Design Study for the Transportation of Heavy Cargo on Hatch Cover of a MPV

Abhishek Kumar

Master Thesis

presented in partial fulfilment
of the requirements for the double degree:
“Advanced Master in Naval Architecture” conferred by University of Liege
"Master of Sciences in Applied Mechanics, specialization in Hydrodynamics,
Energetics and Propulsion” conferred by Ecole Centrale de Nantes

developed at University of Rostock, Rostock
in the framework of the

“EMSHIP”
Erasmus Mundus Master Course
in “Integrated Advanced Ship Design”

EMJMD 159652 – Grant Agreement 2015-1687

Supervisor: Dr. –Ing. Thomas Lindemann, University of Rostock, Rostock

Reviewer: Dr. Zbigniew Sekulski, West Pomeranian University of Technolog
Szczecin

Rostock, February 2018

Declaration of Authorship

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

“Design Study for the Transportation of Heavy Cargo on Hatch Cover of a MPV”

I confirm that:

1. This work was done wholly or mainly while in candidature for a research degree at this University;
2. Where any part of this thesis has previously been submitted for a degree or any other qualification at this University or any other institution, this has been clearly stated;
3. Where I have consulted the published work of others, this is always clearly attributed;
4. Where I have quoted from the work of others, the source is always given. With the exception of such quotations, this thesis is entirely my own work;
5. I have acknowledged all main sources of help;
6. Where the thesis is based on work done by myself jointly with others, I have made clear exactly what was done by others and what I have contributed myself;
7. Neither none of this work has been published before submission nor any part of this work has been published;
8. I cede copyright of the thesis in favour of the University of Rostock, Rostock.

Signed:

Date: 15 January 2018

Contents

List of Tables

List of Figures

List of Symbols:

MPV	Multipurpose vessel
SAL Heavy Lift	Schifffahrtskontor Altes Land GmbH
PK-116	Polar Konsult 116
Lwl	Length of Waterline
Lpp	Length between perpendicular
DNV-GL	Det Nortskey Veritas – Germanischer Lloyd
AutoCAD	AutoCAD design software
CG	Class guidelines
GL	Germanischer Lloyd
PSE	Plane stress element
SWBM	Still water bending moment
FE	Finite element
FEA	Finite element analysis
XML	Microsoft file
NASTRAN	Finite element software
E	Young’s modulus
G	Shear modulus
AP	Aft perpendicular
FP	Fore perpendicular
ZIB	Height of inner bottom from basement
KL	Knuckle line
LT	Line type
LG	Longitudinal girder
LB	Longitudinal bulkhead
LBCA	Longitudinal bulkhead at crane aft
LBCF	Longitudinal bulkhead at forward
LBS	Longitudinal bulkhead at superstructure
kN	Kilo newton (S:I unit)
HP	Bulb profile stiffener
ID	Identity
FL	Floor plate of frame section
W	Web plate of frame section

.BMF	Finite element file extension name in Poseidon
POX	Modelling file extension name in poseidon
HEB	European norms to define beams.
SIV RM	Offshore equipment
LCG	Longitudinal centre of gravity
TCG	Transverse centre of gravity
VCG	Vertical centre of gravity
COG	Centre of gravity
Sub-COG	Substitute centre of gravity
2-D	Two – dimensional
3-D	Three – dimensional
S.No	Serial number
AUTO	Automatic
.GLF	GL frame file
PROD REEL	Offshore equipment such as reel
TWD	Twin deck
WD	Weather deck
HFO SB	Heavy fuel oil starboard side
HFO PS	Heavy fuel oil port side
DO SB	Diesel oil starboard side
DO PS	Diesel oil port side
FWSB	Feed water starboard side
FWPS	Feed water port side
BW PS	Ballast water port side
BW SB	Ballast water starboard side
BD SB	Ballast water starboard side
BD PS	Ballast water port side
GM	Metacentric height
SH- HORIZ	Shear force in horizontal direction
SH – VERT	Shear force in vertical direction
BEN – HORIZ	Bending moment in horizontal direction
BEN – VERT	Bending moment in vertical direction
IMO	International Maritime Organization
CFD	Computational fluid dynamics

HT 36	High tension steel grade
Reh	Yield strength as per GL designation
GL-Frame	Poseidon modelling software
WL	Wavelength

Table of Equations:

$$\frac{A_i}{\sqrt[3]{L_{W,i}/L_{pp}}} = \frac{A_j}{\sqrt[3]{L_{W,j}/L_{pp}}} \tag{1}$$

Ps = p. A+ Pi [kN] (2)

$$A_{s,req} = \frac{P_s}{\sigma_p} * 10\ [cm^2] \tag{3}$$

$$\sigma_p = \frac{\kappa}{S} R_{eH} \tag{4}$$

$$\kappa = \frac{1}{\phi+\sqrt{\phi^2-{\lambda_s}^2}} \tag{5}$$

$$\phi = 0.5[\,1 + n_p * (\lambda_s - 0.2) + {\lambda_s}^2] \tag{6}$$

$$\lambda_s = \frac{l_s}{i_s * \pi}\sqrt{\frac{R_{eH}}{E}} \tag{7}$$

$$i_s = \sqrt{\frac{I_s}{A_s}} \tag{8}$$

ABSTRACT

With the current scenario and as per the demand of the offshore industry market, especially in the Scandinavian coastal region which exhibits shallow draft, SAL Heavy Lift GmbH aims to strengthen its presence by offering heavy lifting of offshore equipment like reels, carousels etc. with their state of the art vessels like PK 116, SAL 161, SAL 183 etc. that possess high stability in shallow draft with large crane and cargo hold capacity. However, heavy cargo with tiny footprints may cause strength problems in the deck area, especially for the PK 116 vessel due to its limited deck strength capacity.

To overcome this hurdle, the company decided to strengthen its weather deck such that it can withstand heavier cargo on top of it without compromising its stability in the shallow draft region.

The work scope of the thesis comprises with the vessel PK116 which was minutely interrogated in all aspects to overcome the above obstacles. First of all, the whole vessel with its entire longitudinal and transversal member including Holes and cut-outs was modelled using the Software “POSEIDON”.

Next, strengthening of the Weather Deck was done by modelling the stanchion using AUTOCAD and later on in the POSEIDON as well. The main task was to design and estimate the number of stanchion which can transfer the load from main deck to the double bottom in an efficient manner. After completion of the vessel modelling, the FE-model was generated using Poseidon and then imported in to the “Shipload” to model mass distribution for calculating the hydrostatic equilibrium. After hydrostatic, linear hydrodynamics were calculated to generate load cases and then finally these nodal loads were imported back to FE software again to compute the global strength analysis of vessel.

After successfully analysing the global strength analysis of vessel, the stanchion design were estimated by calculating the total load on the stanchion members.

At last, Deformation, yield stress and buckling check were done to ensure the stanchion design is withholding the designated load on hatch cover / weather deck and successfully transferring in to the bottom structure.

Keywords: DNV- GL, Offshore, Stanchion assembly, Weather deck capacity, Finite element modelling, stability calculations, cargo positioning, optimisation, buckling check, yield and deformation check.

1 INTRODUCTION

1.1 Preface

In the last three decade, with the world reaching to its development paramount right from economic strengthening to skill interpretation it has seen a drastic evolution in transportation and energy sector and to flourish these sectors to its imperial heights, the Shipping industry has played a vital role in conquering some of the most challenging task on and off the seashore, ranging from installation of some of the heaviest and advanced crude oil drilling platform to Tidal power projects. To make this endeavour successful a ship with heavy cargo carrying capacity which can take bulky equipment and platforms from harbour to the offshore site is required. Therefore, to achieve these projects in an efficient and safer way none other than multipurpose vessel would be the best option.

A Multipurpose vessel is a seagoing ship that is built for the carriage of a wide range of cargoes for e.g. wood, steel, construction equipment, offshore structure, production reel etc. One of the great advantages of MPV is that it can carry different kind of loading on the same voyage. These vessels generally have quite complex design because of their varying operating condition which facilitates it to carry large objects, heavy and utilized bulk cargo etc. Not only, it has heavy cargo carrying capacity but also, it can facilitate easy lifting or rolling of the cargo on board with different types of loading gear.

The MPV's generally has crane installed on it which are designed to transport heavy goods.. However, the crane capacity of these MPV's generally determines the cargo consignment which needs to be handled.

In context of the thesis, one such type of existing MPV named as "**PK-116**" which stands for *Polar Konsult* -116 were investigated for its global strength subjected to heavy cargo loading on top of the hatch cover / weather deck. The vessel is used to carry offshore equipment mostly in the Scandinavian coastal region which exhibits shallow water on some of the loading ports. The vessel with its large crane and long cargo hold capacity has massive success in that region in spite of shallow draft throughout its navigation route. However, heavy cargo with tiny footprints may cause strength problem in the deck area due to its limited deck strength capacity. Therefore, to overcome this hurdle, the company has decided to strengthen its weather deck capacity such that it can withstand heavier cargo on top of it without compromising its stability in the shallow water region.

To achieve the above purpose of strengthening the weather deck, a stanchion (pillar) assembly along with its platforms was designed in such a way that it can transfer the load from weather deck area to the tank top area successfully without compromising any structural failure in the entire vessel. To have an overview of the vessel a longitudinal view of PK-116 is added in the figure-1 along with its principal dimensions in the Table –1.

Table 1: Principal Dimensions

PRINCIPAL DIMENSION OF PK-116:	
Title	**Dimensions**
Lwl	125.745 m
Lpp	123.910 m
Breadth	23 m
Depth (nominal)	11.40 m
Scantling Draught	7.8 m
Block coefficient	0.742
Maximum speed in calm water	15 Knots
Deadweight	9963 tons
Displacement	16742 tons

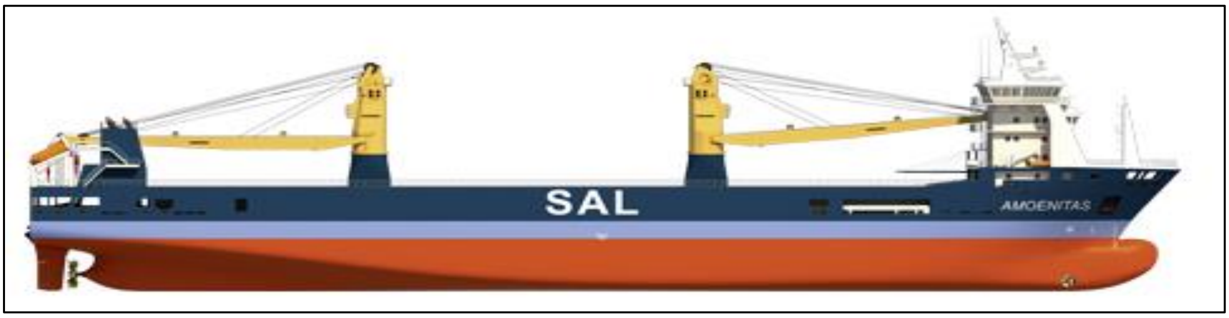


Figure 1: Ship type: PK 116 [Ref: 11]

1.2 Scope of Study

The title of the thesis is **"Design study for the transportation of heavy cargo on hatch cover of a MPV".** The work scope is divided in to various sections and has been explained in the following sequential order:

1.2.1 Study of the DNV-GL rule for global strength analysis of a MPV

Under this section, the rules related to global strength analysis of the vessel will be studied and its related calculations will be performed. But, in purview of the thesis, the Software package for modelling and analysis of vessel would be from DNV-GL, hence all the rules regarding the strength analysis has been already chosen via the software packages. Therefore, no extra calculations have to perform for vindicating these rules and guidelines. The rules and guidelines which have to be followed are presented in the coming sections.

1.2.2 Modelling of the Entire Vessel PK-116 with its hatch cover

To do the global strength analysis, the entire vessel with all its longitudinal and transversal members along with major holes and cut-outs and hatch cover would be modelled using the software **Poseidon**. Since, the vessel already exists so, the AutoCAD drawing of the vessel would be used as a reference for modelling and the aim here, would be to replicate the exact dimensions inside the software such that the realistic model can be achieved at last.

1.2.3 Design of Stanchion Assembly

The main task of thesis would be to design and estimate the number of stanchions which can successfully transfer the load from weather deck to the tank top without compromising any buckling failure. For estimating the design of the stanchion assembly the same software package (Poseidon) would be used. This is the most critical job in the entire thesis.

1.2.4 Load case calculation in Shipload

After finishing the modelling of entire vessel along with hatch cover and its stanchion assembly, the next task would be to do the mass distribution inside the vessel using software **“Shipload”**. Further**,** the wave loading cases were applied on to the vessel to check its hydrostatic equilibrium such that the vessel should be stable and have the correct draft at aft and forward with minimum heel angle. After this the hydrodynamic calculation needs to be done again in the same Shipload software to generate the required nodal forces.

1.2.5 FEA simulation

After generating the nodal forces in shipload, the model would once again brought back to the Poseidon for doing the FE analysis.

1.2.6 Results

At last, the results for deformation, yield stress has to be checked in the hatch cover and stanchion assembly zone of the model along with design feasibility of stanchion assembly, under the permissible value as per DNV-GL rule. In addition to this, buckling of the plate field should be checked in the same analysis zone and a proper result would be published.

2 STUDY OF THE CLASSIFICATION RULES FOR GLOBAL STRENGTH ANALYSIS OF MPV

To achieve the main objective of the thesis which describes as the “Transfer of load from weather deck area to the Tank top area using Stanchion Assembly” in a comprehensive manner, certain guidelines and rules was followed. These guidelines specify the procedure for global strength assessment of multipurpose vessels by means of Finite Element analysis. With the combination of FE analysis and standard rule application it is possible to evaluate complex structures, thus, enabling to do further optimization to achieve greater strength.

The classification society chosen for this assignment was DNV-GL and all the calculation obtained for Bending moment, Shear force, Torsional moments etc. were matched with the standard guidelines provided by the above mentioned class society.

The class rules followed in the thesis in order to carry out the required calculations were as follows:

- DNV-GL-CG-0151 - Class Guidelines
- GL- V – Rules for Classification and Construction Analysis Techniques.
    - Hull Structure Analysis
    - Guidelines for global strength analysis of MPV
- GL – I – Rules for Classification and Construction Ship Technology Seagoing Ships
    - I-1-1 – Hull Structure analysis
        - ➢ Section 2
        - ➢ Section 5
        - ➢ Section 3
        - ➢ Section 17
- Structural Rules for Container ships (I-1-5) Section 5, D 1.2.1.2

The guidelines given by the GL for doing the Global strength analysis mainly comprises of modelling the ship’s structure with its specified boundary conditions where we can apply various load cases to solve system of equations and finally computing the results to match with the associated rules requirement. Although, while going for strength analysis one has to consider overall global deformations and stresses, local deformations and stresses as well as locally induced stresses but, in purview of the thesis, only global deformations and stresses are required to fulfil the main objective.

The prerequisites to perform the Global analysis is to have the entire hull girder and its primary structural members modelled with realistic loading condition such that entire behaviour of complex structures, including interactions between individual components can be taken in to account. In short, the elements which are generally considered for performing the strength analysis are truss elements, beam elements, plane stress elements, plane and shell elements in addition to boundary and spring elements. Also, the stiffness of the structure has to be kept in mind while selection of element types such that we can have the correct stress effect on the structure.

Regarding the loading condition and its respective load cases, it differs from one MPV to another therefore, it is necessary that the loading conditions should be in compliance with the Ship’s loading manual which would be agreed upon with the Classification society.

In general, relevant loads to perform global strength analysis of ship structures can be classified into the following types: ref: [1]

- **Static (Stillwater) loads** from the light ship weight, from the ship's cargo and from the hydrostatic pressures caused by buoyancy and tank contents.
- **Wave induced loads**, i.e. hydrodynamic pressures, loads from accelerated masses and tank contents as well as internal and external hydrodynamic impact forces and other variable loads from the ship's operation e.g. from the action of cranes, stability pontoons, etc.
- Loads on bow and stern structures caused by slamming.

Also, selection of load cases for analysis shall be done in such a way that, with respect to the sum of the forces and moments, either fully balanced load cases are created or realistic sectional forces and / or deformations are obtained at model boundaries and / or supports.

In general, to define loading conditions on the structure two main types of procedure are followed: ref: [1]

- Simplified procedures
- Special procedures

Under Simplified procedures, we have several load cases resulting from defined loading conditions for MPVs such as: ref: [1]

- Harbour loading conditions
- Crane loading conditions
- Loading conditions causing high loads on weather deck hatch covers to obtain seagoing load cases

But, the scope of the thesis lies with the Special procedures which is suitable for the considerations of the wave-induced ship motion and loads. For, specified irregular waves, there are two possibilities to calculate motions and loads: ref: [1]

- Computations in the frequency domain and assessment using spectral method.
- Computations in the time domain and assessment using numerical simulations.

Out of these two procedure, Frequency domain was used with the application of **"SHIPLOAD"** software, where, the first step is to determine the structural response to harmonic elementary waves, in the form of transfer functions which apply for each case of a particular cargo distribution, ship speed and heading relative to the wave direction and these shall be selected with reference to their frequency of occurrence and the structural response to be assessed.

At last, safety against buckling failure is to be determined by considering all calculated stress components in the assessed member area, based on the: ref: [2]

3 DNV-GL RULES AND GUIDELINES FOR GLOBAL STRENGTH ANALYSIS

DNV-GL provides special rule requirements for Global strength analysis of MPV whose main objective is to obtain a reliable description of the overall hull girder stiffness and to compute and assess global stresses and deformation of all primary hull members for specified load cases resulting from realistic loading conditions and wave-induced forces and moments. Generally, global strength analysis is used to calculate the deformation regarding hull girder torsional and transverse strength.

To perform the global strength analysis, a Finite element model of the entire ship is required which would perform the structural adequacy of both longitudinal and transverse primary structure against its deformation and stresses for relevant load cases with in the permissible stresses and buckling.

To perform the Global strength analysis several steps needs to be followed which are depicted as below: ref: [1]

3.1 STRUCTURAL IDEALIZATION

3.1.1 Model Characteristics

To solve the strength characteristics of MPVs, it is necessary to globally model the entire ship structure. Due to the asymmetric structure and the asymmetric loading in seaways, half model are not feasible. The model shall be suitable to capture not only longitudinal and transverse strength aspect, but also structural deformations.

Since, MPVs generally has long cargo hold area with small deck strakes, it possess low global stiffness with respect to torsion and transverse loads, thus, it is important to implement all structural reinforcements that increase the stiffness of the hull. Such reinforcements are e.g., foundation of heavy lift cranes or heavy coaming stays and foundation for hatch cover stopper forces.

Various loading conditions requirement should be kept in mind to evaluate the global strength which is given below:

3.1.2 Heavy mass loading

Regarding generation of loading conditions, the loads should be applied realistically, i.e., heavy loads shall be transferred at correct positions into the ship structure. To achieve this, in some cases auxiliary structures only used for load application are necessary.

3.1.3 Crane loading

For crane load cases, simplified models of crane columns for load application have to be implemented in to the global model. These crane column models shall be able to transfer crane moments and forces from their rotating assembly to the column foundation, requiring correct modelling of the stiffness of all major structural components of the crane column.

3.1.4 Hatch cover loading

The importance of the hatch cover stopper forces on local deformations of the hull, especially for the inward/outward deflections of the coaming, necessitates modelling the hatch covers or implementing an auxiliary system of hatch covers to correctly transfer hatch cover stopper forces at the top of the coaming into the ship structure. Each hatch cover shall be fitted with longitudinal and transverse stoppers according to the hatch cover force plan.

Only large deformations and large contact forces affect the deformation of the hull by keeping the defined clearances at their deflection limiters. Under such conditions, hatch covers transfer forces from one ship side to the other. This leads to a nonlinear problem, which can be solved using contact elements.

Furthermore, on each hatch cover it shall be possible to define loads acting at a prescribed position of the cargo’s centre of gravity.

Also, it is important to note that the Friction forces between hatch cover and coaming are to be neglected for global strength checks, which, enables the determination of maximum coaming deflections as well as maximum stopper forces at deflection limiters.

3.2 Selection of mesh fineness

Mesh size shall be determined according to the scope and kind of structural design and structural results which have to be assessed.

For global strength analysis, the three-dimensional models of the entire hull girder are often meshed coarsely, according to spacing of primary structural components, provided, the element types used here accurately reflect the bending behaviour of the primary structural components.

3.2.1 Definition of principal sections

The principal sections of the MPVs typically shows:

- Longitudinal sections of longitudinal bulkheads, longitudinal walls and longitudinal girders.
- Transverse sections of transverse bulkheads and walls, floor plates, transverse web plates and frames.
- Horizontal sections of the coaming top, main deck, second deck, stringers and inner bottom.
- Crane foundation structures.

The number of transverse sections shown depends primarily on the floor plates arranged in the area of long cargo hold regions. Generally, each floor plate is to be idealized in this region.

3.2.2 Coordinate system and units

A right-handed Cartesian Coordinate system should be used where,

- X measured in the longitudinal direction, positive forward from the aft perpendicular.
- Y measured in the transverse direction, positive from the centreline to the port.

- Z measured in the vertical direction, positive upwards from the baseline.

3.2.3 Element types

In order to decide the element type, we need to know what kind of stresses to be considered while analysing the global effect, as we know, the global model does not analyse the local effects like bending of stiffened plates subject to water pressures. The dominant result is the membrane stress state.

All primary structural members, i.e., shell, inner skin, girders, web frames, horizontal stringers and vertical girders of transverse bulkheads are to be idealized, preferably by four-node plane stress or shell elements. Secondary stiffening members may be idealized by two-node truss or beam elements. High transverse and longitudinal girders can be idealized either by beam elements or by plane stress elements (PSE) for webs and truss elements for flanges.

When using different element types, attention shall be paid to the compatibility of the displacement functions as well as the transferability of the boundary loads and stresses, particularly for the coupling of elements with and without bending stiffness at the nodes. With coarse meshes used for the global analyses, it is beneficial if shape functions of plane stress or shell elements include “Incompatible modes”, which offer improved in-plane bending behaviour of the modelled member. This type of element is required to model web plates with a single element over the full web height in order to calculate the bending stress distribution correctly. But, the disadvantage of the incompatible mode is that the element edges may diverge, causing a Lower stiffness. However, if used in combination with the coarse mesh, these elements realistically reproduce the stiffness of the hull girder.

The element edge aspect ratio shall generally not exceed the value 3.0. The aspect ratio may be exceeded in areas of low stress gradients or where a constant stress distribution over the element possible, Triangular elements with a linear shape function shall be avoided because three-node elements can only represents constant strain or stress. They have no in-plane bending characteristic and are, therefore, too stiff in areas of significant stress gradients. Also, Four-node elements with inner angles below 80 deg. or above 100 deg. between edges shall be avoided as well.

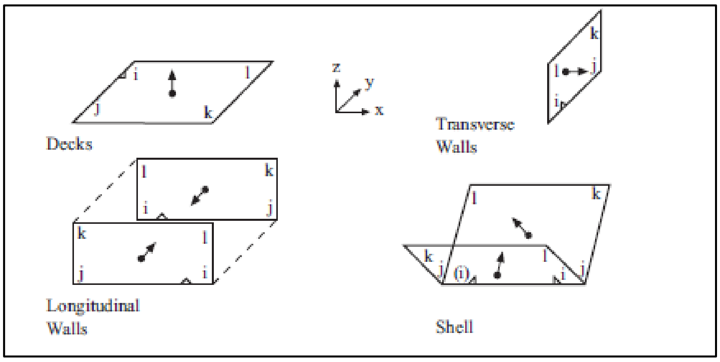


Figure 2: Element orientation and normal vector [ref 1]

3.2.4 Modelling the Structure

The FE model is to be based on gross scantling of the hull structure and for buckling evaluations; the corrosion addition will be deducted. Small secondary components or details that only marginally affect the stiffness can be neglected in the modelling. Examples are brackets at frames, sniped short buckling stiffeners and small cut-outs. Man holes or cut-outs of significant size shall always be considered when calculating realistic shear stresses. A corresponding reduction in element thickness may be considered to reduce the stiffness. Even larger openings corresponding to the element size, such as pilot doors, may be considered by deleting the appropriate elements.

Sometimes, the plate thickness or scantling of profiles, which are not suitable for on element boundaries, shall be taken into account by adapting element data or characteristics to obtain an equivalent stiffness. Plane elements shall generally be positioned in the mid-plane of the corresponding components. As an approximation for thin-walled structures, elements can also be arranged at moulded lines.

To reflect global stiffness behaviour as correctly as possible, plane two-dimensional elements in inclined or curved surfaces shall be positioned at the geometric Centre of the modelled area.

The FE modelling in this thesis has been done in GL-Frame where translatory singularities in PSE structure have been avoided by arranging so-called singularity trusses as shown below:

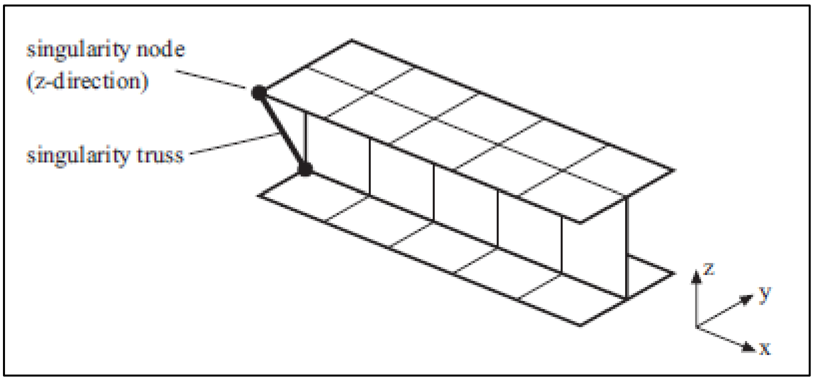


Figure 3: Singularity trusses [ref 1]

For coarse meshes, stiffeners have to be assembled as trusses or beams by summarizing relevant cross-section data. They have to be arranged at the edges of plane stress or shell elements.

3.2.5 Boundary conditions

The boundary conditions have to be applied in to the model in order to eliminate rigid body motions of the entire global finite element model, six supports or boundary elements (springs or high stiffness) have to be arranged for this purpose. As ship and cargo weight are in equilibrium with buoyancy and wave loads, these boundary elements transfer no loads. This has to be checked.

Also, Care shall be taken to locate boundary elements in a way to avoid unrealistic hull deformations.

3.3 Loading conditions

According to the scope of the global strength investigation, several loading conditions have to be generated to represent harbour load cases, crane load cases, and seagoing load cases. But, in purview of the thesis only Seagoing load cases has to be considered.

It should be insured that for all loading conditions the permissible still water values of bending moment, (SWBM), shear force and torsion are not exceeded. The design static torsional moment correspond to the characteristics of MPVs. If the loading manual does not specify values of the static torsional moment, it has to be taken according to GL rule: ref: [3]

General points to be considered for defining the loading conditions are discussed below:

- It should be insured that for all loading conditions the design loads on decks and hatch covers should not be exceeded from permissible.
- Mass distributions of loading conditions shall be selected in a way that possible maximum and minimum bending moments, draughts and lateral double bottom loads are considered.
- Hydrostatic and hydrodynamic calculations need realistic mass distributions with correct positions of centres of gravity to obtain correct still water floating conditions and ship motions in waves. It should be insured that for all loading conditions the minimum required metacentric height (GM) are maintained.

3.3.1 Mass modelling

Under mass modelling, Lightship weight components, such as hull structure, machinery and equipment, outfitting, etc. are the same for every considered loading condition, whereas, consumables, effects and stores will vary little if at all. Other weight groups, such as containers and ballast water shall be defined according to each loading condition.

3.3.2 light ship weight

The light weight of the ship is obtained by applying a material density to the FE elements. Generally, an increased value of light weight tonnage is used to account for the structural components that are not included in the model such as brackets. To match a specified centre of gravity position for the hull structural weight, different material densities can be used for individual element groups.

The remainder of the lightship weight (such as machinery, hatch covers and outfitting) and consumables will be represented by a distribution of nodal masses in relevant regions according to their locations and Centre of gravity.

The mass of each weight group will be adjusted to achieve the correct mass distribution and the correct position of the Centre of gravity. The use of negative nodal masses is to be avoided. The entire mass model shall be in compliance with the considered lightship weight distribution.

3.3.3 Water ballast and tank components

The water ballast and liquid mass in tanks to be represented by nodal masses distributed to the surrounding structure. It is not necessary to include the local pressure distribution in tanks in the global FE analysis.

3.3.4 Container loads and general cargo

Two ways to transfer the Inertia forces of cargo to the appropriate nodes in hull structure are given as follows:

- If forces are transferred to the ship interface nodes prior to a finite element calculation, no explicit auxiliary model is required for the finite element calculation itself to account for the containers.
- If auxiliary systems are used for load application and load transfer, they shall not influence the stiffness of the FE model. This has to be checked by test calculations without loads acting on the auxiliary systems. Hull deformations must not cause stresses and strains in the auxiliary systems. On-deck containers and cargo can be modelled using plane stress, shell or solid elements, which may be connected via the hatch covers to the hull structure by truss elements. The cargo or the hatch covers have to be supported on the coaming using vertically oriented truss elements. At the location of transverse and longitudinal stoppers, the structure of hatch covers will be supported either in the transverse direction only or in the transverse and the longitudinal direction, respectively. The centre of gravity of on-deck cargo has to be correctly represented to obtain realistic heeling moments. If cargo in holds is modelled by an auxiliary system, again special attention shall be paid to the transfer of vertical and horizontal forces to the appropriate nodes in the hull structure in order not to influence the stiffness of the ship.

3.4 Mass distribution for seagoing load cases

3.4.1 Mass distributions with heavy loads on hatch covers

Mass distributions for seagoing load cases with heavy loads on hatch covers are generated to check the transverse strength and deformation behaviour of the hull and the top of the coming under sea conditions causing large roll angles. Normally, these loading conditions have a small metacentric height.

To achieve the correct computation of global strength, a maximum of 80 % of the design loads have to be applied on top of the Hatch covers. The Centre of gravity of loads on hatch covers should be located at a low but realistic vertical position.

To obtain large transverse deformations, loads on hatch covers should be concentrated in the mid area of the hold. Loads on hatch covers at the ends of a hold only marginally influence global transverse deformations

Loads at specified hatch covers have to be defined with regard to following aspects: ref: [1]

- Maximum and minimum possible vertical bending moments have to be defined.
- Maximum and minimum possible inward and outward deflections affected by lateral pressure and bending deformations of the double bottom have to be defined.
- Loading conditions have to be defined for the scantling draught only.

Also, if the ship is designed for two design draughts, corresponding to closed and open weather deck hatch covers, additional loading conditions have to be generated for open top cases. Under these loading conditions, where not all hatch covers are equipped with deflection limiters, maximum stopper forces at deflection limiters of the remaining hatch covers have to be verified.

3.5 Load cases

Two different load cases have to be defined for the relevant loading conditions.

1. Static (still water) load cases
2. Wave- induced load cases.

Generally, static load cases has to be generated for all loading conditions but, the wave-induced load cases has to be derived only for mass distribution of seagoing load cases.

The load case generation is usually done with the help of suitable software programs such as GL-Shipload. Such programs shall consider: ref: [1]

- Mass distributions of loading conditions
- Hydrostatic pressures at the shell
- Wave-induced ship motions and accelerations
- Hydrodynamic pressures at the shell

3.6 Wave load analysis

For doing the wave load analysis, a load generation program is required which can generate loads for static and hydrodynamics cases. In context of this thesis, a program developed by GL called "GL Shipload " has been used which uses strip theory and enables fast simulations of ships in regular waves of different wavelengths, wave height, wave directions, wave phase angles and ship speeds.

Preliminary requirement to define wave loads on the structures are given as follows: ref: [1]

1. The load generation has to be performed with reference to the GL Rules: ref: [4]
2. Seagoing load cases defined with in this guidelines are assessed as combined load cases in accordance with GL Rules. In such cases, only 75% of the maximum vertical wave bending moment has to be considered.
3. The load generation has to be performed for a ship speed of two-third maximum service speed.
4. First of all, we have to define the design wave conditions by taking the most sensitive wave length for vertical wave bending moment combined the smallest wave amplitude which yields the vertical bending moment according to the rules.
5. Based on the design wave obtained, it is assumed that wave amplitude for different wave lengths depend on the cubic root of the wavelength as follows: ref: [4]

$$\frac{A_i}{\sqrt[3]{L_{W,i}/L_{pp}}} = \frac{A_j}{\sqrt[3]{L_{W,j}/L_{pp}}} \tag{1}$$

   Where,

   i – Index for design waves

   j – Index for waves in general

   A – Wave amplitude

   LW – wave length

   Lpp – Ship length between perpendiculars

6. To avoid unrealistic load combinations, it is assumed that maximum wave amplitude and maximum roll angle do not occur simultaneously.

7. In general, for simulations in regular waves a large number of wave situations is systematically analysed by considering different wave lengths, wave heights, wave angles of encounter, wave phases (position of wave crest relative to the ship) and additional roll angles.
8. Relevant load cases for FE analyses generally are to be selected by evaluating sectional forces and moments along the ship’s length for all analysed wave situations. For these load cases, vertical and horizontal wave bending and the torsional moments have to match design values defined in the Rules. For MPVs the transverse acceleration of hatch cover loads is an additional essential item for the load case selection.
9. Selected seagoing load cases shall cover realistic combinations, consisting of vertical and horizontal moments, torsional moments, shear forces and large transverse forces leading to racking conditions.
10. Load case selection has to be done in a way to obtain the largest stress values and stress ranges relevant for fatigue.

3.7 Seagoing load cases

3.7.1 Transverse strength and racking load cases

Out of all the loading cases, only seagoing loading cases have been selected to analyse the Global strength of the vessel. Under this, mass distribution with high loads on hatch covers are generated to check the transverse strength and deformation behaviour of the hull, essentially induced by acceleration of hatch cover loads leading to racking conditions. The racking conditions reach to its extreme conditions when it combined with large roll angles. These loading conditions are generally utilized for analysing the structural design and scantlings of MPVs in addition to its large torsional and horizontal bending moments induced in to the structure.

While setting up the roll angle for racking load cases, it has to be adjusted in such a way that the transverse acceleration of 0.5 g for the inertia forces of hatch cover loads should be approximately equal to the loads defined according to the GL rules: ref: [6]

In this way, it is assumed that calculated stopper forces are in line with the design hatch cover forces according to the rules.

Also, it is common that for each loading condition, the Stillwater load case, the maximum wave hogging load case and the maximum wave sagging load case are generated.

Racking load cases generally, are set up for hogging and sagging waves from ahead (angle of encounter 180 degrees) with respect to variation of side shell pressure. Therefore, load cases with different wave crest and wave trough positions in the area of largest transverse deformations have to be generated. This would ensure the full effect of hydrodynamic pressures on transverse hull strength, hull deformations and stopper forces has been accounted for. Sometimes, due to asymmetry of many MPVs, load cases with roll angles to port and starboard side have to be defined separately.

There are two different cases of loading conditions have to be applied on the basis of length of cargo hold which is described below: ref: [1]

CASE 1: WHEN LENGTH OF CARGO HOLD MORE THAN 40 m

For long cargo hold (more than 40 m), three wave crest and three wave trough positions have to be considered. For prismatic hold geometry, wave crest and wave trough positions can be assumed located at one-quarter, one-half and three-quarter lengths of the cargo hold.

The load case selection described above leads, for a long cargo hold and for one loading condition to 12 racking load cases. These cases arise from six wave phases (positions), one wave length of design wave length, one wave amplitude of 50 percent design wave amplitude, one angle of encounter of 180 degrees and design roll angles to port and starboard side.

CASE 2: WHEN LENGTH OF CARGO HOLD LESS THAN 40 m

Cargo hold less than 40 m is considered as short cargo hold, for this case, only one position for wave crest and trough is recommended. The position has to be estimated at one-half length of the cargo hold. The reduced number of wave phases and roll angles to port and starboard side lead to four racking load cases.

3.8 Model check

Once Finite element model is ready, it has to be checked systematically for the following possible error. ref:[1]

1. Fixed nodes
2. Nodes without stiffness
3. Intermediate nodes on element edges not connected to the element
4. Trusses or beams crossing shells
5. Double elements
6. Extreme element shapes (element edge aspect ratio and warped elements)

In addition to this, geometric descriptions of all elements and its material characteristics have to be verified along with the moments of inertia, section moduli and neutral axis of the complete cross section. For each load case, the sum of forces and reaction forces of boundary elements shall be negligibly small.

To check boundary conditions and detect weak areas as well as singular subsystems, a test calculation run is to be performed. The model should be loaded with a unit force at all nodes for each coordinate direction. This will result in three load cases – one for each direction. The calculated results have to be checked against maximum deformations in all directions. This test helps to find areas of improper connections between adjacent elements or gaps between elements. Substructures can be detected as well.

3.9 Evaluation

3.9.1 Deformation of seagoing load cases

All seagoing load cases have to be evaluated for deformations and stopper forces. Also, maximum design forces and displacements have to be determined.

Generally, the MPVs which have large deck openings, long cargo holds etc., give rise to large deformation under seagoing conditions due to the torsional and transverse loads. These deformations have to be limited in order to assure a safe working ship structure. In particular, severe racking load cases may cause large inward and outward deflections, which make it impossible to design safe working hatch covers without resorting to modifications. An established modification measure is the arrangement of deflection limiters to ensure that predetermined maximum inward and outward deflections do not exceed. The clearance of these deflection limiters shall be large enough to allow hatch cover operations under all harbour conditions.

But, during heavy lifting operations as in the case of this Master thesis, this requirement may be disregarded because the deformation may exceed stopper clearances and hatch covers then transfer the loads. Certainly, under such conditions the hatch covers cannot be moved.

The process of evaluation of Deformation has to be performed in the following ways: ref: [1]

- Calculate inward and outward deflections for seagoing load cases
- Evaluate coaming deflections and specify number and arrangement of deflection limiters
- Calculate stopper forces at deflection limiters by solving nonlinear contact problems and determine maximum design stopper forces
- Evaluate final deformations affected by deflection limiters, determine hatch cover movements on top of coaming and relative displacements between hatch covers and obtain maximum design movements and displacements.

3.9.2 Stresses

Stresses for seagoing load cases have to be assessed on the basis of the permissible nominal stress limits for normal, shear and equivalent stresses as follows:

Normal stress / principal stress–**175**/k [N/mm^2]
Shear stress -**110**/k [N/mm^2]
Equivalent stress -**190**/k [N/mm^2]

Table 2: Material Factor K

Material Factor(K):		
S No.	**R_{eh}(N/mm^2)**	**K**
1.	235	1.0
2.	315	0.78
3.	355	0.72
4.	390	0.66

These values are valid for the evaluation of membrane stresses. For each element of all primary structural components, stress plots shall document maximum stresses.

The stress documentation has to contain information regarding the described structure, the load case, the stress representation and the stress scale.

3.9.3 Buckling strength

Buckling strength of the structure is checked for compliance with GL Rules for ref: [4] with safety factor of S = 1.1.

Biaxial element stresses of the FE model contain the Poisson effect, and stress values are modified according to the specifications in GL Rules: ref: [4]

4 POSEIDON

Poseidon is modelling software developed by Germanischer Lloyd to support the calculation of scantlings for ship structures at preliminary design and construction phases in shipbuilding industry. The scantling criteria are performed according to the

- Germanischer Lloyd Rules for Classification and Construction (in particular to Volume Ship Technology, Part 1, Chapter 1) and
- Direct Calculation Procedures (Finite Element Analysis)

Both procedures use the same descriptions of structures and loads. Also, Poseidon does the assessment of various design variations under economic aspects as well as the optimum utilization of material and the identification of critical structural areas.

POSEIDON offers some of the notable features which are described below: ref: [7]

- POSEIDON is applicable to all types of ships. The program is based on a ship type independent description of all structural components that are relevant for classification. The stored ship model can also be used in connection with the optional class notation RSD (*Rational Ship Design*).
- The structural elements as well as plate and stiffener definitions can be asymmetric to the centre line. Symmetrical members are described efficiently by assigning a symmetry property.
- The designer is always able to control and interact with the running program. In particular, it is possible to jump between different levels in the program hierarchy. Scantling results, protocol files, the on-line reference manual and a large variety of graphical representations can be rendered on screen and printed.
- A hypertext version of the Germanischer Lloyd Construction Rules can be viewed at any time. The text can be browsed and searched.
- POSEIDON can automatically generate Finite Element models. Mesh refinement and other aspects are user controllable. Loads according to GL Construction Rules can be generated and the resulting static equations (as well as free vibration problems) can be solved. Finally, export filters to external Finite Element programs will be provided in the near future.
- To support the assessment of the ship hull during service life, POSEIDON can store measured thickness values and use them to perform re-analyses according to the GL Construction Rules.

As discussed, Poseidon is used to model the Hull structure of a ship. The steps to perform the Hull structures modelling are given as follows:

4.1 Modelling the hull structure

4.1.1 Principal dimensions

The quantities included under principal dimensions are length between perpendiculars, length of waterline, scantling length and breadth, depth, design draught, operational speed, and block coefficient. In addition to this, one can specify information about class notations, or range of service (e.g. Ice class).

4.1.2 Positions in the longitudinal direction of a ship

Generally, framing system is used to define the longitudinal direction of the ship, thereby serving as basis for the exact definition of the location of structural members. It also contains global information. In addition to the frame system, the longitudinal direction can also be defined by relative x-coordinates in relation to the scantling length. Relative coordinates are real values ranging from 0.1 to 0.9, but with a single decimal, with 0.5 L being equivalent to the position of the amidship section.

4.1.3 Longitudinal members

A longitudinal member contributes to the longitudinal strength of the ship. In Poseidon, there is a terminology called Functional Elements which is used to define longitudinal members of a ship. Under this, the user has to assign a unique number to a functional element which would identify, store, list, sort, and present the longitudinal members. The examples for functional elements inside Poseidon are shell, Inner bottom, Deck, Longitudinal girder and longitudinal bulkhead.

4.1.4 Transverse member

Transverse members are defined by so-called *Cells*. In the POSEIDON context this term means a closed contour in the Y-Z-plane, which is specified as a sequence of Functional Elements and points. Each point is characterized by two coordinate values or a coordinate value and a reference to Functional Element. Consecutive points are connected by lines or circular arcs, respectively. Cells, which are completely enclosed by Functional Elements can be determined automatically by POSEIDON, if plate arrangements have been previously defined for all corresponding longitudinal members (resp. Functional Elements). The definition of cells based on Functional Elements guarantees, that the cell geometry is adjusted automatically each time the geometry of an underlying Functional Element changes.

4.2 Plates, holes and profiles

Out of the above three definitions, plate should be assigned first for any structural components before assigning the holes and profiles to any longitudinal and transverse members. Plates, profiles and holes descriptions may be valid for many positions in the ship’s longitudinal direction. The real scope for plates and profiles on longitudinal members is defined by assigning a reference section to the actual cross section. POSEIDON takes over the definition of plates and profiles for a Functional Element from the reference section only, if there is no explicit definition of plates and profiles for that Functional Element at the current cross section. This means in particular, that even if a single plate or a single profile shall be changed, then all plates and stiffeners have to be described explicitly at the current section. (They may be copied to the current section first.)

4.2.1 Plates

Due to the two dimensional view of the structure used by POSEIDON for calculation of scantlings, it is sufficient to describe plates as segments on the moulded geometry or contour of the Functional Element. It is generally defined with the position of Functional element i.e., it is specified in relation to the start of Functional element, end of Functional element or end of preceding plate. Please note, that the orientation of the plate arrangement is always identical to the orientation of the contour of the Functional Element. Also note that for transverse members, it is only possible to define exactly one plate per cell or component.

Plates can be defined by four parameters as follows: ref: [7]

a) Plate thickness
b) Material type
c) The position of the material relative to the moulded line, which is equivalent to the geometry of the functional element.
d) An attribute containing additional information about the plate e.g., whether it shall be corrugated and scantling parameters.

4.2.2 Profiles

Again, due to the two dimensional view of the structure it is sufficient to describe profiles in the longitudinal direction by the position of the profile foot point and the orientation of the web. It should be noted, that the stiffener description for transverse stiffener is different from those of longitudinal ones. Therefore, transverse stiffeners are defined separately.

Properties of profiles are described by the following parameters: ref: [7]

a) **Profile type** : e.g., HP, L, T
b) **Profile dimension**: The dimension has to be specified only, if minimum dimensions are available from design.
c) **Material type:** This is a reference into the list of material types.
d) **Unsupported length**. : Only applicable for profiles in longitudinal direction. This information describes the distance to the next supporting transverse member and is used for scantling calculations in accordance with GL Construction Rules.

4.2.3 Holes

In POSEIDON holes are specified by defining the location of the centre point together with the extent in two directions. For holes on longitudinal members at a fixed cross section, the centre point is specified on the contour of the Functional Element, therefore a single coordinate, either Y or Z, is sufficient. For transverse member, a unique description at a fixed cross section requires a pair of coordinates (Y, Z) for the centre point. In both cases the (absolute) coordinates may be replaced by a quantity, specifying a value relative to the total extension of the Functional Element or transverse member, respectively.

5 GL SHIPLOAD

GL Shipload is an efficient tool developed by **"Germanischer Lloyd „to** generate efficient loading conditions which are mainly due to the acceleration of masses (inertial loads) and from external loads (mainly pressures) for global FE analysis of ship structures. It supports modelling of the mass distribution of ship and cargo and computes static and hydrodynamics pressure due to waves. Both type of loads are combined to yield balanced quasi-static load cases. The program not only supports the generation of load from realistic inertia and wave loads for user supplied wave parameters but also it aids in the selection of relevant wave situations for the global strength assessment based on bending moments and shear force according to GL's rules. Finally, the result it gives is a small number of balanced load cases that are sufficient enough for dimensioning of the hull structures.

One of the main advantages of GL Shipload is that it can be easily applied in a typical design environment e.g. a shipyard, since it only requires a global FE model of the ship as input, but, when it comes to giving output it consists of nodal forces that can be applied in any standard FE program. Realistic loads are applied by performing a first principle non-linear seakeeping analysis. Some of the main feature that makes GL Shipload very user friendly are given as follows: ref: [8]

1. **Convenience:** Clear layout, re-use of data from other programs, copy and paste
2. **Objectivity: Reproducible** result according to the GL guideline for global FE analysis.
3. **Reliability:** Assessment of results by graphical feedback.
4. **Efficiency: Easy** to use and fast to apply.
5. **Transparency:** Any user inputs into the software are stored in an XML file. In this way, data input is transparent to other developers and GL Shipload can be integrated into the other environments.
6. **Flexibility:** Useful for multiple FE systems, data model independent of the hydrodynamic method.
7. **Familiarity:** GL software look and feel.

GL Shipload mainly operates on the basis of nodes, elements, and materials as used by any standard FE program. The program deals with the files for reading information (FE model, hull description), writing results (loads), storing user input data, and for communicating with external programs (hydrodynamics, export for e.g. NASTRAN).

5.1 Principal dimensions

The program starts with entering some principal dimension values mainly for the computation of bending moment rule values according to the rule: [5]. The total vertical bending moment is considered as the sum of the still water bending moment and vertical wave bending moment. Still water bending moment can be entered into or computed according to the rule: ref [5] by considering both Hogging and Sagging loading conditions.

5.2 Shell elements

During, the selection of shell elements, only the outer plating or “shell” of the hull including the stern plate must be identified for the computation of hydrostatics and the application of hydrodynamic loads. For doing the selection of the Shell element two techniques are available first one is “Automatic” and second one is selection by individual element group.

Under **“Automatic”** selection, the outer shell elements are automatically recognized up to the maximum height. The shell must be free from any free edges up to the denoting height and there should not be any openings between any elements.

In case of **“Element Groups”**, the groups corresponding to the outer shell are directly selected by the user with the same condition as in automatic case i.e., it should be free from any free edges and no openings present between two elements where hydrodynamics forces are likely to act.

After execution of the shell element group selection, it would give its output as the outer shell in **3**D-View of the FE – model.

5.3 Frame table

A frame table is defined to allow the addressing of longitudinal positions of the ship structure by frame number rather than coordinate values.

5.4 Hatch cover

The hatch covers are defined inside the model to mainly describe the deck containers on top of it. In other words, the masses and by extension, inertial forces and moments of the deck containers in longitudinal and transverse direction are applied to the nearest specified bearing points. Vertical forces of the deck containers are applied to nodes nearest the forward and after edge of the hatch covers. More importantly, the bearing locations are specified with the direction of load carrying.

