of a beam (adapted from [49])](image)

If considering a longitudinal length equal to 1 unit with a curvature radius, $R$, by triangle similarity, the followed equation is valid:

$$\theta = \frac{1}{R} \frac{\varepsilon}{z} \quad (5.5)$$

Where $z$ is the displacement in vertical direction and $\varepsilon$ is the ratio close to the longitudinal direction displacement of the same point/node. The transverse direction displacement is equal to zero, since,
when bending is applied there are no change in that direction. Therefore, if a certain curvature, \( \frac{1}{R} \) is applied, knowing the initial position of the nodes, is possible to get the final longitudinal and vertical position displacement due to bending moment. The last restriction allows the middle point remains at the same initial location, as happens in real life.

To create vertical bending moment in the ULCS model, only extremes frontiers need to comply with certain conditions while the rest of the structure can move freely, (depending on that restrictions).

There are several different ways to simulate bending moment such as the use of master nodes, (see Figure 5.10). The BVP using master nodes consists in two control points usually located at neutral axis height that relates to the other nodes by a line-rigid beams called umbrella. In this case a rotation would be applied in the control points connected with all the nodes belonging to the same longitudinal location. This way, the nodes follow the rotation imposed on master nodes by the rigid bodies. The method is used in several investigation works [50] [24]. Figure 5.11 on the right shows the implementation of the boundary condition problem by using control points compared with the method used in this dissertation.

\[ \text{Rigid body constrained to control points} \]

**Figure 5.10 – Boundary conditions at fore and aft section for master nodes method**

This dissertation applies the BC in a different way. It restrings the nodes belonging to the fore and aft section of the model as demonstrated in Figure 5.11 and Table 5.1. Whereas frame A has a constraint displacement in longitudinal direction, frame B has a linear displacement on longitudinal direction. This linear displacement is calculated by the method described in Figure 5.9.

Is important to refer that an extra BC was imposed at amid ship. Once the FE model is design for half of the mid ship section, the nodes belonging to amid ship must have no displacement in transverse direction. Otherwise, the centre of the model would change their initial transverse position. All restante displacements and rotations are not constrained.
Note that \( u_x, u_y, u_z \) represents the displacements in longitudinal, transverse and vertical directions respectively while \( \text{rot}_x, \text{rot}_y, \text{rot}_z \) represents the rotations around the longitudinal, transverse and vertical directions correspondingly.

The differences between these methods are represented in Figure 5.12. Here the differences between fore and aft sections for both methods are visible.

Looking at Figure 5.12 on the left side for the implemented method, one frame, (Frame A in the Figure 5.11) stays fixed although it can be move with a null force, (a constant displacement in longitudinal direction). This is to assure the force-balance principle meaning that the total force applied in the structure is null. The opposite frame, (Frame B in the Figure 5.11), as to move around a plane rotation or a linear displacement. Here, the nodes do not move vertically. In Figure 5.12, now on the right side, for control points’ method, one frame has a rotation imposed while the opposite stays fixed although it can be move, (also to assure the force-balance principle).

When comparing both methods, the master nodes method appears to be less genuine. When the rotation is applied on the control point, the distances between the nodes remain constant which does not represent the reality since when the structure is subjected to compression or tension, the thickness’ plates can get higher or thinner correspondently and compensate in the orthogonal direction. That is, the distances between the nodes does not remain constant and can be more together or apart to follow Poisson’s ratio.

<table>
<thead>
<tr>
<th>Plane</th>
<th>( u_x )</th>
<th>( u_y )</th>
<th>( u_z )</th>
<th>( \text{rot}_x )</th>
<th>( \text{rot}_y )</th>
<th>( \text{rot}_z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A - right</td>
<td>Constant/constraint ((F_x = 0))</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>B - left</td>
<td>Linear ( u_x = k \cdot z ) (plane rotation)</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
5.5 Static Analyses in FEM

Structural mechanic problems can be formulated as boundary conditions problems. These problems require the conditions in the boundary of a continuous body and aim to determine the value of the distribution of some quantities inside the body such as displacements, velocities, stresses, or strains [51].

To design the response due to vertical bending moment, in the context of FEA, the finite element equations can be solved by Newton-Raphson’s method, for it has a quick and precise response. This method aims to estimate the roots of a function by calculating the tangent equation line of the function at that point and its intersection with the abscissa axis, to find a better approximation to the root, (see Figure 5.13). Repeating the process, an iterative method is created to find the root of the function. The most basic version starts with a single-variable function $f$ defined by a real variable $x$, the function's derivative $f'$, and an initial guess $x_0$ for a root of $f$. If the function satisfies sufficient assumptions and the initial guess is close, the followed equation is valid:

$$x_1 = x_0 - \frac{f(x_0)}{f'(x_0)}$$  \hspace{1cm} (5.6)

That is a better approximation of the root than $x_0$. Geometrically, $(x_1, 0)$ is the intersection of the $x$-axis and the tangent of the graph $f$ at $(x_0, f(x_0))$: that is, the improved guess is the unique root of the linear approximation at the initial point. The function $f$ is shown in blue and the tangent line is in dark red in Figure 5.12 where $(x_n + 1)$ is a better approximation than $x_n$ for the root $x$ of the function $f$.

The process is repeated as demonstrated in Equation 5.7 until a sufficiently precise value is reached:
\[ x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)} \] (5.7)

Therefore, the idea is to start with an initial guess which is reasonably close to the true root, and then, approximate the function by its tangent line using calculus to finally compute the \( x \)-intercept of this tangent line by elementary algebra. This \( x \)-intercept will typically be a better approximation to the original function's root than the first guess, and the method can be iterated.