5.5 Masses

The intended operation of mass distribution in GL Shipload with reference to the Rules and guidelines are roughly estimated as below: ref: [8]

- Complete ship FE Model divided into element groups is the starting point for steel mass.
- Definition of Box Masses for the remainder of lightship mass.
- Definition of the large tanks by finding their closed volumes in the FE Model. The tanks needed would correspond, for example, to loading conditions which reach the required still water bending moments at design draft.
- Definition of Box Masses for any additional small (and presumably not modelled) tanks in the Machinery space, to achieve the aforementioned loading conditions.
- The assembly of components of the loading condition starting with lightship weight. To compute this assembled mass item, the weighting factors of selected FE groups will be automatically sought to achieve lightship mass and centre of mass.
- The weighting factors of the FE groups will then be fixed, and future loading conditions will use this assembled mass with a factor of 1.

- The fill rate of variable consumable tanks will be set to correspond to the bunkering conditions desired. These conditions will be specified with unique assembled mass items.
- In the case of another ship type, the fill rate of the cargo tanks or weighting factors of cargo box masses are entered to the aforementioned loading conditions. Each loading condition to be computed will be associated with a unique assembled mass item.
- In the end, this approach enables the user to easily manage numerous loading scenarios, with the later assembled mass items.

There are a total of three ways to define the masses in GL Shipload which are given as follows:

1. Box masses
2. Tank masses
3. Container masses

5.5.1 Box masses

Box masses can only be defined by the user on the corresponding nodes of the FE-model according to the prevalent situation which comprises of light weight component such as machinery, equipment, outfitting etc. to represent the distribution of nodal masses in relevant regions as per their locations and centre of gravity. Each box mass is described by one or more rectangular boxes. The nodes with in the defined box or on the box boundary would receive weighted positive mass to reproduce the effect of the component.

Box masses are defined in a Cartesian coordinate system where users can enter the mass value, centre of gravity and boundaries up to which box is designated.

5.5.2 Tank masses

Like box masses, tank masses are also defined by the user with their corresponding nodes of the FE- model. The liquid in tanks is represented by masses corresponding to the local pressure distribution on the bottom and sides of the tanks. There are two methods available to define the tank boundaries in shipload. First one is Surrounding Box method where, each tank can be selected by one or more boxes surrounding all boundaries of the tank. With this input, the program is wise enough and it will search automatically in the FE model for one or more closed volumes inside of the defined box (es). Second method is in box method, where, a point inside the tank is defined which would enable the program to search automatically a single closed volume outward to this point inside the FE model.

5.5.3 Container masses

The container masses are defined according to the numbering system of container bays, rows and tiers corresponding to the ref: [8]

The bays, rows and tiers numbering in the GL Shipload are given as follows:

5.5.3.1 Bays

Bays are counted from the bow to the stern, sometimes with multiple bays for a single hold. Bays are numbered odd or even to represent 20 and 40 foot containers, respectively. They need not be sequential. The input of the hold and deck containers for a given bay must be separated. While not a requirement, bays numbered greater than or equal to 100 are generally used for deck containers, while bay numbers below this value generally used for hold containers.

5.5.3.2 Rows

A row is vertical and counted increasing outboard. On centreline a row is denoted as row 0. All rows with even numbers are on port, with odd numbers or on starboard. Rows that are symmetric on port and starboard also may have odd numbers.

5.5.3.3 Tiers

Tiers are numbered increasing from baseline. All tiers have even numbers, generally with 02 being nearest the baseline. Tiers numbered greater than or equal to 82 are on deck; below this value they are in the holds.

5.6 Assembled mass

The assembled mass item is a combination of all the masses like lightweight, bunker, water ballast, deck cargo and hold cargo together, with an average centre of gravity and radius of gyration automatically computed by the GL Shipload program. It is therefore, represent loading conditions of the whole vessel. These items will be numbered and may be used with factor to specify complete loading condition(s) or other assembled mass item.

5.7 Hydrodynamics

GL Shipload’s intended operation to create hydrodynamic loads is given below: ref: [8]

- Establish a hydrostatic equilibrium based on the mass and centre of mass for the loading Condition.
- Calculate linear hydrodynamic potentials by a strip method.
- Computation of design wave amplitudes from prescribed bending moments.
- Specification of the scan range of wave parameters.
- Computation of pressures/section loads for this scan range with a non-linear correction.
- Selection of load cases (manually or automatically by section load extrema).
- Generation of balanced nodal load cases.

5.8 Linear hydrodynamics

The hydrodynamics calculation begins with the process of strip theory, which is a frequency-domain method for computing hydrodynamic potentials, accelerations and in this case also pressures for regular (harmonic) waves by dividing the hull in to cross sections and thus reducing the three-dimensional computation in to a series of two-dimensional computations. The strip theory is valid for slender bodies like ships.

5.8.1 Reference wave amplitude

It performs the computation of design wave amplitudes and reference wave amplitude to achieve the prescribed bending moments.

5.8.2 Equilibrium

In this section, we can get the display details of the static trim conditions and the sectional forces over the entire ship's length.

5.8.3 Scan

This section is used both to specify and perform one or more scans of various wave parameters. It computes the pressure and section loads for ranges of wave parameters entered.

5.8.4 Select design load cases

Under this, the load cases for creation of nodal forces are defined. Load case title must be unique to identify each load case specifically.

5.8.5 Generate design load cases

It displays a summary of load cases created in the following window.

5.8.6 Load factors

The load factors are set by the program and generally will not need to be changed by the user. For each input line, there is a load group linked with to the global load cases through a positive or negative scaling load factor. In each column, there is one global load case included.

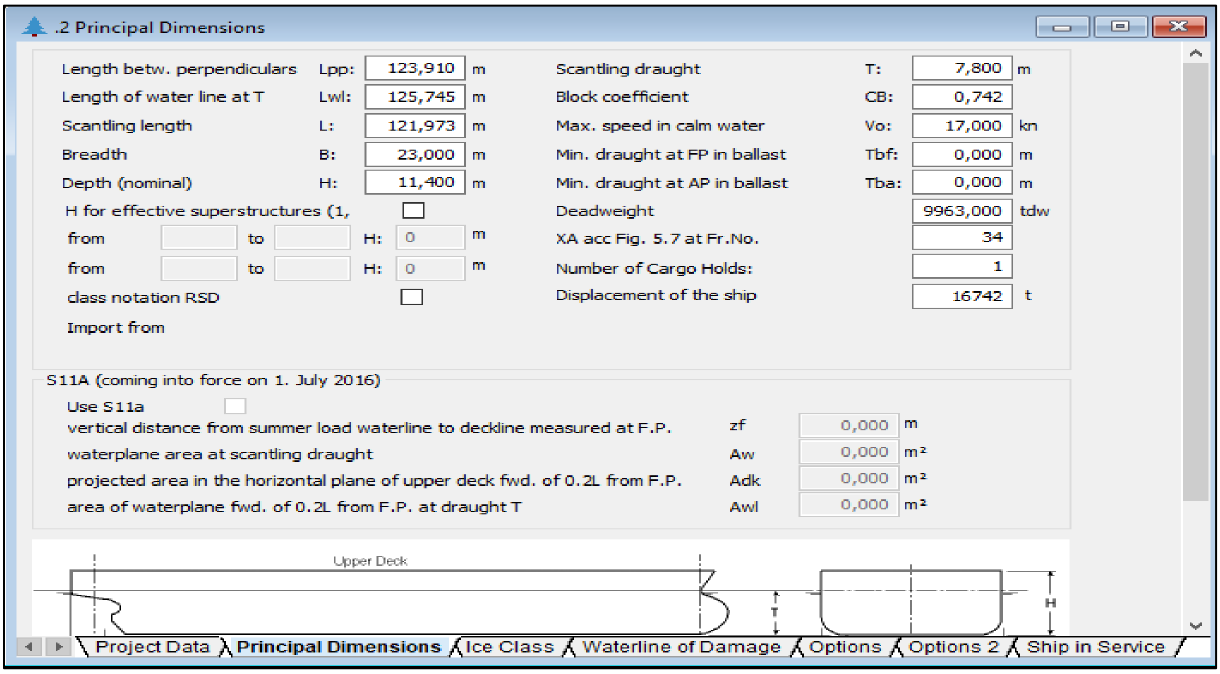
6 MODELLING OF THE VESSEL PK-116

To analyse the Global strength of the vessel, a global model of the entire hull girder with its primary structural components are required. To realize the global model of the MPV, POSEIDON was used. The entire modelling was done as per the explanation given above in the Poseidon section.

Step by step process for modelling the entire hull girder is depicted below to have a broader idea, how Poseidon has been used throughout the course of the thesis.

6.1 Pk- 116 general data

First of all, the general data of the vessel pk-116 was entered in to the respective section of the software, the input data filled here are very important as many calculations depend upon this given input. If not properly filled, it might cause malfunction later on while doing the calculations.



Length betw. perpendiculars Lpp: 123,910 m
Length of water line at T Lwl: 125,745 m
Scantling length L: 121,973 m
Breadth B: 23,000 m
Depth (nominal) H: 11,400 m
H for effective superstructures (1, ☐
from to H: 0 m
from to H: 0 m
class notation RSD ☐
Import from

Scantling draught T: 7,800 m
Block coefficient CB: 0,742
Max. speed in calm water Vo: 17,000 kn
Min. draught at FP in ballast Tbf: 0,000 m
Min. draught at AP in ballast Tba: 0,000 m
Deadweight 9963,000 tdw
XA acc Fig. 5.7 at Fr.No. 34
Number of Cargo Holds: 1
Displacement of the ship 16742 t

S11A (coming into force on 1. July 2016)
Use S11a ☐
vertical distance from summer load waterline to deckline measured at F.P. zf 0,000 m
waterplane area at scantling draught Aw 0,000 m²
projected area in the horizontal plane of upper deck fwd. of 0.2L from F.P. Adk 0,000 m²
area of waterplane fwd. of 0.2L from F.P. at draught T Awl 0,000 m²

Figure 4: Principal Dimension from Poseidon

6.2 Material descriptions

The steel material required to create the vessel was defined in the material section of the software, the user can later on define plate material as per the given requirement. The table below shows the type of Linear Isotropic material defined while modelling this vessel.

Table 3: Type of material used during modelling

Mat. No	E-Modulus [kN/mm^2]	G-Modulus [kN/mm^2]	Material density [Kg/mm^3	Yield stress [N/mm^2]
1.	**206000000**	**79230769**	**8000**	**235**
3.	**206000000**	**79230769**	**8000**	**355**
5.	**206000000**	**79230769**	**0**	**355**

6.3 Frame Table (X-direction)

The Frame table starts with the frame before the transom. For this frame, the frame number and the frame spacing is entered. The last frame of the table is the frame in front of the bow. Therefore, the frame table is defined over the whole length of the vessel. The positions of the other frames are calculated automatically by POSEIDON.

The frame table defined for the vessel PK-116 is shown below:

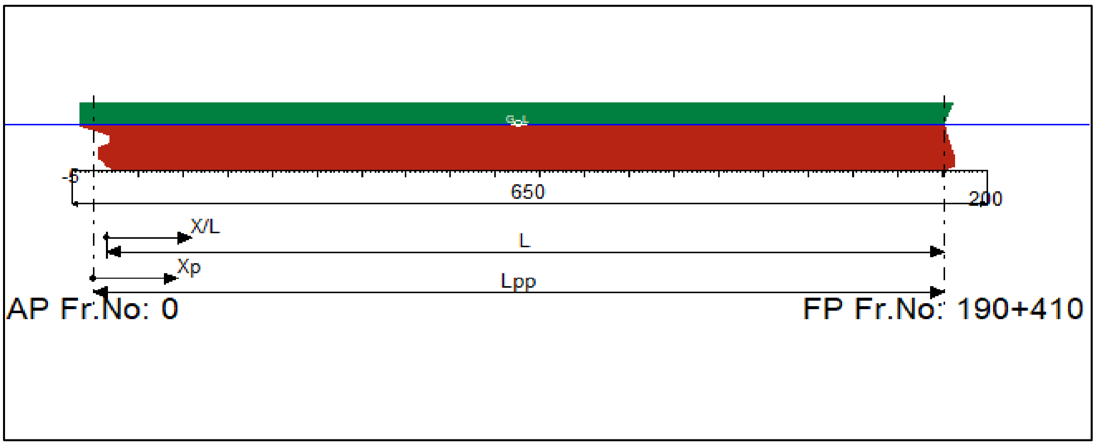


Figure 5: Frame Distance

6.4 Frame Table (Y and Z -direction)

The frame table in Y and Z direction is mainly used to define the elements in longitudinal direction. Here a specific name is given to each of the subsidiary line by keeping some common word as defined for the functional element, for e.g. ZIB was used to define the frame table in Z direction for Inner bottom functional elements, also L0 to Ln was used to define the longitudinal elements in y-direction and Z0 to Zn was used to define the longitudinal elements in Z-directions. For defining the knuckle lines, a special abbreviation with KL was used along with the functional element definitions.

6.5 Cut out contour table

Cut outs are described by a polygon contour. The defined cut-outs can be used for transverse and longitudinal members. The polygon contour of a cut out is defined by a sequence of points and their connections in a local “uvw” coordinate system. Circular segments and straight lines may be used as connections. For straight lines, the radius has to be set to zero. One of the cut out example defined in the contour table is shown below:

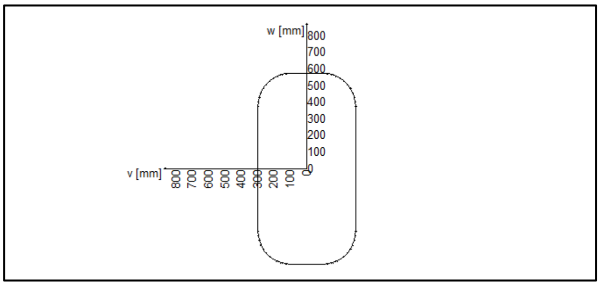


Figure 6: Cut-out Example

6.6 Hull structure

To model the Hull structure in Poseidon, generally the order of modelling to be maintained as follows:

a) Modelling of shell
b) Modelling of major horizontal parts (e.g. weather deck and inner bottom)
c) Modelling of major vertical parts (e.g. inner hull and longitudinal Bulkheads)

6.6.1 Shell modelling

Shell was the first element to model in Poseidon. The geometry of the shell was described by using the coordinates. Since, the vessel PK116 already exists so, the coordinates for the shell geometry were obtained by splitting all 200 frames of the vessel in to the respective coordinates using AutoCAD and then saving that coordinates points in to a text file.

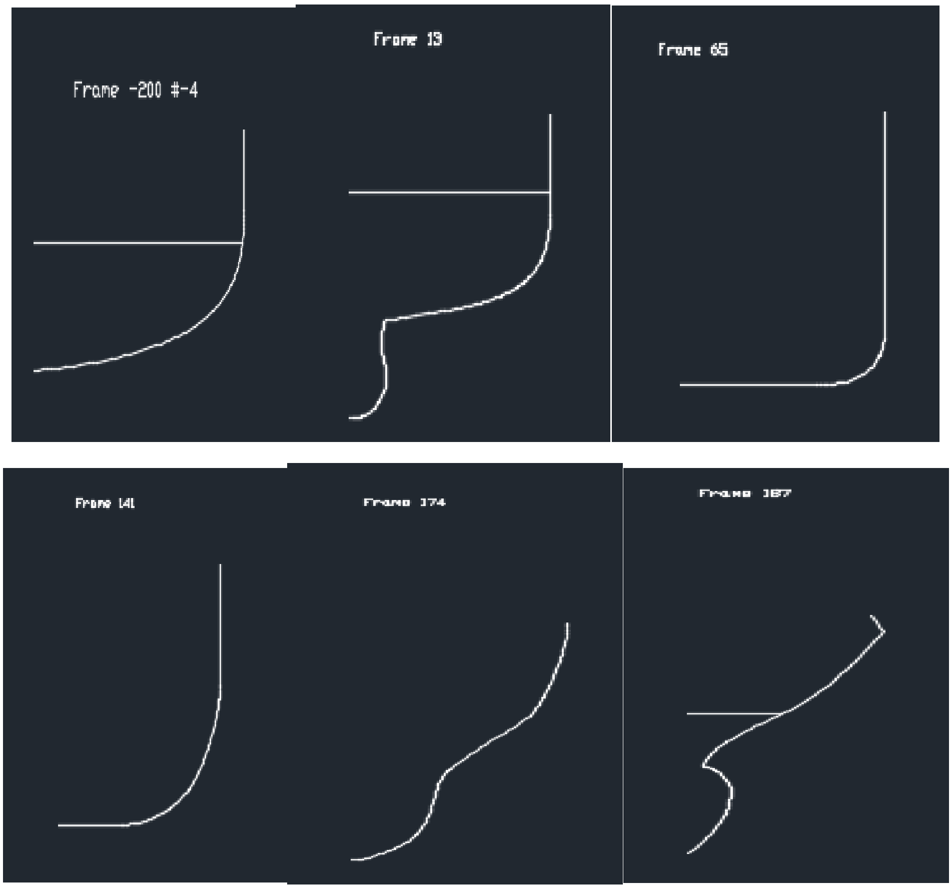


Figure 7: Frame Structural drawing of Aft, Amidship and Forward area

Figure –7 shows some of the frame structural drawing of the vessel PK-116, which were used to generate the coordinates for describing the shell. In Poseidon, the text file which were created with all these coordinates were imported as profile table in .txt format thus, enable to create the shell elements of the vessel. Later on, by taking this shell element as reference all other longitudinal and transverse members were created to complete the modelling.

6.6.2 Longitudinal members

The longitudinal members are the basis for defining the geometry of the entire hull model and it has to be very precise with respect of setting references, intersections in longitudinal direction and structural continuity.

6.6.2.1 Functional elements

The functional elements are developed by reference points in the cross section. They are described by y and z- values in conjunction with the line type LT. Under the functional element, all longitudinal elements such as the Decks, Longitudinal girder, Longitudinal Bulkhead, Coaming, Inner bottom along with the Shell and bow were defined.

While modelling the Functional element, the user can choose in between 1 straight line, 2 Circular, 3 along Functional element and 7 cut out as per the requirement. The interconnection of the Functional elements to each other is of most important due to recognition of elements internal and for further calculation. Also, the minimum distance between functional elements cannot be less than 5 mm.

6.6.2.1.1 Deck structure

DECK SPECIFICATION		
S No	Deck Name	Height from Base (m)
1	Deck 3	5200
2	Deck 2	7900
3	Deck 2.1	8525
4	Deck 1	11400
5	Deck 1.1	12275
6	Deck A	15275
7	Deck B	17975
8	Deck C	20675
9	Deck C1	16190
10	Deck C2	17310
11	Deck C3	19125
12	Deck C4	20300
13	Deck C5	30500
14	Deck D	23375
15	Deck D1	26075
16	Deck E	27275
18	Deck TT	3600
19	CO	14465
20	IB	2300

Table 4: Deck Structure

The definition of decks must include the camber and shear. Regarding the modelling of this vessel, the deck structure was included as per the actual drawing available and it was started by taking the reference through frame by frame going from aft to the forward by following each and every cut outs required for global analysis. Since, there were several decks included inside the vessel so, each of them were named in order of their height from the base or with a standard naming as per the software convention system. Inner bottom and tank top are also included in the deck specifications.

The Deck structure naming mentioned during the modelling with their height from the Base is given below

6.6.2.1.2 Longitudinal girder

The longitudinal girder was also modelled inside the longitudinal member of the Hull structure section; these sections were included as per the drawings of the vessel. The longitudinal girders specifications are shown below:

Table 5: Longitudinal Girder description

LONGITUDINAL GIRDER SPECIFICATIONS:		
S No.	**Girder Name**	**Distance from the centre line(m)**
1	LG00	0
2	**LG**03	1800
3	LG04	2400
4	**LG**05	3000
5	LG06	3600
6	**LG**07	4200
7	LG10	6000
8	**LG14**	**8100**
9	LG17	9900

6.6.2.1.3 Longitudinal plates

The longitudinal plates were defined in each of the functional element described above. All plate butts must be defined even between plates with same plate thickness in the longitudinal directions. In Y-direction each plate strake is required and in X-direction each plate butt.

The plate must be defined at the following areas:

a) Shell
b) Decks and superstructure decks.
c) Cargo hold areas

d) Fore and after peak
e) Sea chests

While defining the plates, the moulded lines must be considered according to the steel plans. If there is no information available, for the shell "right" and for the plates "left" should be used. The design criteria chosen for the plates were as follows:

Table 6: Plate Design Nomenclature

DESIGN CRITERIA OF PLATE:		
S No.	**Plate area**	**Design criteria**
1	Coaming	CO
2	Inner bottom	IB
3	Longitudinal girder	LG
4	Shell	S
5	Weather decks	WD

6.6.2.1.4 Shell plate with bulbous bow

Plates with different thickness were defined along the shell in longitudinal direction according to the shell expansion drawing of the vessel. The shell expansion drawing's specifications for plate thickness was implemented in both Y and Z direction along with the BOW area. The different colour inside the 3-D view of the shell plate and other structures shown below represents different plate thickness at different locations.

3-D view of the shell obtained from Poseidon is shown below:

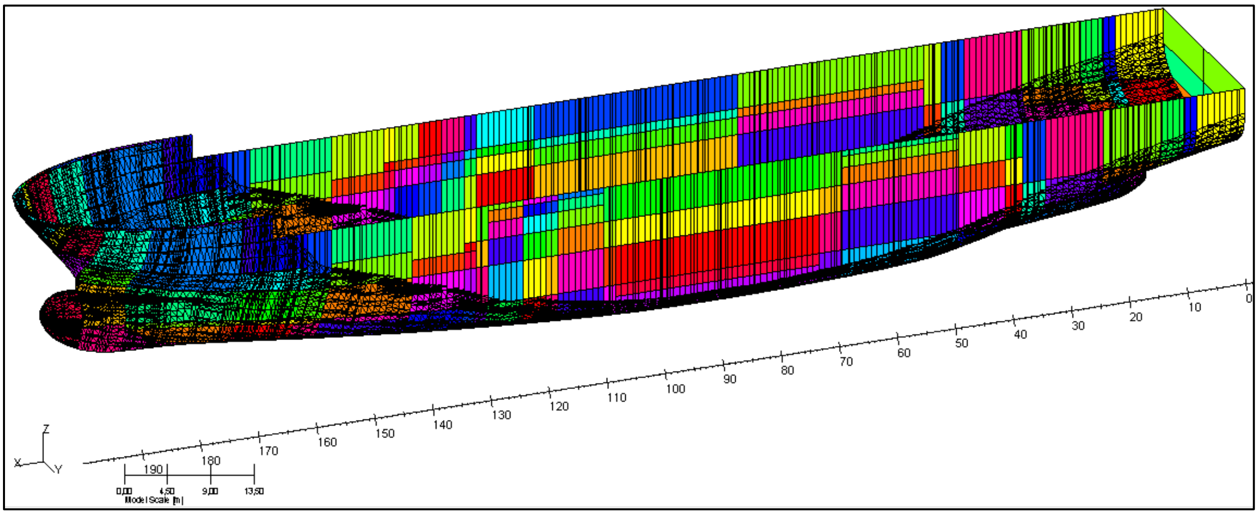
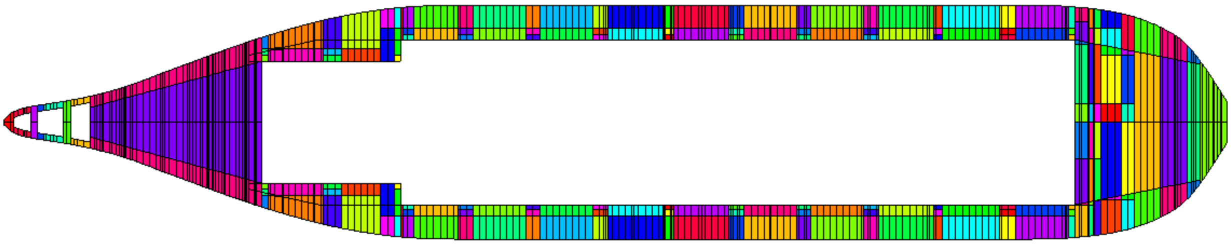


Figure 8: 3D view of Shell Plate

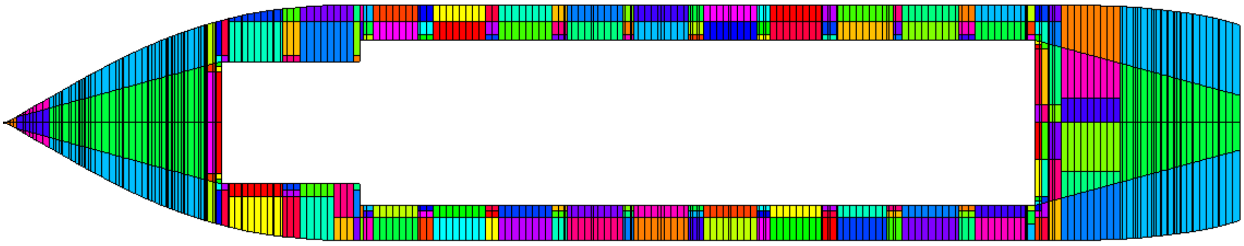
6.6.2.1.5 *Deck plate*

Deck plate was defined along the functional element prescribed for deck strakes. As listed above, there were twenty different kinds of deck surfaces assigned. Out of these, major deck plating's are shown below:

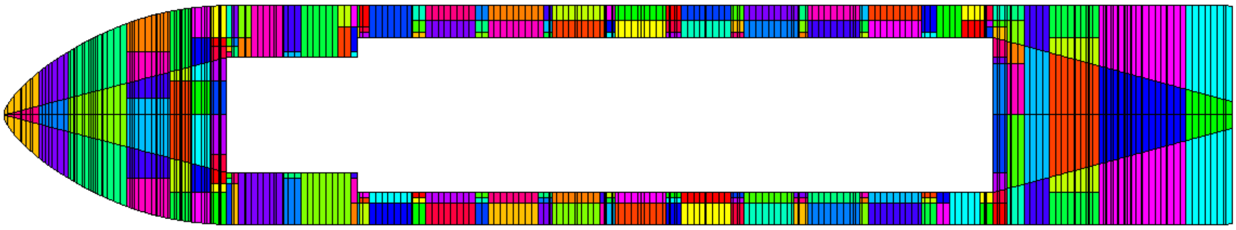
DECK 3



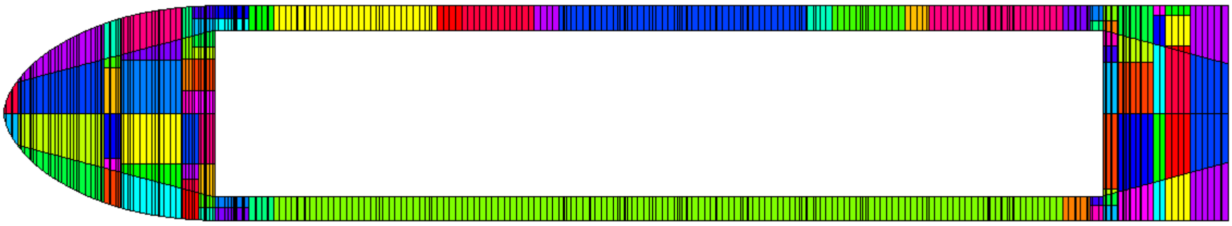
DECK 2



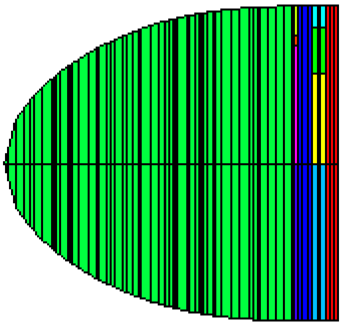
DECK 1



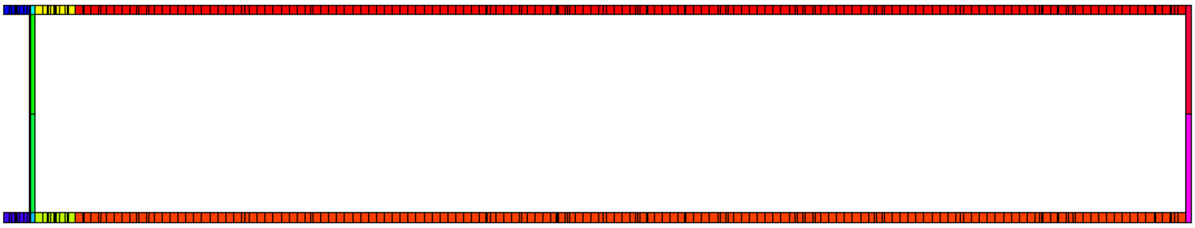
DECK A



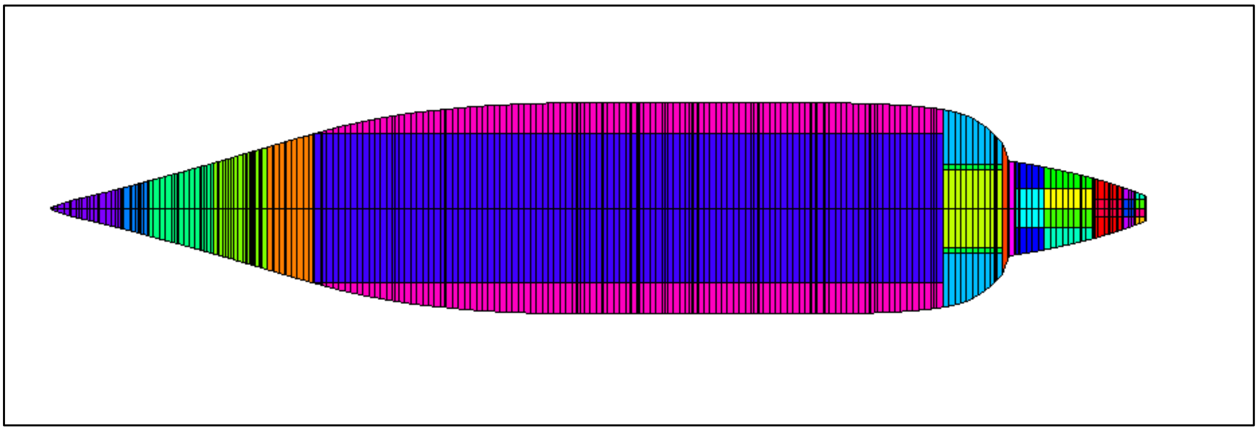
DECK B



HATCH COAMING



INNER BOTTOM



6.6.2.1.6 Longitudinal girder

A total of 9 different kind of longitudinal girders were defined as per the vessel's requirement. While defining the longitudinal elements, the user must keep in mind a good overview, were geometry changes happen to avoid any overlapping of one element with other.

Some of the major longitudinal girders are shown below:

LG00:

LG 03:

LG 07:

LG 14:

6.6.2.1.7 Longitudinal bulkhead

The longitudinal bulkhead was defined according to the drawing of the vessel to strengthen the structure against severe loading and deformations. Various bulkheads modelled inside the ship are tabulated below:

Table 7: Longitudinal Bulkhead

LONGITUDINAL BULKHEAD SPECIFICATIONS:		
S No.	**Bulkhead Name**	**Distance from centre line(m)**
1	LB00	0

2	**LB**01	600
3	LB02	1200
4	**LB**03	1800
5	LB04	2400
6	**LB**05	3000
7	LB06	3600
8	**LB**08	4800
9	LB09	5400
10	**LB**10	6000
11	LB11	6600
12	**LB**12	7200
13	LB14	8100
14	**LB**15.1	8850
15	LB15.5	8950
16	**LB**16.2	9600
17	LB**17**	9900
18	**LB17.1**	9300
19	LB**17.2**	7800
20	**LBCA**	CRANE AFT
21	LB**CF**	CRANE FORWARD
22	**LBS**	SUPERSTRUCTURE

Major bulkheads defined in the longitudinal directions are shown below:

LONGITUDINAL BULKHEAD AT DISTANCE 9900 m FROM CL:

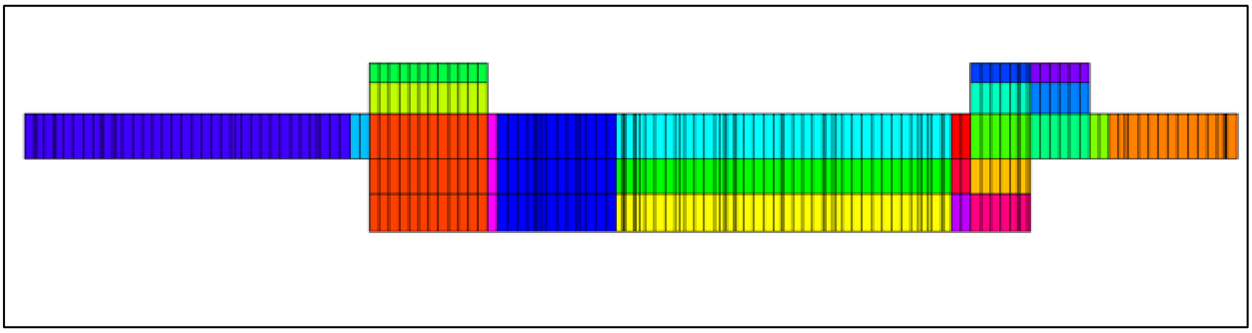


Figure 9: LB 17 Port side

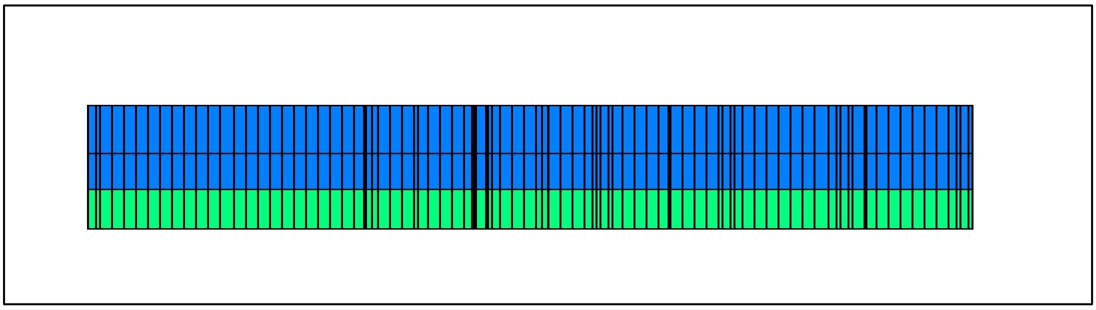


Figure 10: LB 17 Starboard side

LONGITUDINAL BULKHEAD AT DISTANCE 3600 m FROM CL

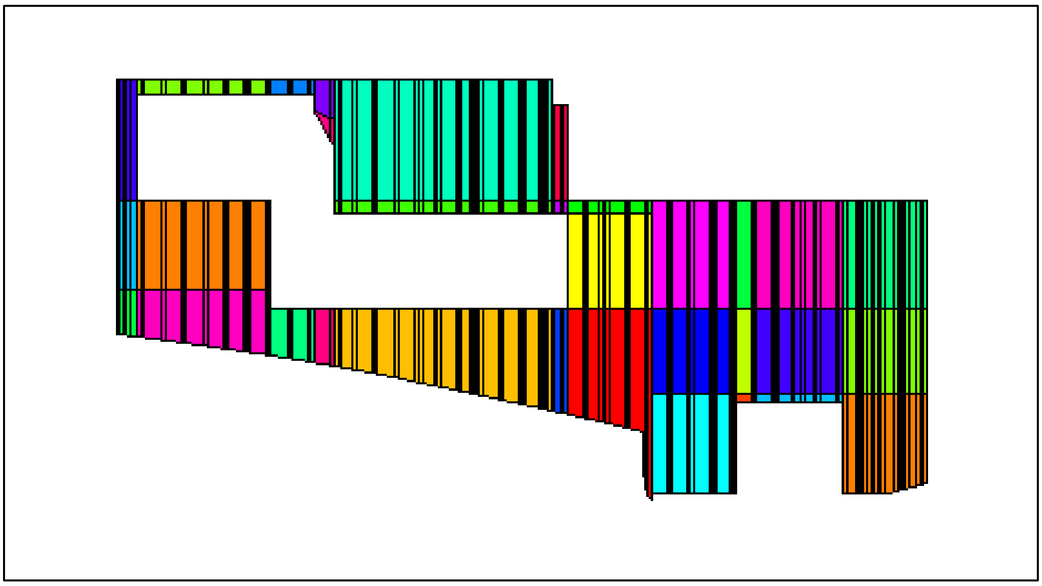


Figure 11: LB 06 Port side

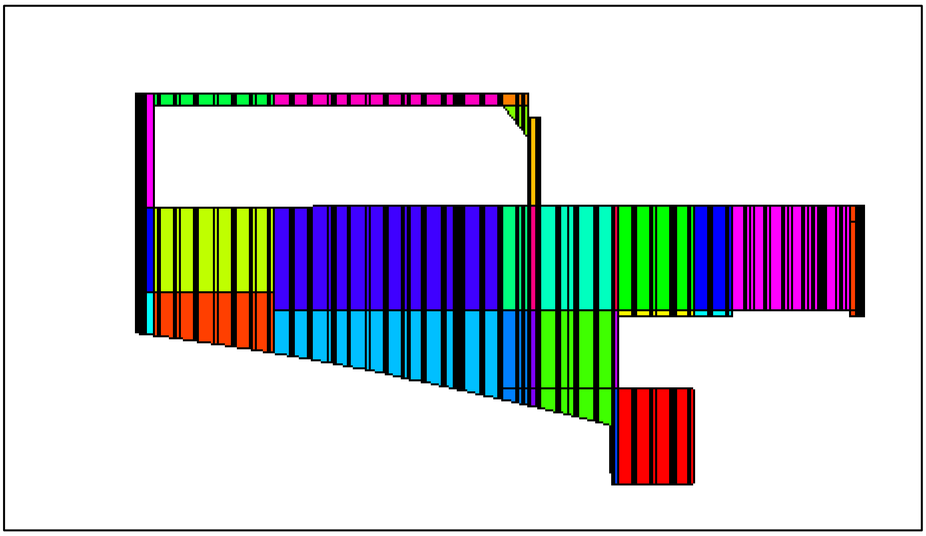


Figure 12: LB 06 Starboard side

LONGITUDINAL BULKHEAD AT DISTANCE 8100 m FROM CL

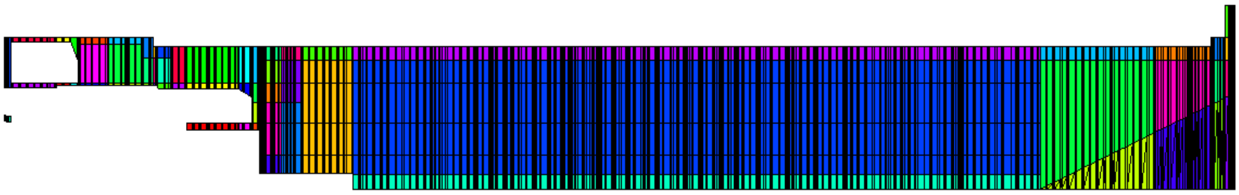


Figure 13: LB 14 Starboard side

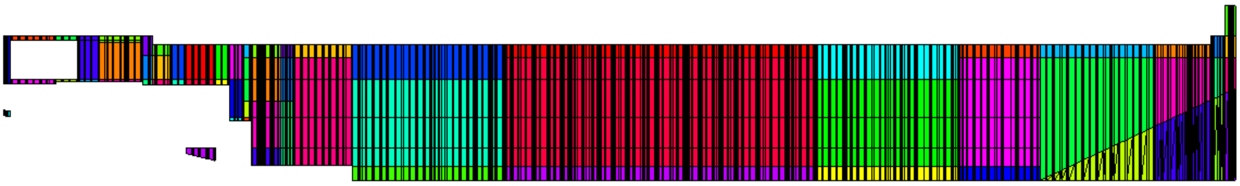


Figure 14: LB 14 Port side

6.6.2.1.8 *Crane structure*

For doing the global strength analysis of the vessel and observing the overall deformations, crane structure with its foundation and boom were also added inside the model as per the drawings. Both the cranes are situated at the port side of the vessel on Deck – A.

A pictorial representation of the crane structure is shown below:

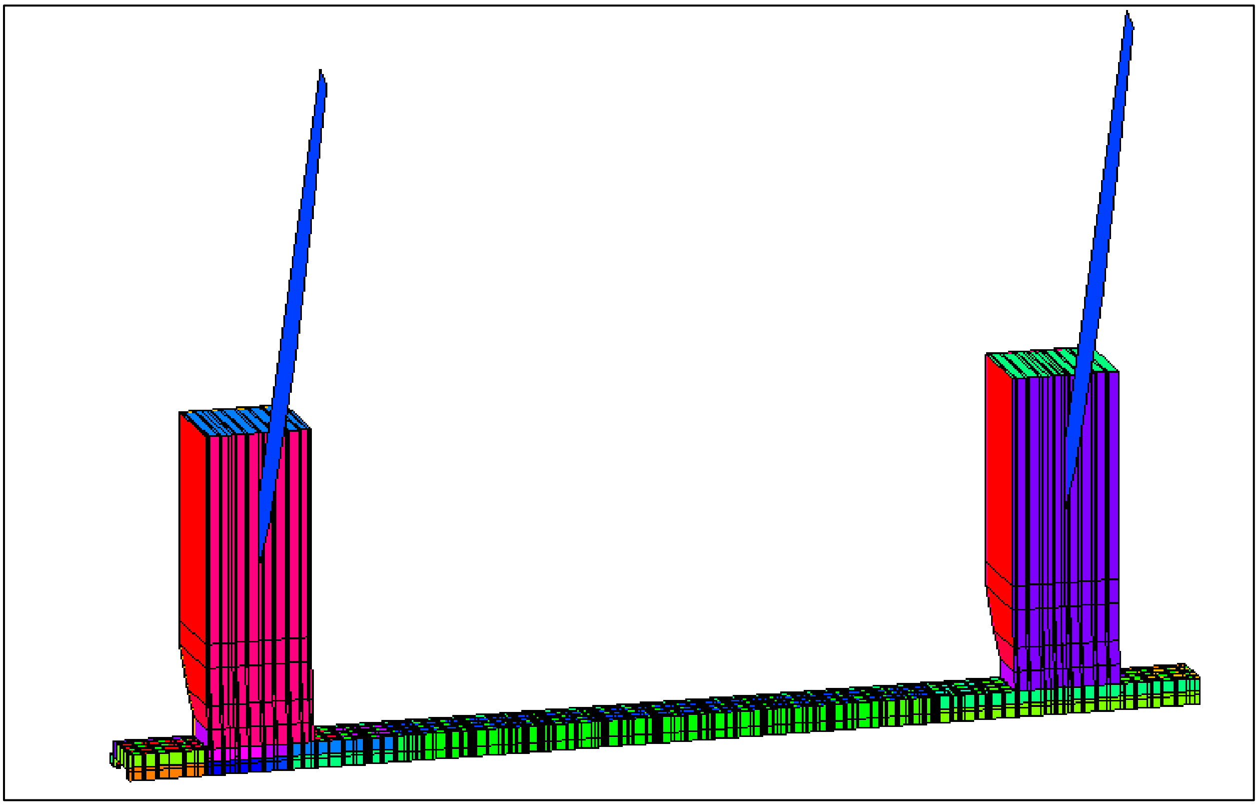


Figure 15: Crane structure with its Boom

6.6.2.1.9 Super structure

The superstructure was also added inside the model to have a more realistic and complete global strength analysis. The superstructure is sitting on the topmost deck of the vessel i.e. Deck – B.

The diagram below shows a 3-d model of the superstructure sitting on Deck – B:

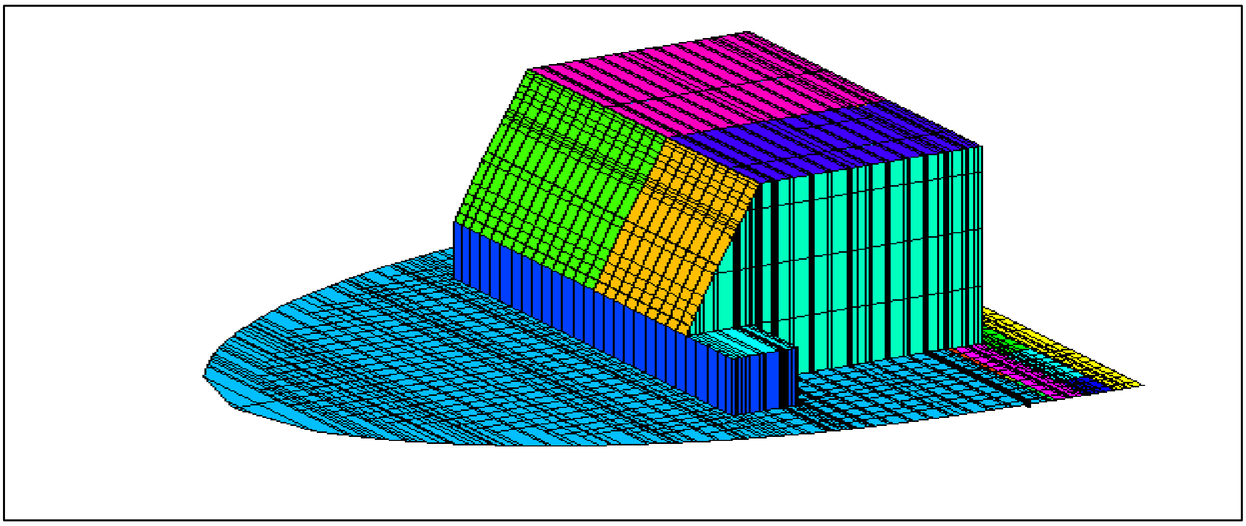


Figure 16: Superstructure at Deck- B

6.6.2.1.10 Longitudinal stiffeners

The longitudinal stiffeners arrangement was done according to the drawings. The naming for the longitudinal stiffeners were followed with the same name as of functional elements, for e.g. the longitudinal stiffeners on DK – A was named with the same name DK – A with the corresponding profile. Various profiles used for the longitudinal stiffeners are tabulated below:

Table 8: Longitudinal stiffener profile

LONGITUDINAL STIFFENERS SPECIFICATIONS:		
S No.	**Profile**	**Dimensional Attributes**
1	HP	140*7
2	**HP**	160*9
3	HP	120*7
4	**HP**	150*15
5	HP	150*12
6	**HP**	120*6
7	HP	220*10

8	**HP**	180*9
9	HP	140*9
10	**HP**	160*7
11	HP	100*6
12	**HP**	120*8
13	HP	100*10
14	**HP**	200*9
15	HP	140*8
16	**HP**	100*7

To have a better view, how the longitudinal stiffeners are arranged inside the model, and a whole arrangement of the stiffeners on DK-A is shown below:

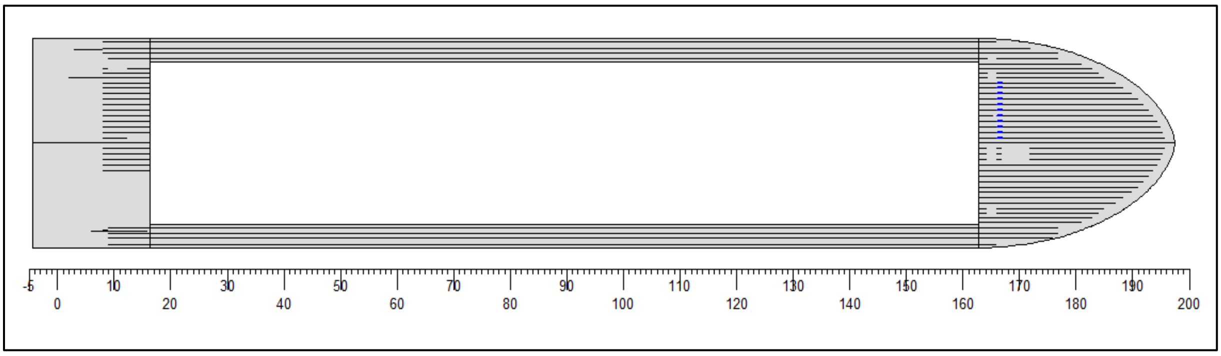


Figure 17: Longitudinal stiffeners arrangement on DK-A

6.6.3 Holes and cut-outs on longitudinal members

All desired holes and cut-outs were modelled efficiently to have a calculated impact on global strength of the vessel. The centre point of the cut-out was used to perfectly place it at the desired location. For modelling the cut-outs same functional elements name were used with X-coordinate starting and ending point. The use of the spacing ‘a’ is recommended to model the cut-outs with Y coordinates. Each cut-out was given with its unique ID which can be defined separately in the Cut-out contour table section of the software. To facilitate easy modelling, a series of cut-outs with similar geometry is also possible by just mentioning the required frame spacing and its longitudinal distance.

To have a better understanding, a pictorial representation of the cut-outs is presented below:

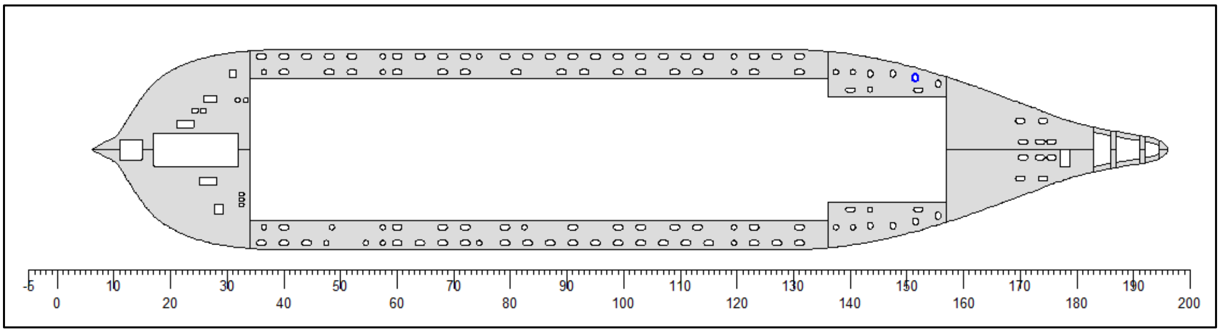


Figure 18: Holes and Cut-outs on DK-3

6.6.4 Transverse stiffeners

Transverse stiffeners were defined on to the transverse frame. The stiffeners are mainly present in the cargo hold area as well as in the fore and aft peak. Generally, the transverse stiffeners are defined along the transverse framing system, decks, longitudinal bulkheads or girders. A framing system with transverse stiffeners along it is shown below:

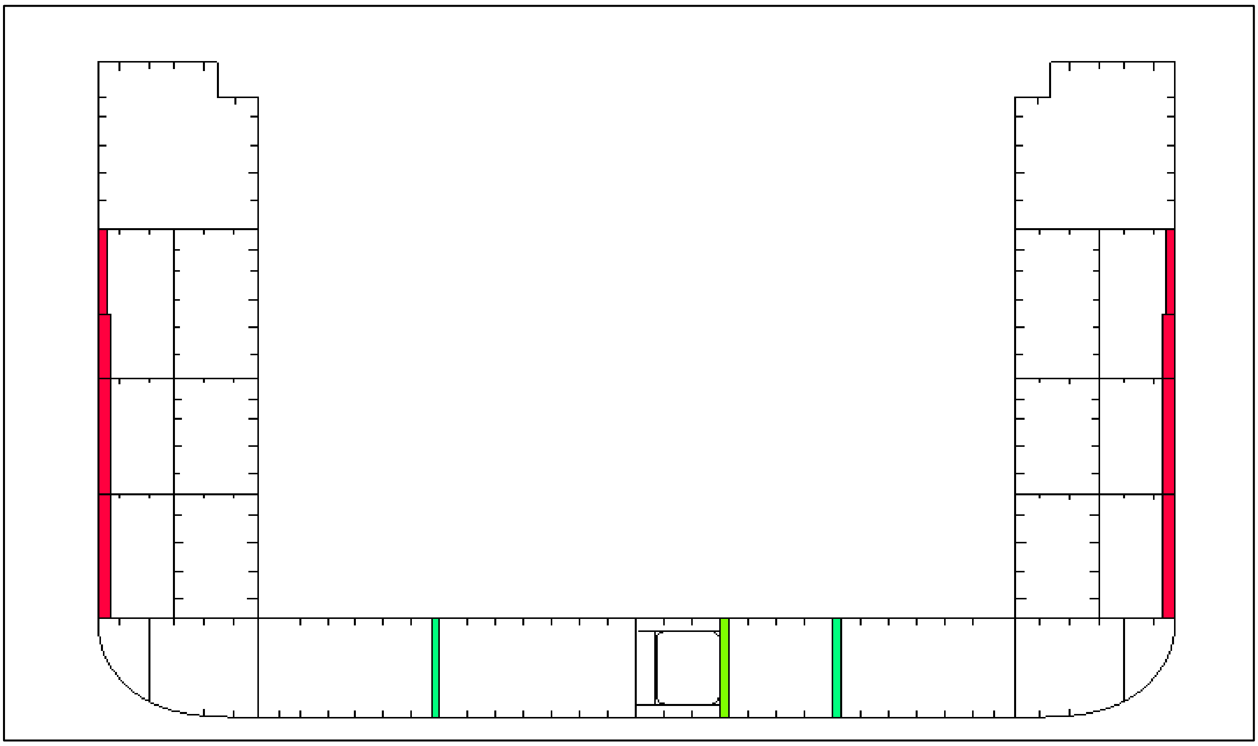


Figure 19: Transverse stiffeners on a Frame in Cargo hold area

6.6.5 Transverse girders

In general, Transverse girders and webs are included in the cargo hold area as well as in the fore and aft peak of a vessel. For changing thicknesses or structure the girder has to be split in to multiple pieces. The field item has to be unique in order to grant that it is interpreted as one girder during calculations. For a transverse girder supported by a stringer, deck, longitudinal bulkhead or girder the definition must be intersected at the support point.

An example of a transverse girder supported by the decks is shown below:

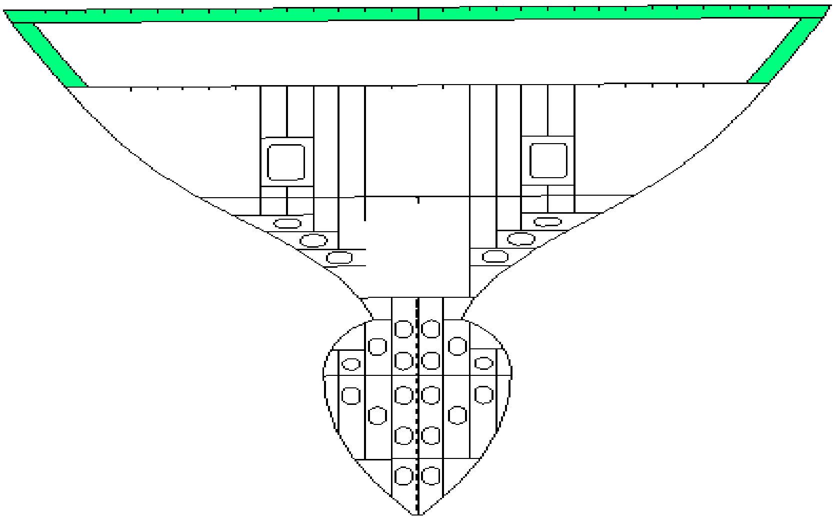


Figure 20: Transverse girder along the Deck

6.7 Transverse members

6.7.1 Transverse web plates

The definition of the transverse web plates is directly done in the transverse plate table as “plate contour” with the given general rules. By looking at the preview graphics of the transverse framing, a new plate is established according to the plate contour. To identify the location of the plates, a fixed shortcut ID is given to each of the plating system for e.g. the plates which are allotted for the flooring system are represented with a shortcut ID of ‘FL’; whereas the one located as web are given with an ID of ‘W’.

There are two different kind of plate definitions used:

6.7.1.1 Regular structure plate

These kind of platings are used at the parallel amidships section where most repeating structure exists. For the floor plates, the naming conventions are basically the same as previously described for the plate contour. The character counting starts from the inside, the frame is the first where the plate is established for e.g. the first plate “FL: 65AP”stands for floor plating at 65th frame on port side similarly, the second plate will have unique ID as “FL: 65BP” where, only the second last letter was changed to specify the second plates. The same tradition was followed for the establishment of web plating as well.

A framing system with regular plating structure is shown below:

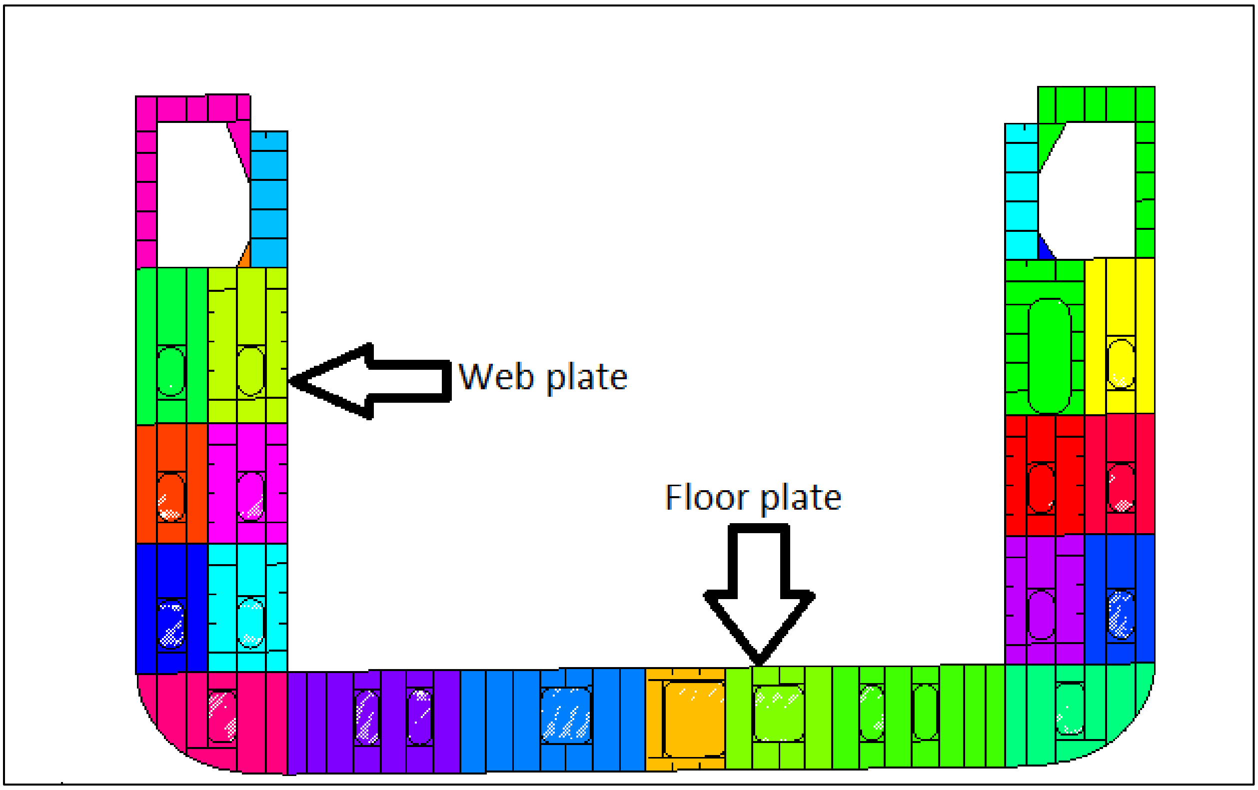


Figure 21: Regular structure plate [frame no -65]

6.7.1.2 *Irregular structure plate*

The plates with a small grade of repetition were modelled with the irregular structure naming. Same method of defining the plates were used as discussed in the regular plating method.

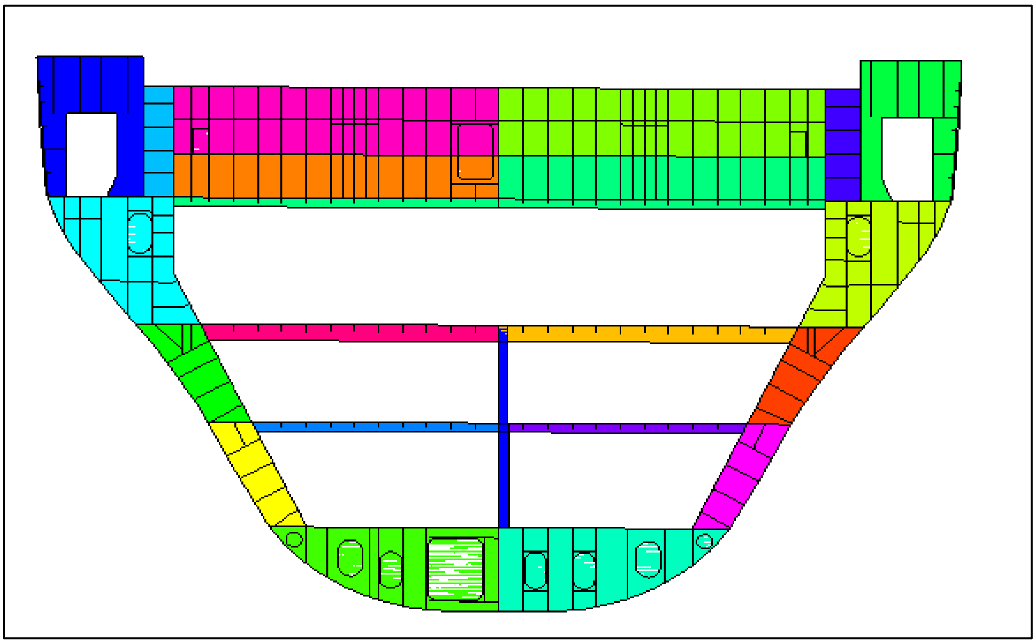


Figure 22: Irregular structure plate [frame no -185]

6.7.2 Transverse stiffeners and cut-outs

The transverse stiffeners and cut-outs are assigned on to transverse plate definitions. Here, the same name as of transverse plates was used to define the transverse stiffeners and cut-outs. One advantage while modelling the transverse stiffener and cut-out is that we can simply copy the definition of transverse plate from one frame to another if we have same definitions for stiffeners and cut-outs at both section.

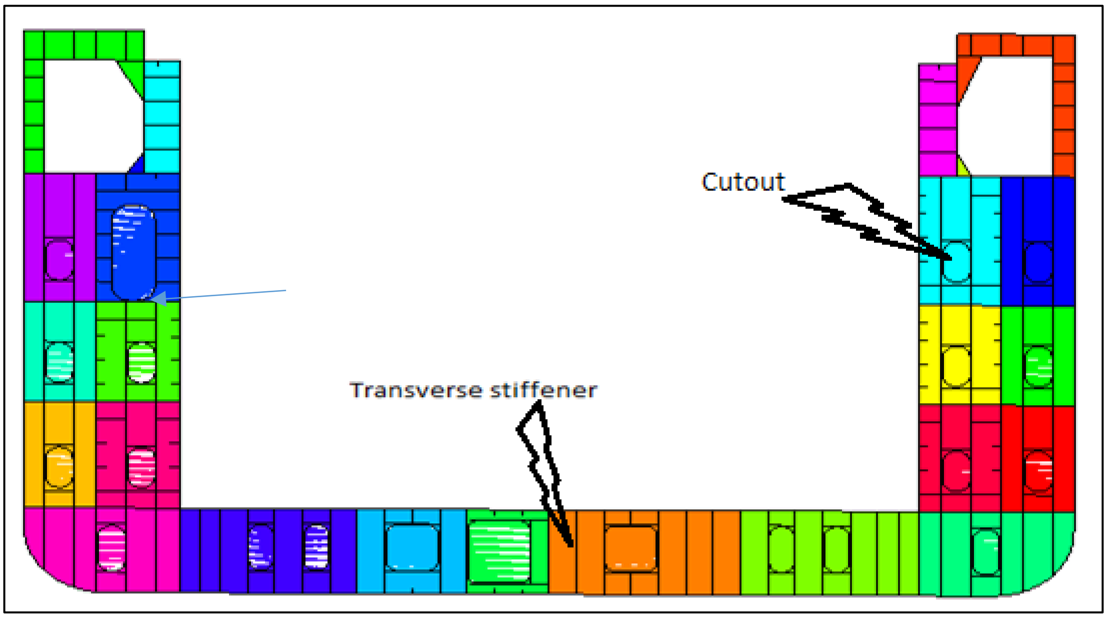


Figure 23: Transverse plate with stiffeners and cut-out [frame no -65]

6.7.3 Transverse bulkhead

There are two numbers of transverse watertight bulkheads in this vessel, one is present at the aft and the other one is at forward with the corresponding frame no as 34 and 157 respectively, in between these two bulkheads the whole area is dedicated for cargo stowage purpose.

Generally, the transverse bulkhead is defined in a separate section of software dedicated to bulkhead modelling, but since, only two bulkheads were present in this vessel so, it was decided to define in the transverse plating section with the desired plate contour. The definitions for the transverse bulkhead plate system followed the same modelling procedure as it was done in case of transverse web plate.

The watertight transvers bulkhead at the fore and aft end is shown below:

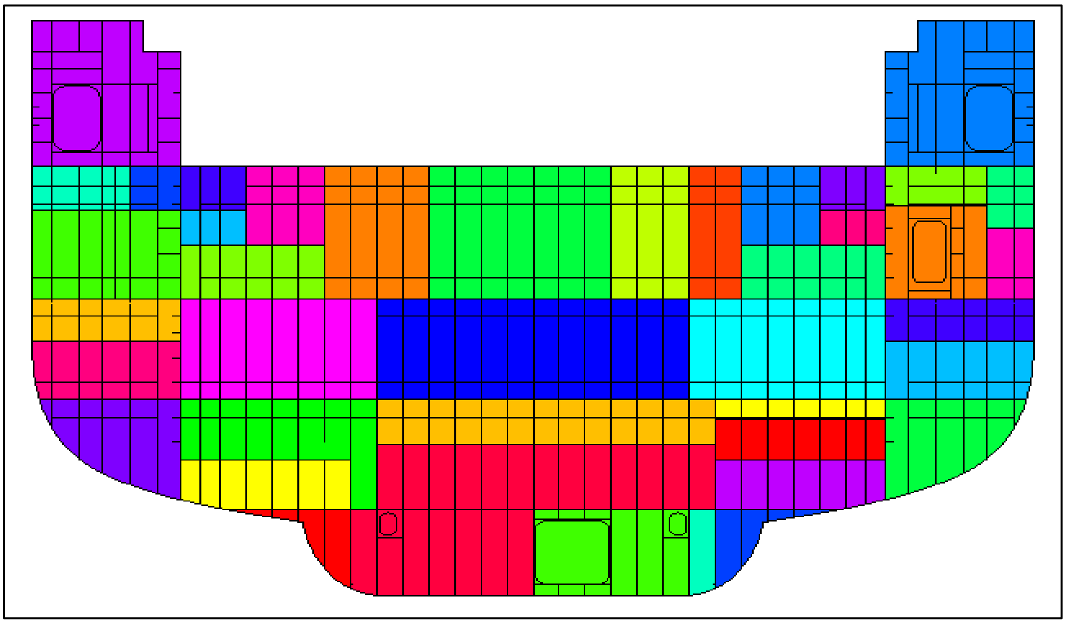


Figure 24: Transverse bulkhead at AFT [Frame no -34]

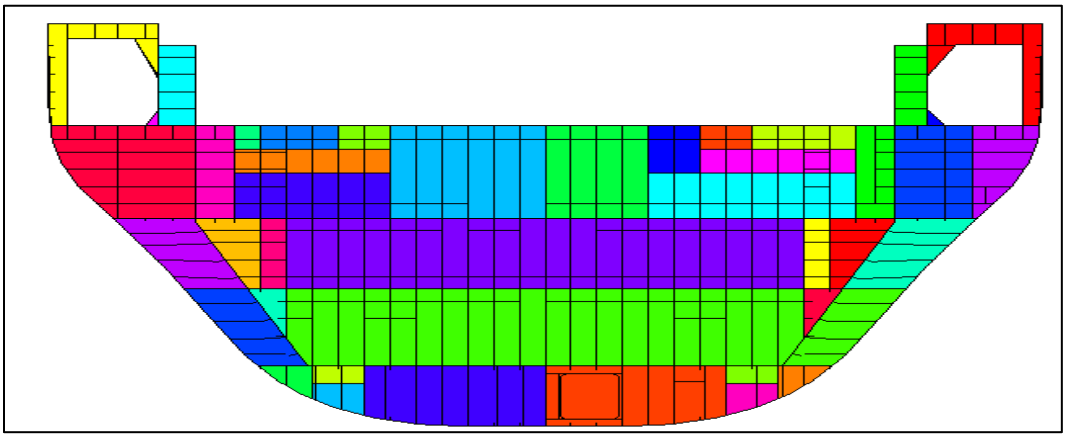


Figure 25: Transverse bulkhead at FORE [Frame no -157]

6.8 Modelling of hatch cover

Out of 15 hatch covers of the vessel *pk-116*, 4 numbers of hatch cover were modelled using Poseidon between 40m to 70 m length in longitudinal direction at the amidships section. According to the nomenclature used for the hatch cover, two odd nos. (7, 9) and two even nos. (8, 10). of hatch cover was modelled in order to carry out the whole carousel weight on top of it.

<u>Main dimensions of Hatch cover:</u>

Odd Hatch cover (7, 9)	Even Hatch cover (8, 10):
Transversal length –17,620 m	Transversal length –17,620 m
Longitudinal length - 6502 m.	Longitudinal length - 6762 m
No. of longitudinal girder – 6	No. of longitudinal girder – 6
No. of Transversal girder – 3	No. of Transversal girders - 3

In GL-Frame Poseidon, the hatch cover was modelled according to the drawings available. First of all, a base plate of uniform thickness was defined with the desired longitudinal and transversal length by choosing the element group name as "Shell". On top of it, the whole plate was distributed in three different kind of thicknesses. For e.g. the thickness at the four corner of hatch cover was different than the thickness at the middle and at the side of the hatch cover. After plate modelling, the longitudinal and transversal girder was also modelled according to the drawing. Except longitudinal length, both kind of hatch cover had the same dimensional requirement.

After modelling, the hatch cover was meshed with a mesh size of 0.5 m in order to import directly in to the bmf file of model.

The figure below shows the even and odd hatch cover FE-model.

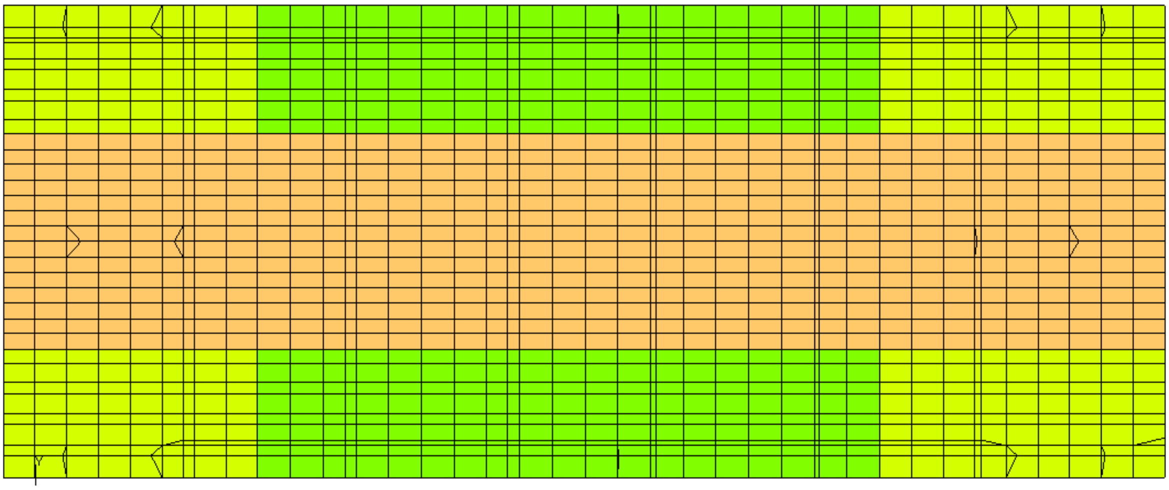


Figure 26: Top view of even hatch cover

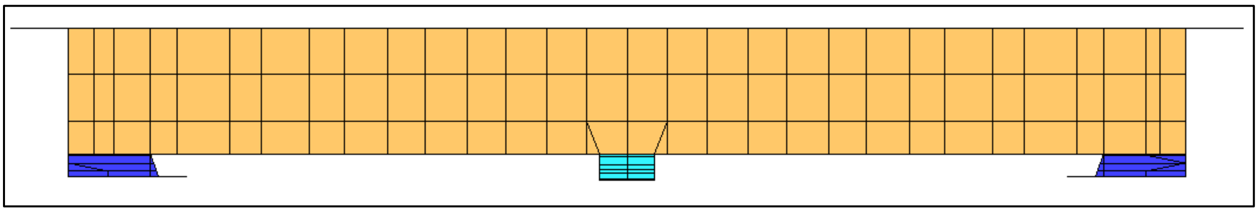


Figure 27: Front view of even hatch cover

Figure 28: Side view of even hatch cover

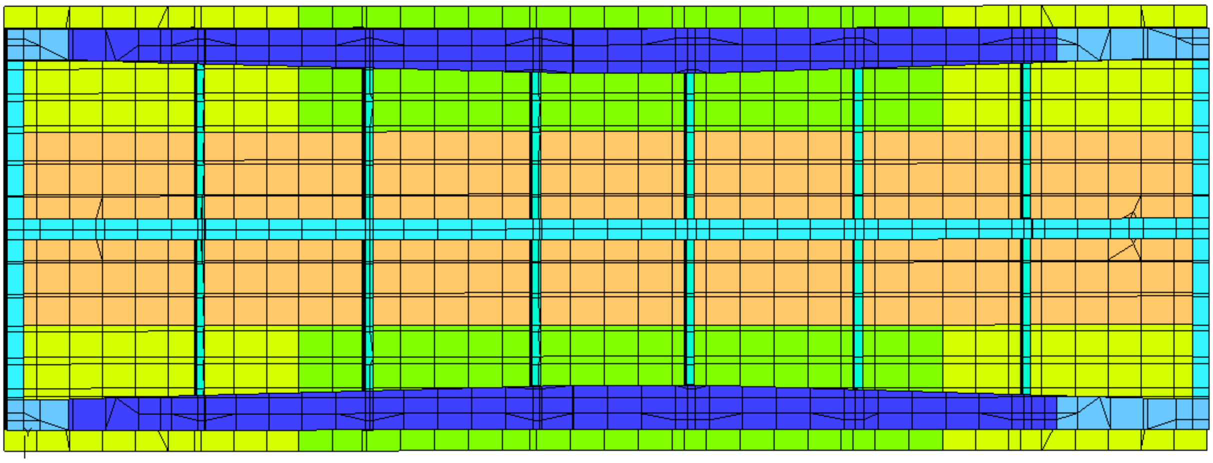


Figure 29: Bottom view of odd hatch cover

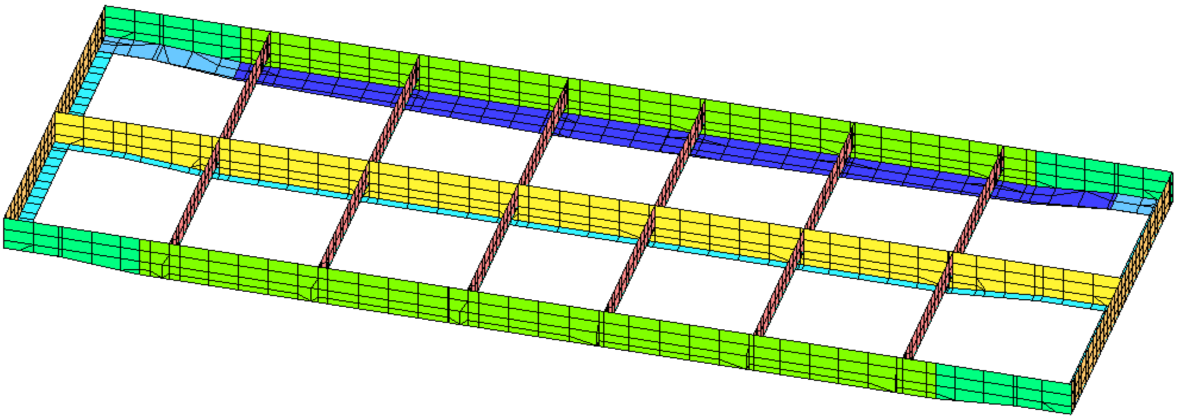


Figure 30: Longitudinal and Transversal Girder of even hatch cover

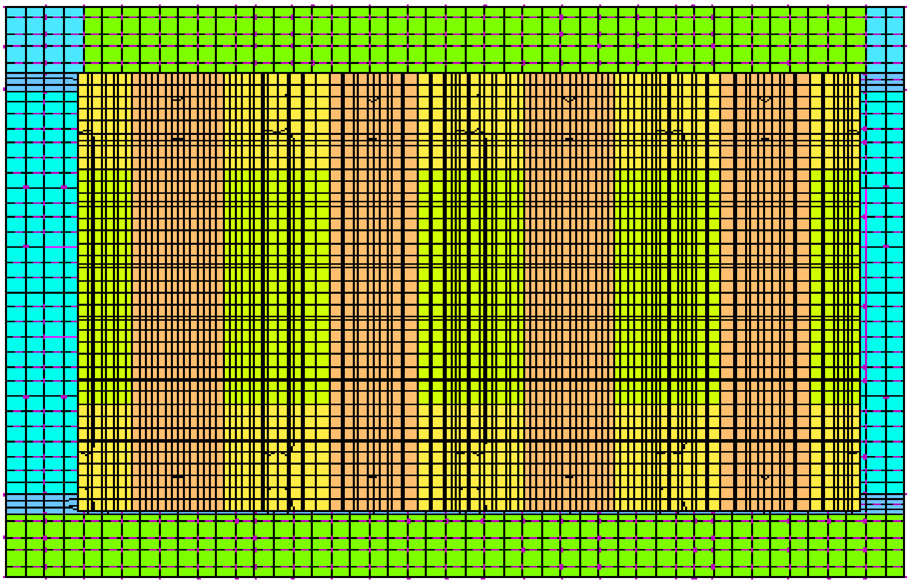


Figure 31: All 4 hatch cover arrangement inside the model

6.8.1 Securing of Hatch cover

The securing of hatch cover needs to be done inside the model because it reflects the same work what hatch cover bearing pad does in reality. To achieve more close to reality for global strength analysis some securing arrangements were done with the help of truss element which transferred the load only in axial direction.

Therefore, all four hatch covers were secured at both port and starboard side. On port side, securing was done in longitudinal and transversal direction whereas on starboard side securing was done in longitudinal direction only.

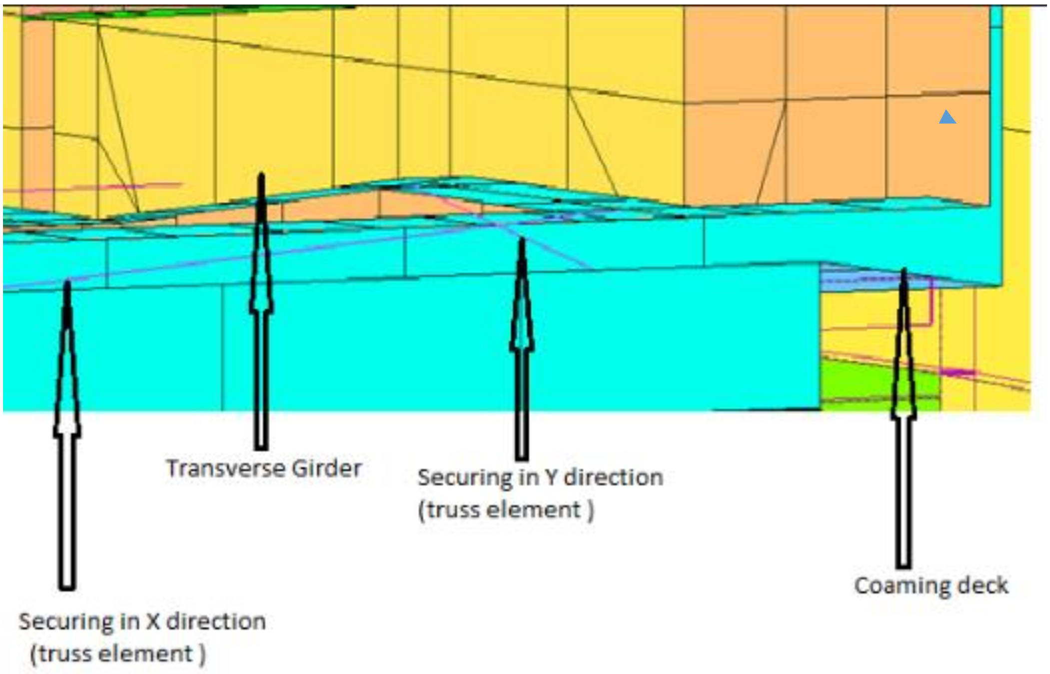


Figure 32: Cross section view of even hatch cover: Securing arrangement at Port side

6.9 Stanchion assembly

6.9.1 Pre- analysis

The main objective of the thesis was to develop a stanchion (pillar) assembly between weather deck and tank top to withstand heavy cargo loading on top of the hatch cover such that it can successfully transfer the load from weather deck to tank top.

Therefore, the main aim was to design an assembly with its platform in an inclusive way. First of all, the strong points were identified on the tank top where load can be transferred without any dislocation of the element. Similarly, for the upper section, the nodes at the intersection point of longitudinal and transversal girder of hatch cover were selected. Next task was to choose the type of element which can withstand the load and correctly transfer it to the bottom structure without getting buckled.

The Figure-33 shows the final Stanchion assembly setup which was developed after repetitive and regressive investigation.

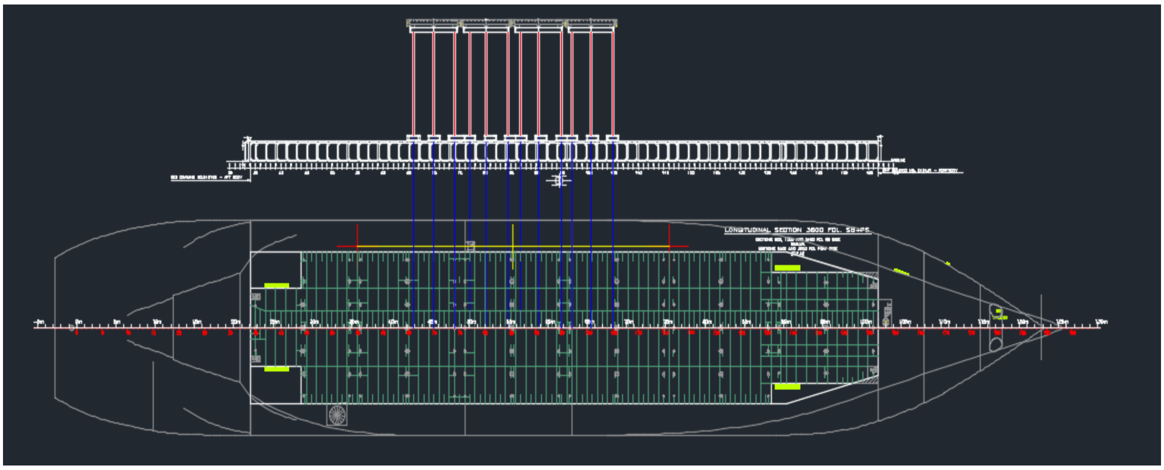


Figure 33: Set up of stanchion assembly with tank top

In the above figure –33, the deck which is green in colour represent Tank top with various longitudinal and transversal girder on it, the intersection point of both girders / stiffeners at the tank top indicates the strong point on which the stanchion assembly platform would be installed. The uppermost structure in white colour represents 4 numbers of hatch cover. The red colour structure is the stanchion trusses which would transfer the axial load and the one in blue shows the connection point between stanchion platform and tank top.

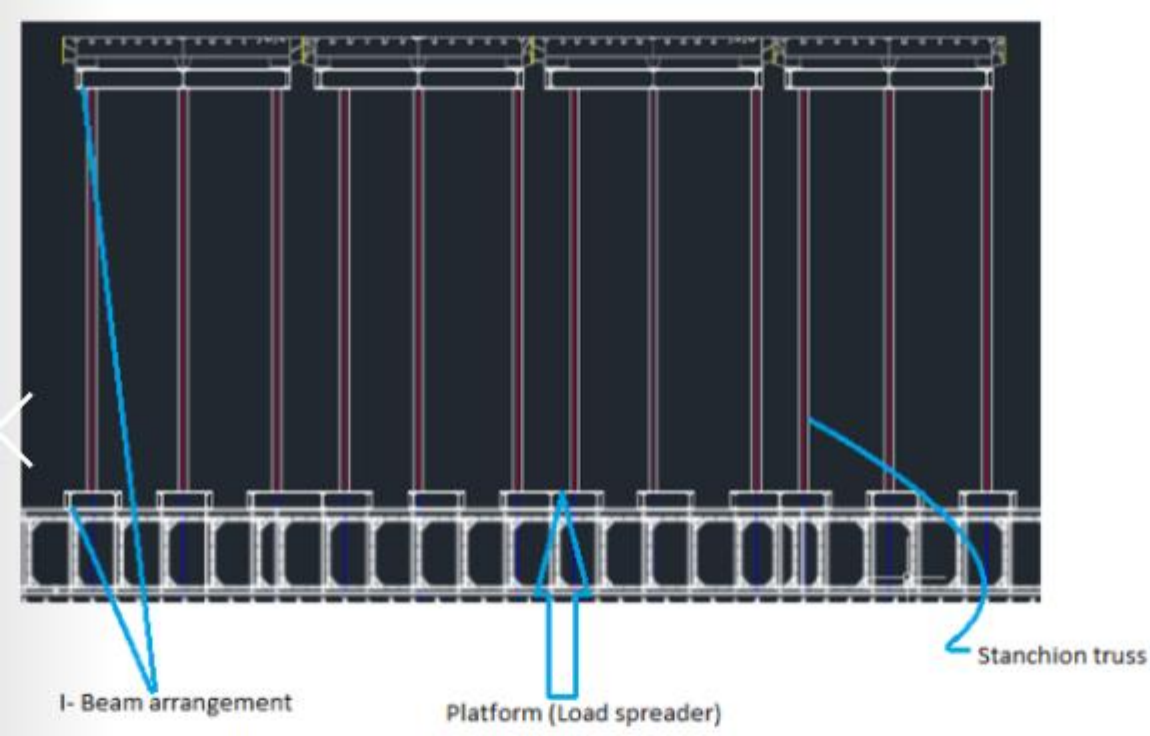


Figure 34: 2-D view of the stanchion assembly

As can be seen in the Figure- 34, there are 9 platforms sitting on the tank top comprising of adequate number of I- beams and four upper platforms attached to the hatch cover with the same configuration of I-beams. Both the platforms are connected with truss element.

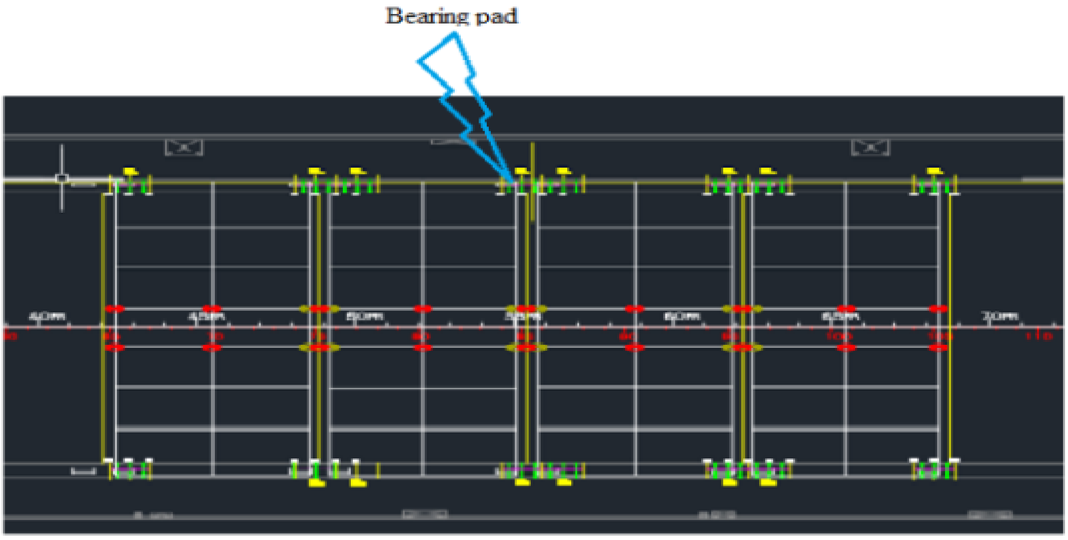


Figure 35: Location of Hatch cover bearing pads

Figure-35 shows the exact location of hatch cover bearing pads. In Poseidon model, it was not modelled like a bearing pad's rather it was augmented by securing the hatch cover in longitudinal and transverse direction at port side and only longitudinal direction at starboard side. While implementing the securing arrangement it was found that there were not many exact strong location present inside the Poseidon model to transfer the load from hatch cover to the coaming top. Therefore, extra bracket was added inside the model to transfer the load from hatch cover to the deck via securing arrangement as shown in the Figure-36:

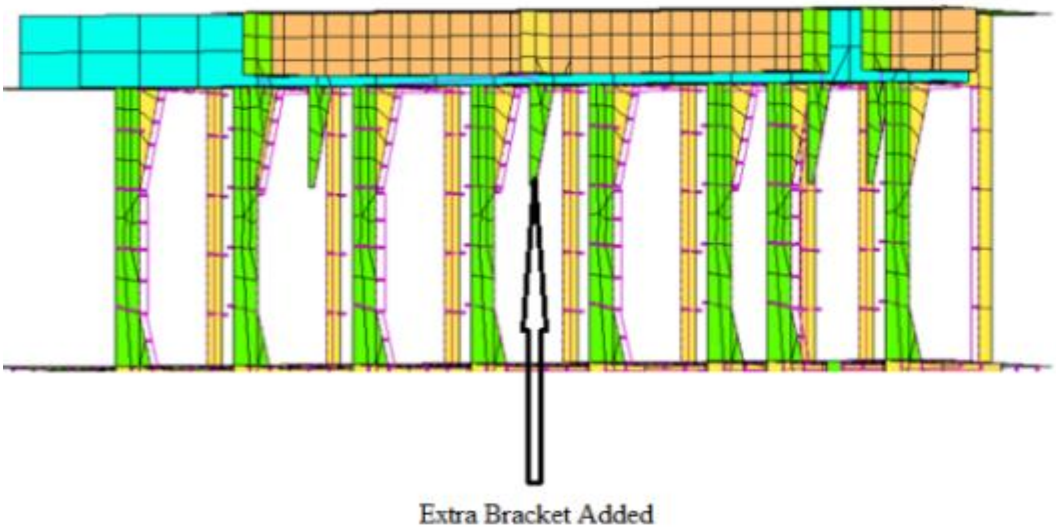


Figure 36: Cross section view with extra bracket

6.9.2 *Modelling of stanchion assembly*

As like other modelling, the stanchion assembly was also modelled in Poseidon GL-Frame. The drawing which was referred for modelling was the same as shown in Figure-33. First of all, the bottom platform was modelled according to the figure aforementioned then, extreme care was taken while placing the platform on to the tank top such that the load transfer can be more realistic and with least deformation of the structure. Therefore, the nodal point for load transfer was chosen just above the longitudinal girders. Very similar approach was applied for positioning of the upper platform, the intersection point of longitudinal and transversal girder was chosen for connection of upper platform to the hatch cover.

Next task was to replicate the above contemplation in to the GL- frame. To have the exact load transfer point, the nodes from the FE-model was identified and according to the nodes X, Y, Z coordinate, same nodes were created in GL- Frame again for the stanchion assembly by varying the coordinates only in the Z direction. In this way a collection of nodes representing the platform was created which, later on were connected with the help of elements like beams and trusses.

The beams which were defined for the stanchion assembly was I- Beam and the dimensions was chosen by referring some of the existing beams inside the vessel in that area, although, these dimensions are superficial and would be tested after application of various load cases in Shipload.

There were two kinds of I-beam defined in GL-Frame; one was HEB500 and the other HEB700 under the “Cross section” definitions.

Table 9: Type of I-beam

I-BEAM DEFINITIONS :			
S No.	**Title**	**Flange (mm)**	**Web(mm)**
1.	HEB500	300*28	444*14.5
2.	HEB700	300*32	636*17

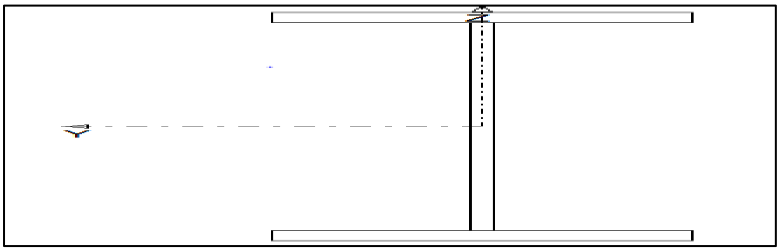


Figure 37: Profile HEB 700

Other than I-beam, truss element was also added inside the assembly between the two platforms to transfer the axial load. Similar to I-beam, some superficial dimension was also added for the truss element which would be tested again by several load cases in Shipload. At the moment, an area of **100,000 mm^2** was prescribed at each of the stanchion trusses, stopper X truss and stopper Y truss with **material no. 5** as per material description added in **subchapter 6.2.**

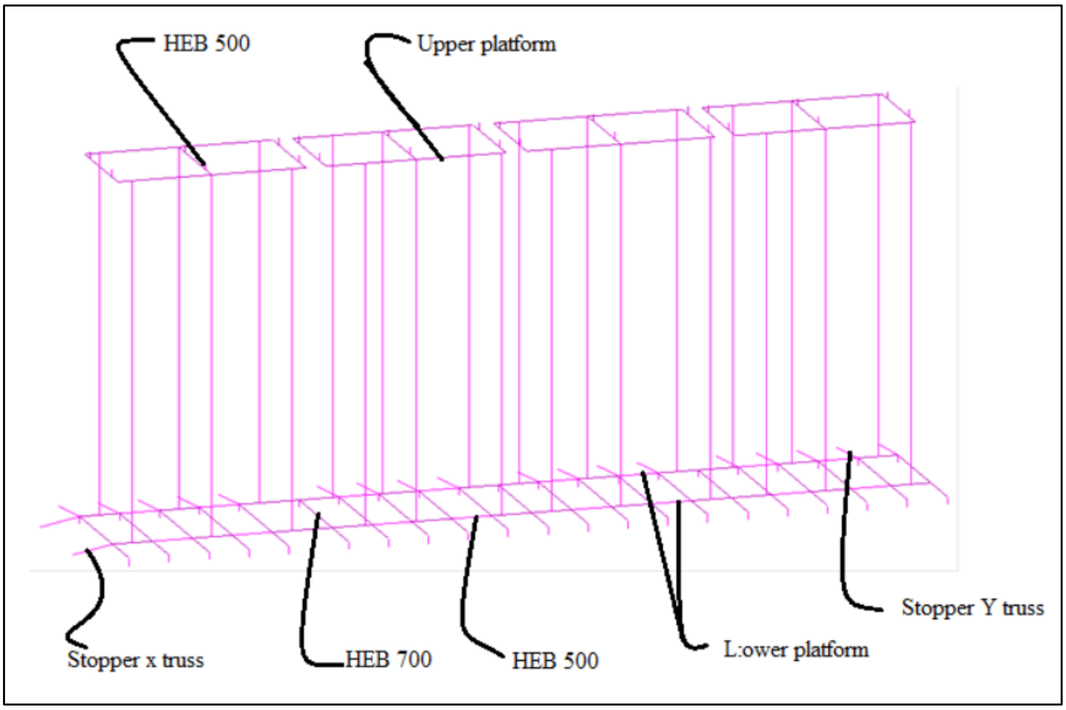


Figure 38: Stanchion Assembly

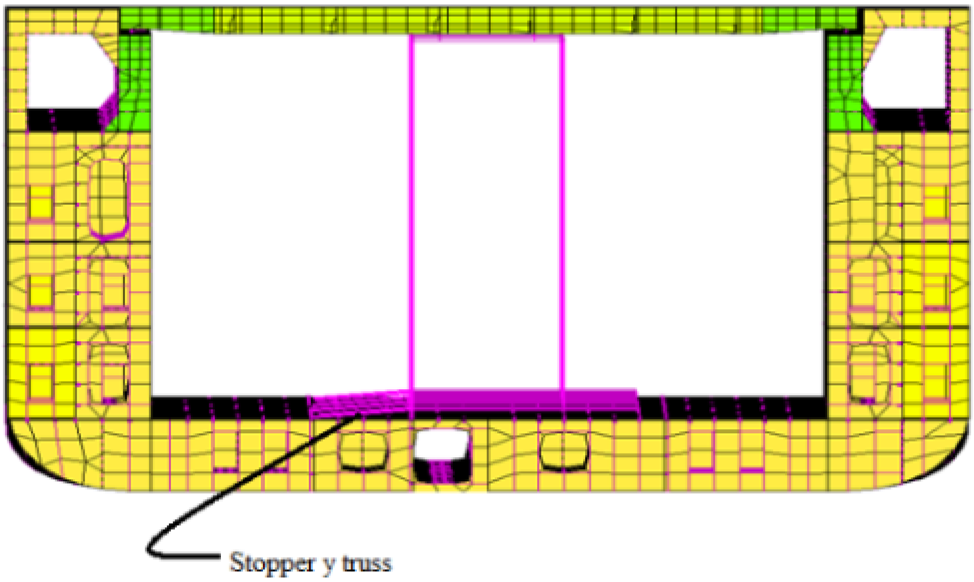


Figure 39: Front view of stanchion assembly

In Figure-38, we can see that there is an extra arrangement done by adding Stopper truss in Y and X direction, these trusses were added to the platform in order to avoid any big deformation in the model near the big opening.