Hence, in FEM the static analysis can be solved using Newton-Raphson’s method. However, due to its formulation, this method is not a good choice in cases where the stiffness matrix of the structure is not purely positive well marked as shown in Figure 5.14.

So, the Newton’s method cannot accurately predict the solution after a limit point is reached. It obstructs the analysis of problems that exhibit instabilities in the form of buckling, for instance. This is, when the problem under consideration exhibits one or more critical points in conditions of a simple mechanical loading-unloading problem, a critical point could be understood as the point at which the loaded body cannot support an increase of the external forces and an instability occurs [52].

---

Figure 5.13 - Newton-Raphson's method (taken from [53])
These kinds of problem can be addressed using the arc length method, which solves non-linear systems of equations with efficiency and accuracy, as showcased in Figure 5.14.

With this method, a curve in the plane can be approximated connecting a finite number of points on the curve by using line segments. If a function given by $\lambda = f(x)$ represent a smooth curve on the interval from $[a, b]$, the arc length of the function $f$ from $[a, b]$ is given by the following equation:
\[ S = \int_a^b \sqrt{1 + (f'(x))^2} \, dx \]  

(5.8)

The arc length method has its own limitations. In most cases Equation 5.8, above, cannot immediately archive to satisfactory result as in the cases shown in Figure 5.16. In these, it is possible to understand that curves are changing the sign curvature route/path where a system is unstable under load control, (snap–through instability), under displacement control, (snap–back instability) and under both, displacement and load control.

![Figure 5.16](image-url)

Figure 5.16 - (a) Snap–Through, (b) Snap–Back, (c) Through and Snap-Back instabilities (taken from [55])

Therefore, the arc-length method is a very efficient method in solving non-linear systems of equations when the problem under consideration exhibits one or more critical points. It is also quick and a precise method [56], despite its limitations.

To improve these methods, an equation is added to the original non-linear governing equation of the problem, to obtain a solution point along the path. Step-by-step and changing the value of a parameter contained in the constraint equation, the solution path can be traced by both Newton and arc-length methods. Beginning with a known solution, further solutions of the extended system of equations for specified values of path parameter are calculated step by step, until one reaches a target point. These methods typically require suitable starting values.

Therefore, the ideal graph cumulative iteration number vs absolute convergence norm on ANSYS is as in Figure 5.17:

![Figure 5.17](image-url)

Figure 5.17 - Typical convergence graph
While ANSYS steps through non-linear analysis, the Newton and the arc-length methods iterates to find a solution. Initially, when the problem is relatively linear, very few iterations are needed but, when the solution is highly non-linear, or is not converging, more iterations are required.

This graph relates to forces and moments which values corresponds to the solution vector for the DOF’s elements being used. For each parameter, there are two curves plotted:

- CRIT curve refers to the convergence criteria force value. The default value is the square root of the sum of the squares of the applied loads;
- L2 curve refers to the L2 Vector Norm of the forces. It is the square root of the sum of the squares of the difference between the calculated internal force at a particular DOF and the external force in that direction.

For each substep, ANSYS iterates until the L2 value gets below the CRIT value. Once this occurs, it is considered that solution is within tolerance of the correct solution and it moves on to the next substep. Once it gets converged, the solution stops with the time defined (Time = 1, for example) [44].

Therefore, to design the static analysis and the convergence solution complying with Newton and arc-length methods, the ANSYS software code is provide as follows:

```plaintext
!--------------------------- STATIC ANALYSIS ---------------------------!
/SOL ! Enters the solution processor
ANTYPE,0,new ! Static analysis (not buckling)
/STATUS,solu ! Provides a solution status summary
time=1 ! Time at the end of the load step
NROPT,unsym ! Specifies the Newton-Raphson options in a static/full transient analysis
NSUBST,20,50,20 ! Number of substeps (used, maximum, minimum)
ARCLEN,1.2 ! Activates the arc-length with a curvature radius of 1.2
NLGEOM,ON ! Non-linear geometry solution supported (large-deflection effects)
OUTRES,all,all ! Controls the solution data written to the database
SOLVE ! Solve the Finite Element Model
FINISH ! Finish the solution
```

Figure 5.18 – Static analysis using Newton and arc-length methods (ANSYS code)

### 5.6 Corrosion Effects

In marine structures, corrosion is a natural process resulting in a spontaneous reaction between the metal and its environment which results in the degradation of the material. Metal corrodes easily in seawater and quickly loses strength which may result in structural failure. The more the environment is susceptible to corrosion, the greater the added thickness of the elements in that area.

Along the time, vessels tend to lose their initial thicknesses (plates and stiffnesses) due to corrosion, making the ultra large container ships (and others) less resistant when submit to vertical bending moment and other efforts. To realize if the ship continues to remain within the International Association of Classification Societies norms considering the corrosion effects, a study of the progressive collapse...
behaviour of FE model is made. With this, would be possible to evaluate the behaviour of the structure in these conditions and find out the weakness spots to improve the ultra large container ship in study.

To be approved by IACS, the model of the structure created in FE needs to fulfil with the limits given by the design bending moment in Chapter 3.1.2. For that, it is necessary to study the model results after corrosion. The values initially given in Table 4.2, 4.3 are the results for the final thickness, i.e., after corrosion addition thickness. This means that, to do the FEM analysis, the thickness corresponding to corrosion, given by Figure 5.19 for different zones, will have to be subtracted to final thickness.

<table>
<thead>
<tr>
<th></th>
<th>Tank/hold region</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Within 1.5 m</td>
<td>Elsewhere</td>
</tr>
<tr>
<td></td>
<td>below weather</td>
<td></td>
</tr>
<tr>
<td></td>
<td>deck tank or</td>
<td></td>
</tr>
<tr>
<td></td>
<td>hold top</td>
<td></td>
</tr>
<tr>
<td>Ballast tank 1)</td>
<td>3.0</td>
<td>1.5</td>
</tr>
<tr>
<td>Cargo oil tank only</td>
<td>2.0</td>
<td>1.0 (0) 2)</td>
</tr>
<tr>
<td>Hold of dry bulk cargo carrier 3)</td>
<td>1.0</td>
<td>1.0 (3) 3)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Tank/hold region</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Within 1.5 m</td>
<td>Elsewhere</td>
</tr>
<tr>
<td></td>
<td>below weather</td>
<td></td>
</tr>
<tr>
<td></td>
<td>deck tank or</td>
<td></td>
</tr>
<tr>
<td></td>
<td>hold top</td>
<td></td>
</tr>
<tr>
<td>Ballast tank 1)</td>
<td>2.5</td>
<td>1.5 (1.0) 2)</td>
</tr>
<tr>
<td>Cargo oil tank only</td>
<td>2.0</td>
<td>1.5</td>
</tr>
<tr>
<td>Hold of dry bulk cargo carrier 3)</td>
<td>2.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Hold of dry bulk cargo carrier 4)</td>
<td>1.0</td>
<td>0.5 (0) 3)</td>
</tr>
<tr>
<td>Hold of dry bulk cargo carrier 5)</td>
<td>0.5</td>
<td>0.5</td>
</tr>
</tbody>
</table>

1) The term ballast tank also includes combined ballast and cargo oil tanks, but not cargo oil tanks which may carry water ballast according to Regulation 14(2), of MARPOL, 73/78.
2) The figure in brackets refers to non-horizontal surfaces.
3) Other category space denotes the hull exterior and all spaces other than water ballast and cargo oil tanks and holds of dry bulk cargo carriers.
4) Hold of dry bulk cargo carriers refers to the cargo holds, including ballast holds, of vessels with class notations Bulk Carrier and Ore Carrier, see Pt.5 Ch.3 Sec.3.
5) The figure in brackets refers to webs and bracket plates in lower part of main frames in bulk carrier holds.

Figure 5.19 – Corrosion addition in mm (taken from [57])
Chapter 6

Results Analysis

This chapter compares the numerical calculations with the model results in order to assess the validity of the model. It also includes a detailed analysis and discussion of the results obtained in the ANSYS software, considering different levels of initial imperfection.
6.1 Design Vertical Bending Moment

To accomplish the design vertical bending moments according to classification society rules, several characteristics of the ULCS of 10 000 TEUS were estimated with Equation 3.7 for $300 \leq L_P \leq 350$ and Equation 3.8 for a container ship. The design speed of the vessel is taken from an article about propulsion trends of container vessels [58]. Table 6.1 summarizes these results.

<table>
<thead>
<tr>
<th>Block coefficient, $C_b [-]$</th>
<th>0.674</th>
</tr>
</thead>
<tbody>
<tr>
<td>Typical design speed, $v [\text{knots}]$</td>
<td>22.000</td>
</tr>
<tr>
<td>Wave load coefficient, $C_w [-]$</td>
<td>10.750</td>
</tr>
</tbody>
</table>

Therefore, the design results for still water and wave induced for sagging and hogging conditions amidship section are shown in Table 6.2, below.

| Design still water bending moment amidship (Sagging), $M_{SO}$ [N.m] | $-4.74E+09$ |
| Design still water bending moment amidship (Hogging), $M_{SO}$ [N.m] | $5.96E+09$ |
| Design wave induced bending moment amidship (Sagging), $M_{WO}$ [N.m] | $-8.02E+09$ |
| Design wave induced bending moment amidship (Hogging), $M_{WO}$ [N.m] | $6.79E+09$ |

It is possible to conclude that design wave induced bending moments have higher values compared with the still water bending moments: unsymmetric cargo loading over forward and aft of the ship has less impact than unsymmetric hydrodynamic wave loading on both extremes of the vessel.

When compared, still water and wave induced bending moments for hogging and sagging conditions, was concluded that hogging moment is higher than sagging moment for still water, and on the contrary, sagging moment is higher than hogging moment for wave induced. This means that unsymmetrical cargo loading over forward and aft of the ship has more impact when the overall weight is greater near the bow and stern, and also that unsymmetric hydrodynamic wave loading on both extremes of the vessel has more influence when two wave’s crests pass simultaneously through the bow and stern. In conclusion, the total sagging and hogging moments are quite similar, with a value of: $M_T = 1.28 \times 10^{10} \text{ N.m}$.

When compared with the beam theory and FEM, it is important that all calculations are made for the same conditions, this is, all parameters must be calculated for half of the midship section. The FEM analysis was designed for half of the midship section to facilitate the solution scores (to take less time to draw results), so, the total sagging and hogging moments for half of the midship section is given: $M_T = 6.38 \times 10^9 \text{ N.m}$. 
6.2 Beam Theory

The section properties for the ULCS of 10 000 TEUs are presented in Table 6.3. The calculation of these section properties is in detail on Appendix A.

Table 6.3 – Section properties

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Area, A [m²]</td>
<td>2.991</td>
</tr>
<tr>
<td>1st Moment of Area, S [m³]</td>
<td>35.115</td>
</tr>
<tr>
<td>Inertia Moment about zz axis, Izz [m⁴]</td>
<td>792.292</td>
</tr>
<tr>
<td>Neutral axis height, hNA [m]</td>
<td>11.741</td>
</tr>
<tr>
<td>Moment of Inertia about neutral axis, I [m⁴]</td>
<td>380.021</td>
</tr>
<tr>
<td>Section Modulus about the keel, ZK [m³]</td>
<td>32.368</td>
</tr>
<tr>
<td>Section modulus about the deck, ZD [m³]</td>
<td>24.582</td>
</tr>
</tbody>
</table>

Therefore, the initial yielding moment, the plastic moment and the maximum curvature given by IACS and beam theory for half of the midship section are in Table 6.4, bellow.