Furthermore, to have a more elaborative view of the system, the stanchion assembly with all 4 hatch cover is shown below:

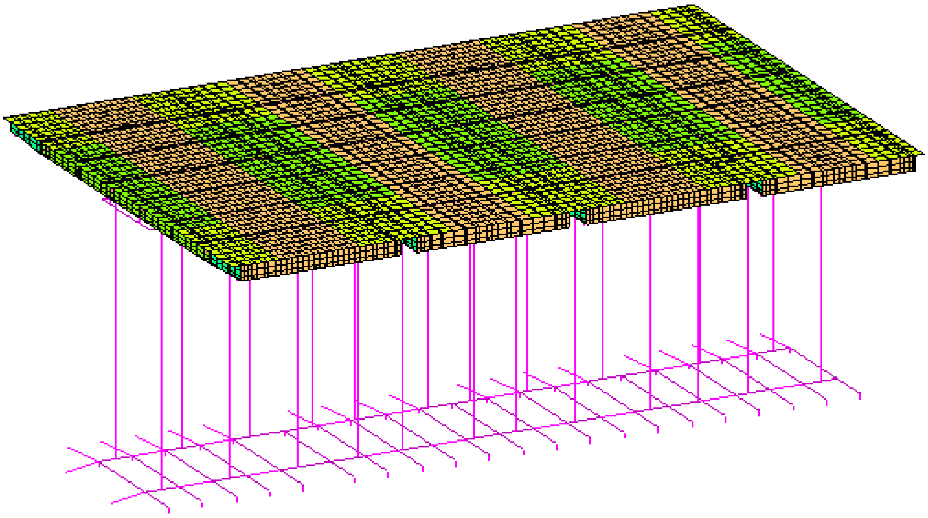


Figure 40: Hatch cover with Stanchion assembly

6.10 Auxiliary system

The Auxiliary systems were developed for all the heavy offshore cargos and equipment. The system was required only for those masses which were heavier than 200 tons and defined in the Shipload as Box masses. Since, these masses were generally in the cargo hold region except engine so, it was difficult to have a proper nodal mass distribution for these equipment. If the nodal masses were not computed correctly than it may lead to improper nodal force generation. Therefore, to cover up those heavy masses with the adequate nodal mass distribution these auxiliary systems were defined.

The auxiliary systems were modelled for the following equipment:

a) Main engine
b) Production reel #3
c) Production reel #4
d) Reel #6 SIV RM
e) Carousel

6.10.1 Modelling approach for auxiliary system

In general, the approach of modelling for all the equipment was same except Carousel. The procedure followed in case of carousel was much more complicated than the other four systems. In case of first four equipment, after modelling them as box masses in Shipload, its centre of gravity in longitudinal direction was known, taking this LCG as reference point various truss element with an area of **100.000 mm^2 and material no- 5** as per definition in **subchapter-6.2** were modelled between LCG and to the strong nodal point at the base of these box masses. The strong nodal point means the point which is both supported by a longitudinal and transverse member. Figure-41 shows the Box mass structure of Reel #6 with its nodal point arrangement.

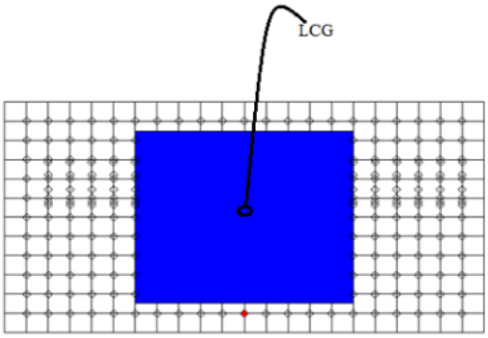


Figure 41: Box mass: Reel #6 with nodal point

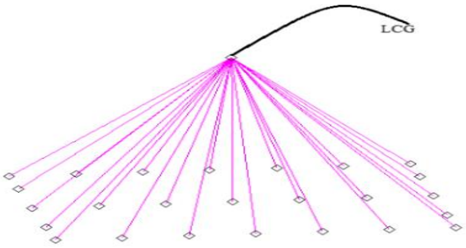


Figure 42: Auxiliary system of Reel #6

Figure –42 represents auxiliary system of reel #6 with its LCG and the strong nodal points to serve the aforementioned purpose.

In addition to Reel #6, the auxiliary system for the other three equipments was modelled in the same way and was added in the main model for strength analysis purpose.

6.10.2 Carousel auxiliary arrangement

Carousel in other words is a collection of all heavy cargo item at one place such that it can be moved from one place to another in an intact manner if required. Wish thesis perspective, the carousel is placed at the top of hatch cover almost in the middle of the vessel. To observe it in a better way, a stowage plan with the sectional view is shown in the figure-43 and 44 where a roundabout item represent the carousel.

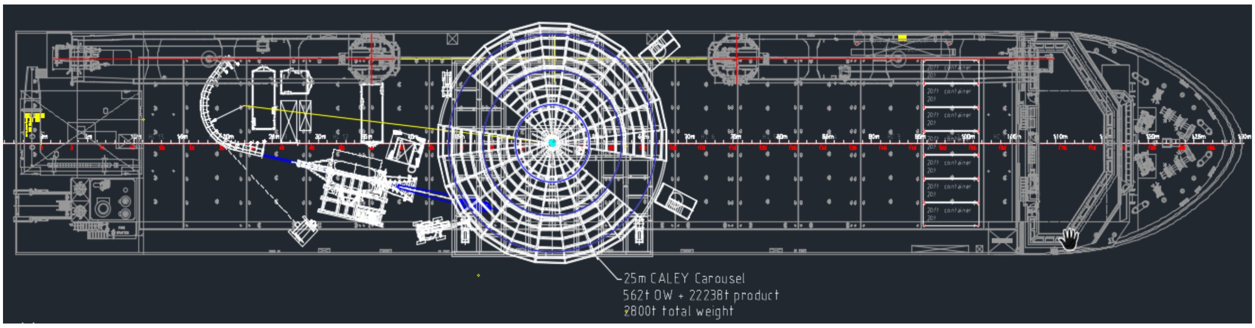


Figure 43: Plan view: Stowage plan

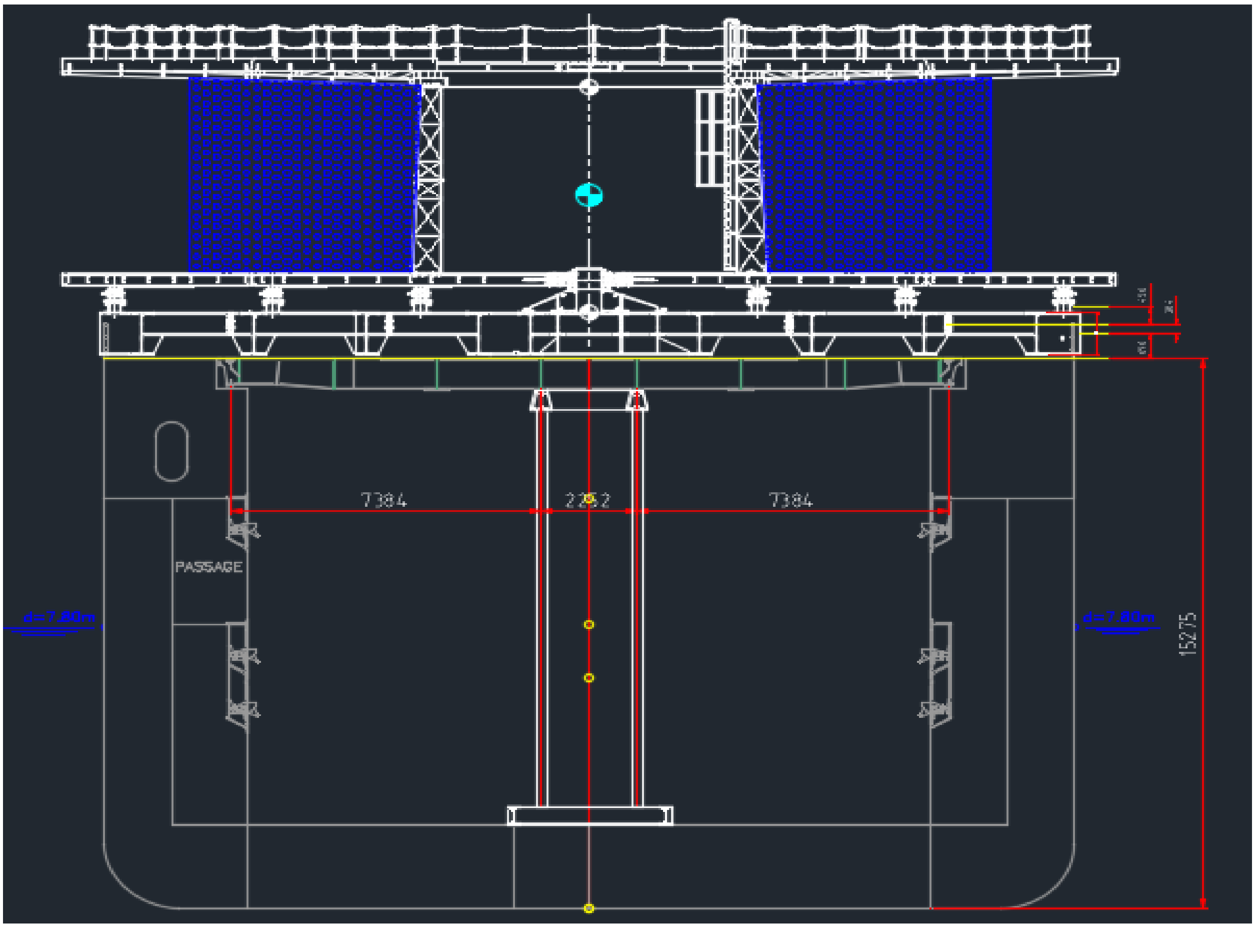


Figure 44: Section view: Complete carousel assembly

The carousel total mass were taken as **2800 tons** which in fact is a collection of many offshore equipment. Mainly, the carousel mass includes

a) Umbilical
b) Basement
c) Turning
d) Hydraulic winch
e) Deck deflector
f) Over boarding chute
g) Umbilical ship loose item
h) Spare parts in a container box.

All the above items together weighs around 2800 tons and were taken as a box mass in shipload to analyse its acceleration when the vessel would be subjected to wave loading conditions.

To have a more realistic approach while analysing the load cases the box mass was tightened using auxiliary system to place it at the desired position.

The modelling of auxiliary system was quite a complex process and utmost care was taken while defining the beams and trusses for the system.

6.10.3 Modelling of auxiliary system for carousel

The carousel assembly which was placed on the hatch cover includes all the parts which is mentioned above, under carousel mass items.

For modelling this assembly, the whole section was studied and the structures were split in to geometrical shapes such that it can be easily presented inside the GL-Frame Poseidon. Since, the carousel is a roundabout structure so, it was decided to divide in to a polygonal shape.

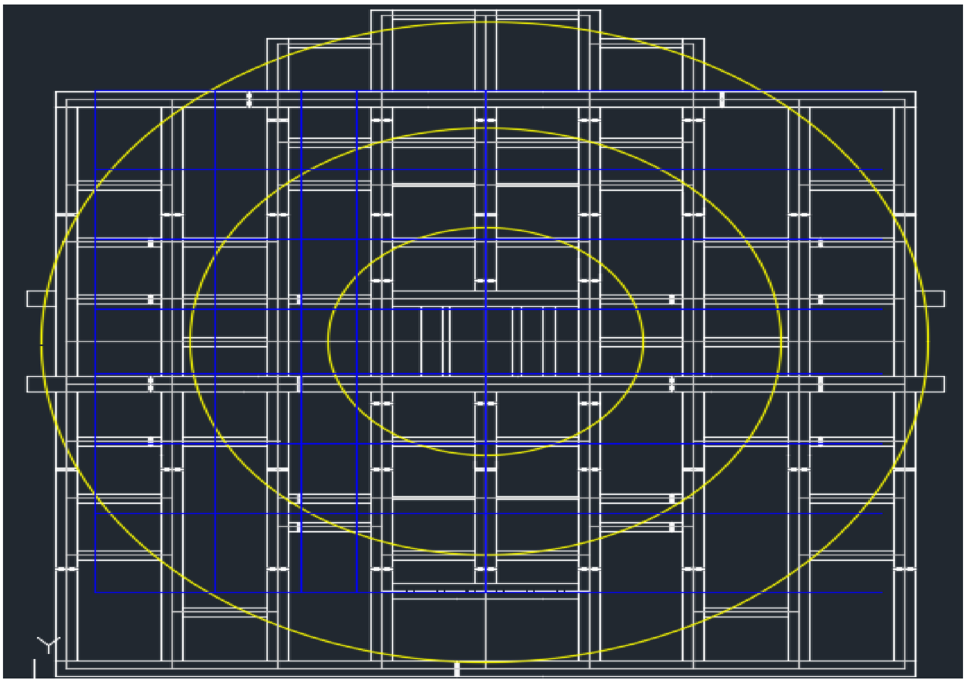


Figure 45: Carousel basement and turning tracks

Figure- 45 shows the carousel basement which is divided in to various smaller parts as mentioned below:

a) Basement
b) Basement Inner track
c) Basement Mid track
d) Basement Outer track
e) Basement truss in X direction
f) Basement truss in Y direction
g) Basement truss in Z direction
h) Truss Umbilical

Now, the next task was to replicate the above assembly in Poseidon by defining the various sections.

6.10.3.1 Basement

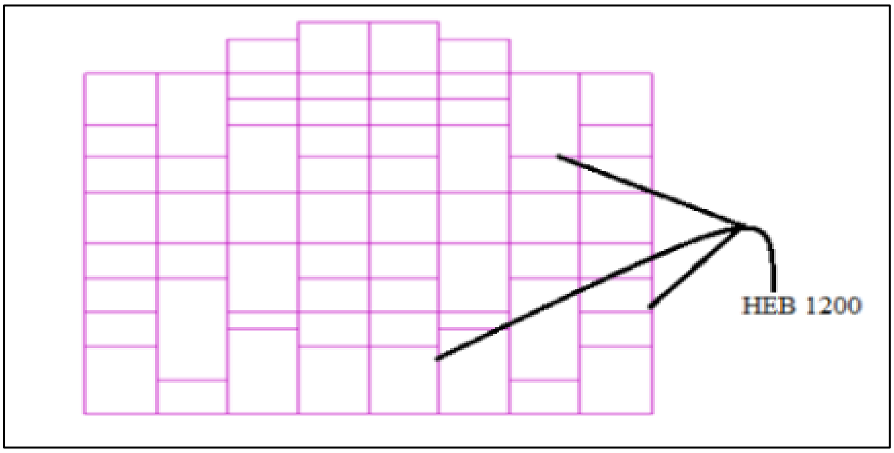


Figure 46: Basement of carousel

The basement was modelled in the same way as it was done in other auxiliary system. First of all, by referring the AutoCAD drawing the exact coordinate at each point was identified and then this coordinate value was put inside the software as nodes. After creating all the nodes, it was joined with an I-Beam element HEB1200. Only one type of I-beam element was used here at all the sections. Further, the Inner, Mid and Outer tracks were defined by considering the Umbilical structure and its mass distribution throughout the basement. Also, the basement was extended up to the Deck- A at both port and starboard side to make it better with load distribution in to the hull structure. The element definition is shown below in the table –10.

Table 10: Beam element

I-BEAM DEFINITIONS			
S No.	**Title**	**Flange (mm)**	**Web(mm)**
1.	HEB1200	1200*17	400*35

6.10.3.2 Basement inner track

There were a total of three tracks defined to estimate the correct COG of the carousel mass such that it can evenly distribute the cargo loadings. The inner track was defined as Dodecagon (12 side geometrical shape) by dividing all the sides at 30° angle and making it as COG. The element used here was I-beam profile HEB300.

Table 11: Beam element for all Inner, Mid and Outer track

I-BEAM DEFINITIONS			
S No.	**Title**	**Flange (mm)**	**Web(mm)**
1.	HEB1200	300*15	300*30

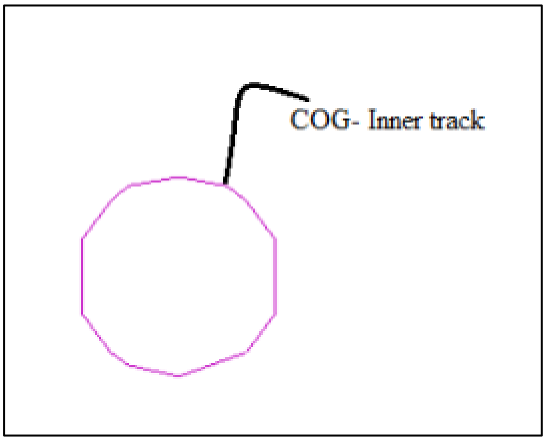


Figure 47: Inner track

6.10.3.3 Basement mid track

Similar to Inner track, the basement was defined for mid track also with a Dodecagon shape, again by dividing the side equally at 30°. This time, the COG were named as **"Sub-COG".** Later on, during the modelling Sub-COG were used as the defining COG to connect all the trusses from this to the other COG of Inner track, outer track and strong point of Basement.

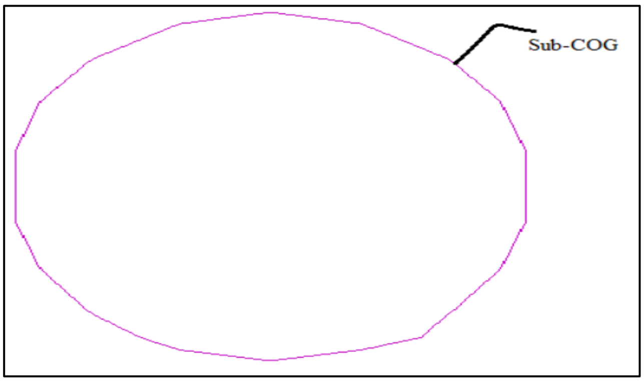


Figure 48: Mid track

6.10.3.4 Basement outer track

The outer track represents the track till which the whole umbilical is spread out. Similar to other two tracks it was also assigned with a Dodecagon shape. It covers the whole basement of the carousel and its dedicated COG was joined with the Sub-COG to provide the required shape.

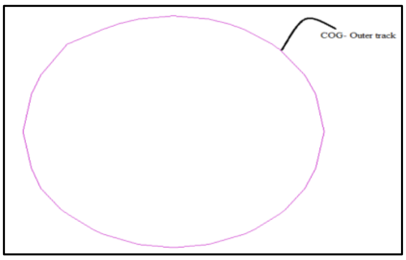


Figure 49: Outer track

6.10.3.5 Basement truss in x, y and z direction

The truss element was defined in X, Y and Z direction to support the basement by fixing to either the hatch cover or weather deck strong nodal points. When cargo would be loaded on to the basement, truss element would transfer axial load in respective directions. The area chosen for these truss element were **100,000 mm^2 with material no -5 as per definition in subchapter -6.2.**

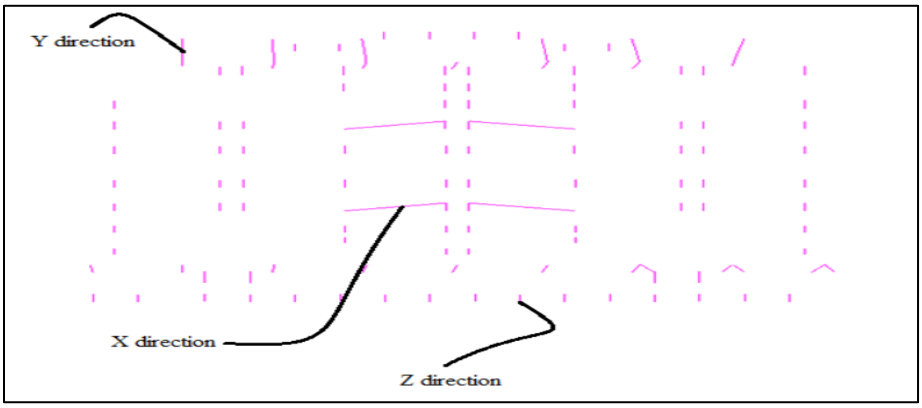


Figure 50: Basement truss in X, Y and Z direction

6.10.3.6 Truss umbilical

Umbilical is an offshore structure which can be as long as 7-8 kilometre. These umbilical needs to be transferred from harbour to the working site. To transfer it in an efficient manner these Umbilical are split in to many part and loaded on to the basement where it has been tightened enough for safe and secure traveling. The auxiliary system introduced here was to have a desired nodal mass distribution.

Truss umbilical with an area of **100.000 mm^2 and material no- 5** as per definition added in **subchapter 6.2** were added between Sub-COG and COG of Inner and Outer track in addition to the strong nodal point of Basement. Figure-51 shows the truss Umbilical arrangement.

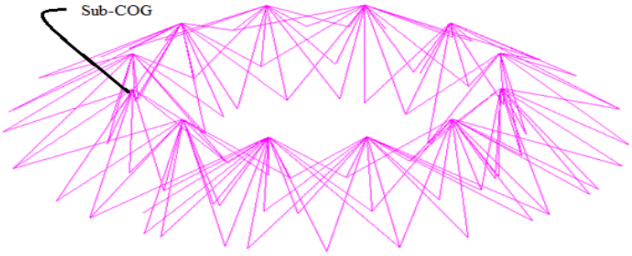


Figure 51: Truss Umbilical

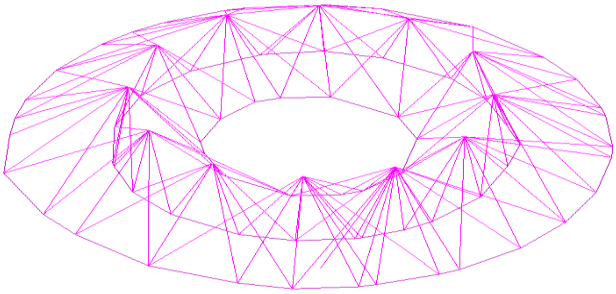


Figure 52: Truss Umbilical with Inner, Mid and Outer track

Figure –52 gives a clear picture of how these truss umbilical element were joined with the various Basement tracks.

6.10.3.7 Final assembly

Figure – 53 represents the final integration of all the above element. This whole assembly was imported in to the .bmf file of main model to do further analysis.

Finally, it can be seen that the same model was replicated as given in Figure-45 (which were planned in AutoCAD drawing). Thus, we can say that the model with carousel assembly can go forward for further load case analysis.

Also, Figure-54 shows the sectional view of the Carousel assembly imported inside the main model.

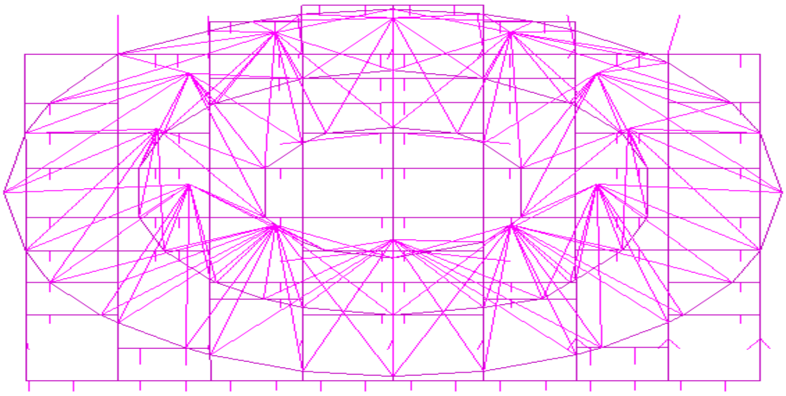


Figure 53: Whole carousel assembly

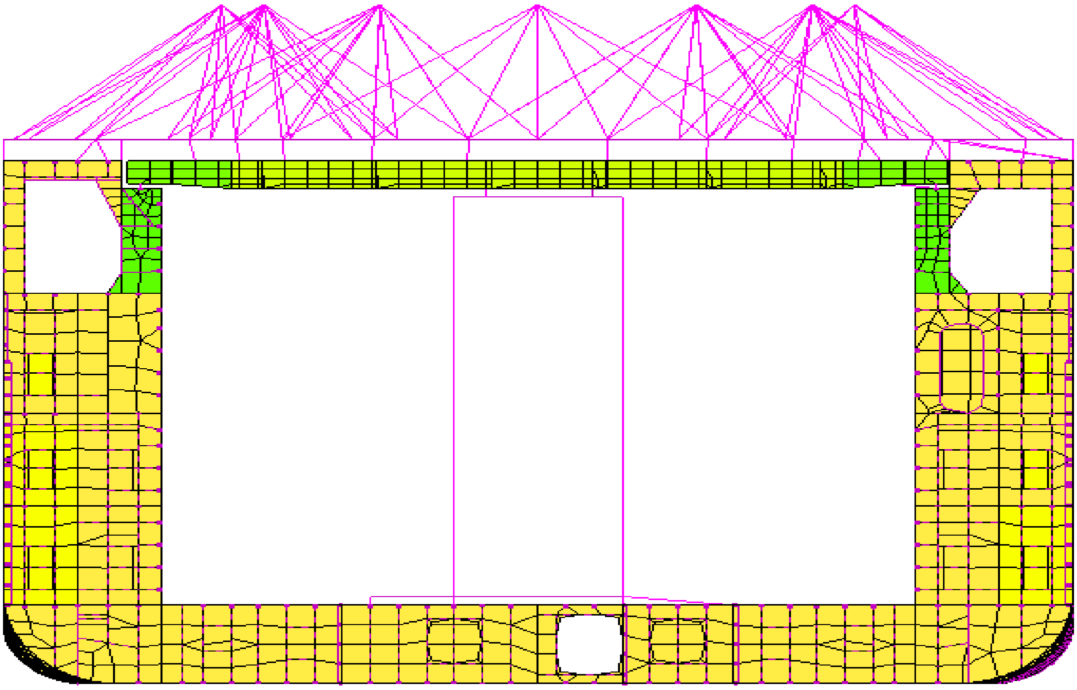


Figure 54: Sectional view: Carousel assembly on top of hatch c

7 FINITE ELEMENT MODELLING

Finite element analysis is a powerful tool for doing the structural analysis of a 2-D or 3-D models. In context of the thesis, the ship model was generated using Poseidon and also the Finite element analysis was done in the same software.

Without going much deeper in to the detail about modelling, the main focus in this section would be to proceed from the pre-processing part and ahead.

Once the model was ready in all aspects i.e. all the longitudinal, transversal members and cut-outs were placed at the correct position without any overlapping and double elements then, the pre-processing steps were carried out.

Under pre-processing, First of all, the model name and number were defined. After defining the name and number of the model Meshing of the model were carried out.

7.1 Mesh tolerance

7.1.1 Definition

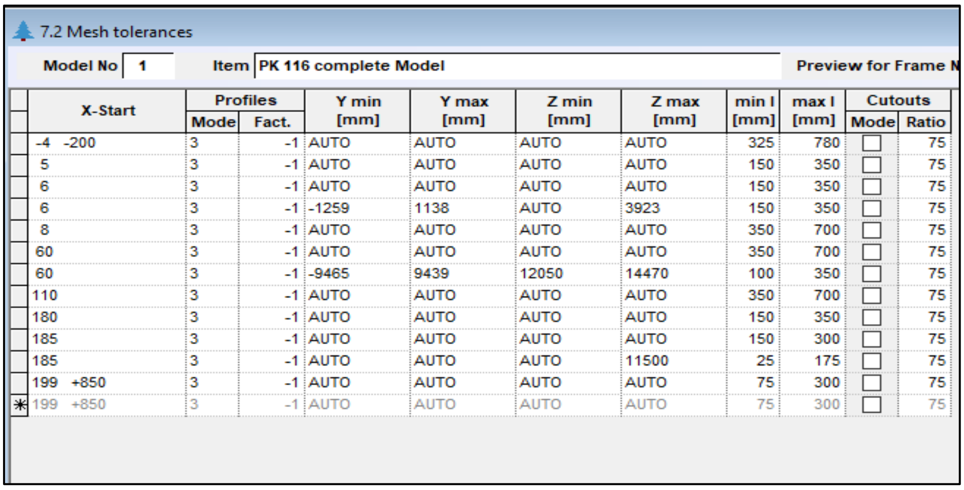
This section is aimed at defining parameters needed to control the automatic mesh generation. These parameters include:

a) Range of generation
b) Maximum and minimum edge length
c) Local degree of detail

The last parameter describes the way in which stiffeners were modelled. One of the main advantages here is that all parameters could be varied through the entire length of the ship which means that different values can be assigned for different frames. Furthermore, it was possible to specify different areas of parameters at the same longitudinal position.

7.1.2 Mesh rule

There were certain rules and obligation on which mesh sizes were defined, the mesh size were taken greater than or equal to the stiffener spacing such that the model can have more rectangular meshes than any other shapes which will make the calculation more feasible. Therefore, to explain it in a better way, the mesh tolerance table has been added below:



Model No 1
Item PK 116 complete Model

X-Start	Profiles		Y min [mm]	Y max [mm]	Z min [mm]	Z max [mm]	min l [mm]	max l [mm]	Cutouts	
	Mode	Fact.							Mode	Ratio
-4 -200	3	-1	AUTO	AUTO	AUTO	AUTO	325	780	☐	75
5	3	-1	AUTO	AUTO	AUTO	AUTO	150	350	☐	75
6	3	-1	AUTO	AUTO	AUTO	AUTO	150	350	☐	75
6	3	-1	-1259	1138	AUTO	3923	150	350	☐	75
8	3	-1	AUTO	AUTO	AUTO	AUTO	350	700	☐	75
60	3	-1	AUTO	AUTO	AUTO	AUTO	350	700	☐	75
60	3	-1	-9465	9439	12050	14470	100	350	☐	75
110	3	-1	AUTO	AUTO	AUTO	AUTO	350	700	☐	75
180	3	-1	AUTO	AUTO	AUTO	AUTO	150	350	☐	75
185	3	-1	AUTO	AUTO	AUTO	AUTO	150	300	☐	75
185	3	-1	AUTO	AUTO	AUTO	11500	25	175	☐	75
199 +850	3	-1	AUTO	AUTO	AUTO	AUTO	75	300	☐	75
199 +850	3	-1	AUTO	AUTO	AUTO	AUTO	75	300	☐	75

Figure 55: Mesh size

As shown in the Figure –55, there were different mesh sizes defined throughout the entire length of the ship. First column in the above figure indicates the frame no starting point and its following row indicates the frame number at which this mesh size will end and so on. The second column has two partition one is Mode and the other one is Factor, there were a total of 4 options with number 1,2,3,4 available for mode column but, number 3 fits exactly as per the current scenario. With Number 3 options, plates are modelled as shell element, nodes were generated for the trace curves of the stiffener and the stiffeners were modelled as Beam elements. Similarly factor, -1 was defined which means that nodes, generated from the y-z frame table are ignored, when no other edge is situated on this node.

Next step was to set up the area on which these meshing has to be defined. To do so, the range in the transverse and vertical sections were implemented either by just putting AUTO or by defining it with particular area value.

The most important task is to have a correct meshing size and to do so, there are some specific rules defined which are as follows:

a) Minimum edge length (mm): It should not be less than**' a/2**', where**' a'** is frame spacing.
b) Maximum edge length (mm): it should not be more than **1.2 a**, where**' a'** is frame spacing.

The last column is Cut-out, it has two part, first one is mode which can be kept in 'uncheck' condition if the calculations followed GL rules and the second part is ratio between the larger opening lengths to the maximum edge length.

7.1.3 Principle for the generation of fe nodes

For the automatic generation of FE nodes, the following principles were applied. A node on a functional element shall be generated, if

a) The angle, enclosed by any three consecutive points of the contour differs by more than 10° from the straight angle or
b) If thickness changes at the start or end of a plate, respectively, or
c) Stiffeners are present and the value of the parameter mode is equal to 3 or 4 respectively.

Figure-56 represents the 2D view of Frame no -65 with its nodal point.

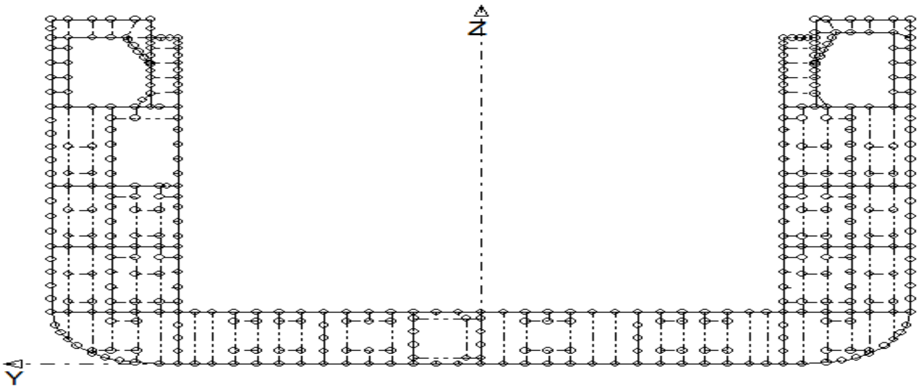


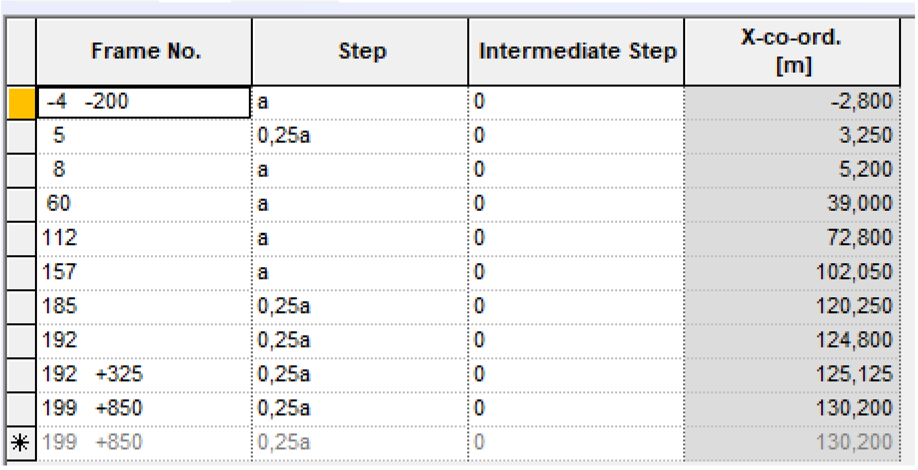
Figure 56: 2D –View of the Meshing [Frame – 65]

7.1.4 *Generate FE model*

After putting the mesh tolerance the FE- model were generated. The mesh size which is shown in the Figure- 55 were obtained after a number of iteration with different mesh sizes and then, inspecting the generated model again and again.

In this section, the overall range for the generation of the FE model was specified in the longitudinal direction of the ship in the same way, as for transverse sections. Also, the domains were defined with different step sizes in terms of the framing system.

As usual, longitudinal frame positions were given by a frame number, an absolute coordinate value, or an offset value and with variation in sizes a high flexibility for the automatic mesh generation were ensured.



	Frame No.	Step	Intermediate Step	X-co-ord. [m]
	-4 -200	a	0	-2,800
	5	0,25a	0	3,250
	8	a	0	5,200
	60	a	0	39,000
	112	a	0	72,800
	157	a	0	102,050
	185	0,25a	0	120,250
	192	0,25a	0	124,800
	192 +325	0,25a	0	125,125
	199 +850	0,25a	0	130,200
✳	199 +850	0,25a	0	130,200

. Figure 57: Frame table: FE model generation

Figure -57 shows the frame number distribution for the entire vessel to generate the FE model. After a rigorous exercise and repetitive calculation, the frame table shown above was obtained where, the first column indicates the frame no, the second column specify the number of steps in to which the frame distance (a) were divided. The third column could be used only if an additional cross section is required and the last column of x – coordinate was automatically generated by Poseidon corresponding to its frame number.

At last with the GO command, the generation of FE model were started for the relevant transverse and longitudinal members of the model.

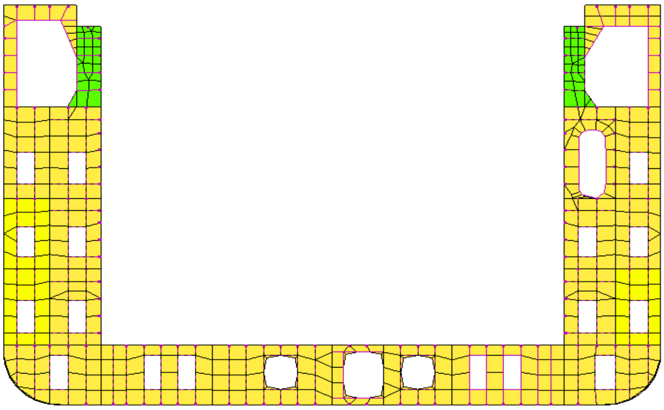


Figure 58: 2D View: Final mesh generation of Frame 65

The Figure-58 shows an example of the final mesh size in frame -65 located at load distribution area (amidships section).

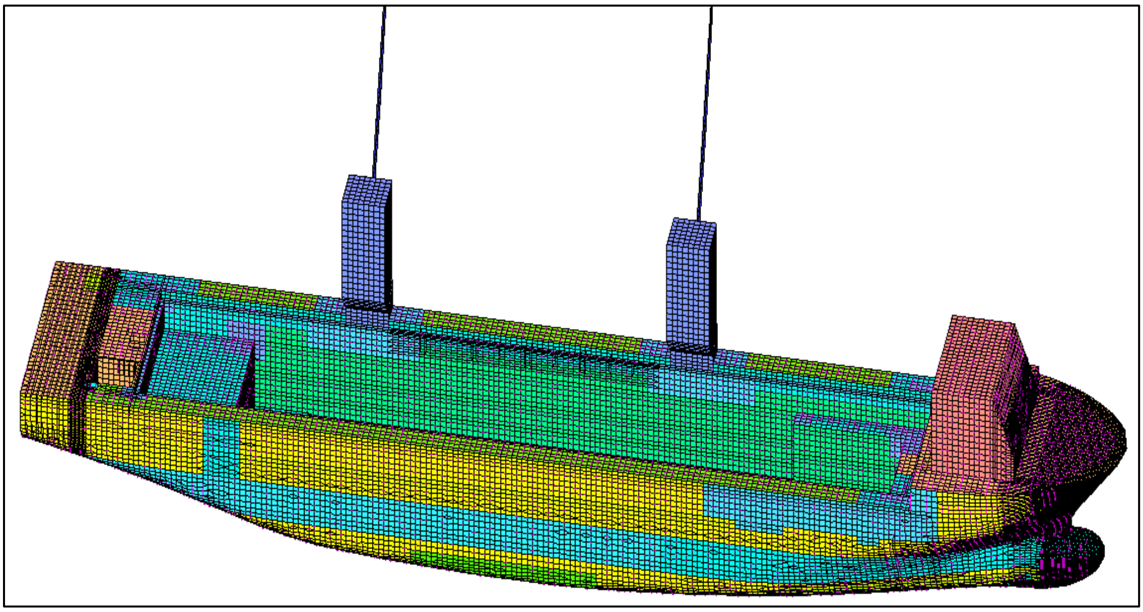


Figure 59: 3D View: Full FE model

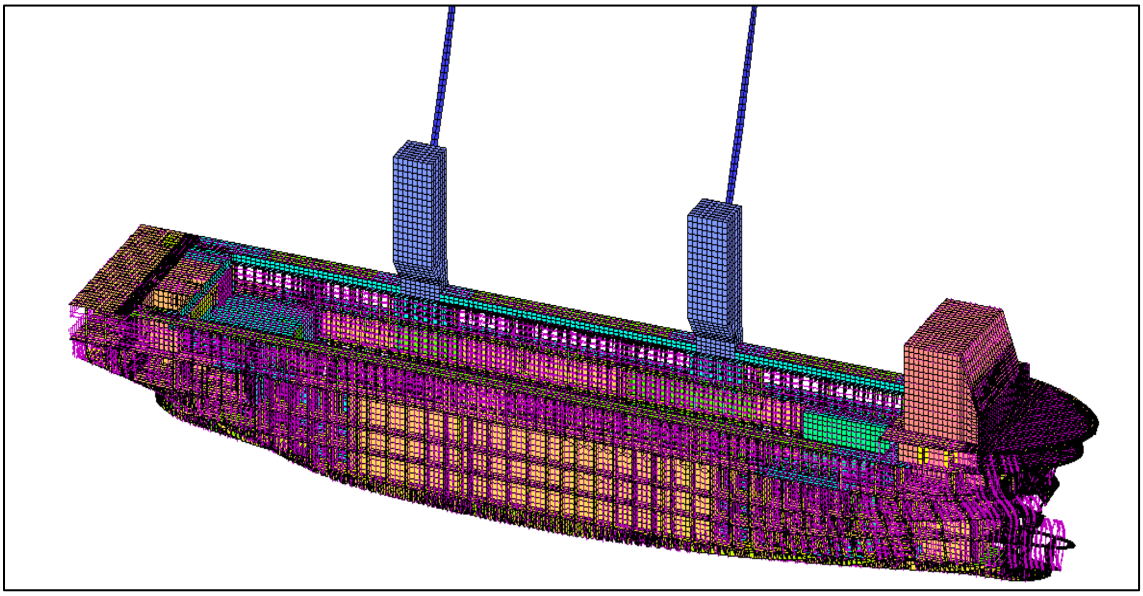
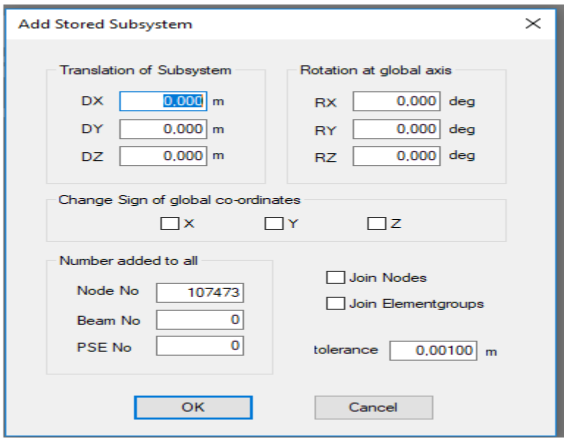


Figure 60: Structural view of the Entire model

7.1.5 Import of auxiliary system to main FE model

As discussed in the subchapter – 6.10, there were a total of 5 auxiliary system generated using GL-frame Poseidon which was imported in to the main model to fulfil the main objective of the thesis.

Importing of the auxiliary system was done in .bmf file of the main model by going in to the section “FILE” and then clicking the “IMPORT” button for .glf file which would result in a window shown below:


Add Stored Subsystem

Translation of Subsystem
DX 0.000 m
DY 0.000 m
DZ 0.000 m

Rotation at global axis
RX 0.000 deg
RY 0.000 deg
RZ 0.000 deg

Change Sign of global co-ordinates
☐ X ☐ Y ☐ Z

Number added to all
Node No 107473
Beam No 0
PSE No 0

☐ Join Nodes
☐ Join Elementgroups

tolerance 0.00100 m

Figure 61: File Import window: .glf

The Figure-61, shows a window which were popped up after pressing the Import button, as can be seen, the auxiliary system which were modelled can be imported by shuffling it in the translational direction with its exact X, Y and Z coordinate and also rotating it, in any direction if required. But, in our case, the auxiliary system were defined with its exact coordinate positions inside the model. Therefore, there were no shuffling required and the values in all these tabs were kept as **"zero".**

Further, the box was checked for join nodes and join element groups to import it correctly.

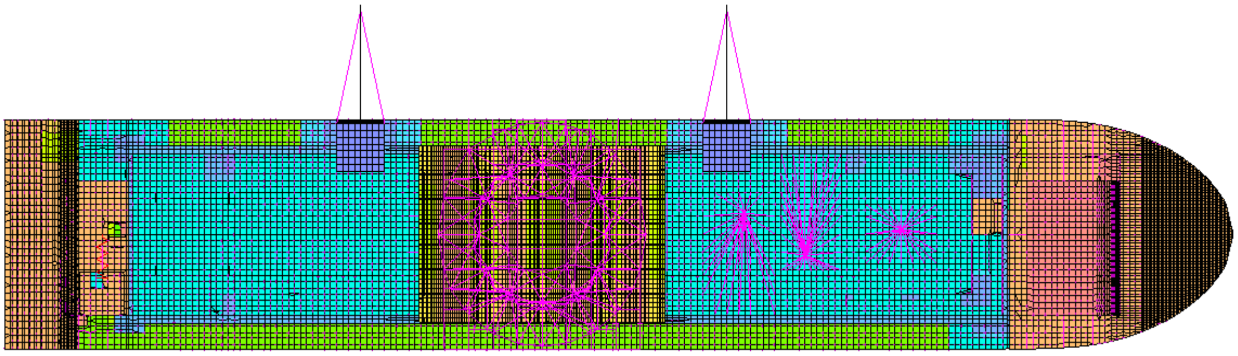


Figure 62: Collective view of Auxiliary system imported inside the model

The truss and beam element in Pink colour represents the auxiliary system shown in Figure-62.

7.1.6 Boundary conditions

Boundary conditions were defined to balance all the forces by assigning a different element group in the FE-model as:

a) Define a new element group.
b) Under that element group define the node number which was selected.

An element group named as **"Boundary Elements"** was created under which a total of three nodes were chosen in the entire vessel. Out of the three nodes, two nodes were chosen at the aft and one node was selected at the forward section of the model.

The nodes exact coordinates are mentioned below:

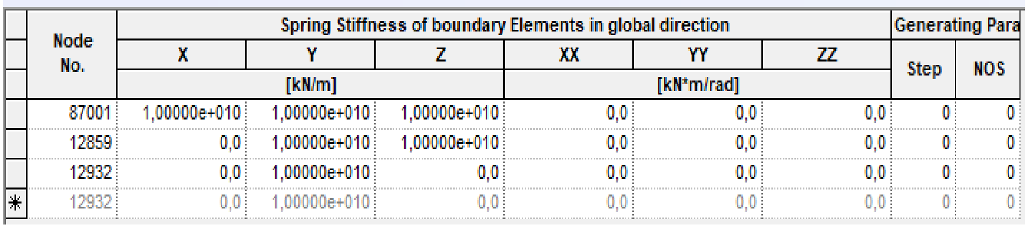
Table 12: Boundary element Nodes

NODES POSITION							
S No.	**Node no**	**AFT(m)**			**FORWARD(m)**		
		X	**Y**	**Z**	**X**	**Y**	**Z**
1.	87001	-	-	-	111.15	0	0
2.	12859	7.8	0	0	-	-	-
3.	12932	7.8	0	15.275	-	-	-

After successful selection of nodes, the boundary elements in the form of spring elements with stiffness for 6 degrees of freedom were elected. The spring element which were chosen for boundary element has stiffness value, **1*e^10 KN/m.**

As per global strength analysis rule for fixation of boundary element direction, At Fore section of the vessel, boundaries were fixed in all three coordinates (x, y, and z) but, at the aft area the boundaries were fixed in Y and Z direction on the weather deck and only in Y direction on the shell element.