Table 6.4 – Yielding and plastic moments, and maximum curvature for the ULCS

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Yielding Moment, My [N.m]</td>
<td>8.73E+09</td>
</tr>
<tr>
<td>Plastic Moment, MP [N.m]</td>
<td>1.17E+10</td>
</tr>
<tr>
<td>Maximum curvature, kMax [m⁻¹]</td>
<td>0.00016</td>
</tr>
</tbody>
</table>

6.3 FEM Geometry

In order to have a clear perception if the geometry of the model made in ANSYS software is well designed or nor, it is important to analyse the parameter values that the software can provide. This can be achieved by comparing the theoretical values of the moment of inertia about neutral axis and of the neutral axis height previously calculated in Table 6.3 with those of the model. These values are taken from the ANSYS software by using the command “asum,fine” which provides the statistics geometry of the whole structure. These are in Table 6.5.

Table 6.5 – FEM section properties

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of Inertia about neutral axis, I [m⁴]</td>
<td>382.260</td>
</tr>
<tr>
<td>Neutral axis height, hNA [m]</td>
<td>11.742</td>
</tr>
</tbody>
</table>

As can be observed in Table 6.3 and Table 6.5, the figures are quite close. Even so, the relative error between these parameters, calculated by beam theory and FEM, are needed to evaluate the error rate.
between them to check if it is permissible (10% or less). So, the relative error, \( \text{relative error} = \frac{(\text{measured} - \text{real})}{\text{real}} \) is calculated, in which the measured is the value taken from FEM and the real is the value taken from beam theory, and is presented in Table 6.6.

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of Inertia about neutral axis, ( I ) [m(^3)]</td>
<td>0.0059</td>
<td></td>
</tr>
<tr>
<td>Neutral axis height, ( h_{NA} ) [m]</td>
<td>0.0001</td>
<td></td>
</tr>
</tbody>
</table>

So, the error is less than 10% meaning that FEM can be considered well dimensioned. Most of the error can be explained by the approximation of the curvature on the bilge to plane plates in FEM and, also by the error associated in ANSYS software itself, which made its own simplification (e.g., in FEM plates were considered a 2D shell with thickness). Even so, the results are adequate.

### 6.4 Vertical Bending Moment on FEM

Two simulations are made in this sub-chapter: one for sagging bending moment and another for hogging bending moment. A curvature of \( 0.000625 \text{m}^{-1} \) and \( -0.000625 \text{m}^{-1} \) are imposed for sagging and hogging conditions, respectively. Then, the bending results are observed to see if the outcomes designed by FEM analysis are within the parameters given by IACS and to see if the yielding moment, and the plastic moment limits calculated by beam theory are respected. Validation is then carried out.

Then, is important to take some conclusions about bending moments’ results, namely the: sagging and hogging curvature-line curve and their evolution in the elastic and plastic regime. It is also essential to verify whether they correspond the expected results.

Finally, the finite element model for sagging and hogging conditions is discussed and explored with some pictures taken from the ANSYS software where is possible to visualize the scale of tension and compression that the half of the midship section is sustaining.

The plot of the curvature-line curve, (curvature versus bending moment), for sagging and hogging conditions is given in Figure 6.1 following by Tables 6.7 and 6.8 where the results of the elastic regime and the error between the beam theory and the FEM for hogging and sagging conditions, respectively, is presented.

Concerning about the plastic regime Table 6.9 describes the plastic moment in relation to the maximum plastic moment achieved for the model for hogging and sagging conditions, correspondingly. Table 6.10 shows the difference between the initial yielding moment and IACS rules design moment for hogging and sagging conditions, respectively.
Figure 6.1 – Moment-curvature curve for sagging and hogging conditions

Table 6.7 – Beam theory and FEM results in elastic regime in hogging condition

<table>
<thead>
<tr>
<th></th>
<th>Beam Theory</th>
<th>FEM</th>
<th>Relative Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial yielding moment [N.m]</td>
<td>8.73E+09</td>
<td>8.77E+09</td>
<td>1%</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
<td>0.000111</td>
<td>0.000114</td>
<td>3%</td>
</tr>
<tr>
<td>Structural modulus [N.m²]</td>
<td>7.83E+13</td>
<td>7.58E+13</td>
<td>3%</td>
</tr>
</tbody>
</table>

Table 6.8 - Beam theory and FEM results in elastic regime in sagging condition

<table>
<thead>
<tr>
<th></th>
<th>Beam Theory</th>
<th>FEM</th>
<th>Relative Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial yielding moment [N.m]</td>
<td>-8.73E+09</td>
<td>-8.36E+09</td>
<td>4%</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
<td>-0.000111</td>
<td>-0.000107</td>
<td>4%</td>
</tr>
<tr>
<td>Structural modulus [N.m²]</td>
<td>7.83E+13</td>
<td>7.71E+13</td>
<td>2%</td>
</tr>
</tbody>
</table>

Comparing the initial yielding moment for hogging and sagging conditions correspondent in Tables 6.7 and 6.8, is possible to verify that whereas for hogging condition the yielding moment is quite close (1%), for sagging moment the error is around 4%.

Then, analysing the linear gradient in elastic regime between beam theory and FEM results, is possible to recognize that strength lines are almost overlapping having a difference of 3% and 2% for hogging.
and sagging moments, correspondingly. This likely means that the sub-steps established in the Newton method do not exactly predict the yield point. It is also notorious that the inclination is lightly lagging. This could be due to the imperfections in the FEM which do not precisely match the theoretical calculations, which is common. So, the elastic regime and yielding moment of the FEM analysis seems satisfactory getting relative errors under 10% (between 1% and 4%).

Following tables are concern about the elastic/plastic regime, where is possible to verify that the maximum plastic moment achieved is considerably lower than the plastic moment calculated by beam theory.

Table 6.9 – Plastic moment by beam theory and maximum plastic moment achieved for hogging and sagging conditions

<table>
<thead>
<tr>
<th></th>
<th>Beam Theory Plastic Moment [N.m]</th>
<th>FEM Max. plastic moment achieved [N.m]</th>
<th>Diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hogging</td>
<td>1.17E+10</td>
<td>1.01E+10</td>
<td>16.6%</td>
</tr>
<tr>
<td>Sagging</td>
<td>-1.17E+10</td>
<td>-8.36E+09</td>
<td>40.5%</td>
</tr>
</tbody>
</table>

Although the non-linear curve line is imperfect (something which is later discussed), is possible to say that the maximum moment achieved for hogging and sagging conditions stays under the plastic moment calculated by beam theory. In sagging condition, the distance from the plastic moment (around 41%) is higher than for hogging condition which is about 17%.

Concerning about design value for the vertical bending moment given by IACS, (see Table 6.10) is possible to verify that it lies significantly below the yielding stress, inside the elastic regime (above 30% for both conditions). This proves that the structure complies with the norms given by classification society rules and, for that reason, in this concern, it fulfils the specified requirements.

Table 6.10 – Initial yielding moment vs IACS

<table>
<thead>
<tr>
<th></th>
<th>IACS [N.m]</th>
<th>Yielding moment [N.m]</th>
<th>Diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hogging</td>
<td>6.38E+09</td>
<td>8.77E+09</td>
<td>38%</td>
</tr>
<tr>
<td>Sagging</td>
<td>-6.38E+09</td>
<td>-8.36E+09</td>
<td>31%</td>
</tr>
</tbody>
</table>

With regards to the discontinuity of the curvature of the hogging moment, is crucial to have a look at the convergence solution graphs for sagging and hogging moments. Starting with the hogging bending moment, Figure 6.2 shows the convergence solution by the imposed curvature.

This convergence solution started with a stable line’s convergence, and then, the time began to decline slowly until it turned negative and finally down abruptly, which means high values of plasticity, and the geometric shape of the structure changes and cannot converge to a solution. Due to this, the final curvature vs bending moment is created and, a huge leap is seen once the hogging moment values became negative. Since those negative values are not possible, they were deleted, remaining only the positive values of bending moment.
In a similar manner, it is possible to conclude that the plastic regime was not achieved for the sagging bending moment result in Figure 6.1. The convergence solution graph is in Figure 6.3.

Once more the critical and vector norms, forces and moments are stable to a certain point. From there, the critical points drop, the convergence solution is not found and a gap in the bending graph occurs. This means that, once the plastic regime was achieved the structure did not bear the bending imposed.

As previously mentioned, the ship responds less strongly to sagging conditions which is illustrated in Figure 6.1 where hogging bending moment achieve a higher value of bending moment than sagging bending moment.

![Figure 6.2 – Convergence solution for hogging bending moment](image)

![Figure 6.3 – Convergence solution for sagging bending moment](image)

In this dissertation, only the nodal solutions for $x, y, z$ - component of stress, von Misses stress, elastic and plastic strain for $x, y, z$ - components are taken into consideration when relevant. So, to analyse the figures from the FEM in sagging and hogging conditions, the following nomenclature is used: $SMX =$ maximum stress; $SMN =$ minimum stress; $DMX =$ maximum displacement.
Therefore, for the study of the structure’s reaction when submitted to hogging condition the following figures are taken:

Figure 6.4 shows the longitudinal stresses where the positive tensions are seen in the upper deck, and the negative tensions are observed in the bottom. The blue tonners represent places in compression and the red ones represents very low tensions stresses due to plastic yielding stresses in the upper deck (where the dark red represents the yield stress). It can be seen the curvature turned up in the bottom which leads to the rotations of the side of the FEM.

In Figure 6.5 it is already possible to see that from neutral axis height down, the structure is in compression while from the neutral axis height up it is in tension, as expected. Is also evident the initial imperfections on outer and inner side and bottom and upper deck. The inner side plates are subjected to a higher stress than outer side plates. Is possible to realize that the imperfections on the upper plates are not evident as the rest which is due to the change of the thickness of the shell in that position, which is much larger and by consequence. Also, is possible to see higher levels of stress in the corners of the hatch coaming where symmetry is visible.
On the right image of Figure 6.5 is possible to observe a red point with a high level of stress near to the symmetry line which will be relevant when plastic strain is showed on Figures 6.6, 6.7.

Which concerns about the elasticity in $x$ direction, is possible to verify that the initial imperfections are well distributed with regular spaces and symmetry.

![Figure 6.6 – $x$ component of elastic strain for hogging condition](image)

The following figures show most of the structure model does not have plastic strain along the $x$ direction instead upper deck, hatch coaming and part of the plate joined to the upper deck.

![Figure 6.7 – $x$ component of plastic strain for hogging condition](image)

Here is possible to perceive the beginning of yielding lines which began near the hatch coaming and go along the outer side shell. Once these lines appear far from the neutral axis height and due to higher yield stress and thickness of plates and stiffeners in that location, is possible that they have more difficult to extend (Poisson coefficient) going into plastic regime earlier.