Figure-63 shows, how the boundary element were exactly defined for the model with its node number, spring stiffness and direction of boundary fixation.



	Node No.	Spring Stiffness of boundary Elements in global direction						Generating Para	
		X	Y	Z	XX	YY	ZZ	Step	NOS
		[kN/m]			[kN*m/rad]				
	87001	1,00000e+010	1,00000e+010	1,00000e+010	0,0	0,0	0,0	0	0
	12859	0,0	1,00000e+010	1,00000e+010	0,0	0,0	0,0	0	0
	12932	0,0	1,00000e+010	0,0	0,0	0,0	0,0	0	0
*	12932	0,0	1,00000e+010	0,0	0,0	0,0	0,0	0	0

Figure 63: Boundary element definition

Please note that the rotation around the global X,Y,Z direction were not considered here that's why the field is empty in the above table.

A pictorial representation of Boundary conditions is shown in the Figure -64

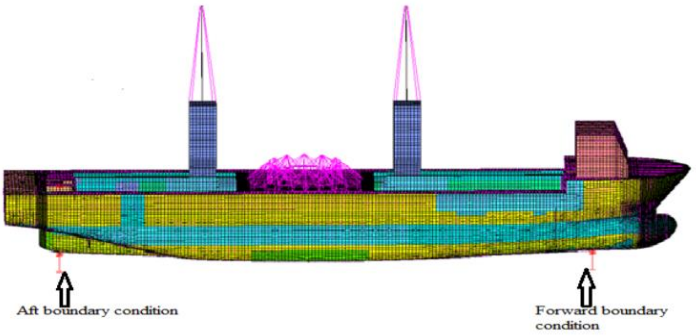


Figure 64: Boundary Conditions

8 SHIPLOAD ANALYSIS

As mentioned earlier, SHIPLOAD is an integrated load generation tool for FE analysis developed by Germanischer Lloyd. Without going again in to the step by step explanation which is already covered in the earlier part of this report, the main focus here would be to get in to the procedure which were followed to generate the nodal forces that can be applied in to the Poseidon FE program to do the global strength analysis of the vessel.

In short, the steps which were followed to achieve the required target is depicted below:

1. Import of the FE model
2. Modelling of the mass distribution (steel weight, equipment, ballast, cargo)
3. Computation of the still water floating position
4. Computation of the hydrodynamic potentials by strip method
5. Selection of load cases
6. Generation of balanced nodal load cases.

8.1 Fe model import

In context of the thesis, the FE-model was generated using Poseidon with desired meshing as explained in the chapter FE- Model. The model was imported under Main particular category of Shipload in .bmf file format which was generated from Poseidon. After model import, Shell elements were calculated which include the whole shell element with transom plate, stern plate and bow.

The next task was to generate the .xml file which was required to perform the hydrostatic and hydrodynamic calculations.

8.2 Frame table

Frame table was defined in the same way as explained in the subchapter 6.3.

8.3 Hatch cover

Generally, hatch covers are not explicitly modelled by finite elements (as they must not contribute to the overall stiffness of the model), but, in purview of the thesis, four Hatch covers were modelled between frame no 61 to 108 in Finite element file in order to put the carousel load on it. Other than these four hatch covers, all other hatch covers were defined in the Shipload file for the load applications of Deck Containers.

A pictorial representation of hatch covers inside the shipload file is shown below:

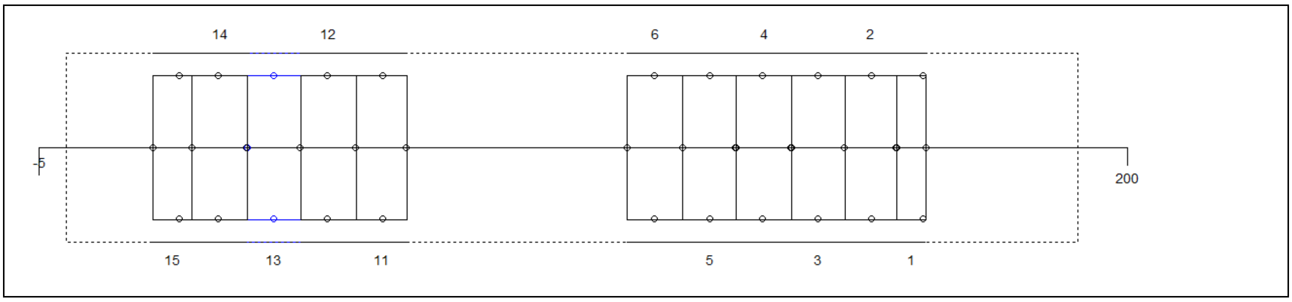


Figure 65: Hatch cover arrangement in Shipload

8.4 Mass distribution

Proper mass distribution in a ship is responsible for better stability, typical components of a mass distribution includes steel weight, equipment, accommodation, bunkering, water ballast and cargo. While some components differ for different loading conditions for e.g. bunkering for departure and arrival conditions whereas some of the components do not change in any condition such as ship lightweight. Therefore, to have a better view on mass distribution a Block diagram representing different cases is shown below:

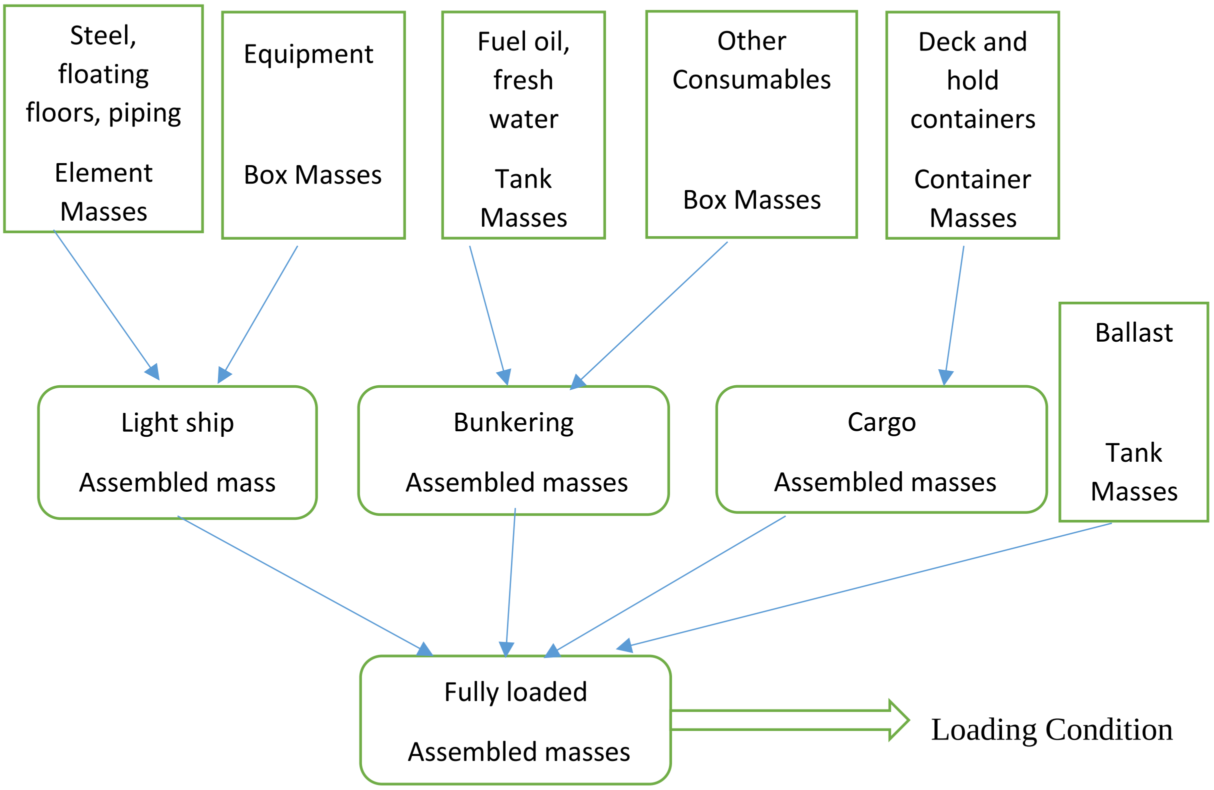


Figure 66: Building blocks of typical loading conditions of a vessel

8.5 Category of mass distribution

a) Element group masses
b) Box masses
c) Tank masses
d) Container masses
e) Assembled masses

8.5.1 Mass distribution technique in shipload

In Shipload, the mass items are represented internally by mass matrices. A mass matrix relates nodal loads to nodal accelerations. Nodal accelerations are derived from the computed hydrodynamic rigid body accelerations under which the translational rigid body acceleration is directly applied to all nodes, whereas rotational rigid body acceleration is converted to translational nodal accelerations that depend on the distance from the axis of rotation. Generally, all mass matrices couple the nodes to which they are connected.

If we can look one by one on how different category of masses are represented than we can see that the Nodal masses from element group masses and box masses are represented by diagonal mass matrices, Container mass matrices actually couple the bearing nodes in the FE model. The tank masses would couple all wet nodes with in the tank and require a more complex computation but, since the coupling has only a minor effect on the global strength examination, static tank masses are excluded from the mass matrix approach and dynamic tank masses are treated as nodal masses.

8.5.2 Element group masses

Element group masses were computed from the structural masses of the vessel through its FE-model. In FE-model, each element group belongs to one and only one element group, and all elements of a specific element group have the same element type for e.g. truss, beam, plain stress, shell, boundary etc.

Element group masses were computed from the element geometry and the associated material density without further user input. But, later on these masses were manipulated by changing some factors under the assembled mass item to achieve the required target of total masses.

8.5.3 Box masses

The purpose of box masses is to distribute the prescribed total masses to a spatial region which is described by either a single coordinate box (a rectangular region specified by two diagonally opposing point) or a combination of coordinate boxes.

Under box masses, various offshore equipment such as production reel, cable reel, Carousel Umbilical, Engine, Crane structure, Superstructure with other essential equipment such as mooring assembly, anchor chain, steering gear, ship gear etc. masses were estimated. Also, some general supply material which is spread throughout the vessel was accumulated as box masses to achieve the target mass. In addition to all these masses, a cumulative weight of 200 tons was kept at the aft portion of ship to accommodate propeller, funnel, gearboxes etc. as box mass.

The phenomenon behind the Box mass distribution is that the mass is distributed as homogenously as possible. In this way, nodes in regions of higher nodal density will get less mass per node than nodes in regions of lower nodal density.

A comparison of FE- structure, box mass application and nodal mass distribution of superstructure is shown below:

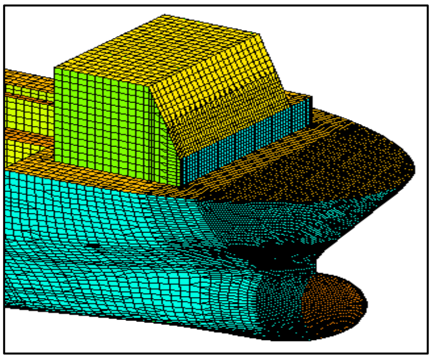


Figure 67: FE- Structure of Superstructure

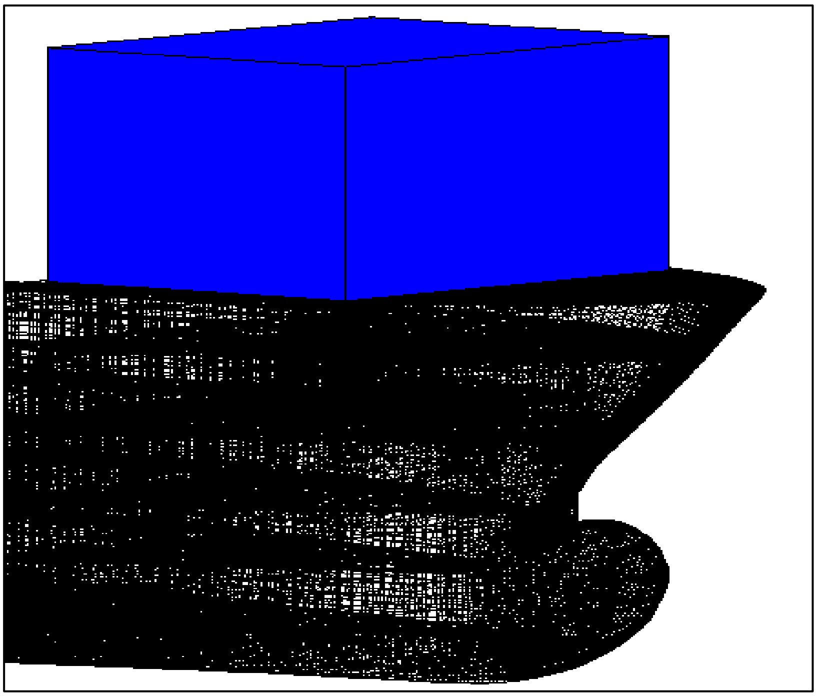


Figure 68: Box mass of Superstructure

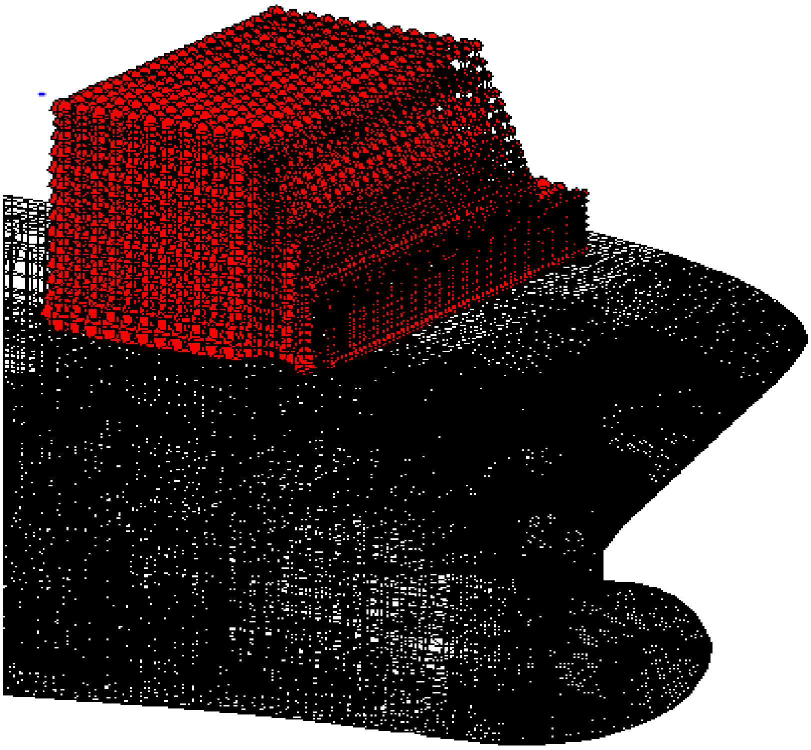


Figure 69: Nodal mass distribution of Superstructure

Table 13: Box Mass in Shipload

BOX MASSES DESCRIPTION

S No.	**Title**	**Mass (T)**	**LCG (m)**	**TCG (m)**	**VCG(m)**
1.	Prodreel #3 1,7	184,2	77,3	2,1	8,3
2.	Prodreel #4 1,8	184,2	83,8	-2,1	8,3
3.	Reel #6 SIV / RM 1,9	155,9	94,3	0	7,05
4.	Carousel	2800	55,4	0	19,83
5.	Engine	270	15,5	0	5,5
6.	Genset 1	24	15,6	-4,7	9,25
7	**Genset 2**	**24**	15,6	-9,45	9,25
8	Crane foundation #2	275	35,75	10	24
9	**Anchor Fore SB**	**35**	**120**	**-2,5**	**15,76**
10	Anchor Fore PS	35	120	2,5	15,76
11	**Mooring assembly**	**25**	**120**	**2,5**	**15,76**
12	Steering Gear	125	-0,65	1,25	9,3
13	**TWD / WD support 2,1**	**50**	**55,73**	**0**	**8,3**
14	Superstructure	450	112,5	-1,5	22,7
15	**Constant supply**	**350**	**5**	**1**	**12,9**
16	Ship gear	35	100	0	3
17	**Supplies**	**250**	**15**	**0**	**11,5**
18	Aft Structure	200	5,2	1	9

8.5.4 Tank masses

All the tanks, comprising of Fresh water, Diesel oil, HFO, Ballast water etc. were installed inside the FE-model by implementing them in the tank masses section of shipload. Tanks were defined as surrounding boxes where the combination of all closed cells that lie completely inside the box constitutes the tank.

Other data which were used for tank load distribution was its X, Y, Z coordinate according to the original drawings. For application of tank loads in to the FE- model, the Shipload software is wise enough to identify (topologically) closed regions in the FE-model.

8.5.4.1 Phenomenon behind tank mass distribution

8.5.4.1.1 Hydrostatic tank pressure

For computation of hydrostatic pressure, first of all, the actual mass distribution with in the tank was calculated by finding the position of its free surface in the still water floating condition of the ship then, the resulting hydrostatic pressure was transferred to the FE-model by forces that are perpendicular to the tank walls and increases linearly with the distance from the free surface.

8.5.4.1.2 Dynamic tank pressure

The dynamic tank pressure resulting from the ship's acceleration minus gravity was transferred to the model as nodal masses. The same phenomenon which was used for Box masses was implemented here with a restriction to the selection of node at tank wall elements below or directly above the tank's free surface.

A typical example of Tank mass distribution for Ballast water tank no – 6-1 Starboard side is shown below:

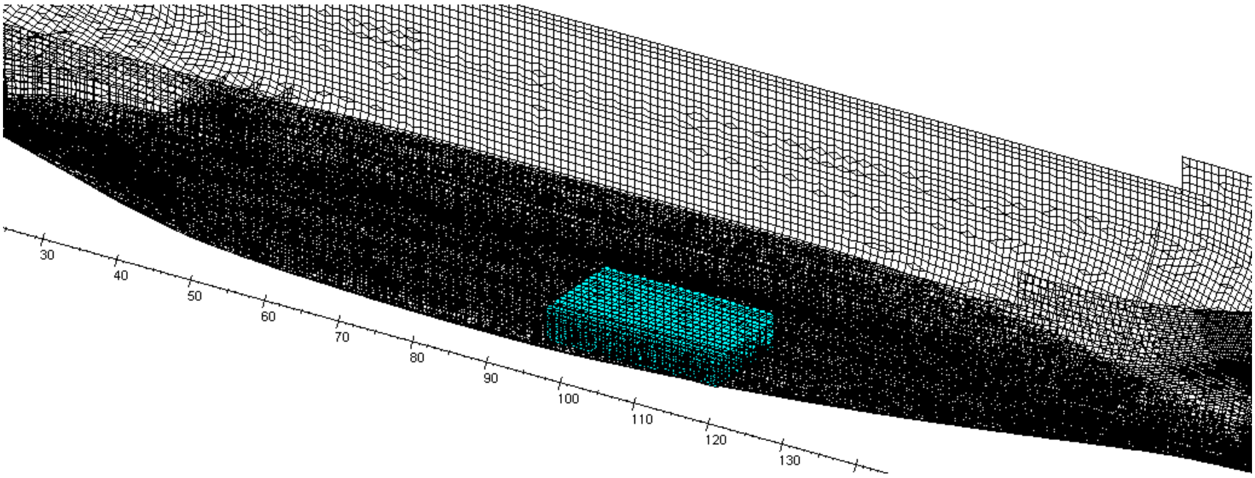


Figure 70: Ballast water tank 6-1 SB from Shipload

The tank mass distribution from Shipload is tabulated below:

Table 14: Tank Mass in Shipload

TANKMASS DESCRIPTION					
S No.	**Title**	**Volume (m3)**	**LCG(m)**	**TCG(m)**	**VCG (m)**
1.	FW SB	87,36	111,8	-1,2	5,1
2.	FW PS	87,36	111,8	1,2	5,1
3.	DO SB	29,74	8,24	-7,57	6,94
4.	DO PS	29,74	8,24	7,57	6,94
5.	HFO 7-2 SB	223,59	56,22	-9	6,85
6.	HFO 7-2 PS	137,59	56,22	9	5,1
7	**HFO 3-1 PS**	**93,92**	104,86	2,79	3,81
8	BW 3-2 SB	146,92	104,77	-5,84	5,17
9	**BW 4-2 SB**	**330,91**	**95,14**	**-8,65**	**8,3**
10	BD 4-1 SB	170,19	96,04	-2,89	1,2
11	**BD 4-1 PS**	**116,37**	**95,96**	**3,8**	**1,22**
12	BW 5-2 SB	325,59	83,84	-9,33	4,53
13	**BD 5-1 SB**	**250,17**	**83,46**	**-4**	**1,17**
14	BD 5-1 PS	193,66	83,45	4,9	1,17
15	**BW 5-2 PS**	**289,3**	**83,18**	**-9,62**	**4,64**
16	BW 6-2 SB	505,14	69,84	-9,76	5,94
17	**BD 6-1 SB**	**266,41**	**69,55**	**-4,05**	**1,15**
18	BD 6-1 PS	207,2	69,55	4,95	1,15

19	**BW 6-2 PS**	**419,11**	**69,84**	**9,91**	**5,17**
20	BD 7-1 SB	266,41	56,55	-4,05	1,15
21	**BD 7-1 PS**	**207,21**	**56,55**	**4,95**	**1,15**
22	BD 8-1 SB	229,98	41,93	-4,05	1,15
23	**BD 8-1 PS**	**178,86**	**41,93**	**4,95**	**1,15**
24	BD 9-2 SB	280,07	29,44	-9,68	4,78
25	**BD 9-2 PS**	**280,09**	**29,44**	**9,68**	**4,78**
26	BD 9-1 PS	124,68	28,74	5,97	1,75
27	**BD 9-1 SB**	**261,64**	**28,7**	**-3,94**	**1,44**

8.5.5 Container masses

The container mass items represent hold or deck container bays. In the container mass section of the Shipload only the container arrangement is specified. The average mass per container and the vertical centre of gravity are specified later, during the input of the assembled mass items.

There were two different kinds of container defined, one was 20 foot and the other one 40 foot as per the requirement. Tween decks, spooling tower, Beams for tween deck, carousel equipment and some loose container items were defined as 20 ft. container whereas the Spooling tower were defined with 40 ft. container inside the model.

The naming for 20 ft. container was done with odd number and even number was used to define 40 ft. container.

8.5.5.1 Definition of container mass

First, a container bay is defined by its longitudinal centre of gravity and the aft and fore positions at which horizontal loads should be applied to the structure. Secondly, container stacks were defined by their lateral centre of gravity, their vertical support and their vertical extent in terms of upper and lower tier id. For deck containers, tier id greater than or equal to 82 were used.

8.5.5.2 Calculation of nodal mass for container

For nodal masses, the degree of freedom varies for hold and deck containers. They also have difference in strength factor. The strength analysis for hold container involves all lateral loads to the aft and fore transversal bulkheads independent of the still water floating position and the hydrodynamic acceleration.

For deck containers, hatch cover definitions control to which structural nodes are applied. Vertical loads were applied at the aft and fore edges of the hatches at the lateral positions of the corners of the container stack. Horizontal loads were applied at the stopper positions. Load resulting from containers that completely or partially overlap the hatch cover were applied to the nearest node on deck.

An example of deck container with 20 foot (Blue colour) and 40 foot (Orange colour) sizes are shown below:

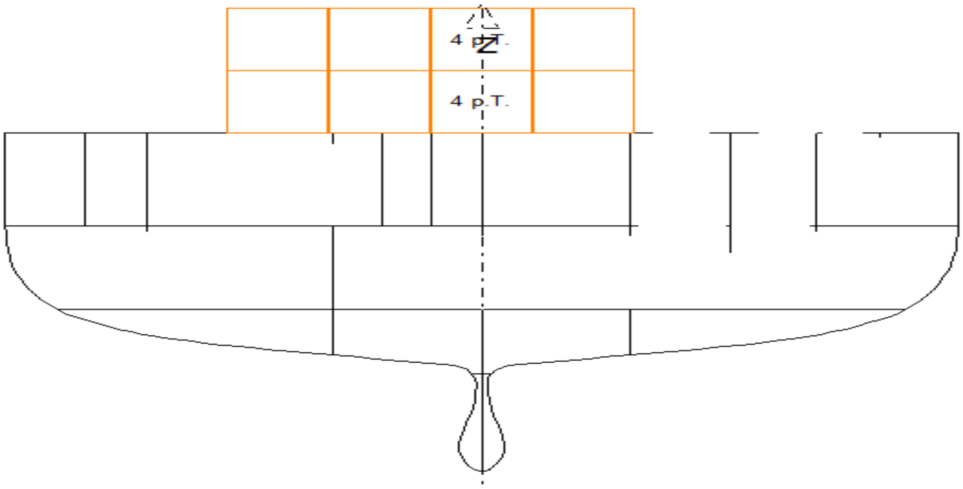


Figure 71: 40 foot container loaded as Stability pontoon at Aft

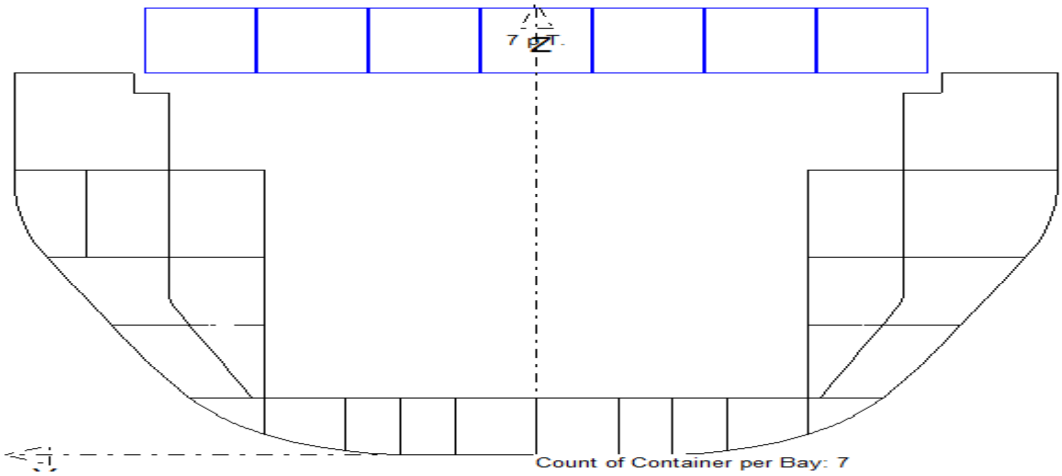


Figure 72: 20 foot container loaded as loose container item at the Fore

8.6 Assembled masses

The assembled mass item is the combination of element group, box, tank and container masses to form a mass distribution. Under the assembled mass item, the contribution from individual basic mass items are controlled by factors for structural and box masses and level and densities for tank masses. For container masses, an additional parameter was given for vertical centre of gravity other than its factor value.

8.6.1 Calculation of assembled masses

The computation of total mass, centre of gravity and tensor of inertia for one or for all assembled mass items takes into account any dependencies on other assembled mass items and perform calculations in correct order.

In context of the thesis, the total assembled mass item was taken from the stability booklet of PK-116 where each and every mass including Box, Tank and Container masses were taken into account to calculate the correct draft at fore and aft with almost zero trim and heeling angle. In short, the intact stability of the vessel was achieved with proper distribution of masses and a decent GM value was obtained.

After achieving the intact stability of vessel with LOADMASTER (stability software), the priority was to achieve the same conditions and distribution of masses in shipload and try to match the draft, trim and heel angle beside GM. To achieve the same GM and centre of gravity in shipload, the factor for element group were adjusted by fixing the target total mass and centre of gravity. In this way, the element group factor got changed accordingly to achieve the required total mass and centre of gravity as shown in the table below:

A comparison between Stability booklet data and shipload data for COG, Mass, GM and Draught are tabulated below:

Table 15: Comparison Table

Data Comparison between Stability booklet and Shipload:									
S No.	**Title**	**Mass (T)**	**LCG(m)**	**TCG(m)**	**VCG(m)**	**GM (m)**	**Heel (deg)**	**Draught Aft(m)**	**Draught Fore(m)**
1.	Stability Booklet	17009.1	60.06	0.01p	9.48	0.92	0.0p	7.822	7.774
2.	Shipload	17009	60.04	0.0p	9.56	0.92	0.009p	7.840	7.807

After mass distribution, the next step is to do the hydrostatic and hydrodynamic calculations to generate the required nodal forces which can be imported to the FE-Analysis software for global strength analysis. Hydrostatic and hydrodynamic calculations were performed internally by the Shipload. The internal steps of computation by the software is briefly explained below:

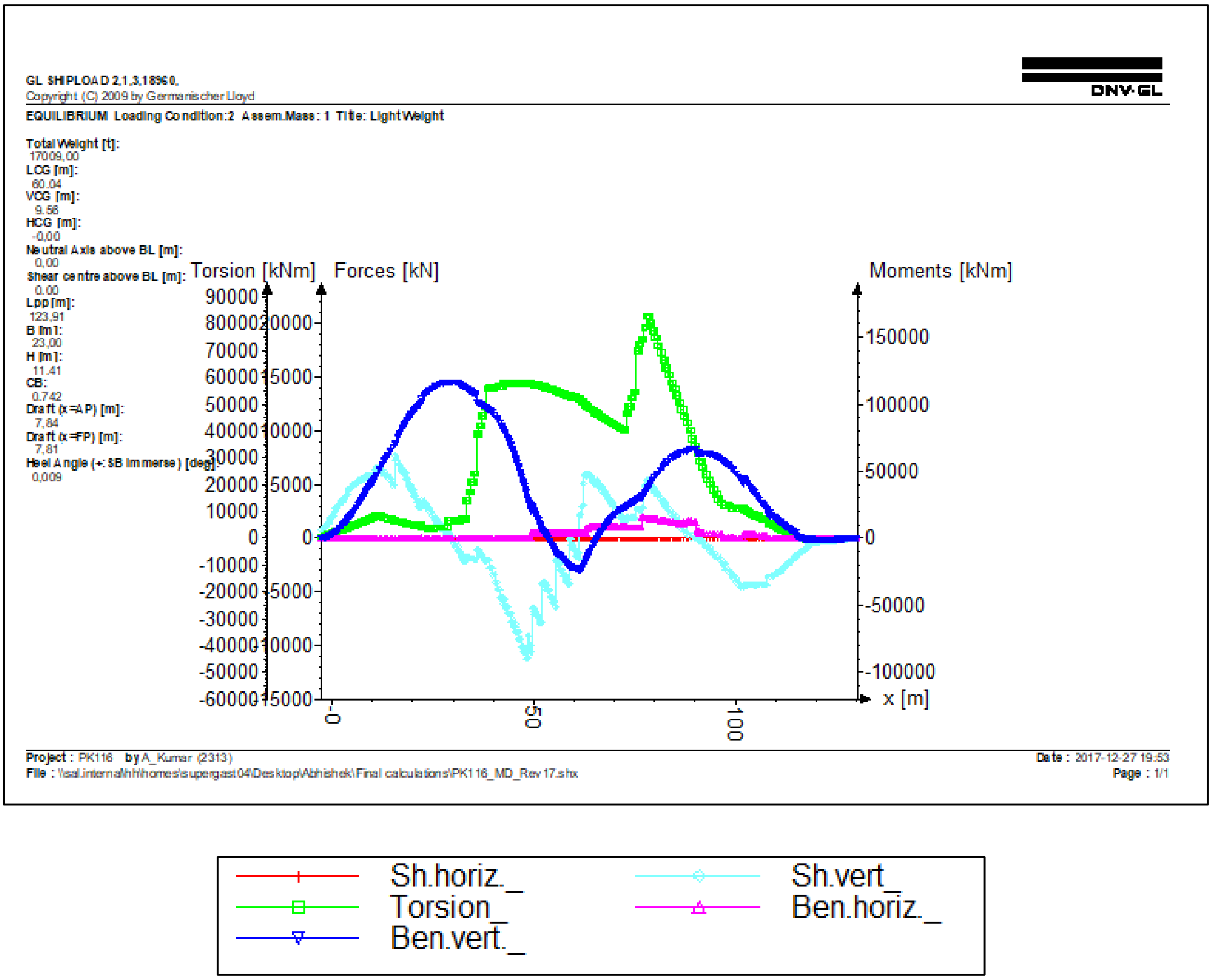
8.6.2 Hydrostatics

For the correct application of hydrostatic pressure and for the determination of trim and heeling angle, the floating position in still water must be found. In Shipload, static trim is computed by a Newton iteration of draught, trim angle, and heel angle until hydrostatic equilibrium is achieved, i.e. until the buoyancy forces and moments balance the gravity forces and moments of the mass distribution. Buoyancy forces are computed by integration of hydrostatic pressure over the hull described by the shell elements, gravity forces are obtained by multiplication of the mass distribution with the current gravity vector (in ship coordinates). The Jacobian matrix, which is required for the Newton iteration, is computed numerically by finite differences.

The hydrostatic calculations were done under the category "Main" by just pressing the hydrostatic equilibrium button. The output obtained from this calculation was draught at the aft and fore perpendiculars and the heel angle. To achieve the correct draught and heel angle, a number of adjustment in the mass distribution section was done and hydrostatic equilibrium was computed again and again until the desired value for draught and approx. zero heeling was achieved.

A graphical representation of Shear force, Bending moment and Torsion as per the given mass distribution is shown in the Figure no – 73.

In the diagram below, we can see that the vertical bending moment is maximum at 25 meter and 80 metre in longitudinal direction of vessel whereas the vertical shear force is maximum at the 50 metre in longitudinal direction. The horizontal shear force and bending moment is almost zero throughout the vessel. Moreover, it has maximum torsion near 75 metre in longitudinal direction of vessel. The diagram obtained in hydrostatic equilibrium case was compared with the stability program diagram (figure no- 77) for shear force and bending moment to verify the mass distribution in shipload. After comparison, it was found that the shear force and bending moment distribution in both (shipload and stability program) the cases were almost similar thus, verifying the mass distribution of shipload. After achieving the hydrostatic equilibrium, hydrodynamic calculations were started.


GL SHIPLOAD 2,1,3,18960,
Copyright (C) 2009 by Germanischer Lloyd

EQUILIBRIUM Loading Condition:2 Assem.Mass: 1 Title: Light Weight

Total Weight [t]: 17009,00
LCG [m]: 60,04
VCG [m]: 9,56
HCG [m]: -0,00
Neutral Axis above BL [m]: 0,00
Shear centre above BL [m]: 0,00
Lpp [m]: 123,91
B [m]: 23,00
H [m]: 11,41
CB: 0,742
Draft (x=AP) [m]: 7,84
Draft (x=FP) [m]: 7,81
Heel Angle (+: SB immerse) [deg]: 0,009

Project : PK116 by A_Kumar (2313)
File : \\sal.internal\hh\homes\supergast04\Desktop\Abhishek\Final calculations\PK116_MD_Rev17.shx
Date : 2017-12-27 19:53
Page : 1/1

Figure 73: Hydrostatic equilibrium case

8.6.3 *Hydrodynamics*

Using the “linear” hull description from hydrostatics, hydrodynamic potentials were computed by Strip method. The strip method is a frequency domain method for computing hydrodynamic potentials, accelerations and in this case also pressure for regular waves by dividing the hull in to cross sections (strips) and thus reducing the three – dimensional computation to a series of two dimensional computations. The strip theory is valid for slender bodies like ships.

Moreover, from the hydrodynamic potentials and the global mass data of the selected mass distribution, hydrodynamic pressures can be computed for arbitrary wave parameter combinations such as wave height, wave length, wave direction, phase angle and ship’s speed.

As discussed above, the strip theory was used to compute hydrodynamic potentials. The strips which were selected to do this computation is shown below:

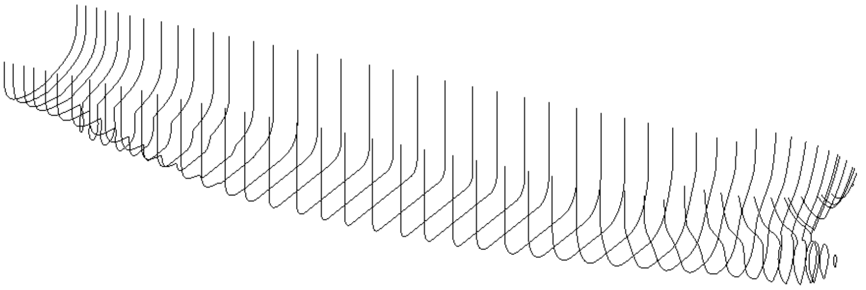


Figure 74: 3D View of the model for computing hydrodynamic potentials

8.6.4 *Selection of load cases*

The definition of loads is one of the most important steps in a global strength analysis of a ship. Several design sea states with different wave heights, lengths and headings was investigated systematically for the application of loads in a realistic way.

Procedure for selection of load case is explained in detail under the chapter 10.

9 STABILITY ANALYSIS

The stability calculation was done using a program called "LOADMASTER": The loadmaster was developed by a firm called "*Kockum Sonics*" dedicated to calculate Intact stability by executing the proper mass distribution for the vessel "PK-116".

9.1 Stability criteria of a vessel

If a rigid body, when subject to a small disturbance from a position of equilibrium, tends to return to that state it is said to possess positive stability or to be in a state of stable equilibrium. If however, following the disturbance, the body remains in its new position, then it is said to be in a state of neutral equilibrium. If following the disturbance the excursion from the equilibrium position tends to increase then the body is said to be in a state of unstable equilibrium or to possess negative stability.

A freely floating undisturbed body in a still liquid is acted upon by two resultant vertical forces, the upward force of buoyancy and the downward force of weight. If the body is in equilibrium, the resultant forces must be equal and opposite and in the same vertical line.

Stability is generally considered to refer to the moment that is brought into action when a ship is inclined by the action of some external force. This moment is caused by the movement of the centre of buoyancy, although, of course, the stability would be changed if the position of the centre of gravity were changed. For the seaworthiness of an undamaged vessel it is sufficient to investigate the stability in the transverse direction.

The parameter which needs to observe while doing the stability calculations is briefed below in the Figure - 75

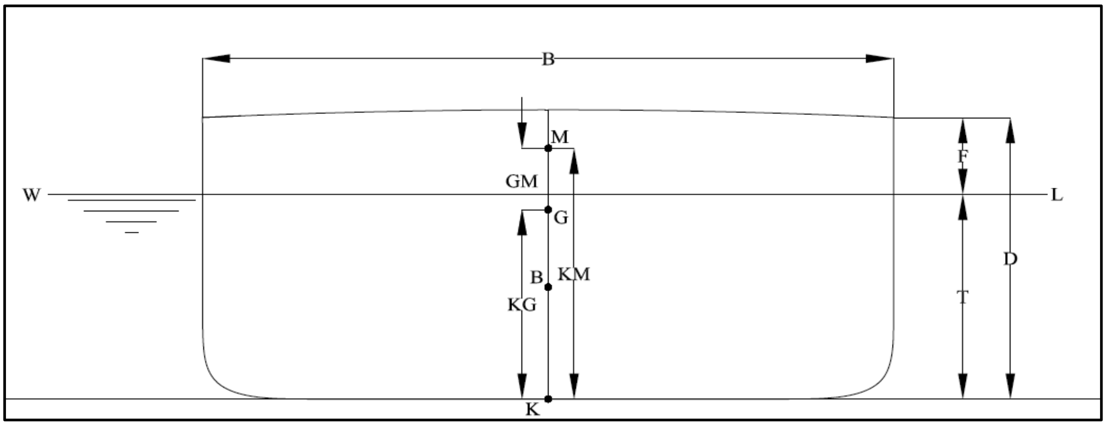


Figure 75: Metacentric height

9.2 Calculations using loadmaster

Loadmaster implements all the masses inside the vessel to calculate stability. The program already includes Lightweight of the ship whereas the Dry cargo, General cargo and the storage tanks masses can be varied according to the Voyage conditions. Furthermore, the calculation was done according to IMO rules using propeller immersion method for summer ocean water case.

In context of the thesis, all masses in the stability program were adjusted according to the mass distribution of shipload such that the draft and GM can be matched. Later on, to cross verify the mass distribution in Shipload, the draft at the fore, aft, heel angle and GM were matched between the two.

A comparison between the Loadmaster and Shipload data is already presented in table –15under the chapter Assembled mass.

Mass distribution in Loadmaster is tabulated below:

Table 16: Loadmaster Mass Distribution

MASS DISTRIBUTION IN STABILITY PROGRAM:					
S No.	**Title**	**Weight (T)**	**LCG(m)**	**TCG(m)**	**VCG(m)**
1.	Light ship	7049.7	59.35	0.91p	10.20
2.	Moving Structures	-318	60.22	0.00	10.95
3.	Input Cargo	4120.5	59.52	0.01p	16.38
4.	Dry stores	635	14.17	0.55s	11.77
5.	Heavy oil	616.8	79.26	1.59s	5.60
6.	Diesel oil	50	8.24	0.00	6.93
7.	Fresh water	171.2	111.80	0.00	5.10
8.	Ballast water	4633.9	64.46	1.04s	2.78
9.	Dead weight	10277.4	60.55	0.61s	9.02
10.	Total	17009.1	60.06	0.01p	9.48

The GZ- Curve from and shear force, bending moment diagram from the stability program is shown below:

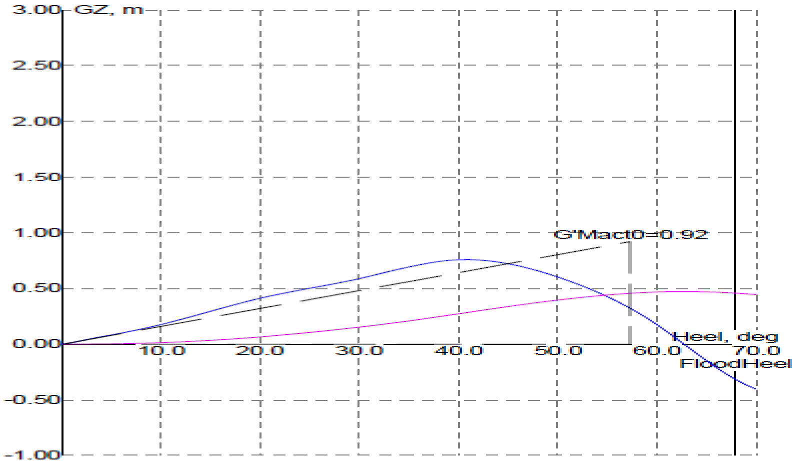


Figure 76: GZ - Curve

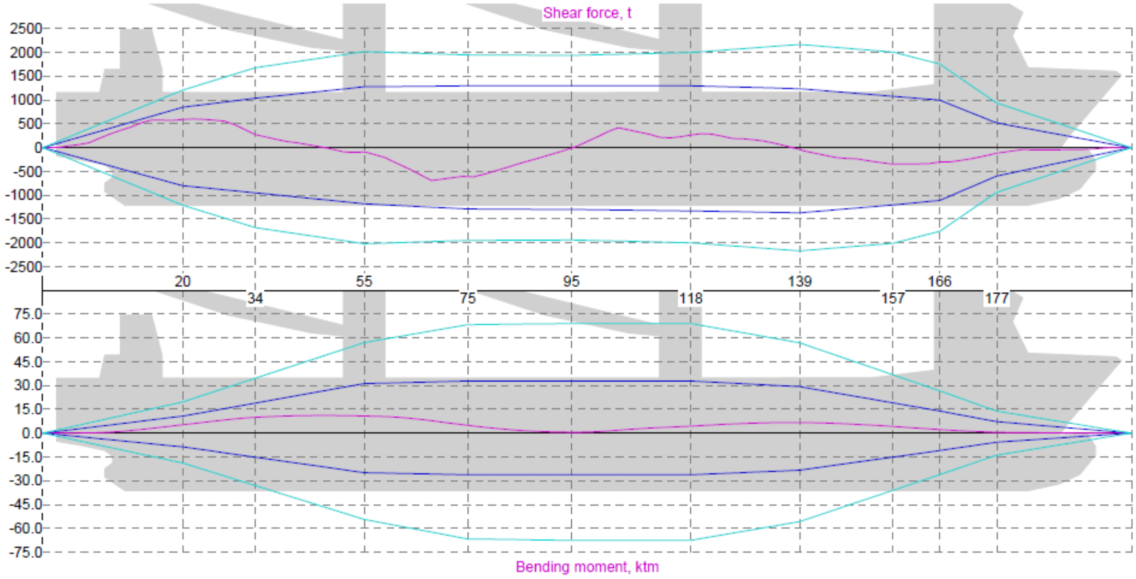


Figure 77 Shear force and bending moment distribution in stability program

10 LOAD CASES

10.1 Input values for Shipload

One of the great strength of Shipload is that the User can principally select the waves required for the strength analysis manually by specifying the height, length, direction and phase angle of the wave and the speed of the ship.

Banking on this great competency of the software, two critical load cases were selected for performing the global strength analysis of the vessel. First one was Beam sea case and the other one was Head sea case. Load cases were obtained by replicating the same loading conditions in a hydrodynamics software *ABB Octopus.* The output data from Octopus in the form of wave amplitude, wave direction and wavelength at stipulated speed of 12, 75 knots in head sea and zero knots in beam sea case became the input load cases for Shipload application.

"*Please note that the work which was done in Octopus is out of the scope of this thesis".*

General wave parameter obtained from Octopus for applying in to the shipload is tabulated below:

Table 17: Wave parameter

Final load case for shipload							
S No.	**Title**	**Wave amplitude (m)**	**WL/Lpp**	**Wave direction (°)**	**Wave phase angle(°)**	**Add.Roll angle**	**V/ Vmax**
1.	Beam sea	9.59	7.25	90	216	0	0
2.	Head sea	5.71	1.24	180	216	0	0.85

The wave specification data shown in the above table was implemented inside the Shipload to generate various load cases in order to do the strength analysis of the vessel. The load cases generated in the first attempt was way far from the target acceleration value of Carousel thus, it was decided to do more iteration with the wave phase angle and wave amplitude (if necessary) in order to achieve the target acceleration value obtained from *ABB OCTOPUS.* After as many as 20 design load cases with 20 different phase angles, somewhat closer value was achieved near 210° degree phase angle in both beam and head sea cases.

Another iteration was done between 210° to 220° phase angle in search of target acceleration value, finally at 216° for both beam and head sea case the target acceleration felt very closer thus, these load case was finally chosen for further evaluation. A comparison between the Acceleration value from Octopus and shipload is tabulated below:

Table 18: Comparison of acceleration for Carousel

COMPARISON OF ACCELERATION:		
Title	**Beamsea(y direction)**	**Headsea(z direction)**
Octopus	2.61	3.09
Shipload	2.65	3.19
Deviation (%)	1.5	3.2

Referring to the table –18, the target acceleration value was almost achieved in Shipload as compare to the *ABB Octopus* for carousel. The acceleration obtained here referred to the carousel **COG (55.84, 0, and 19.83)** as per the stability program (Load master) and shipload mass distribution. The acceleration value has a deviation of 3.2 % foe head sea case and 1.5 % for beam sea case which is acceptable considering the two most critical cases chosen for the analysis.

The acceleration comparison shown in the table -18 vindicates that the whole model with hatch cover and carousel load is working well when the vessel is subjected to various wave loading conditions thus, it allowed to move ahead for global strength analysis by importing the model in to the Poseidon FEA software and observe whether the Stanchion assembly has been able to cope up with the stresses or not. Detailed explanations for stress evaluation has been discussed in the "Result" chapter of this thesis.

A graphical representation showing the acceleration value, bending moment, shear force and torsion along with the pressure distribution on the Hull for both critical cases are shown below:

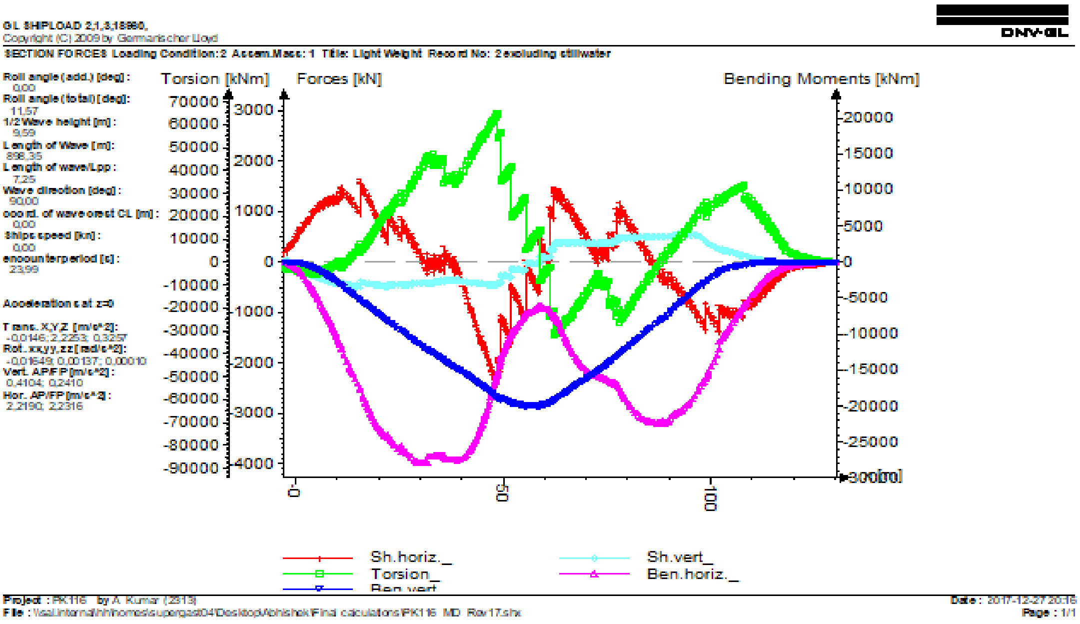


Figure 78: Beam sea load case

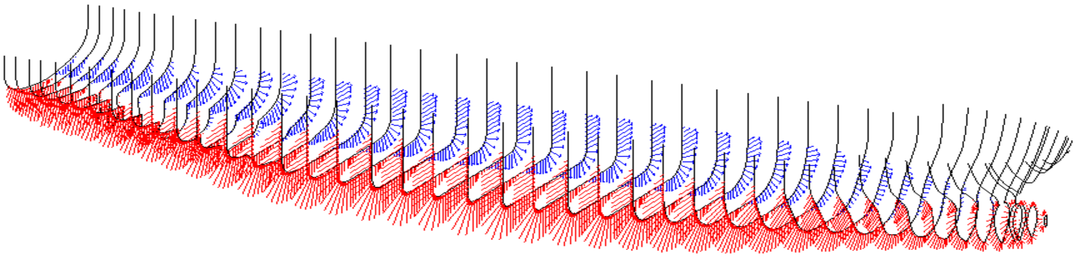


Figure 79: Beam sea case: Pressure distribution along the Hull

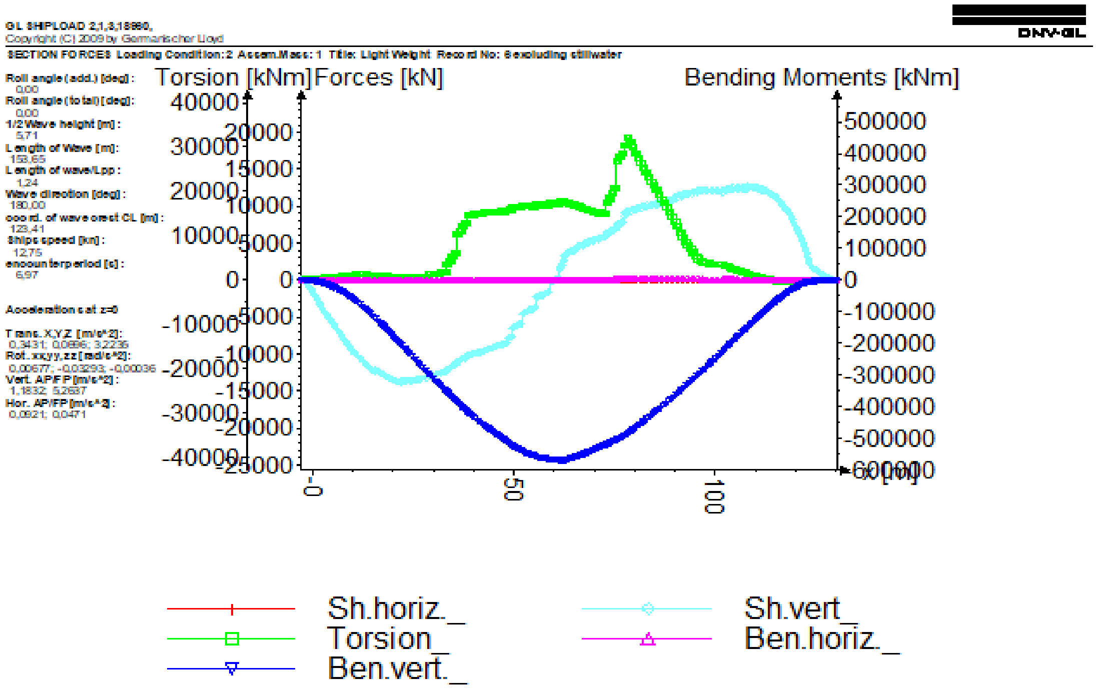


Figure 80: Head sea load case

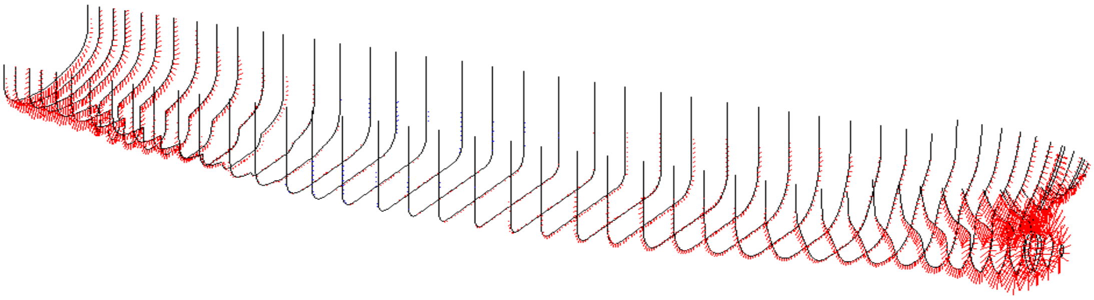


Figure 81: Head sea case: Pressure distribution along the Hull

The bending moment, shear force and torsion diagram of vessel with its pressure distribution for both beam and head sea cases are shown in the figure – 78; 79; 80; 81.

10.1.1 Graph analysis for Beam sea case:

Figure-78 shows that the horizontal bending moment is minimum at the amidship section between 40 to 60 metre in longitudinal direction which lies in the **analysis zone (where hatch cover and stanchion assembly is located),** whereas it has the maximum value at the start and end of the analysis zone. The vertical bending moment is maximum in the same analysis zone. Looking at the horizontal shear force curve, it has maximum value at 45 metre in longitudinal direction again lies at the start of the analysis zone whereas the vertical shear force is more or less constant in the amidship section with minimum value lying in the middle of the analysis zone. The Torsion is maximum at 45 m in longitudinal direction of the ship again at the start of the analysis zone. One notable point from the diagram is that the horizontal shear force and torsion both are maximum at the same location. The water pressure on the hull for Beam sea case is maximum at the starboard side of the vessel throughout in the shell and it should be so because the wave is coming from that direction only.

10.1.2 Graph analysis for Head sea case:

Figure-80 shows that there is almost no shear force and bending moment in horizontal direction whereas the vertical shear force is maximum at 30 m and 110 m distance in longitudinal direction and vertical bending moment is maximum at the amidships section which comes under analysis zone. The torsion is almost constant between 35 m to 70 metre which comes under analysis zone and it has a sudden increase at 77 metre in longitudinal direction of the vessel. The water pressure on the hull for head sea case is maximum at the bow region and it should be so because the wave is coming from that direction only.

10.2 Generation of nodal loads

For all selected waves, hydrodynamic pressures are converted to nodal loads acting on the shell of the FE model. These loads are combined with the unit load groups resulting from (dynamic) mass acceleration and the static load groups (weight and buoyancy) into balanced load cases suitable for the FE analysis.

Hydrodynamic pressures (acting on the hydrodynamic shell representation) are converted to nodal loads (acting on the FE shell representation) in a way that the total forces and moments remain identical. Finally, these nodal loads can be appended to the FE-model by just storing the load groups and load factors in to the BMF format thus, enabling it to do the global strength analysis in the Poseidon FEA software.

10.3 Advantage of using shipload

a) A more elaborate interpolation routine that distributes pressure homogenously to FE nodes is provided with Shipload.
b) Shipload integrates the necessary input such as global FE-model, all necessary input for generation of pressure and inertial loads under one environment to compute global strength analysis.
c) Particular care has been given to an efficient and comfortable input of the mass distribution for several loading conditions. This includes cargo, outfitting and tanks.

d) Due to the use of an efficient and proven hydrodynamic method, the loads for hundreds of different wave situations can be generated in an interactive session.
e) Non-linear correction is applied to obtain realistic pressures also above the still water line.

11 RESULTS

After modelling the whole vessel, hatch covers, carousel assembly and auxiliary system, the wave load cases were applied to know what kind of impact it is evoking on the entire vessel especially, in the amidships area.

A series of FE load groups based on the hydrostatic equilibrium computation were obtained by providing the unit load groups for masses and tanks in each of six degrees of freedom, which was scaled with accelerations based on subsequent hydrodynamic calculations. Then, these load groups were embedded in the FE load cases. The static load groups such as Buoyancy, weight and tanks are generated in addition to the unit load groups.

After application of several wave loading cases in Shipload *as* discussed in chapter- 10, the load factors were generated in the form of acceleration inside shipload. These acceleration were matched with the acceleration value given by a hydrodynamic analysis software ABB Octopus. Later on, these generated load factors / load groups were imported in to the FEA Poseidon for further analysis.

In Poseidon FEA software, the model was transferred and the desired boundary conditions were applied to calculate the reaction forces. **The main aim here was to match the action forces (acting on the carousel) applied on each nodes in Shipload to the reaction forces (from carousel to weather deck and then to the Inner bottom through stanchion assembly) obtained from Finite element analysis.** The comparison for these action and reaction forces would be elaborately discussed in this section ahead.

11.1 Load analysis area

Figure-82 (shown below) represents the cross sectional area located at the amidships section which includes 4 number of hatch covers, stanchion assembly, carousel assembly, Basement etc. This cross-sectional area needs to be investigated properly against its deformation when subjected under the prescribed load cases. Each and every element including beams, trusses, shell were minutely checked with their corresponding axial force, stress, bending moment and torsion.

All the results associated with this area are discussed in the coming subchapters.

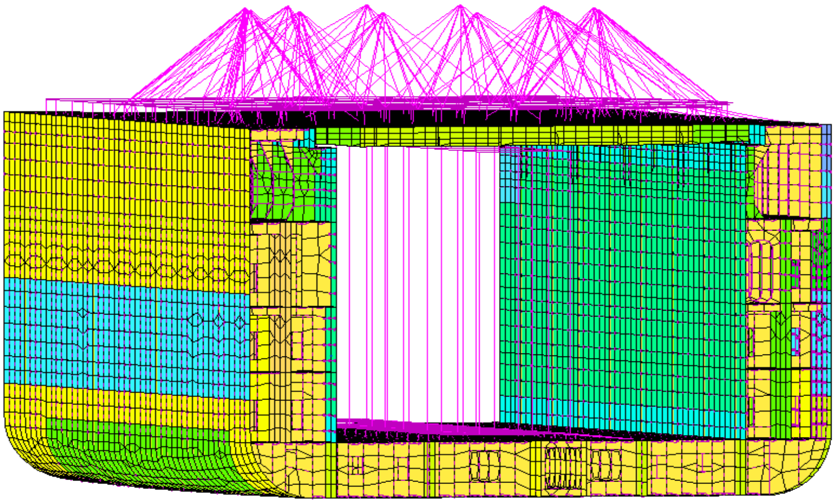


Figure 82: Cross sectional view: Load analysis area

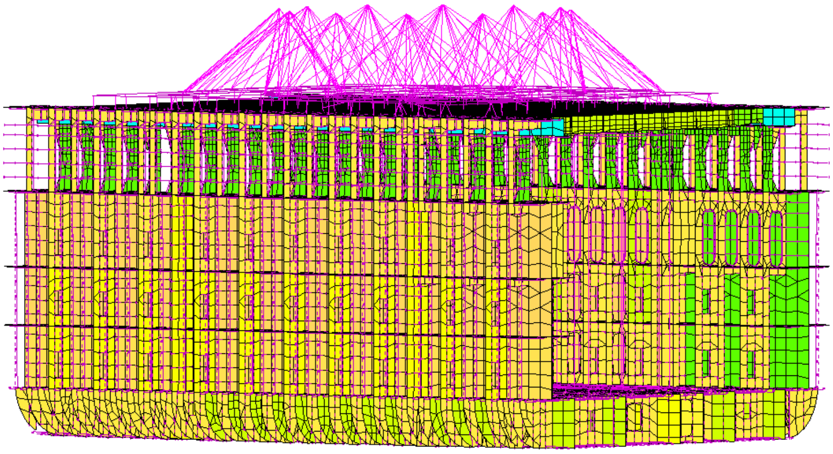


Figure 83: Structural view of load analysis area

11.2 Deflection checking

After importing the model in to FEA Poseidon and application of boundary conditions, the vessel was checked against big deflection by solving the **Static Analysis in FEA software.** The FEA software used its internal program to build matrix, equations and did the condition checking across all the elements and nodes throughout the vessel.

The Final results obtained after static analysis is presented below:

1) **Summary:**

NO. OF NODAL POINTS: 131676

NO. OF BEAM ELEMENTS: 78946

NO. OF PLANE STRESS ELEMENTS: 153588

NO. OF BOUNDARY ELEMENTS: 3

NO. OF P.S.E. MATERIAL: 4

NUMBER OF EQUATIONS: 789915

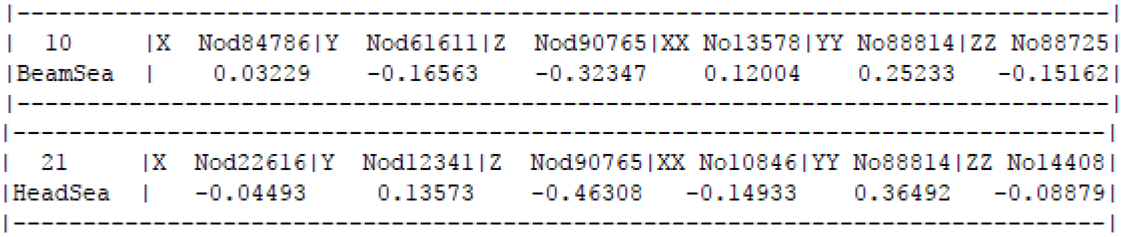
2) **Stiffness matrix diagonals:**

MIN DIA=0.2001E+01

MAX DIA=0.4905E+11

COND.CHECK= 0.1937E+03

3) **Displacement and Rotation in X,Y,Z direction:**



```
|-------------------------------------------------------------------------------|
|  10      |X  Nod84786|Y  Nod61611|Z  Nod90765|XX No13578|YY No88814|ZZ No88725|
|BeamSea   |    0.03229    -0.16563    -0.32347    0.12004     0.25233   -0.15162|
|-------------------------------------------------------------------------------|
|-------------------------------------------------------------------------------|
|  21      |X  Nod22616|Y  Nod12341|Z  Nod90765|XX No10846|YY No88814|ZZ No14408|
|HeadSea   |   -0.04493     0.13573    -0.46308   -0.14933     0.36492   -0.08879|
|-------------------------------------------------------------------------------|
```

As can be seen from the output of static analysis which is also called as global strength analysis of the vessel, the total number of nodes generated by the FEA tool with total number of elements and equations are presented under summary section. An internal programming used to generate Stiffness matrix diagonals is also mentioned above.

The outcome of the Static Analysis is the displacement and rotations values in respective coordinates. The displacement given above is in meter.

The table for displacement and rotations shows the final value but, before reaching to this least possible value of displacement and rotations, lot of checking's were done in terms of deletion of free nodes, deletion of any hanging element like truss, removal of double elements etc. Finally, after insuring all the elements and nodes at proper position the values for displacement and rotation were obtained as shown above.

The notable point from displacement and rotation table is that the maximum displacement is **463 mm** which is occurring at **node no 90765** located near the superstructure and the minimum deflection is **31.8** mm which is occurring at node no. **84786** located also at the fore part of the vessel. The deflection at the other location of vessel such as cargo hold regions were found okay. The rotation value shown in the above table is also reasonable. Therefore, we can say that the all the elements and nodes inside the FE model is correctly oriented and placed at the desired position which allows us to go for further analysis.

Further, after checking the model for deflections, it can be moved ahead to look what is happening with the action and reaction nodal forces on the concerned elements in amidships section and check the overall deformation of the vessel.

11.3 Interaction between Carousel and vessel structure

Here, the carousel assembly with its basement is resting on the hatch cover. The nodal nodal forces resulted from the Shipload wave loading cases were distributed on each of the Sub-COG of carousel basement which is shown in the figure below.

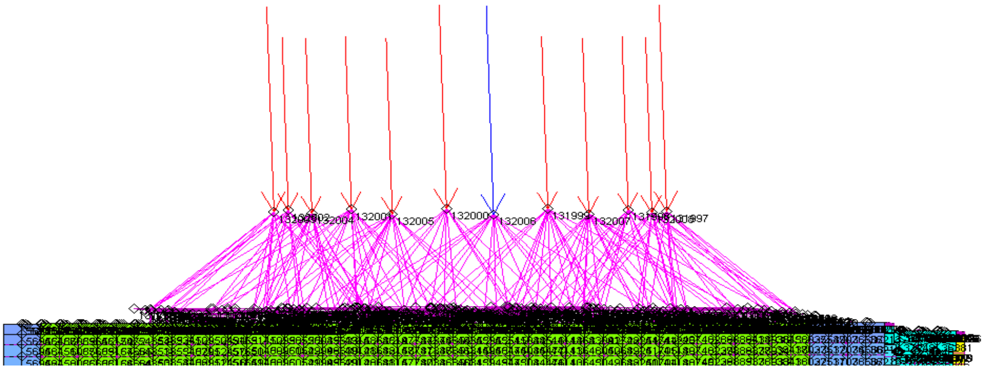


Figure 84: Nodal force distribution on Sub-COG of carousel

In Figure-84, there were a total of **12 nodes** on which these forces were disturbed which otherwise also called as nodal force. These 12 nodes are:

a) Node no. 131997
b) Node no. 131998
c) Node no. 131999

d) Node no. 132000
e) Node no. 132001
f) Node no. 132002
g) Node no. 132003
h) Node no. 132004
i) Node no. 132005
j) Node no. 132006
k) Node no. 132007
l) Node no. 132008

The forces in X, Y and Z direction were distributed on all these nodes. The load group which were distributed automatically by the FEA software are Load group no. **20,65,45,44,59,39,63**. Each node with all these load group had some forces distributed in all three direction. A summary table has been added in ***Annexure-1*** with all the nodal forces applied on each nodes against its load group.

After determining the nodal forces, next step was to evaluate the impact of these nodal forces on the carousel basement or in other words we can say that how much would be the reaction forces coming up from the application of action forces. To make the system stable and keep it at the desired location the action and reaction forces should be equal or with acceptable deviation.

Total nodal forces acting on each of the above nodes against all load groups is shown in table –19, for individual nodal loading refer ***Annexure – 1.*** As can be seen from table no -19 the maximum force is occurring in the Z direction and it is of compressive nature also, which means that, most of the nodal load which is acting on the sub-COG of carousel is transferring in to the bottom structure of vessel, which would certainly help in justify the purpose of this thesis.

Table 19: Total nodal force

Total nodal Force including all load groups		
S No.	**Direction**	**Force (kN)**
1.	Px	58322.6
2.	Py	102376.7
3.	Pz	-1803231

Table 20: Load Factor from Shipload

LOAD FACTOR (Global Load Case):				
S No.	**Load group**	**Load Title**	**Beam sea**	**Head sea**
1.	20	C001 X	1	1
2.	65	C001 Y	0,01293	-0,34357
3.	45	C001 Z	-2,28765	-0,09271
4.	44	C001 XX	-0,38608	-1,17608
5.	59	C001 YY	0,0173	-0,00683
6.	39	C001 ZZ	-0,0012	0,03298
7.	63	SHR 1 2	-0,00015	0,00036

11.4 Action Forces (Forces acting on carousel)

The action forces were calculated one by one on each of the aforementioned nodes against all its load group in Shipload and later on it was imported in to the FEA software for strength analysis.

The total action forces computed in each of the Beam and Head sea cases are presented in table – 21.

Table 21: Action Force:

Total Action Force:						
Title	**Beam sea(Load factor)*Px**	**Beam sea(Load factor)*Py**	**Beam sea(Load factor)*Pz**	**Head sea(Load factor)*Px**	**Head sea(Load factor)*Py**	**Head sea(Load factor)*Pz**
Force (kN)	-32	-7726	-28898	905	156	-36412

11.5 Reaction Forces (From Carousel)

The reaction forces are summarized by taking the axial loads applied on trusses. The trusses were used to support the carousel. The reaction forces were computed against all the action forces acting on the sub-COG of carousel basement.

In table-21, since, the action forces are negligible for Beam sea*Px and Head sea*Py case, so, it was not considered while calculating the reaction forces. Other cases are summarized below in the table-22, whereas individual forces on each truss element of hatch cover and basement are given in ***Annexure-1***.

Table 22: Reaction Force (from carousel)

Total Reaction Force:				
Title	**Beam sea in Y direction**	**Beam sea in Z direction**	**Head sea in X direct**	**Head sea in Z direction**
Force (Kn)	7626	-30592	817	-37257

The reaction forces in Y direction for Beam sea case were obtained by summing up the total loads for hatch cover and Basement in Y direction and for Z direction it was calculated by summing up the loads on Basement in Z direction. Similar approach were used to calculate the reaction force for Head sea in X and Z direction by summing up the respective loads of hatch cover and Basement as summarized in the table-22. A detail explanation of individual forces is attached in ***Annexure - 1.***

The reason behind summing up the carousel and hatch cover reaction forces in x and y directions and only carousel forces in z direction for head and beam sea case was because, in beam sea case the wave has impact in transverse direction and in head sea case the wave impact would be in longitudinal direction so, all the forces (both hatch cover and carousel) which are acting along X and Y direction was summed up while for Z direction only the carousel force was added.

11.5.1 Comparison of Action and Reaction Forces

Comparison of action and reaction forces from table – 21 and 22 with its deviation is explained in the table-23. A detail explanation of this comparison is attached in ***Annexure-1.***

Table 23: Comparison table

COMPARISON:				
Title	**Beam sea in Y direction**	**Beam sea in Z direction**	**Head sea in X direc**	**Head sea Z direction**
Action Force (Kn)	-7726	-28898	905	-36412
Reaction Force (Kn)	7626	-30592	817	-37257
Deviation (%)	1.30	5.86	10.78	2.32

In table-23, we can see that there is a maximum deviation of **10.78 %** for Head sea case in X direction, although the difference percentage is little bit higher but, considering the total difference of loads i.e. around **100 KN or 10 tons** is relatively small when compared to other direction whereas, the other deviation is less than **6%**, which is acceptable.

The reason for these deviation could be because, nearly all of the supports do not remains in parallel to the vessel origin once it is under axial loading, there is a chance that it might get deflected with some angle from its original position resulting in to higher load summation. These deviation should be brought as minimum as possible to reach a perfect balancing conditions.

11.6 Global strength analysis of vessel and checking of stanchion assembly feasibility

Finally, after modelling the whole vessel with its longitudinal and transversal members along with holes and cut-outs in Poseidon, and load cases application in Shipload, the model was run through the FEA analysis to check its global strength and stanchion (pillar) assembly feasibility under the heavy cargo masses placed on the hatch cover.

In purview of this thesis, the load analysis zone is only the **"Amidships section" in** cargo hold region between **Frame number 61 to 110 (approx. 32 meter area)** in longitudinal direction of the vessel. So, all the strength analysis would be concentrated in this area only. Since, the stanchion assembly with hatch cover and carousel is located in this area therefore, all the respective checking was done in this zone only.

All strength checks of vessel and stanchion assembly were performed for both Beam and Head sea cases in various steps as described below:

a) Allocation of high values of forces of truss element for basement in x, y, z direction.
b) Checking of all the truss element of Basement in Z direction with regards to tension loads.
c) Checking of maximum allowable load that can be carried by Hatch cover supporting system in vertical and horizontal direction.
d) Strength verification of all the Beams involved in Stanchion assembly.
e) Checking of loads on each of the truss element of stanchion assembly.
f) Allocation of high stress area in analysis zone.

11.6.1 Allocation of high values of forces for basement in x, y and z direction

Z direction:

The axial force value was checked for trusses of carousel basement in z direction for both beam and head sea case and it was found as **-16 tons** for **element no.21** in head sea case as shown in figure-85. The element was located between the carousel basement and hatch cover no -8 at 52.13 m from aft of the vessel at port side.

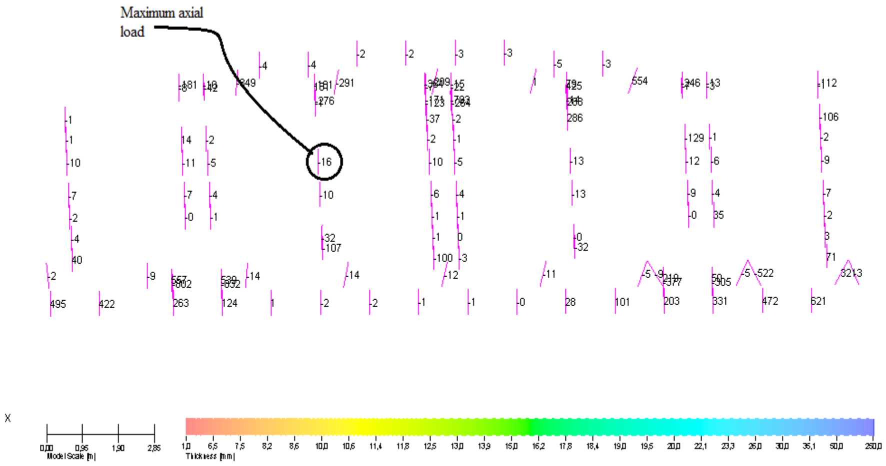


Figure 85: Maximum axial load on carousel basement.in Z direction [Head sea]

Y direction:

Similar to z direction, the maximum axial force was also checked in y direction of the carousel basement in both beam and head sea case. A table presenting the axial force in both beam and head sea direction is given below:

Table 24: Axial load of basement Y

Basement Y:		
Element no.	**Beam sea load (kN)**	**Head sea load (kN)**
1	1110.2	195.4
2	356.1	-195.3
3	690.8	314.6
4	949.5	608.9
5	3264.3	859,6
6	342.4	-1745.6

From table-24, it can be seen that the element no 5 in case of beam sea, have a very high tension load compare to other element of basement Y, this element is located between carousel basement and Deck A at 47.45 m from aft of the ship near hatch cover no- 7 at port side. In head sea case also, the element no- 6 has high compressive load located between carousel basement and Deck – A at 63.52 m from aft of the ship near hatch cover no – 9 at port side. The reason behind the high forces in Beam sea case would be elaborately discussed in the Deformation (Chapter -12) section of the vessel

X direction:

The axial forces in X direction of the basement would also be checked in both beam and head sea cases.

Table 25: Axial load of basement X

Basement X:		
Element no.	**Beam sea load (kN)**	**Head sea load (kN)**
1	232.1	127.6
2	-259.4	-581
3	-887.4	-1287.3
4	946.2	726.1

The maximum compressive load found was -1287.3 KN for element no – 3 in case of head sea. This element was located at 58.8 m from the aft of the ship at centre of carousel basement on hatch cover no. – 8 at port side.

11.6.2 Checking of truss element for tension loads

The truss element in Z direction of Basement was thoroughly checked for tension loads, as much as possible tension loads should be avoided due to the fact that the basement is just resting on Hatch cover/vessel and it is not welded to the deck therefore, tension load cannot be allowed in any case. Further, when the truss element was checked, it was found that out of 129 truss elements, 25 truss elements were under axial load with a maximum load of around 45 tons in 3 cases, around 15 tons in 4 cases and rest of the case was under 5 tons as tabulated in ***Annexure-1.*** Therefore, it was decided to make the area **"zero"** for those truss element such that it would not take part in the calculation. The process was executed and found that the other truss element with designated area took more load (compression) and managed the Basement well on top of hatch cover.

11.6.3 Checking of maximum allowable load for hatch cover supporting system

The maximum allowable load for hatch cover supporting system was obtained from the hatch cover operation manual: ref: [8], which is written below:

1) Maximum allowable value in Z direction -1425 kN
2) Maximum allowable value in Y direction -2328 kN
3) Maximum allowable value in X direction -931 kN

By taking these values as the maximum limit, the Hatch cover supporting systems were checked inside the model and it was found that, in Beam sea case all the loads are well below these three limiting values, but in Head sea case, there were three elements which had crossed the above limit.

The elements which were crossed the above limits are element no. **5 and 7** for Hatch cover supporting system in Z direction. Out of these element, element no. 5 was located between hatch cover 9 and coaming deck at port side as aft bearing and element no. 7 was located between hatch cover 10 and coaming deck at port side again as aft bearing. The other element was **no- 26** located between hatch cover 9 and coaming deck in X- direction at port side with a maximum load of **1016 kN.** In Y direction, none of the element had loading above their maximum allowable limit.

.Therefore, it can be said that these three element were found overloaded among all hatch cover supporting systems. Later on, these elements would be checked for any overloading or buckling in the coming section by visualizing in the diagram.

11.6.4 Strength verification of I-beams of stanchion assembly

The strength check was performed for all the I-Beams used inside the stanchion assembly by referring GL rule. As per *the GL rule: ref [3]* - the permissible stress allowed is **0.8*R_{eh}** where **R_{eh}** is the maximum yield stress (N/mm^2). The steel used here is **HT36** with the **yield stress of 355 N/mm^2**. Therefore, the maximum allowable stress would **be 0.8*355 = 284 N/mm^2.**

Now, the next task was to go through all the beam elements of stanchion assembly to check whether the stress values are less than 284 N/mm^2 or not. After, thorough checking of all the Beam elements in both beam and head sea case, it was found that none of the Beams in either stanchion or carousel assembly had crossed this permissible stress value. **Maximum stress value found was 264 N/mm^2 in stanchion assembly for element no.63 in Head sea case.**

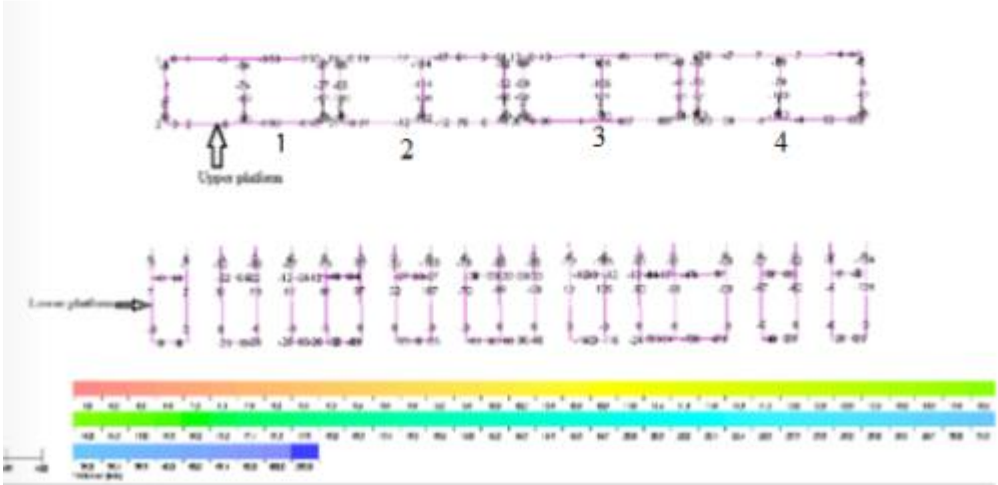


Figure 86: Stress plot of stanchion assembly beams in head sea case [Head sea]

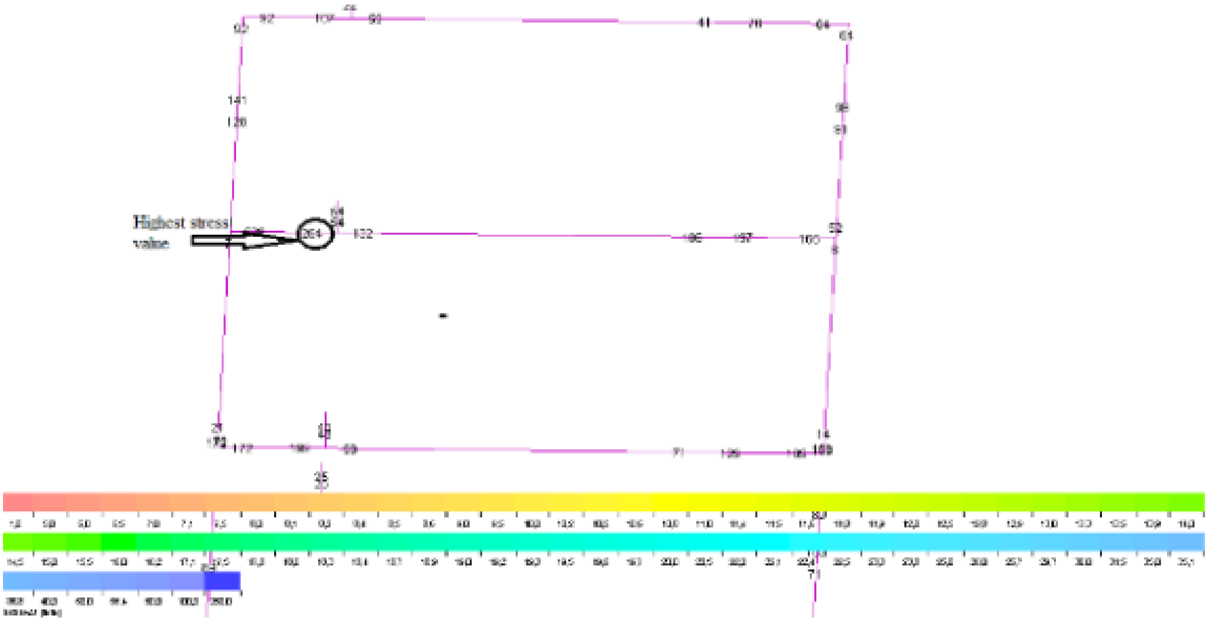


Figure 87: Highest stress allocation on upper platform no -3 in head sea case

Figure –86 shows the stress distribution of stanchion assembly beams at its upper and lower platform with numbering allotted as 1,2,3,4 for upper platforms. Figure –87 shows the highest stressed beams of 264 N/mm^2 (although under critical stress value) on upper platform no- 3.

11.6.5 Checking of loads for stanchion truss

All the stanchion truss was checked for its maximum loading value in head and beam sea case. The maximum compressive load found on stanchion truss was 13 tons on stanchion truss no -5 and 78 in head sea case. Truss no- 5 was located under hatch cover no -7 and truss 78 was located under hatch cover no-9 between the stanchion assembly platforms.

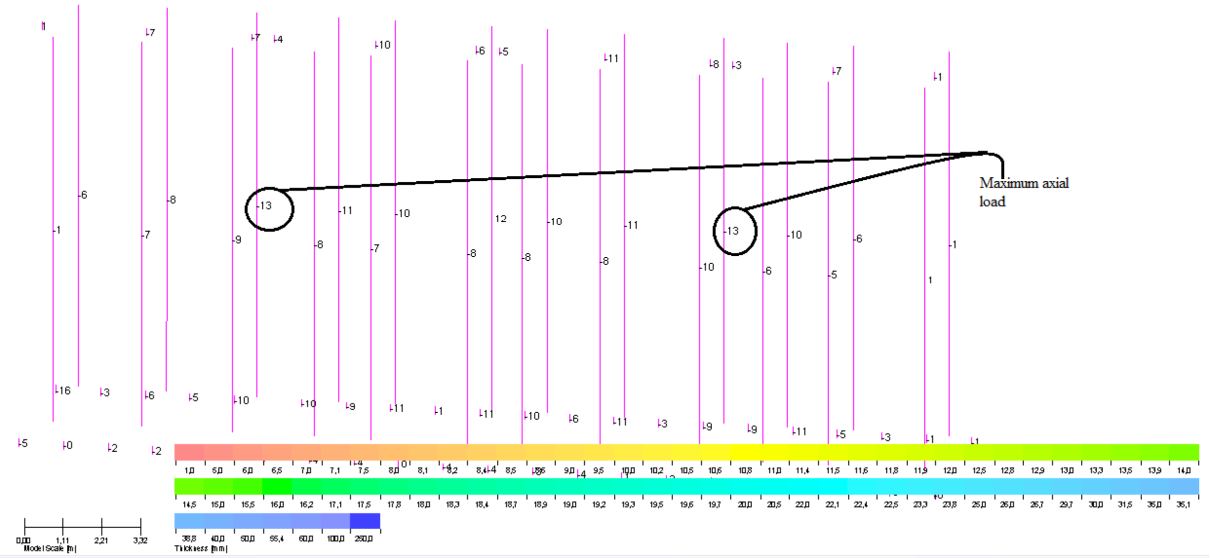


Figure 88: Location of maximum axial load in stanchion truss [Head sea]

Further, the stanchion trusses were checked for its buckling in the “Design of stanchion truss” section (chapter no- 14) of this report.

11.6.6 Allocation of high stress area in analysis zone

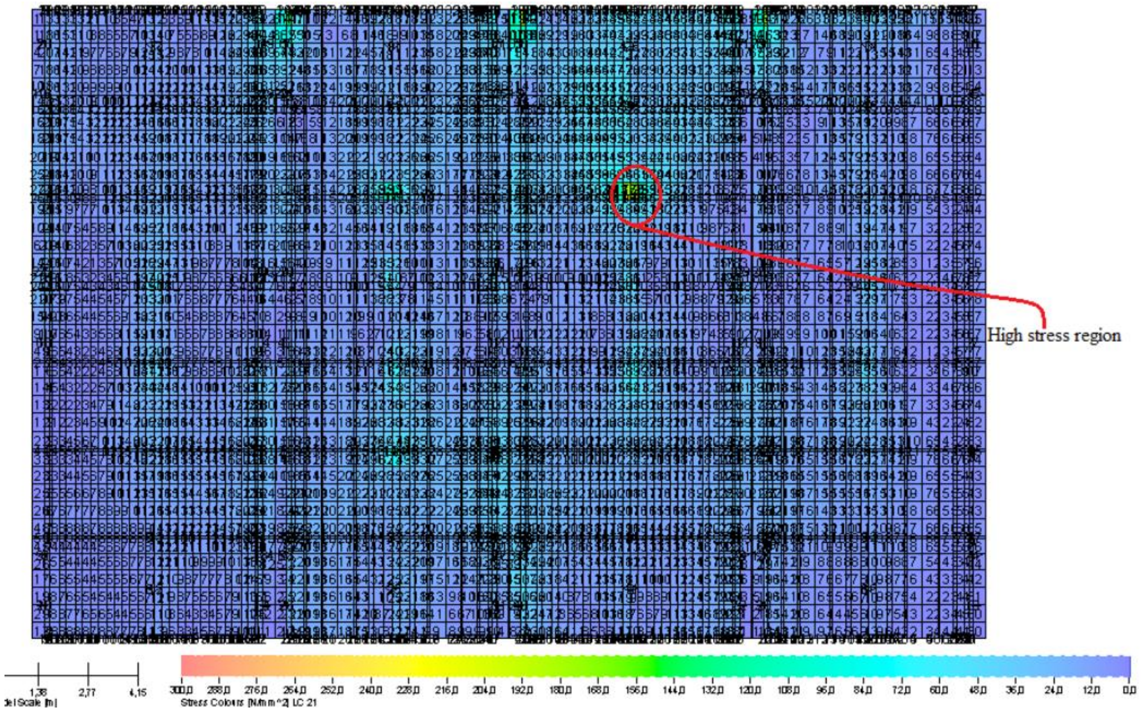


Figure 89: High stress area on hatch cover plate in head sea case

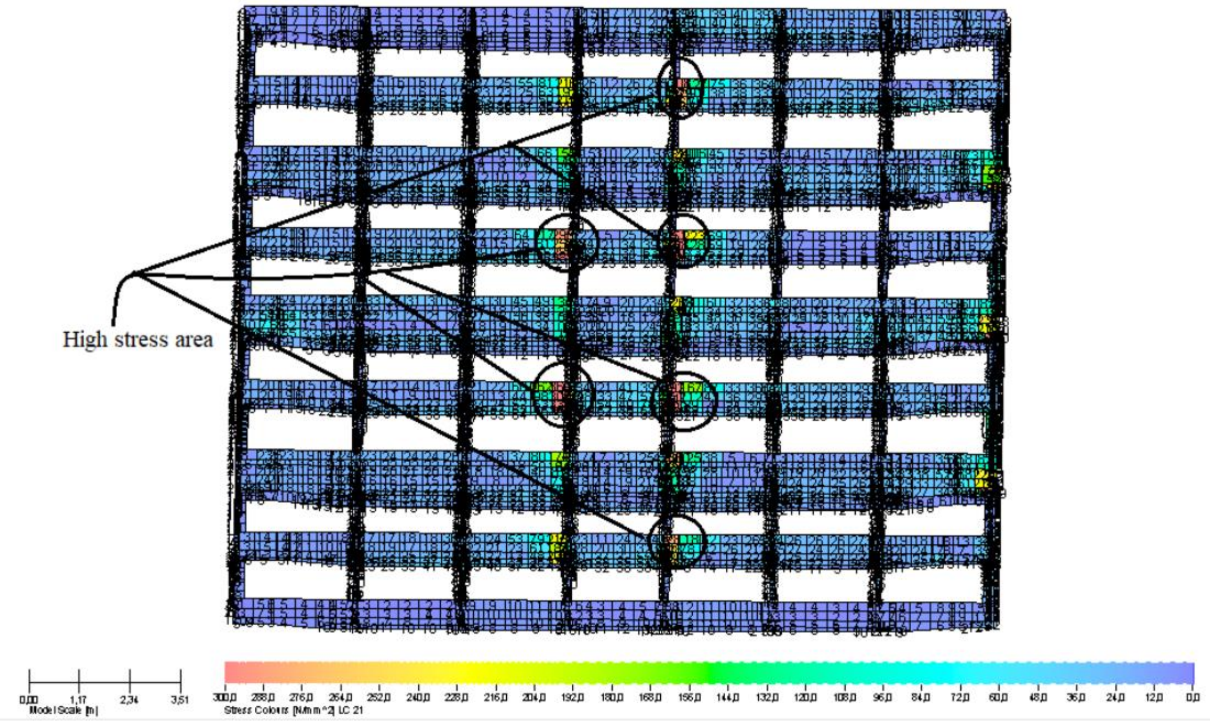


Figure 90: High stress area on hatch cover girder in head sea case

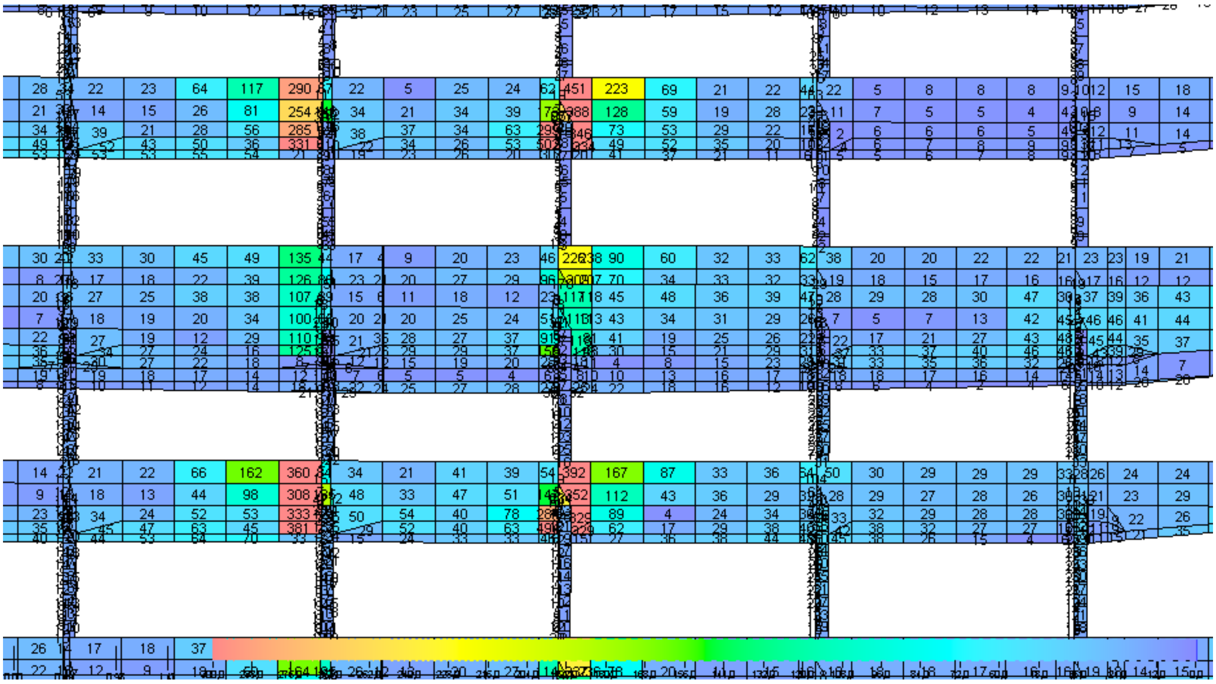


Figure 91: Zoomed view of the high stress area in head sea case

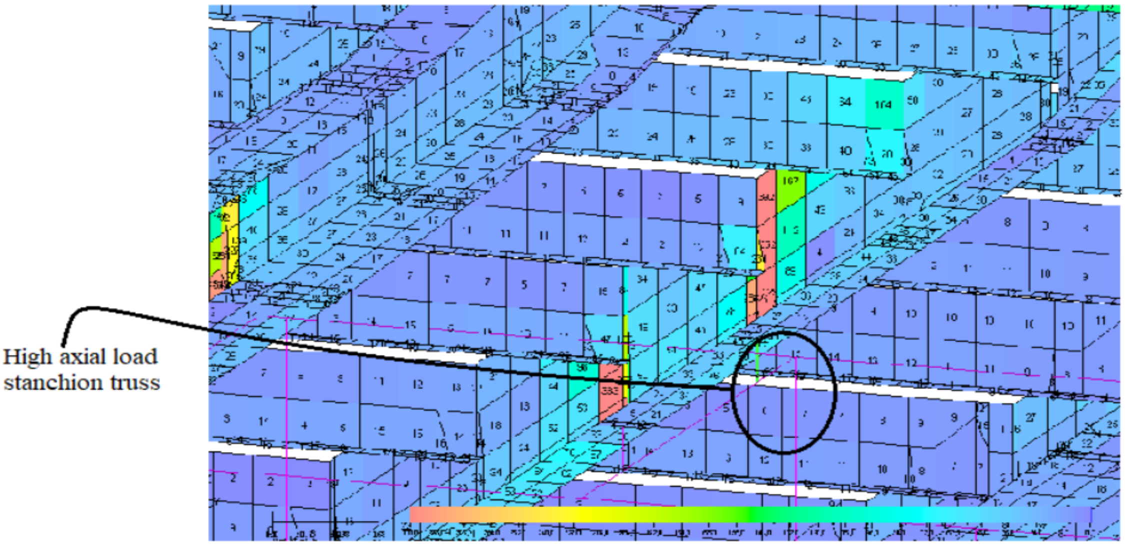


Figure 92: Sectional view: stanchion truss with maximum axial load in head sea case

Figure-89, 90, 91, and 92 shows all 4 hatch covers with allocation of high stress area. The stress value was checked in both beam and head sea case, but head sea case found more critical and vulnerable to stress. Therefore, head sea case was chosen to allocate all high stress area in the analysis zone between frame no- 40 to 130. Here, only the hatch cover and stanchion was shown because these places has higher stress value than other part of vessel. The high stress area of the other element of model would be checked under yield stress checking section of this report. Also, figure-92 shows the cross section view of the exact location of high stress area in hatch cover web plate with the stanchion truss.

The area which is blue in colour are well under the permissible yield stress and with red is overstressed area. Since, most of the area is covered in blue, so, we can say that the stress values are highly acceptable.