Finally, when considering the plastic strain in $z$ direction for hogging condition, is possible to conclude that the double bottom also shows plasticity. When comparing with the $x$ component of plastic strain, is visible that in $z$ direction the stresses are more notorious in the hatch coaming and upper deck. In inner bottom shell there are small levels of non-symmetric plasticity with some spots that goes through the frames in opposite direction.
After observing and commenting the results for hogging conditions, the same is done for the sagging condition. Figure 6.8, contains the x component results for sagging.

Figure 6.9 shows the longitudinal stresses where the positive tensions are seen in the bottom, and the negative tensions are observed in the upper deck. It can be seen the curvature turned down in the bottom which leads to the rotations of the side of the FEM.
Contrary to the hogging condition, in sagging moment condition the deck is in compression and the bottom is in tension. It is possible to verify that more distant to the neutral line axis, higher the stress on the structure’s model. Also, the initial imperfections are more considerable on the outer bottom shell than on the inner bottom shell, even though it is thicker.

From component of stress in z direction, it is possible to see an area with high levels of stress, located on the second highest of outer side shell where a difference between the thickness is very dissimilar. Therefore, it is possible that the model structure will break in this area and it is thus advisable to increase this plate thickness there.

![Figure 6.11 – z component of stress for sagging condition](image)

In terms of elasticity, (see Figure 6.12) it is possible to see areas with higher tension between stiffeners and the girder plates. Still in the outer bottom, it is possible to verify two yellow points, which match the points of bigger distortion seen in Figure 6.8, on the left side. Once more, stresses are higher the further away they are from the neutral axis.

![Figure 6.12 – x component of elastic strain for sagging condition](image)

With respect to plasticity on x direction, (Figure 6.13), only the corners of the hatch coaming suffered plasticity strain.
Unlike the behaviour of the structure model in plastic strain on x direction, on z direction only two spots in the outer side shell have any plasticity. These belong to the area affected with higher level of stresses in z direction in Figure 6.14.

When analysing the model closely, using a higher scaler factor, it was verified that in the outer bottom, between centre and first girder the structure is deformed, (see Figure 6.15).
Is possible to visualise that the distance between first and second girder are more distant than the others. This fact could be the cause of the weakness of the model in this location. Other conditions, like the thickness of plates and longitudinal stiffeners could be a cause.

6.5 Influence of Initial Imperfections

As mentioned, is relevant to consider the behaviour of the structure model when submitted to different levels of initial imperfections.

To verify the effect of the initial imperfections on the structure, two levels of maximum geometric imperfections were considered: average and severe levels. The model without imperfections was also considered. The plot moment-curvature curve for these conditions is presented in Figure 6.16 following by Tables 6.11 and 6.12 where the results are given for hogging and sagging conditions, respectively. It also presents the difference between no imperfections and average level of imperfections results and average level of imperfections with severe level of imperfections figures.

![Figure 6.16 – Moment-curvature curve for different levels of initial imperfections](image)

---

56
Table 6.11 – Hogging moment results for different level of initial imperfections

<table>
<thead>
<tr>
<th></th>
<th>No imperfections</th>
<th>Average level</th>
<th>Diff.</th>
<th>Severe level</th>
<th>Diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yielding moment [N.m]</td>
<td>8.92E+09</td>
<td>8.77E+09</td>
<td>2%</td>
<td>8.17E+09</td>
<td>7%</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
<td>0.000113</td>
<td>0.000114</td>
<td>1%</td>
<td>0.000112</td>
<td>2%</td>
</tr>
<tr>
<td>Structural modulus [N.m2]</td>
<td>7.92E+13</td>
<td>7.58E+13</td>
<td>4%</td>
<td>7.21E+13</td>
<td>5%</td>
</tr>
<tr>
<td>Maximum plastic moment achieved [N.m]</td>
<td>1.01E+10</td>
<td>1.01E+10</td>
<td>1%</td>
<td>9.45E+09</td>
<td>7%</td>
</tr>
<tr>
<td>Maximum curvature achieved [1/m]</td>
<td>0.000150</td>
<td>0.000170</td>
<td>13%</td>
<td>0.000205</td>
<td>17%</td>
</tr>
</tbody>
</table>

Table 6.12 – Sagging moment results for different level of initial imperfections

<table>
<thead>
<tr>
<th></th>
<th>No imperfections</th>
<th>Average level</th>
<th>Diff.</th>
<th>Severe level</th>
<th>Diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yielding moment [N.m]</td>
<td>-8.04E+09</td>
<td>-8.32E+09</td>
<td>4%</td>
<td>-6.89E+09</td>
<td>21%</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
<td>-0.000101</td>
<td>-0.000107</td>
<td>5%</td>
<td>-0.000093</td>
<td>15%</td>
</tr>
<tr>
<td>Structural modulus [N.m2]</td>
<td>7.93E+13</td>
<td>7.76E+13</td>
<td>2%</td>
<td>7.43E+13</td>
<td>4%</td>
</tr>
<tr>
<td>Maximum plastic moment achieved [N.m]</td>
<td>-9.67E+09</td>
<td>-8.36E+09</td>
<td>16%</td>
<td>-7.05E+09</td>
<td>19%</td>
</tr>
<tr>
<td>Maximum curvature achieved [1/m]</td>
<td>-0.000130</td>
<td>-0.000107</td>
<td>21%</td>
<td>-0.000095</td>
<td>13%</td>
</tr>
</tbody>
</table>

The first thing that becomes clear, is that as the curvature line increases, the distance between the graphs increases as well. That is, as the bending moment increases, the linear gradient in elastic regime for different imperfections gets distant from each other.

For higher initial imperfections, the yielding moment (dashed lines) is reached earlier and the structure withstands the bending less (7% and 21% in severe level of initial imperfections for hogging and sagging respectively). In addition to the initial yielding bending moment increasing for the average level and without imperfections, it is quite notorious that the curvature remains practically the same for the two cases in that point (2% and 4% for hogging and sagging conditions, respectively). From Tables 6.11 and 6.12 is notorious that for sagging conditions, the values’ differences are quite higher in elastic regime between average and severe level (4% to 21%).

Is possible to conclude that the ship response less strong to sagging conditions than hogging conditions and for sagging conditions, they end shortly after entering the plastic regime. The difference between the levels of initial imperfections were expected since a structure with no imperfections can hold better when submitted to vertical bending moment than a structure that has imperfections.

The curve lines in elastic/plastic regime have very dissimilar terminations when compared. While the curvature at non-linear regime for a severe level appears complete, the remain curves are not. For severe level of initial imperfections, the convergence solution was found and did converge, (see Figure 6.17).
In order to evaluate the finite element model for the three different levels of initial imperfections, the nodal solution results are carried out and compared regarding only relevant solutions.

Figure 6.18 – \(x\) component stress for hogging condition (0 Imperfections)

Figure 6.19 – \(x\) component stress for hogging condition (average level of initial imperfections)
Is possible to verify that the initial imperfections have a higher applied level of stress and the imperfections which were not visible in Figure 6.18 on the left side are now prominent. Also, comparing the severe initial imperfections model results with the average initial imperfections on the upper deck, is possible to conclude that with a higher level of imperfections they are no longer at the same size or symmetry as is observed in Figure 6.20. This may be since with a certain applied curvature, the stresses are so higher, that the upper deck deforms as much as with the highest level of imperfections.

Concerning elasticity, in Figures 6.21, 6.22 is possible to observe that the structure model imperfections for average level are uniform and does not merge to the near imperfection’s stresses whereas, for severe level of initial imperfections, the stress are no longer regular or uniform, rather they tend to join and merge with each other. It is also visible that, for severe initial imperfections, there are concentrated stresses in the frames. These are a continuation of the stresses applied on the inner bottom shell. This is confirmed in Figure 6.21, where there are no evident stresses applied while frames have notorious higher stresses between stiffeners and amid ship plate.
In terms of plasticity, when the structure model has no imperfections, only the hatch coaming appears to have heighter levels of stress, lower than the demonstrated in Figure 6.24. For this reason only plastic strains were evaluated in concern for average and severe initial imperfections. It is possible to recognise that the structure model with severe initial imperfections presents a greater area of higher stresses applied, on the hatch coaming, on the left side of Figures 6.24, 6.25. This means that the yielding lines are more prominent. On the right side of Figures 6.24, 6.25, is possible to observe that, unlike what happens for average initial imperfections, the lines of higher stresses appear to follow a path line. In conclusion, the results matched expectations, that higher levels of imperfections lead to more deformation of the structure model.
In conclusion, the results matched expectations, that higher levels of imperfections lead to more deformation of the structure model.

6.6 Corrosion Margin

The approval of the structural design by the regulatory authorities and, in particular, by the International Association of Classification Society's standards must be complied, not under construction conditions, but by deducting the margin for corrosion. Therefore, follows the moment-curvature curve graph in Figure 6.26 before and after removing the corrosion margin for pure vertical bending moment. Table 6.13 and Table 6.14 presents the results for both cases where the differences for hogging and sagging moments are provided, respectively.

![Moment-curvature curve before and after removing the corrosion margin](image)

Figure 6.26 – Moment-curvature curve before and after removing the corrosion margin
Observing Table 6.13 and Table 6.14 is possible to conclude that, in general, FE model under construction conditions, achieves higher values for the yielding moment and maximum yielding curvature (until near 20% highest results), meaning that the structure withstands less to bending after the margin corrosion is applied. The yielding moment considering the corrosion margin is achieved long before than under constructed conditions (under 15% and 20%). With regard to the linear gradient in elastic regime is possible to recognize that FE model under constructed conditions presents a higher slope than the structure after the margin corrosion is applied, between around 4% and 7% for hogging and sagging, respectively.

Concerning to the elastic/plastic regime, the difference is more preeminent for sagging conditions than hogging conditions in terms of maximum plastic moment and curvature achieved. This can be verified in Tables 6.13 and 6.14 where for hogging condition the difference goes till 6% and for sagging condition goes only till 3%.

Since knowing that corroded elements represent loss of thickness, (which means a loss of area), the bending moment which is directly proportional to the moment of inertia, which in turn, is directly proportional to the total area of the elements’ structure, is lower for a minor area. This means that results corresponded as expected. As mentioned, the approval of the structural design by the regulatory authorities must be complied by deducting the margin for corrosion. Table 15 present the results for initial yielding moment and the IACS standards for hogging and moment conditions, respectively.