Figure no.- 93,94,95 and 96 shows the truss element for various section which were discussed above for high compression force values, but after looking at the specific element it was found that these elements in spite of 10 tons more load on it, can withstand successfully without any major buckling. Therefore, we can say that these trusses are theoretically withstanding the given compressive loads and upholding it successfully.

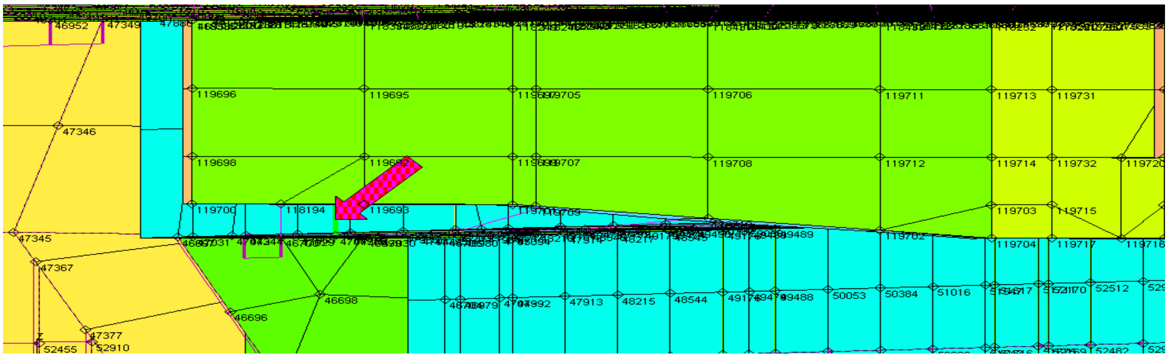


Figure 93: Pink Arrow: Truss Element no. – 5 between hatch cover – 9 and coaming deck at port side in z direction

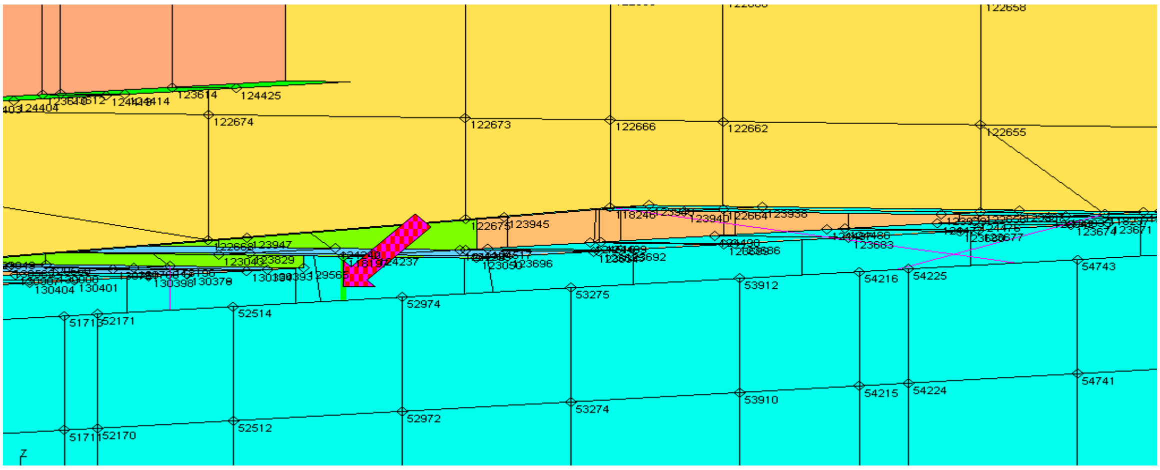


Figure 94: Pink Arrow: Truss Element no. – 7 between hatch cover – 10 and coaming deck at port side in Z direction

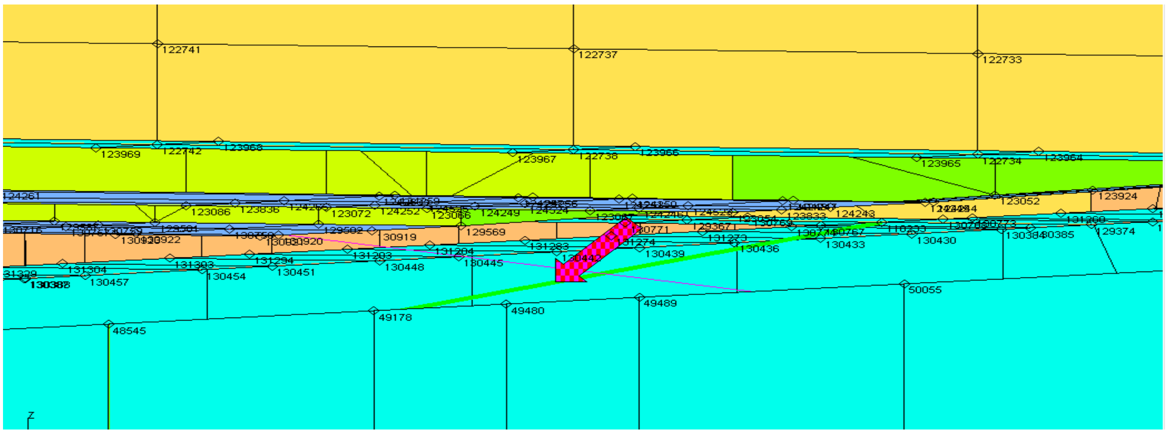


Figure 95: Pink Arrow: Truss Element no. – 26 between hatch cover 9 and coaming deck at port side in x direction.

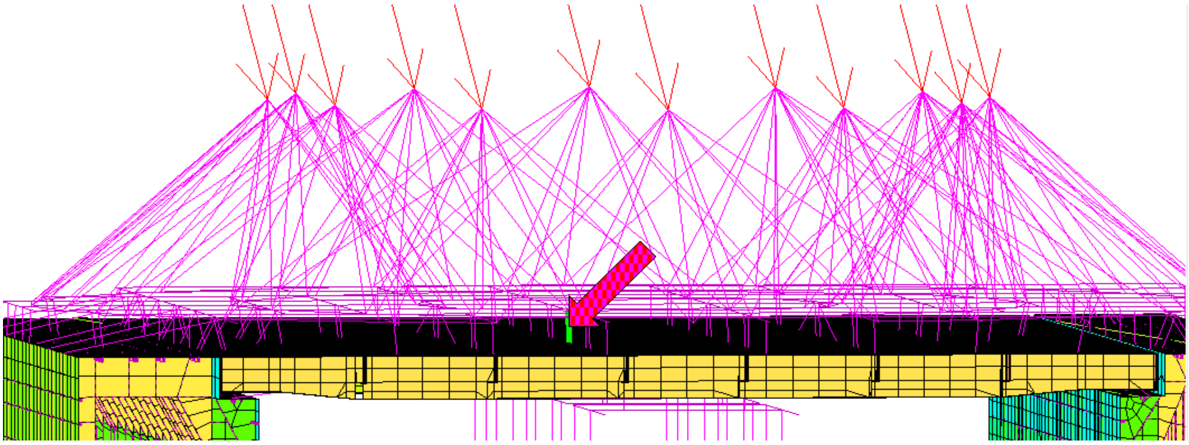


Figure 96: Pink Arrow: Truss Element no. – 21 between hatch cover 8 and basement at port side in Z direction

12 DEFORMATION

After applying both the Beam and Head Sea load case on to the model, the results were obtained and the global deformation of the vessel was captured to present in the coming diagram.

Head sea load case provides vertical bending moment which would cause sagging in the model that means it will have compression at upper deck and tension at the lower portion(below neutral axis). Similarly, the beam sea case would provide horizontal bending moment that means tension in the upper deck region and compression in the lower portion of the ship.

Some of the deformation diagram for the major functional element is shown below:

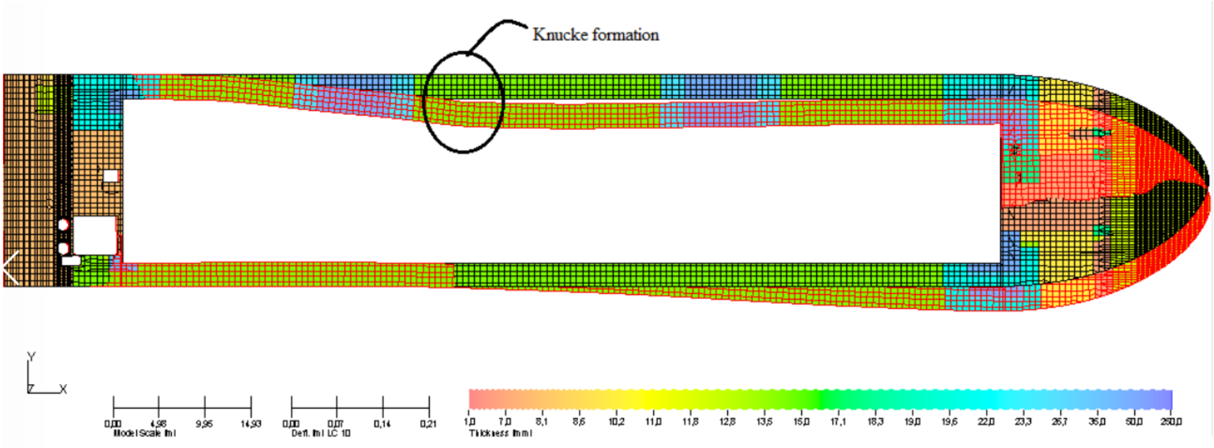


Figure 97: Deformation of DK-A [Beam sea case]

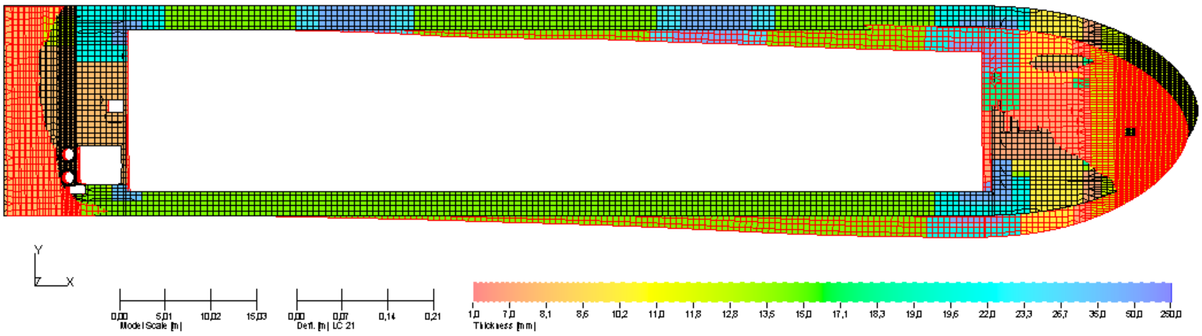


Figure 98: Deformation of DK-A [Head sea case]

.

Figure-97 represents deformation of DK-A, we can see that the deformation is higher at the port side of vessel in Beam sea case. Because of this high deformation it induces a high load in that area as presented in the table –24. The deformation is higher in the aft area compare to the fore portion, the reason behind it, would be unequal distribution of loads for Basement Y trusses and higher loads at aft portion because of the presence of aft stopper in the form of bulkhead. Because of this watertight bulkhead, DK-A at aft becomes more stiff compare to the fore portion also creates knuckle in that region. Another reason could be the presence of crane foundation in that area which makes the Deck stiffer thus, load distribution at port and starboard side becomes unequal.

Figure-98 shows the deformation in head sea case for DK-A. When deformation was measured on model case inside the software, it was found as 50 mm.

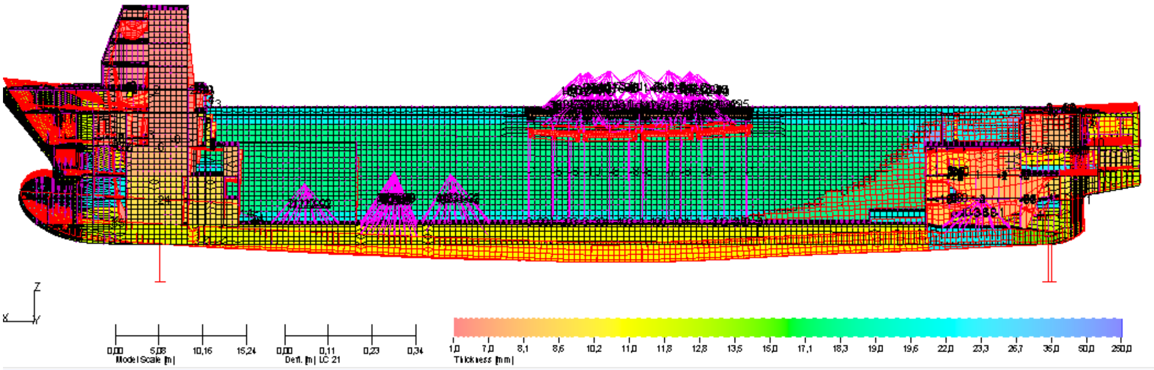


Figure 99: Global Deformation of model [Head sea case]

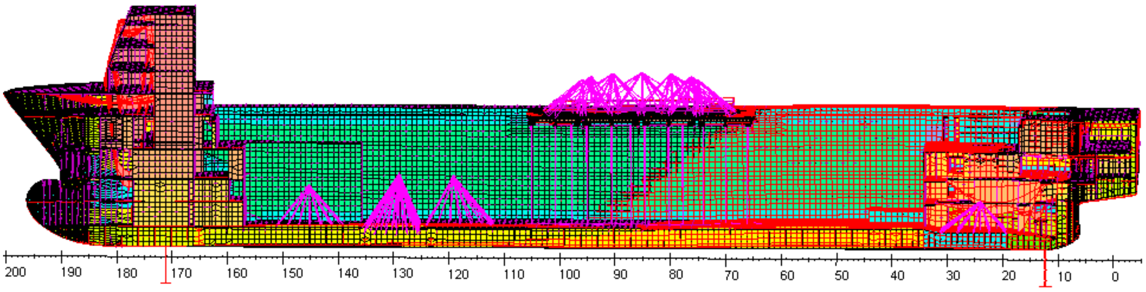


Figure 100: Global Deformation of model [Beam sea case]

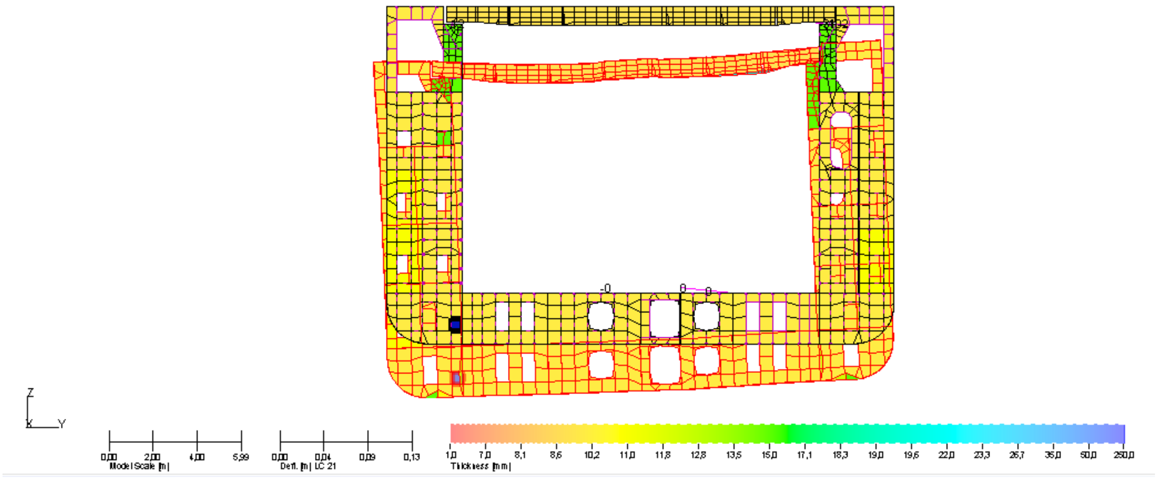


Figure 101: Deformation of Frame section no – 88 with hatch cover, looking from aft [Head sea]

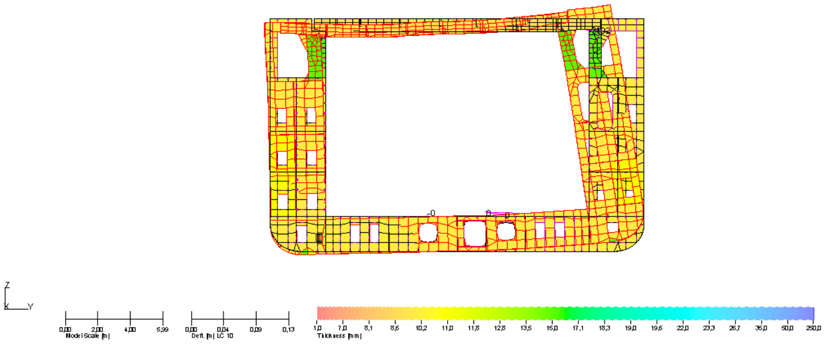


Figure 102: Deformation of Frame section no – 88 with hatch cover, looking from aft [Beam Sea]

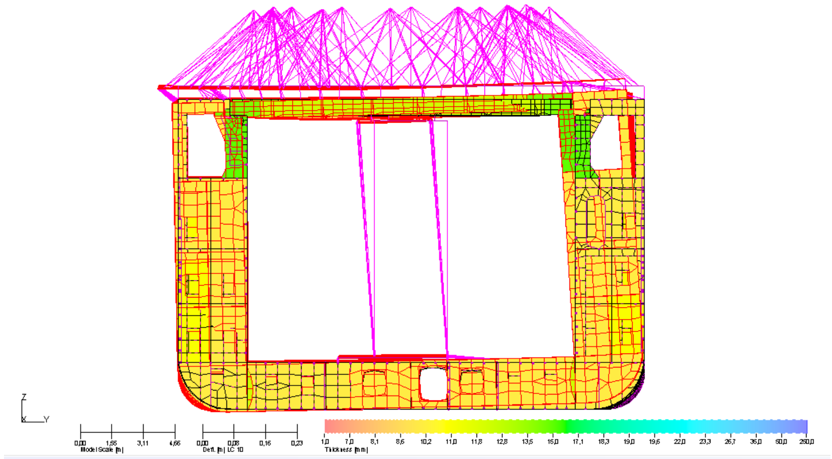


Figure 103: Sectional view: Deformation of amidship section for Beam sea load case, looking from aft

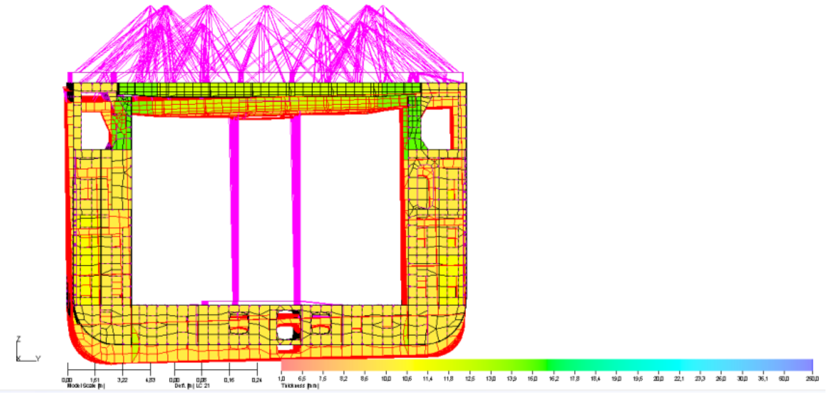


Figure 104: Sectional view: Deformation of amidship section for Head sea load case, looking from aft.

Figure-103 and 104 represents the deformation in amidship section of the vessel with hatch cover and carousel assembly. The relative displacement between hatch cover and side shell coaming in beam sea case was approximately 40 mm as per measurement by model scale in the software.

13 YIELD STRESS CHECK OF MODEL

Stress distribution checks were performed throughout all the functional elements in longitudinal and transverse direction for beam and head sea case.by taking the area between frame no – 40 and 130.. A margin of 20-25 frames were taken on either side of the analysis zone which covers stanchion assembly and hatch covers as shown in the coming figure. In this section, only those cases (either beam or head sea or both) were considered for respective functional element which possess higher stress value.

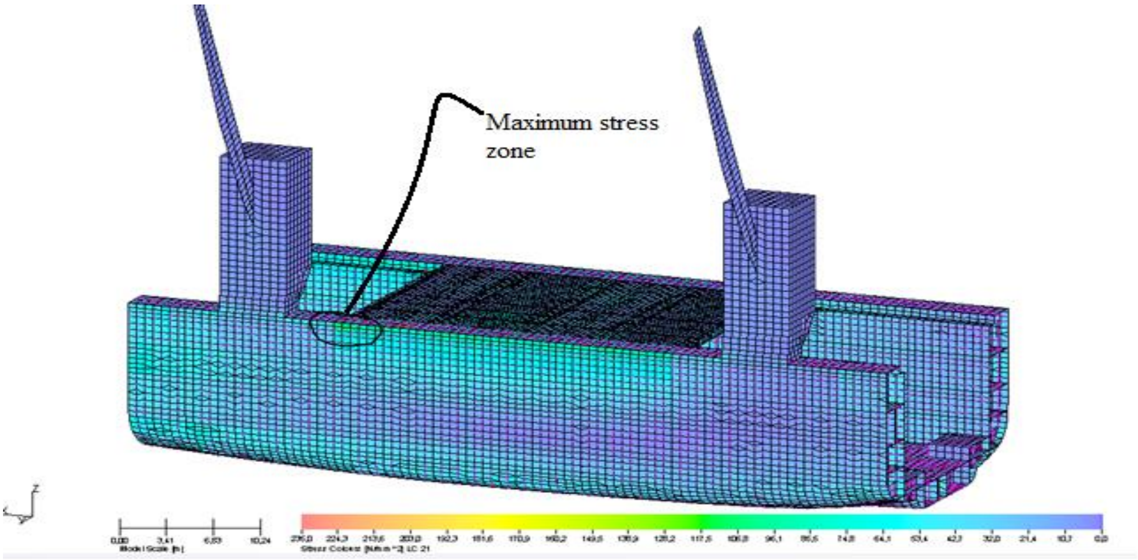


Figure 105: Stress distribution of shell at Port side [Head sea]

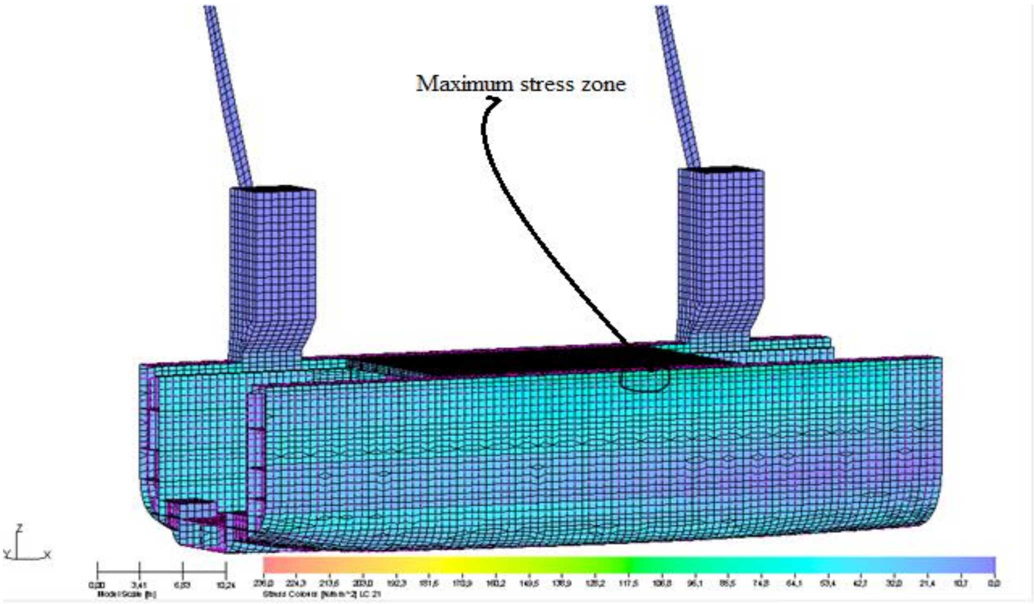


Figure 106: Stress distribution of shell at Starboard side [Head sea]

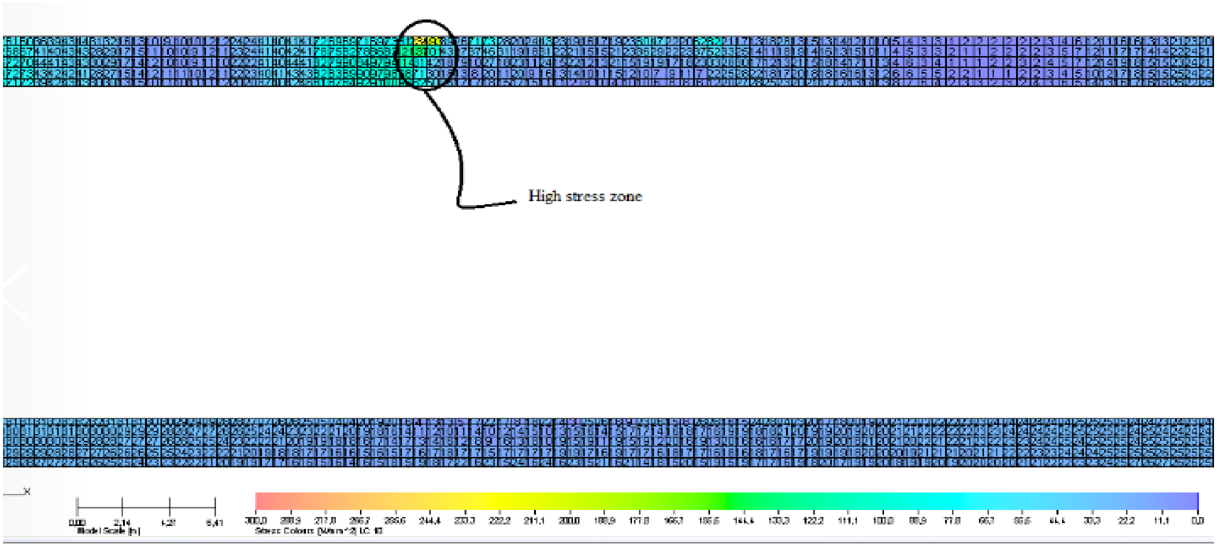


Figure 107: Stress distribution of DK-A [Beam sea case]

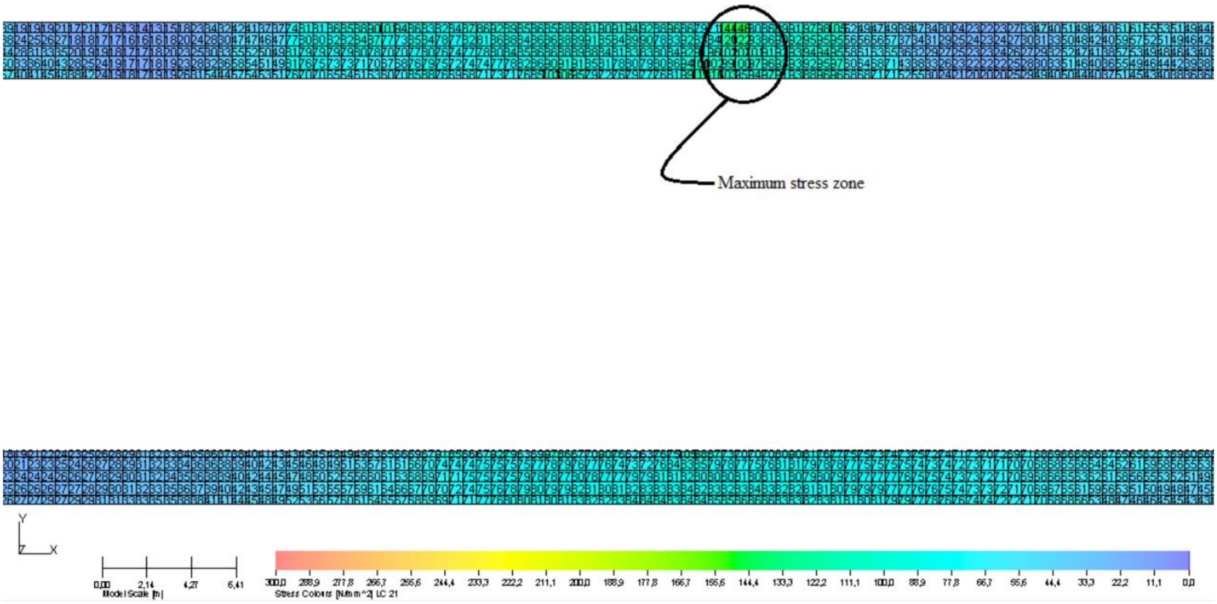


Figure 108: Stress distribution of DK-A [Head sea case]

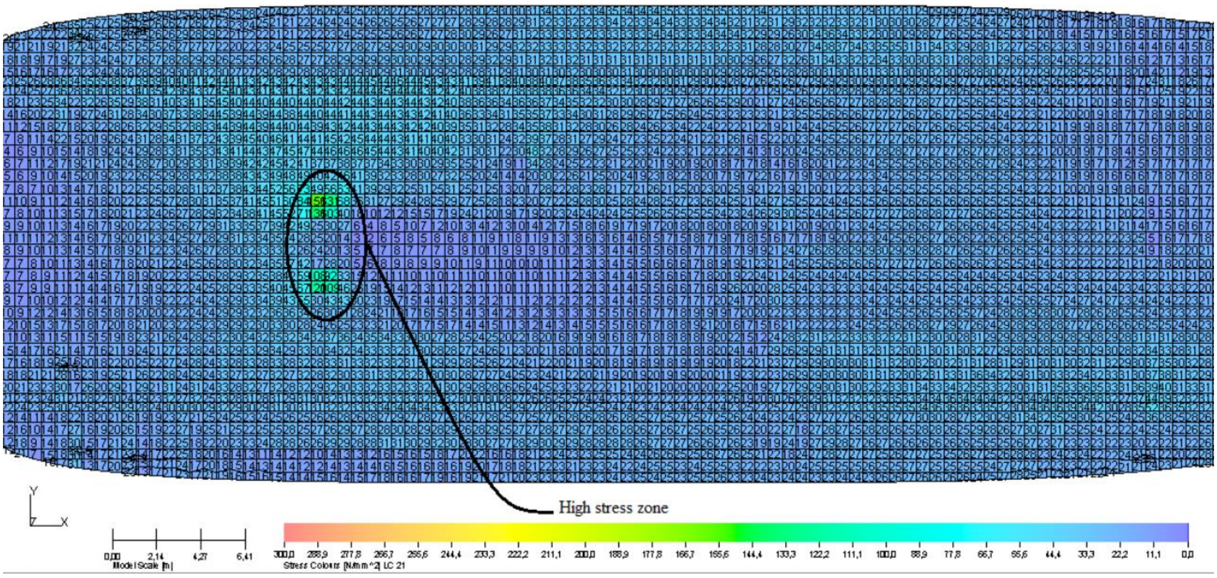


Figure 109: Stress distribution of Inner Bottom [Head sea case]

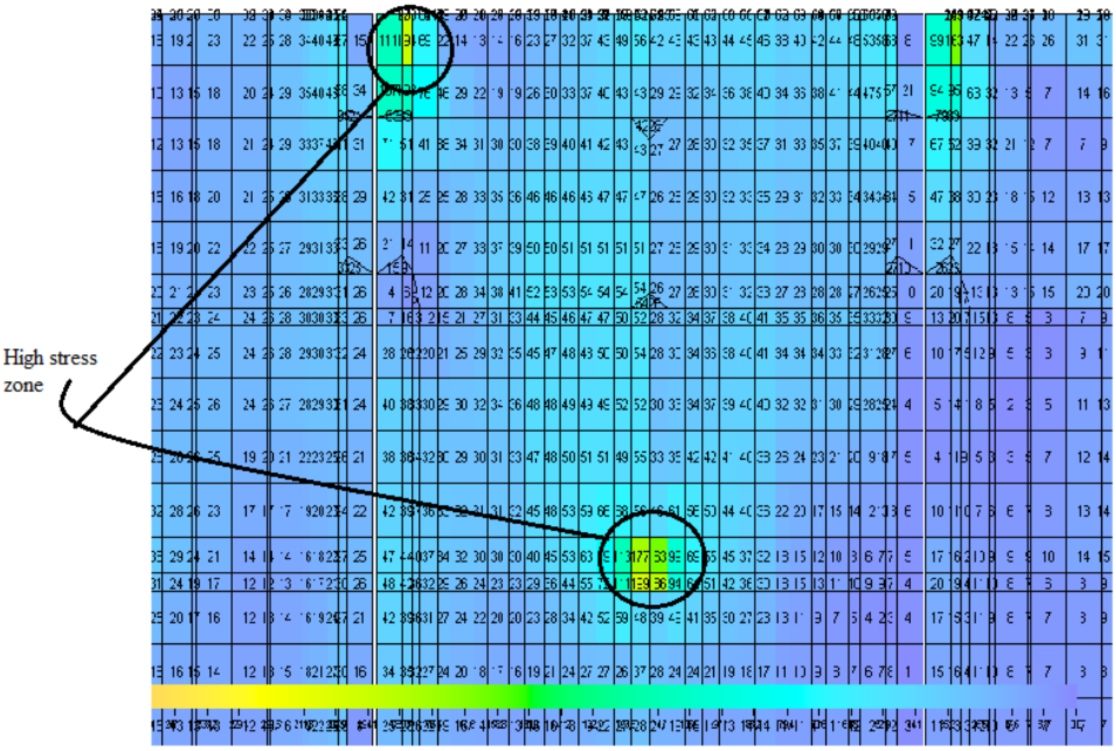


Figure 110: Stress distribution on hatch cover plate [Head sea case]

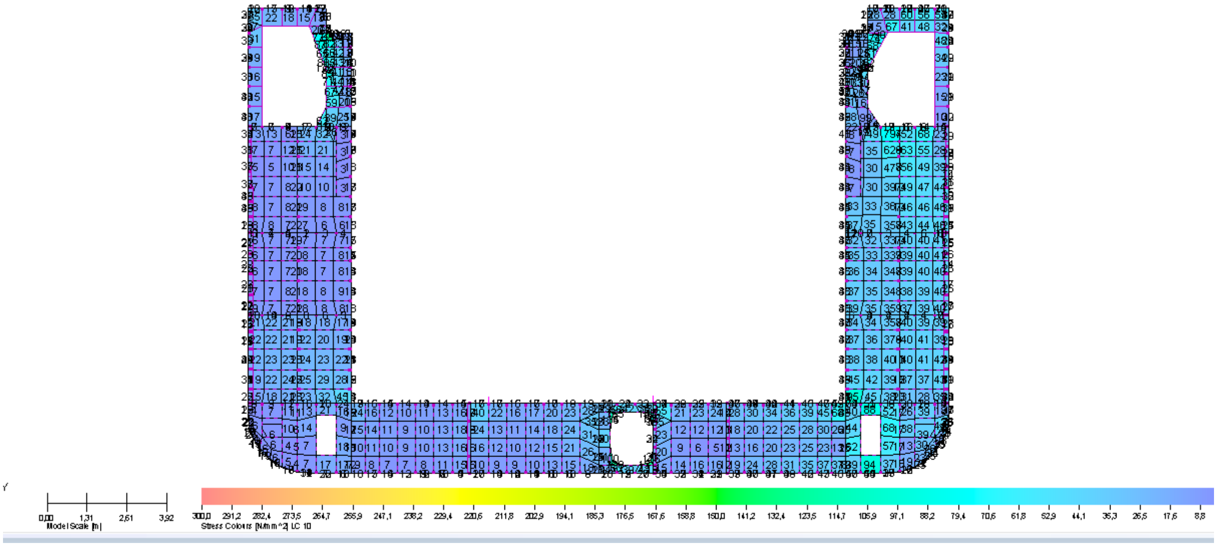


Figure 111: Stress distribution transverse frame no. 76 [Beam sea case]

Figure –105,106,107,108,109 and 110 shows the stress distribution of various functional element of the model. The stress checks were done throughout the model between frame no -40 to 130 and various high stress zones were identified. Except DK-A beam sea case and hatch cover plate head sea case, all other stress in the above diagram was under critical yield stress value of 188 N/mm^2.

In DK-A beam sea case, the value was gone up to 234 N/mm^2 and in hatch cover plate head sea case the value of stress was 194 and 199 N/mm^2 at two places. The reason behind the overstress on hatch cover plate would be, due to the restrictions possessed by the Carousel Basement –X direction trusses. and on DK-A could be the high stopper forces of carousel basement in Y direction.

Also, figure-111 shows the stress distribution of a transverse frame which is located in the analysis zone of hatch cover and stanchion assembly. Here, all the stress values are well within the critical yield stress.

14 BUCKLING ANALYSIS:

14.1 General introduction

After doing the strength analysis of vessel, buckling analysis was done especially in the cargo hold region. Since, all the above analysis was done between Frame no-61 to 110 where, the heavy cargo loading is applied on the hatch cover, it was decided to go for the buckling analysis with 30 % more length on either side of analysis zone i.e. between frame no -40 to frame no-130 (same length were taken for yield stress checks) to observe the effect in a broader and better way.
The program used for the buckling analysis was Poseidon same as used for modelling and FE analysis. Section 11 of the Poseidon software named as "**FE-analysis buckling"** were used for this purpose. The program section is intended for a systematic buckling assessment of plates and stiffeners of longitudinal and transverse members as per the GL rules. In context of the thesis, only Plate buckling would be evaluated and its results will be discussed in the coming subchapter.
The only constraint to have a successful buckling analysis is that the Poseidon modelling and the generated finite element model should have the same name convention and its element geometry.

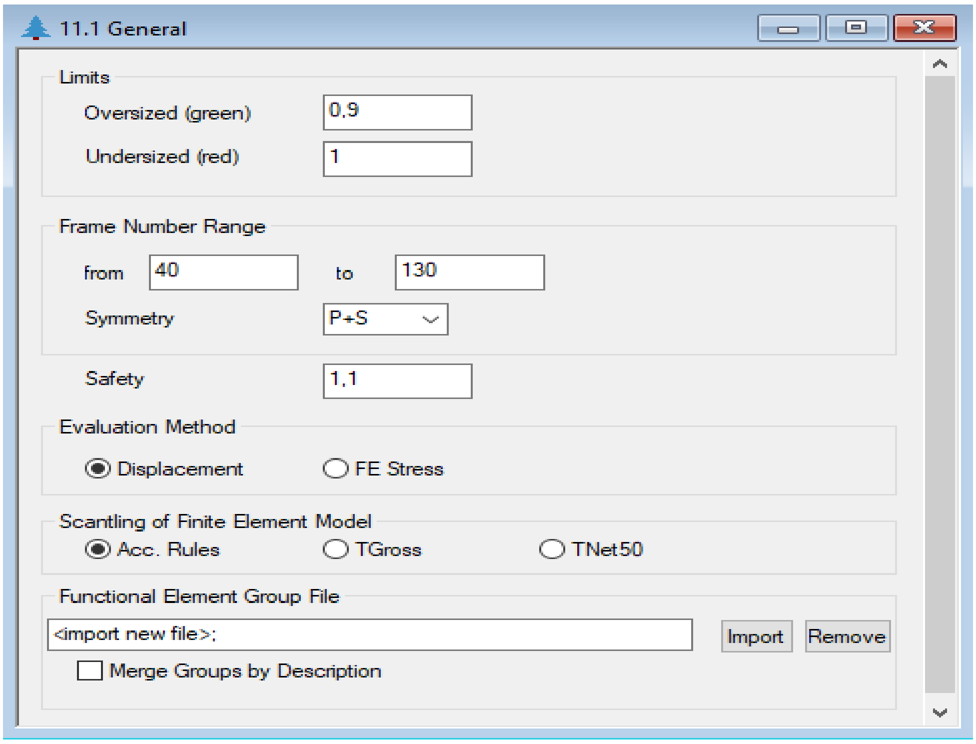
One very important point to be considered during buckling analysis is that, the systematic buckling analysis is not applicable for buckling fields in way of opening without edge reinforcement. To make it happen, a subsequent post process is required and this is out of scope of this thesis. Although, this is a major concern while doing the buckling analysis of an Oil tanker which has big openings.

14.2 Steps for buckling analysis

a) Import of file in BMF format
b) General Data
c) Overview
d) Load definition
e) Pressure Load groups
f) Result analysis.

14.2.1 File import

The buckling analysis was performed in the main modelling file of Poseidon (.pox). First of all, the BMF file which was generated with all the load cases from Shipload was imported in to the main modelling file under section ***1.1 General data***. After, importing successfully, next step was to go directly to the section 11- FE- Analysis Buckling. Under section 11, some general data have to be filled as shown in the Figure below:



11.1 General

Limits

Oversized (green) 0,9

Undersized (red) 1

Frame Number Range

from 40 to 130

Symmetry P+S

Safety 1,1

Evaluation Method

● Displacement ○ FE Stress

Scantling of Finite Element Model

● Acc. Rules ○ TGross ○ TNet50

Functional Element Group File

<import new file>;

☐ Merge Groups by Description

Figure 112: General Data for Buckling Analysis

14.2.2 General Data

As shown in the Figure-100, some general parameters has to be set before proceeding to the buckling analysis.

a) **Limits** – Here, oversized and undersized limits can be specified as per the GL rule. The limit sets here would be used to calculate and evaluate the usage factor obtained for different element cases. For e.g.- if the usage factors are

  - Blue in colour, it means fields and stiffeners is less than the oversized value.
  - Green in colour, it means fields and stiffeners are between oversized and undersized value.
  - Red in colour, it means fields and stiffeners are greater than the undersized value.

b) **Frame number range** – Here the frame number would be defined for beginning and end of the buckling assessment region.

c) **Symmetry** – Buckling assessment side can be defined here by picking port or starboard or both.

d) **Safety-** Factor of safety to be defined as per classification rules for global strength analysis, ref [3] in this case 1.1 was chosen as a general case.

e) **Evaluation method** - There were two evaluation methods present here, first one was Displacement method and the other one was FE-stress method. In this case, Displacement method was chosen because this method is intended for buckling assessment of cargo hold models (finite element size <= buckling field size).

f) **Functional element group file** – An external file was loaded here for grouping of functional elements to one new functional element name, if this option is used then the new names were available for buckling assessment. Since, in our case, BMF file is already imported with the designated element name so, the field was left vacant.

14.2.3 Overview

This program section shows the computed buckling field geometry for all longitudinal and transverse members.

14.2.3.1 Longitudinal memebers

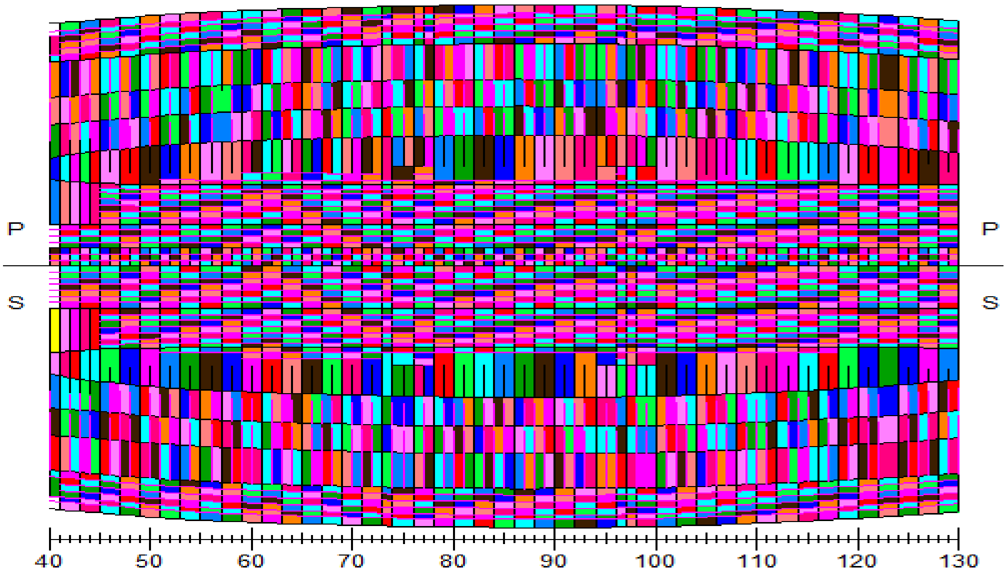


Figure 113: Buckling field geometry of Shell

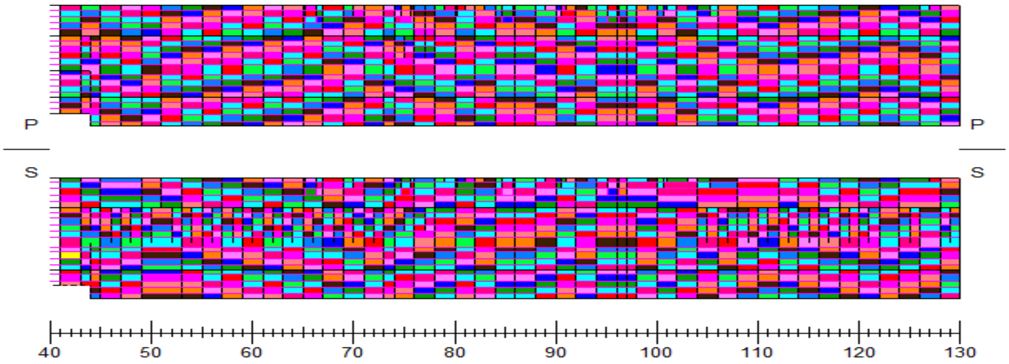


Figure 114: Buckling field geometry of Longitudinal Bulkhead no-14

Figure-113 and 114 shows an example of buckling field geometry distribution of a shell and longitudinal bulkhead – 14. Using this field geometry, the buckling analysis were performed by taking the load cases and pressure distribution from the BMF file.