<table>
<thead>
<tr>
<th>Table 6.10 – Hogging results before and after removing the corrosion margin</th>
</tr>
</thead>
<tbody>
<tr>
<td>Corrosion addition</td>
</tr>
<tr>
<td>---------------------</td>
</tr>
<tr>
<td>Yielding Moment [N.m]</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
</tr>
<tr>
<td>Linear gradient in elastic regime [N.m2]</td>
</tr>
<tr>
<td>Maximum moment achieved [N.m]</td>
</tr>
<tr>
<td>Maximum curvature achieved [1/m]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 6.11 – Sagging results before and after removing the corrosion margin</th>
</tr>
</thead>
<tbody>
<tr>
<td>Corrosion addition</td>
</tr>
<tr>
<td>---------------------</td>
</tr>
<tr>
<td>Yielding Moment [N.m]</td>
</tr>
<tr>
<td>Maximum yielding curvature [1/m]</td>
</tr>
<tr>
<td>Linear gradient in elastic regime [N.m2]</td>
</tr>
<tr>
<td>Maximum moment achieved [N.m]</td>
</tr>
<tr>
<td>Maximum curvature achieved [1/m]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 6.12 – Initial yielding moment without corrosion addition vs IACS</th>
</tr>
</thead>
<tbody>
<tr>
<td>IACS [N.m]</td>
</tr>
<tr>
<td>------------------</td>
</tr>
<tr>
<td>Hogging</td>
</tr>
<tr>
<td>Sagging</td>
</tr>
</tbody>
</table>
Is possible to verify that the structure without the corrosion addition fulfil with the IACS standards. For sagging condition, the initial yielding moment is closer to the IACS rules limit than for hogging condition. Still, both present to be within the rules.
This chapter finalises the dissertation, presenting the achievements of the study and recommendations of further works.
8.1 Achievements

The present dissertation had as main objective to study the behaviour of an ultra large container ship of 10 000 TEUs when submitted to pure bending moment (sagging and hogging moments) using the finite element method. To do that, half of the midship section of a ULCS was modelled in ANSYS software using finite element method. To verify if the FEM complied with the classification society rules and was within the parameters of the beam theory, in which the hull girder is idealized as a hollow thin-wall box beam, the moment-curvature curve for sagging and hogging conditions were designed. The moment-curvature curve of the structural model when subjected to these efforts was analysed yielding important data from which conclusions were drawn. The influence of initial imperfections on local plates and stiffeners was also considered. The progressive collapse behaviour for the vertical bending moment of the structural model was analysed without imperfections and for severe and average levels of imperfections. The progressive collapse behaviour after considering corrosion margin was analysed and compared with the structural model under construction conditions.

The progressive collapse behaviour of the model proved that the ULCS design by FEM fulfils with the IACS standards, remaining in the elastic regime of the progressive collapse behaviour with a satisfactory distance of the yielding moment (38% and 31% for hogging and sagging conditions, respectively).

Concerning the elastic-regime, the finite element model delivered fair results when compared with the beam theory, with relative errors inferior to 5%. Hence, one must conclude that the finite element model analysis is within the parameters calculated by beam theory, meaning that the BC led to satisfactory results.

The moment-curvature curve shown that the model response is more susceptible to sagging conditions than hogging conditions. In general, sagging response ends right at the beginning of the plastic regime whereas the hogging response produces results inside the non-linear analysis.

When the model is subjected to higher levels of initial imperfections, the less it can support the imposed vertical bending moments. I.e., for higher initial imperfections, earlier the yielding moment is reached and less the structure withstands the bending applied.

At the outer bottom of the structure, between the central and the following girder, the bottom area shows some weakness, i.e., deformations. This issue arises from the increased distance between the two girders, which is not the same as the remaining spaces between the others even though the plate is thicker. Also, at the outer side of the structure, under the hatch coaming, the structural model present deformations. This is a consequence of the disparate thicknesses in this area.

The structural model with the corrosion margin subtracted fulfils with the IACS rules standards (19% and 15% for hogging and sagging conditions, respectively).
8.2 Further Work

Although the work done accomplished the presented goals, there is room for improvement, both in the available methods and in their application. This sub-chapter aims to list relevant items in the topic that should be further researched.

First, it is essential to improve the results of the vertical bending moment-curvature curve. This dissertation considers 2D finite element meshing with thin to moderate thicknesses. Knowing that the hatch coaming of the structural model does not fit with last two conditions, a 3D finite element meshing would be more appropriated in this zone to improve the results.

A study comparing the method used in this dissertation with the IACS and master nodes methods would be of interest.

Finally, it would be interesting to combine the vertical bending moment with torsional moment. Since ultra large container ships have low resistance to torsion and vertical bending moment due to their large open decks, a study about these combined moments would be of great relevance.
References


[17] I. Senjanović and Y. Fan, “A higher-order theory of thin-walled girders with application to ship


Structures In the Post Yield Region,” International Society of Offshore and Polar Engineers,


response of an ultra large containership in high seas compared with experiments,” Ocean


[25] C. Wang, J. Wu and D. Wang, “Design similar scale model of a 10,000 TEU container ship through
combined ultimate longitudinal bending and torsion analysis. Applied Ocean Research,” Applied


Kawamoto, P. Kujala, J. K. Paik, S.-R. Cho and U. e. S. B. C. Rohr, “Ultimate Strength (Committee
III.1),” in 14th International Ship and Offshore Structures Congress (ISSC), Tokyo, 2000.

Naar, Y. Pu, P. Rigo, Z Q Wan, “Ultimate Strength (Committee III.1),” in 16th International Ship

[29] T. Yoshikawa, A. Bayatfar, B. J. Kim, C. P. Chen, D. Wang, J. Boulares, J. M. Gordo, L. Josefson,
M. Smith, P. Kaeding, P. Jensen, R. Ojeda, S. Benson, S. Vhanmane, S. Zhang, X. Jiang,
“Committee III.1 - Ultimate Strength,” in Proceedings of the 19th International Ship and Offshore
Structures Congress (ISSC), Portugal, 2015.

container ship subjected to asymmetrical bending loadings,” Marine Structures, vol. 58, pp. 172-
198, 2018.

[31] A. Tatsumi and M. Fujikubo, “Ultimate strength of container ships subjected to combined hogging
69, 2020.