14.2.3.2 Transversal members

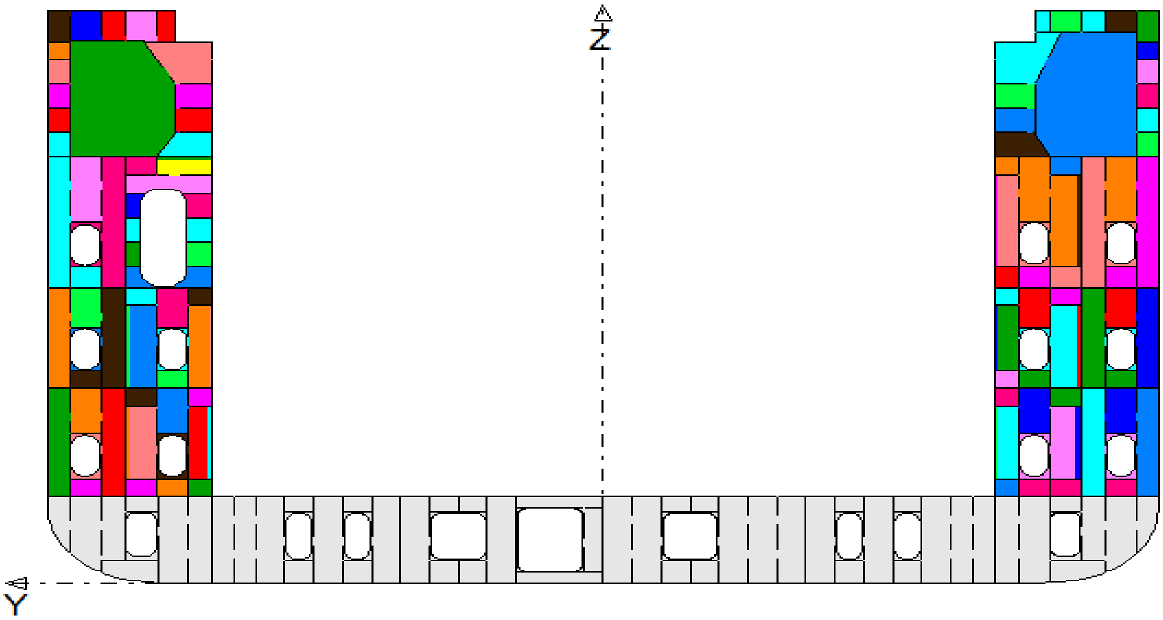


Figure 115: Buckling field geometry of web plate for Transversal frame -73

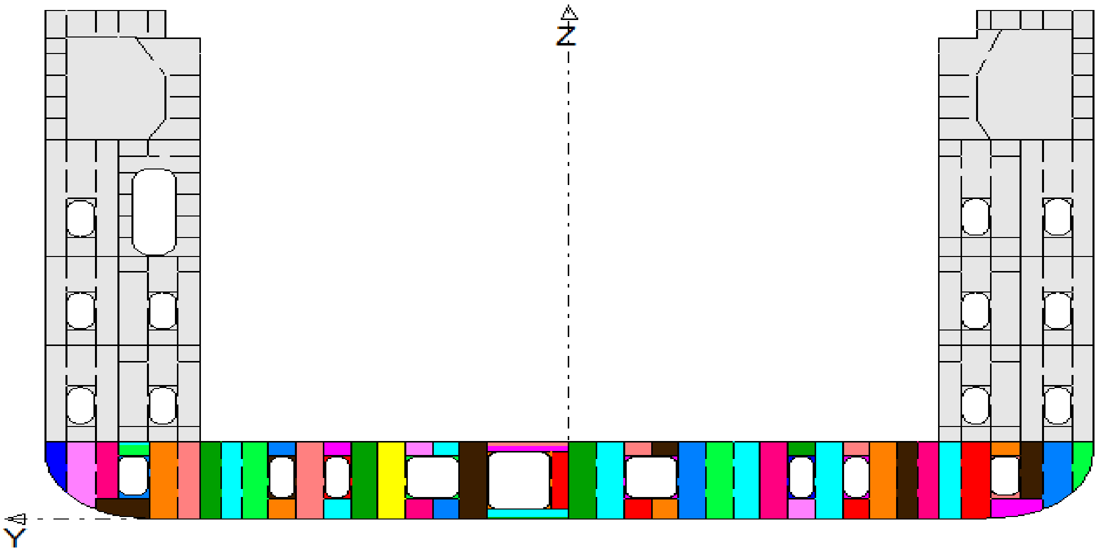
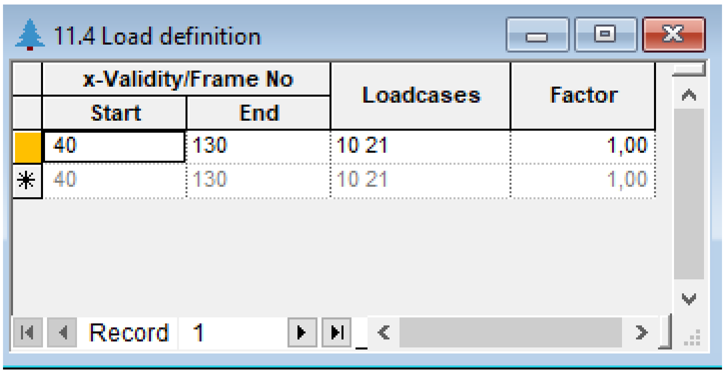


Figure 116: Buckling field geometry of floor plate for Transversal frame-73

Figure – 115 and 116 shows the transversal member with its web and floor plate buckling field geometry, the same load definition and pressure distribution were applied here as of longitudinal members.

14.2.4 Load definition

Under load definition, once again the validity region for buckling assessment were defined by prescribing the beginning and end frame number. Frame no – **40** and **130** were put with load cases as **10 and 21** as per the BMF file load case description for Beam and Head sea case. Also, a factor equal to **1** were applied for scaling FE results for the given X- validity range as shown in the figure-117.

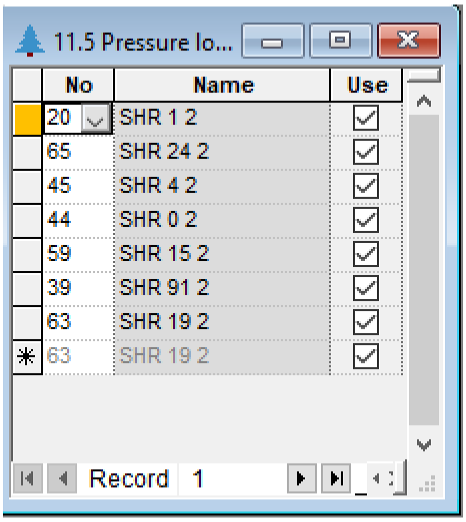


	x-Validity/Frame No		Loadcases	Factor
	Start	End		
	40	130	10 21	1,00
✳	40	130	10 21	1,00

Figure 117Load definition

14.2.5 Pressure load groups

The pressure load group is the same load group which were obtained after doing the hydrostatic and hydrodynamic analysis in shipload as given in table-19. Once again these load groups were included for doing the buckling analysis as shown in the figure-102.



	No	Name	Use
	20	SHR 1 2	☑
	65	SHR 24 2	☑
	45	SHR 4 2	☑
	44	SHR 0 2	☑
	59	SHR 15 2	☑
	39	SHR 91 2	☑
	63	SHR 19 2	☑
✳	63	SHR 19 2	☑

Figure 118: Pressure Load group

14.2.6 Buckling analysis results

14.2.6.1 Longitudinal members (single buckling field)

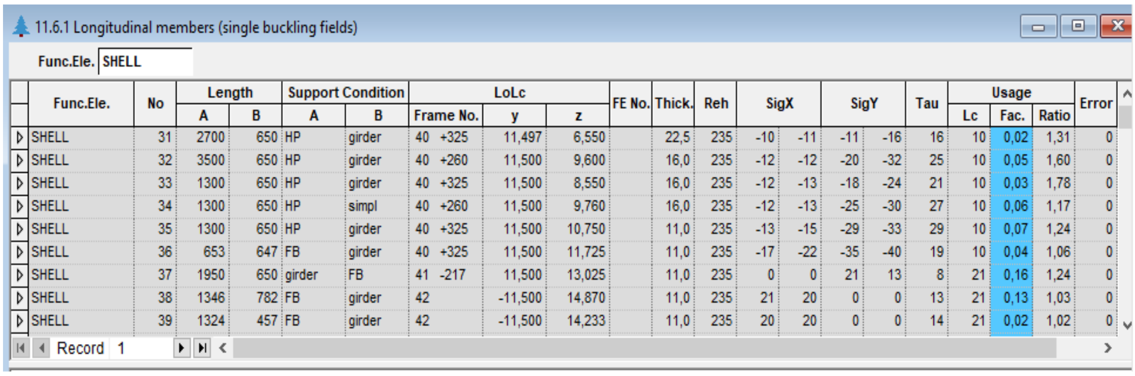
This section presents the single buckling field results for all longitudinal members. The buckling assessment is based on the rules and stresses derived from the finite element results. Since, we are considering the single buckling field here. Therefore, the undersize limit should be defined as per the DNVGL rule ref: [4] which states that for a single buckling field it's value should be **equal or less than 1** this is the reason why the oversized value was fixed to 0.9 and undersize value to 1. Now, to have the appropriate buckling assessment, all elements should lie under the oversized limit or between oversize and undersize limit. The best perspective for analysis would be the value less than the oversized value.

14.2.6.1.1 General approach for buckling assessment

In Poseidon, under the result section there is a term called **"Usage Factor"**, which helps in identifying the buckling assessment field quickly by just observing the color of the field. For example if color of the buckling field is:

- **Blue – Usage factor is less than the oversized value**
- **Red - Usage factor is greater than the undersized value**
- **Green – Usage factor is between undersized and oversized value**
- **Purple – Usage factor could not be determined**

14.2.6.1.2 Result analysis



11.6.1 Longitudinal members (single buckling fields)

Func.Ele. SHELL

Func.Ele.	No	Length		Support Condition		LoLc			FE No.	Thick.	Reh	SigX		SigY		Tau	Usage			Error
		A	B	A	B	Frame No.	y	z									Lc	Fac.	Ratio	
SHELL	31	2700	650	HP	girder	40 +325	11,497	6,550		22,5	235	-10	-11	-11	-16	16	10	0,02	1,31	0
SHELL	32	3500	650	HP	girder	40 +260	11,500	9,600		16,0	235	-12	-12	-20	-32	25	10	0,05	1,60	0
SHELL	33	1300	650	HP	girder	40 +325	11,500	8,550		16,0	235	-12	-13	-18	-24	21	10	0,03	1,78	0
SHELL	34	1300	650	HP	simpl	40 +260	11,500	9,760		16,0	235	-12	-13	-25	-30	27	10	0,06	1,17	0
SHELL	35	1300	650	HP	girder	40 +325	11,500	10,750		11,0	235	-13	-15	-29	-33	29	10	0,07	1,24	0
SHELL	36	653	647	FB	girder	40 +325	11,500	11,725		11,0	235	-17	-22	-35	-40	19	10	0,04	1,06	0
SHELL	37	1950	650	girder	FB	41 -217	11,500	13,025		11,0	235	0	0	21	13	8	21	0,16	1,24	0
SHELL	38	1346	782	FB	girder	42	-11,500	14,870		11,0	235	21	20	0	0	13	21	0,13	1,03	0
SHELL	39	1324	457	FB	girder	42	-11,500	14,233		11,0	235	20	20	0	0	14	21	0,02	1,02	0

Figure 119: Buckling result for longitudinal members

Figure –119 details the result of buckling assessment of one of the longitudinal members of the model, for e.g. shell. In the above diagram, the column **"Usage Factor and Error"** were the two most important parameter to analyze the buckling result. As mentioned before, the basis to assess buckling field completely depends upon the color coding of the usage factor. Blue color was considered as the most appropriate and suitable color to assess the buckling field as per GL rule. If Blue color appears that means, the buckling assessment is on right track and the element in that region can cope up with the compressive stress for single buckling field. Therefore, the main objective while doing the assessment was to check Usage factor with as much Blue color coding as possible to have the best results for a member. Other than usage factor, error code were also checked.as per GL rule and Poseidon software assessment coding, there are 18 different types of error possible from the buckling results. Instead of going through all the error code, only those error code which were present in case of this particular model assessment are mentioned in table -24.

Table 26: Error code explanation for longitudinal and transversal members

ERROR EXPLANATION:		
Error code	Tool tip	Meaning
1	Buckling Field not finished.	No buckling fields could be computed for the referring functional element group in the specified frame number range
4	No element for buckling field found	No finite elements were found belonging to the considered buckling field. E.g. In case of FE group name and Functional element name mismatch.
5	Cannot designate support condition	Buckling field has a complicate boundary condition which could not be mapped to an appropriate boundary condition used for the buckling check
8	Field is not rectangular.	Considered buckling field is not rectangular. An equivalent rectangular was taken for the buckling check.
10	Error in field displacement calculation	The stress computation based on the displacement method failed. This may be caused by a geometric mismatch between the Poseidon geometry and the FE mesh.
12	Von Mises stress exceeds Reh.	The von Mises stress already exceeds the yield stress of the buckling field
18	Hole data on buckling field.	Buckling fields includes hole. The buckling assessment must be done manually.

Next task was to go through all the longitudinal members which were located in the buckling assessment zone (frame no 40 to 130) to observe whether these members are able to withstand the compressive force coming from the heavy cargo loading on hatch cover under the two most critical load cases or not. The longitudinal members present in that zone are:

a) Shell
b) Deck A
c) Deck 1
d) Deck 2
e) Deck 3
f) Coaming deck
g) Inner bottom

h) Longitudinal Bulkhead 14
i) Longitudinal Bulkhead 15.1
j) Longitudinal Bulkhead 15.5
k) Longitudinal Bulkhead 17
l) Longitudinal Girder at centre
m) Longitudinal Girder 03
n) Longitudinal Girder 07
o) Longitudinal Girder 14
p) Longitudinal Girder 17

One by one all of the above member were checked against the buckling and some of them are shown in the coming figures.

a) Shell:

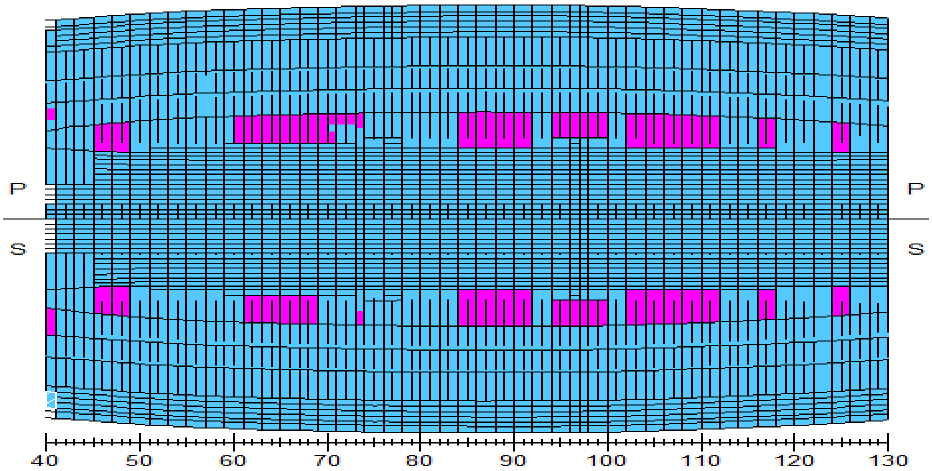


Figure 120: 2D view: Buckling result of Shell

Figure-120 shows the buckling result of shell on both port and starboard side of the vessel. The buckling assessment result with blue color is within the limit and its related stress is under the critical value calculated by displacement method.. Hence, we can say that the area covered in blue can easily withstand any buckling failure whereas the area in purple means that Usage factor could not be determined in this zone. Therefore, we need to investigate the reason behind it.

Out of 3327 buckling field in figure-120 there were 80 cases where the field has purple color that means, these 80 field's usage factor could not be determined. These 80 fields mainly has three kinds of error code i.e.4, 8 and 10 as described in the table –26..When we look at these error code, it can be seen that they are generally due to the mismatch between Poseidon modelling and FE meshing, basically related to its name convention and geometry. These errors are those kind of errors which highly depends upon Software's internal approach for doing computation whereas the error with code no -8 is a problem solving technique where the software automatically defines the rectangular buckling field for any element if it has any difficulty to introduce the buckling field..

Analysis of Undetermined Buckling field:

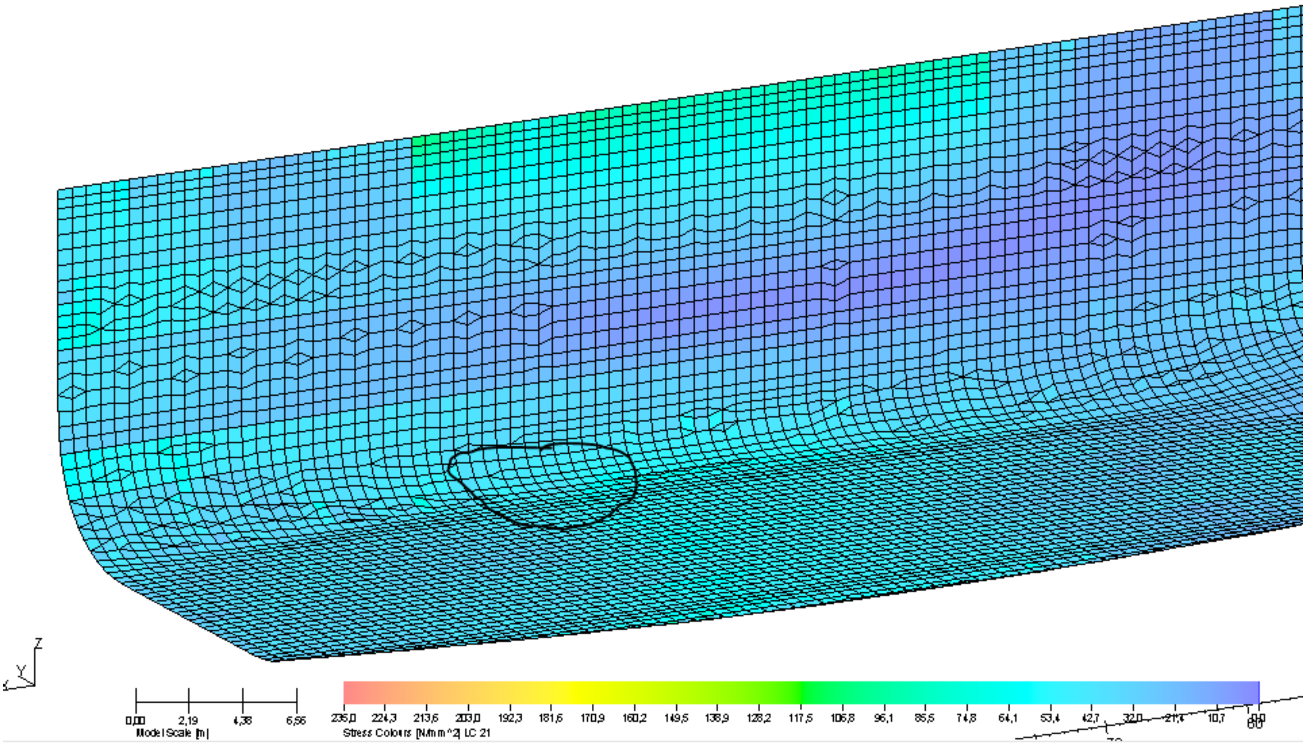


Figure 121: Shell plate with stress color distribution at Starboard side

Figure –121 shows the shell plating in 3d format with its stress distribution color pattern. Blue color pattern means the stress values are very much less and it is well below the critical value of stress. Now, it was decided to look in to those area where the buckling field has purple color as shown in figure -120. This area is located at the curved plating of shell structure as marked with a round shape in above figure. The stress checks were done in FEA software at these purple color pattern to verify whether these areas are under what amount of stresses, and the result obtained from FEA software were quite satisfactory. Hence, we can say that, although we cannot get the desired buckling field in that area but it is not overstressed either. .Since, the vessel already exists and it was approved by GL with very high vertical bending moment as per the class guidelines, hence, we can say that in spite of non-determination of buckling field in those purple area due to geometric unacceptance by the software it can withstand any buckling failure and can be accepted in our analysis.

Similarly, all other longitudinal members mentioned above were checked against buckling, and the result in most of the cases were satisfied with all buckling field color in Blue except DK-A and DK-3 Therefore, it was decided to look in to the DK-A and DK-3 separately.

b) DKA:

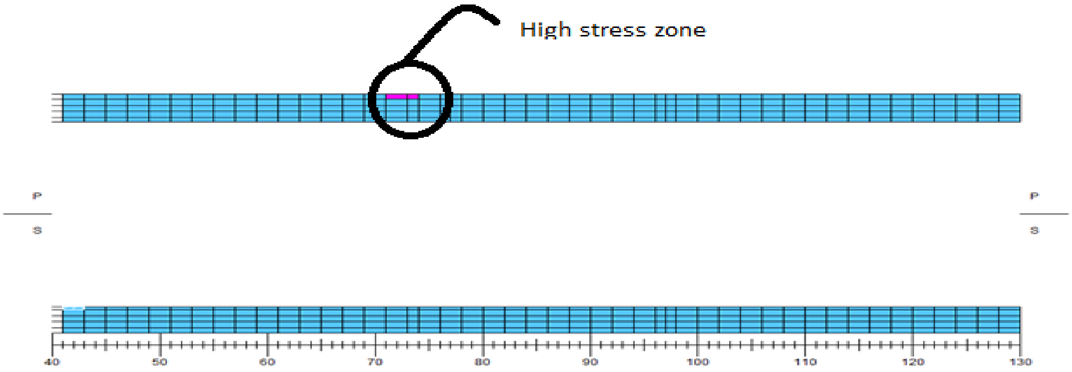


Figure 122: 2D view: Buckling result of DK-A

Figure-122 shows the buckling result of DK-A at both port and starboard side. The approach of analysing the buckling were same as explained in shell, but in this case, there were two fields where Usage factor could not be determined with error code of 12 which states that the Von Mises stress exceeded R_{eh}, At both field, the critical yield stress were 188 N/mm^2 but the Von Mises stress obtained here was 210 and 302 N/mm^2, which is much bigger than the critical value. The high stress value in these two fields are due to the carousel basement stopper in Y direction which is overloaded in Beam sea case. These buckling field can later be assessed by post processing approach.

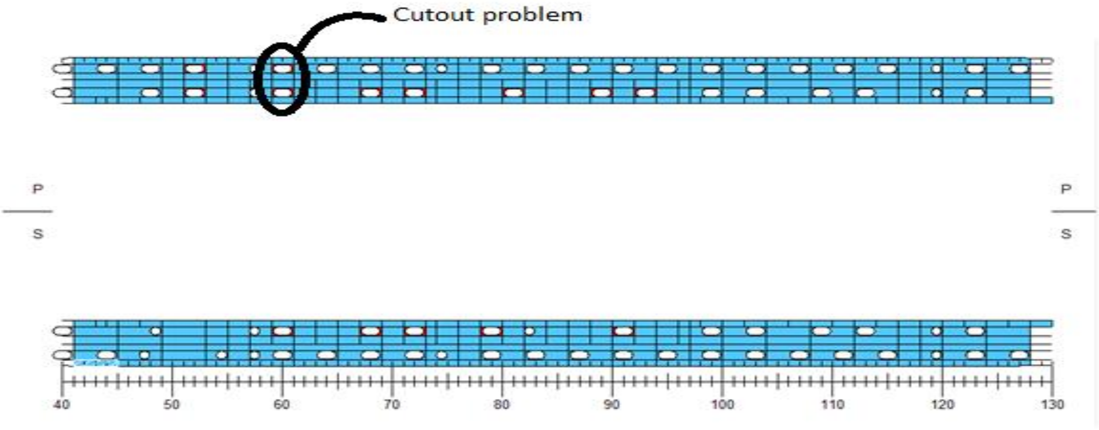


Figure 123. 2D view: Buckling result of DK-3

Figure-123shows the buckling result of DK-3 at both port and starboard side. The approach of analyzing the buckling was same as explained in shell, but in this case, the buckling analysis was not successful at the cut-outs section due to the software inadequacy as explained in the introduction part of this section. Therefore, nothing much can be done in this case to determine the buckling field. Other than hole, buckling of the plate is very much under control.

14.2.6.2 Transversal members (single buckling field)

This section presents the single buckling field results for all transverse members. Here also, the buckling assessment were based on displacement method. The undersize and oversize limits were the same as it was in case of longitudinal members. The method of analysis and the approach to do the buckling assessment were same as it was for longitudinal members.

The buckling analysis of transversal member becomes more important for the Beam sea case compare to head sea because in beam case the transversal frame will have a direct impact of incoming wave subjected to heavy loading on hatch cover. Therefore, it's imperative to check whether these transverse frame can withstand the horizontal bending or not.

All the transversal plate field between frame no- 40 to 130 were investigated one by one against buckling and it was found that the buckling results were quite satisfactory except few cases where it was a mismatch between the Poseidon modelling and FE mesh regarding geometry and name convention. There were few case where bucking field could not be assessed due to the holes without edge reinforcement as discussed in case of longitudinal members .Except to this area at starboard side (red colour buckling field in below figure), everything related to Von Mises stress calculation, shear stress calculation, pressure computation looked very much under the critical range. Thus, we can say that the buckling assessment for the transversal frame in the heavy loading area on hatch cover were found highly reasonable and satisfactory.

Few frames from the Analysis zone with its buckling field is shown below:

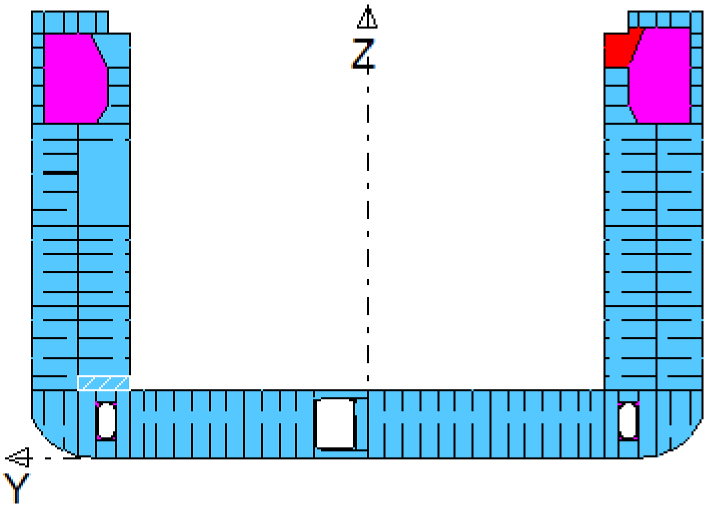


Figure 124: Frame no -76

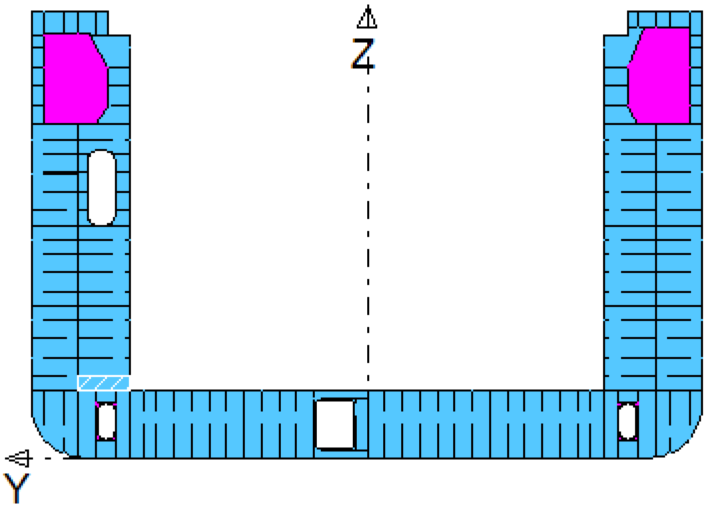


Figure 125 : Frame no -97

Figure – 124 and 125 shows the two transverse frame from the Hatch cover loading zone with its buckling field. As shown in above figure, most of the buckling field is blue in colour which suggest that the Usage factor is well below the oversized limit value that means the plate field can withstand the compressive stress successfully even under the severe Beam sea load case. The purple field at the top of the frame has error code 1 which states that the functional element is missing in that region and it is rightly so, because there is nothing defined in that region for this model. So, it can be ignored.

Other important observation in the Frame -76 is the Red field at the top corner on starboard side.. It means that the buckling field computed here is greater than the undersized limits and it can be vulnerable towards buckling thus, it’s identified as critical zone. There were as many as 10 frames with frame no- 73, 76, 78,80,82,85,86,88,99,102 which has this kind of oversized value at the same spot. Hence, all these areas were kept under critical zone for buckling.

15 FINAL DESIGN OF STANCHION TRUSS:

15.1 Main objective

This was the last objective of this thesis, after doing yield stress, deformation and buckling check of the model and the stanchion assembly, the final design of stanchion trusses with its suitable cross section were proposed by referring GL rule: ref : [9]

First of all, the maximum normal stress were obtained from the FE- model based on the hypothetical area of 100,000 mm^2 defined during the modelling of stanchion truss. The lone purpose of defining this hypothetical area was to get the maximum axial load.

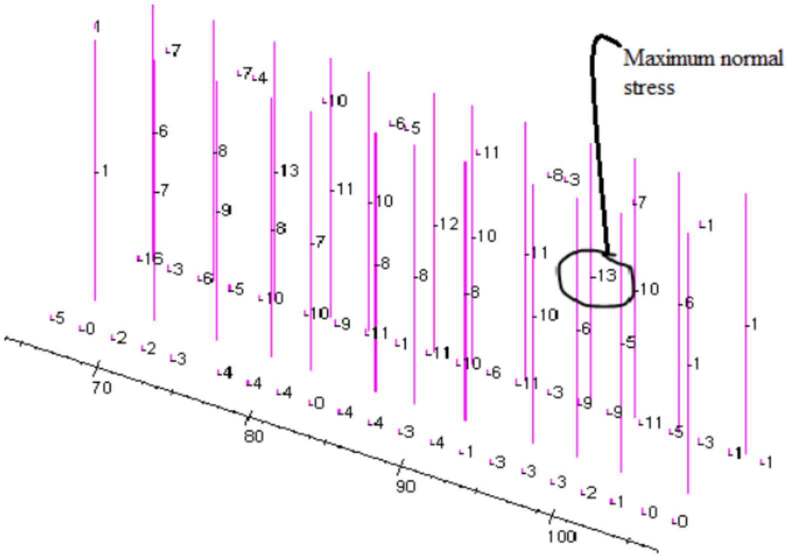


Figure 126: Maximum normal stress [Head sea case]

After finding the maximum normal stress, the corresponding maximum axial load were obtained again by referring the FE-model. The maximum axial load found was **-1295 kN**. Based on this maximum compressive load, the load on a pillar is to be calculated by referring the GL rule: ref: [9] described below:

The load on a pillar Ps is to be calculated by the following formula:

$$Ps = p.\ A + P_i\ [KN] \qquad (2)$$

Where,

- p: pressure [kN/m2] on load area for one pillar
- A: load area [m2] for one pillar
- Pi: load [kN] from pillars located above the pillar considered

In context of the thesis, the maximum compressive load on a pillar is already obtained as mentioned in the above paragraph. Also, due to the building tolerance and possible variation of load transfer, an additional skew load factor of 1.3 shall be considered as per class guideline: ref: [5]. Hence, the total load on the pillar (Ps) is given as:

$$Ps = -1295*1.3 \text{ kN}$$

$$= -1683.5 \text{ kN}$$

After calculating the load on a single pillar, the required scantling sectional area $A_{S,req}$ of pillars needs to be calculated. So,

$$A_{s,req} = \frac{P_s}{\sigma_p} * 10\ [cm^2] \tag{3}$$

Where,

- Ps: Load on pillar according to equation (1)
- $\sigma_{p:}$ Permissible compressive stress [N/mm^2]

Again,

$$\sigma_p = \frac{\kappa}{S} R_{eH} \tag{4}$$

Where,

- κ : reduction factor, defined as:

$$\kappa = \frac{1}{\phi + \sqrt{\phi^2 - \lambda_s{}^2}} \tag{5}$$

Where:

- ϕ: characteristic value, defined as:

$$\phi = 0.5[\ 1 + n_p * (\lambda_s - 0.2) + \lambda_s{}^2] \tag{6}$$

Again,

Factor defined as:

$n_{p\ =\ 0.34}$ for tubular and rectangular pillars

And

S: safety factor, defined as:

S = 2.00 in general

One more value λ_s was calculated by referring the GL rule ref : [9] as:

$$\lambda_s = \frac{l_s}{i_s * \pi} \sqrt{\frac{R_{eH}}{E}} \qquad \text{With } \lambda_s \geq 0.2 \tag{7}$$

Where,

- $l_{s:}$ length [cm] of the pillar
- λ_s degree of slenderness of the pillar
- $i_{s:}$ radius [cm] of gyration of the pillar, defined as

$$i_s = \sqrt{\frac{I_s}{A_s}} \tag{8}$$

Where,

- I_s :Moment of Inertia [c.m^2] of pillar
- A_s :Sectional area [cm^2] of the pillar

To find out the degree of slenderness of the pillar (λ_s), one has to define the cross sectional area of the trusses. It was decided to choose the hollow pipe as the desired profile by referring the profile table from standard Eurocode handbook ref : [10] .The proposed stanchion truss diameter and thickness were determined after an iterative process where many profile dimensions ranging from 267 to 419 mm diameter and 16 to 17.5 mm thickness were checked and finally the dimension for a hollow pipe along with area and moment of Inertia were selected by referring the same profile table ref : [10] and it is presented below in the table –27.

Table 27: Stanchion truss profile

Stanchion truss profile:				
Profile name	**Wall thickness (mm)**	**Diameter (mm)**	**Area (cm^2)**	**Moment Inertia (cm^4)**
Hollow pipe	16	406,4	196	37449

To calculate the required sectional area $A_{S,req}$ (eq.2), all the consecutive formulas from equation 3 to 8 were calculated and its final value is mentioned below:

Starting from equation 8, here the moment of inertia and the sectional area of pillar was taken from table-27.

From here all length and areas would be expressed in cm and cm^2 respectively to match with the GL rule reference units, Again, going back to the required scantling area calculation, we have:

$$i_s = \sqrt{\frac{I_s}{A_s}}$$

I_s= 13.82 cm

From I_s value, λ_s value was calculated according to equation (7), hence,

λ_s = 1.14

From λ_s value, ϕ was calculated using equation (6) hence,

$$\boldsymbol{\phi = 1.31}$$

From ϕ value, K was calculated using equation (5) hence,

$$\mathbf{K = 0.58}$$

From K value, σ_p value was calculated using equation (4) hence,

$$\boldsymbol{\sigma_p = 103.37 N/cm^2}$$

From σ_p value, finally, the required scantling area was calculated using the equation (3)

$$\mathbf{A_{S,req} = 47.4\ c.m^2}$$

Here, we can see that the required scantling area was calculated as **47.4 c.m^2**. This scantling area would be the required design area of the stanchion truss which can be implemented.. Hence, we can say that the hypothetical area of 100,000 mm^2 which was taken at the design stage for calculating the axial load can be replaced now with this scantling area. Also, **the total number of stanchion trusses to successfully transfer the load was 24.**

Finally, we can say that the design proposal of stanchion assembly between the weather deck and tank top looks very promising and it can be forwarded for next phase of analysis.

At last, it can be said that the goal of the thesis to do the global strength analysis and design of stanchion assembly to transfer the load from top of hatch cover/weather deck to bottom structure looks good except the high tension load for basement stopper in y direction (Table – 24, subchapter -11.6.1)in beam sea load case.

16 CONCLUSION

The thesis topic “Design study for the transportation of heavy cargo on hatch cover of a MPV” was more intended towards the existing and real case scenario where the vessel (*pk-116*) which was used for analysis is already existing and to make this vessel more efficient in heavy cargo handling an additional stanchion (pillar) assembly design was proposed between weather deck and tank top, such that it can efficiently transfer the load from top to bottom. After vessel modelling, the most critical task was to propose a design of stanchion assembly for an efficient load transfer, and to do this, 5-6 possible scenario were analysed before lastly arriving to the final case as discussed in this report. Without going much in to the modelling detail, the major findings are summarized below:

- Two critical load cases for Beam and Head sea were provided by the company personnel.

The loading condition is based on:

- The feasibility of the stanchion assembly which was investigated by analysing the compressive load on the stanchion trusses. The load range for all these trusses were found in between -30 kN to -1295 kN for head sea case. And for beam sea case, the load range on stanchion truss was in between -30kN to -1000kN.
- Since the main analysis zone was amidships section where the stanchion assembly was modelled along with hatch cover, hence, an area 30% more towards the aft and forward region of this analysis zone were chosen to do the yield stress and buckling checks.
- The results for yield stress throughout the analysis zone of global model was found well within the critical yield stress of 188 N/mm^2 except two places on hatch cover plate and one place at DK-A plate at port side, where the whole cargo load was applied. At hatch cover the yield stress was 6-11 N/mm^2 more than the critical value and on DK-A it was around 50 N/mm^2 more than the critical value. The material grade at both these places were **NT24** with a yield stress of 235 N/mm^2. The overstressed region on hatch cover plate can be sorted out by reinforcing some plates at that location.
- The deformation was measured in plan view and longitudinal view of model by taking the reference from model scale given inside the software. In plan view, it was observed that there is a knuckle formation near 40 m distance from the aft of the vessel in beam sea case due to bulkhead positioning very near to it. The deformation of model in longitudinal direction was measured for head sea case in z direction and its value was nearly 50 mm. Another observation was again for beam sea case, where the relative displacement between hatch cover and side shell coaming was around 40 mm.
- The buckling check was performed by Displacement method in the same zone as mentioned above for yield stress check, the single buckling field assessment of global model towards its main functional elements such as Decks, Longitudinal bulkhead and Longitudinal girders were done and it was found that most of the elements are well below the buckling failure except 2 element at DK-A shown by the Red colour pattern inside the software, other critical area was located between transverse frame no- 73 to 102 at starboard side on top of the frame precisely at frame no. 73,76,78,80,82,85,86,88,99,102.
- The upper and lower platform of stanchion assembly also called as load spreader were checked for its utilization. The element used to build this platform was HEB 700 and

HEB 500 I- beam section. A thorough check up of overstressed area at both upper and lower platform was done and it was found that the **element no-63** located at the upper platform of stanchion assembly underneath the intermediate girder of hatch cover – 9 at port side had 262 N/mm^2 and 210 N/mm^2 of stress values in head and beam sea case respectively. These high values are still under the critical value of yield stress for material HT36 which is used for the I-beam section, the critical yield stress of HT36 is 284 N/mm^2. Thus, we can say that, even the size of the I-beam is small but due to lower utilisation of region all its values are in permissible zone.

- After stanchion assembly, Hatch cover supporting structure was investigated at both port and starboard side for any overloading in beam and head sea case. The maximum load found was -1600 kN in head sea case for element no – 7 located as aft bearing of hatch cover no -10 at port side in Z-direction. All other supporting structure in x, y and z direction was found reasonable.
- Next was checking of loads for carousel basement, in X, Y and Z direction, here the maximum value for beam sea case was 3264 kN in Y direction located between carousel basement and DK-A near hatch cover no -7 at port side. This high force value was due to the fact that there is a watertight bulkhead located nearby in that region at aft side which is making DK-A more stiff as shown in the deformation section of this report. For head sea case, the maximum load was occurring at basement Z for element - 8 near hatch cover no. – 8 at port side of vessel.
- After assessing all above checks, the stanchion truss cross sectional area was defined using GL rule ref: [9]. by considering the maximum axial load of -1295kN with a skew load factor of 1.3. The total number of stanchion truss added in the assembly was 24.

Recommendation for future work:

The proposed design of stanchion assembly is very much efficient in fulfilling all its desired objectives without much hassle and it can be considered for implementing inside the existing vessel but, before doing so the high loads generated by the hatch cover supporting members should be sorted out efficiently by doing modification in existing hatch cover bearing pads such that it can distribute the loads equally in all directions.
There are always some ways to improve the existing design and few among them would be:

- To add intermediate Beam at the middle of the stanchion trusses between the two existing platform such that it can distribute the axial load more comprehensively while transferring it to the tank top. With this arrangement more cargo load can be applied on top of the hatch cover.
- Bracings can also be added in Y-Z and X-Z plane in order to distribute the loads coming from top of the hatch cover to tank top.

17 ACKNOWLEDGEMENTS

I owe my deepest gratitude to my company supervisor *Mr. Jonathan Schmidt*, Senior Project Engineer, *Mr. Arthur Stuckert*, Project Engineer and *Mr. Helge Rathje,* Senior Project Engineer at SAL Heavy Lift GmbH, whose encouragement, guidance and support from the initial to the final level enabled me to complete my Internship work. I heartily thank them for giving me an opportunity to work on a company project and providing all possible assistance in terms of knowledge and resources during the Internship.

I am grateful to my University of Rostock professor Dr. - *Ing Thomas Lindemann,* for providing me an invaluable chance to work on an Industrial internship. His positive, enthusiastic and inspiring attitude with timely assistance in terms of knowledge enabled me to have sound understanding of my thesis. Without his imperative guidance I would not have succeeded in my endeavour.

It was undoubtedly a tremendous experience working with the highly qualified, competent and supportive professional of the SAL Heavy Lift GmbH. I would also like to express my special thanks to *Mr. Karsten Behrens*, General Manager, *Engineering Dept.* at SAL Heavy Lift GmbH for his consistent support in terms of administrative and organizational work.

Also, I would like to express my regards towards DNV-GL personals who had guided me with the modelling and analysis approach of the respective software used during the Internship tenure.

In the end, I would like to thank my family members, my parents, my brother and my sister who had given me moral support throughout my thesis regime.

This thesis is developed under the frame of European Master Course in Integrated Advance Ship Design named EMSHIP for the European Education in Advance Ship Design.

Ref.: 532404-EM-1-2012-1-BE-ERA MUNDUS-EMMC

18 REFERENCES

[1] GL- V- Rules for Classification and Construction Analysis Techniques.
  a. Hull structure analysis
  b. Guidelines for global strength analysis of MPV
[2] GL – I – Rules for Classification and Construction Ship Technology Seagoing Ships
[3] I-1-1 – Hull structure analysis Section 2
[4] I-1-1 – Hull structure analysis Section 3
[5] I-1-1 – Hull structure analysis Section 5
[6] I-1-1 – Hull structure analysis Section 17
[7] POSEIDON Fundamental by DNV- GL Group.
[8] GL Shipload Manual by Germanisher Lloyd
[9] GL– I – Rules for Classification and Construction Ship Technology Seagoing Ships, Section – 10.C for pillars.
[10] Euro code hand book for profile table : Klockner Konsttuktions handbuch page no-157 to 159
[11] http.:// www.sal-heavylift.com/

19 ANNEXURE- 1

NODAL FORCES:

LG	Node	Px	Py	Pz	B.S- Px	B.S -Py	B.S- Pz	H. S - Px	H. S- Py	H. S - Pz
20	131997	-0,1	-2,1	-2603,6	0	-2	-2604	0	-2	-2604
65	131997	260,4	0	0	3	0	0	-89	0	0
45	131997	0	260,4	0	0	-596	0	0	-24	0
44	131997	0	0	260,4	0	0	-101	0	0	-306
59	131997	0	-5164,1	0	0	-89	0	0	35	0
39	131997	5164,1	0	-12654,9	-6	0	15	170	0	-417
63	131997	0	12654,9	0	0	-2	0	0	5	0
20	131998	-0,2	-1,7	-2199,5	0	-2	-2200	0	-2	-2200
65	131998	220	0	0	3	0	0	-76	0	0
45	131998	0	220	0	0	-503	0	0	-20	0
44	131998	0	0	220	0	0	-85	0	0	-259
59	131998	0	-4362,7	748,9	0	-75	13	0	30	-5
39	131998	4362,7	0	-10893,5	-5	0	13	144	0	-359
63	131998	-748,9	10893,5	0	0	-2	0	0	4	0
20	131999	-0,3	-1,6	-2198,9	0	-2	-2199	0	-2	-2199
65	131999	219,9	0	0	3	0	0	-76	0	0
45	131999	0	219,9	0	0	-503	0	0	-20	0
44	131999	0	0	219,9	0	0	-85	0	0	-259
59	131999	0	-4361,2	1296,7	0	-75	22	0	30	-9
39	131999	4361,2	0	-11436,6	-5	0	14	144	0	-377
63	131999	-1296,7	11436,6	0	0	-2	0	0	4	0
20	132000	-0,4	-1,8	-2601,9	0	-2	-2602	0	-2	-2602
65	132000	260,2	0	0	3	0	0	-89	0	0
45	132000	0	260,2	0	0	-595	0	0	-24	0
44	132000	0	0	260,2	0	0	-100	0	0	-306
59	132000	0	-5159,7	1767,8	0	-89	31	0	35	-12
39	132000	5159,7	0	-14416	-6	0	17	170	0	-475

63	132000	-1767,8	14416	0	0	-2	0	-1	5	0
20	132001	-0,3	-1,4	-2201,3	0	-1	-2201	0	-1	-2201
65	132001	220,1	0	0	3	0	0	-76	0	0
45	132001	0	220,1	0	0	-504	0	0	-20	0
44	132001	0	0	220,1	0	0	-85	0	0	-259
59	132001	0	-4364,5	1297,7	0	-76	22	0	30	-9
39	132001	4364,5	0	-12943,1	-5	0	16	144	0	-427
63	132001	-1297,7	12943,1	0	0	-2	0	0	5	0
20	132002	-0,2	-1,3	-2199,1	0	-1	-2199	0	-1	-2199
65	132002	219,8	0	0	3	0	0	-76	0	0
45	132002	0	219,8	0	0	-503	0	0	-20	0
44	132002	0	0	219,8	0	0	-85	0	0	-259
59	132002	0	-4359,3	746,8	0	-75	13	0	30	-5
39	132002	4359,3	0	-13474,5	-5	0	16	144	0	-444
63	132002	-746,8	13474,5	0	0	-2	0	0	5	0
20	132003	-0,1	-1,5	-2595,7	0	-2	-2596	0	-2	-2596
65	132003	259,5	0	0	3	0	0	-89	0	0
45	132003	0	259,5	0	0	-594	0	0	-24	0
44	132003	0	0	259,5	0	0	-100	0	0	-305
59	132003	0	-5145	0	0	-89	0	0	35	0
39	132003	5145	0	-16137,6	-6	0	19	170	0	-532
63	132003	0	16137,6	0	0	-2	0	0	6	0
20	132004	0	-1,3	-2199,4	0	-1	-2199	0	-1	-2199
65	132004	219,8	0	0	3	0	0	-76	0	0
45	132004	0	219,8	0	0	-503	0	0	-20	0
44	132004	0	0	219,8	0	0	-85	0	0	-259
59	132004	0	-4359,3	-746,8	0	-75	-13	0	30	5
39	132004	4359,3	0	-13474,5	-5	0	16	144	0	-444
63	132004	746,8	13474,5	0	0	-2	0	0	5	0
20	132005	0,1	-1,4	-2201,8	0	-1	-2202	0	-1	-2202
65	132005	220,1	0	0	3	0	0	-76	0	0

45	132005	0	220,1	0	0	-504	0	0	-20	0
44	132005	0	0	220,1	0	0	-85	0	0	-259
59	132005	0	-4364,5	-1297,7	0	-76	-22	0	30	9
39	132005	4364,5	0	-12943,1	-5	0	16	144	0	-427
63	132005	1297,7	12943,1	0	0	-2	0	0	5	0
20	132006	0,2	-1,8	-2602,6	0	-2	-2603	0	-2	-2603
65	132006	260,2	0	0	3	0	0	-89	0	0
45	132006	0	260,2	0	0	-595	0	0	-24	0
44	132006	0	0	260,2	0	0	-100	0	0	-306
59	132006	0	-5159,7	-1767,8	0	-89	-31	0	35	12
39	132006	5159,7	0	-14416	-6	0	17	170	0	-475
63	132006	1767,8	14416	0	0	-2	0	1	5	0
20	132007	0,1	-1,6	-2199,5	0	-2	-2200	0	-2	-2200
65	132007	219,9	0	0	3	0	0	-76	0	0
45	132007	0	219,9	0	0	-503	0	0	-20	0
44	132007	0	0	219,9	0	0	-85	0	0	-259
59	132007	0	-4361,2	-1296,7	0	-75	-22	0	30	9
39	132007	4361,2	0	-11436,6	-5	0	14	144	0	-377
63	132007	1296,7	11436,6	0	0	-2	0	0	4	0
20	132008	0	-1,7	-2199,8	0	-2	-2200	0	-2	-2200
65	132008	220	0	0	3	0	0	-76	0	0
45	132008	0	220	0	0	-503	0	0	-20	0
44	132008	0	0	220	0	0	-85	0	0	-259
59	132008	0	-4362,7	-748,9	0	-75	-13	0	30	5
39	132008	4362,7	0	-10893,5	-5	0	13	144	0	-359
63	132008	748,9	10893,5	0	0	-2	0	0	4	0
Total:		58323	102376,7	-180323,1						
Total action Force				kN	-32	-7408	-28898	868	156	-36412
Weight. Umblical:		2800 t	Total acti Force	t	-3	-755	-2946	88	16	-3712
Gravity				g	0,00	-0,27	-1,05	0,03	0,01	-1,33

Accl‘n	m/s²	-0,01	-2,65	-10,32	0,31	0,06	-13,00
Accl‘n in Z				-0,51			-3,19

In above table,, B:S represent Beam sea, H:S represent head sea, total force written in saffron is calculated after considering the load factor generated from shipload (as mentioned in subchapter 11.3)

COMPARISON OF ACTION AND REACTION FORCE FOR BEAM SEA:

Comparison:	
Action Loads on Sub-COG in y-direction(KN)	-7408
Own weight of HC Panels (abt 30 tons per panel x and)	-317
Total Action loads (KN)	-7726
Deviation between action and reaction (%)	1.3

Sum Loads @ HC Support		Sum Loads @ basement Y	
Element No.	[kN]	Element No.	[kN]
30	808	1	1047,7
31	265,9	2	272,9
32	140	3	583,6
33	340,2	4	755,7
		5	3154,9
		6	256,6
Total	1554 kN		6071 kN
Total Reaction Loads in Y-Direction (KN			=1554+6071
			7626

Sum Loads @ Basement Z	
Element No.	[kN]
1	143,5
2	-24,8
3	-248,1
4	-596,6
5	-776,1
6	-2,2
7	-63,5
8	-53,1
9	-558,3
10	-893,6
11	87,3
12	-932
13	-88,9
14	-325,9
15	-406,1
16	-144,7
17	196,1
18	-445,7
19	-0,1
20	-39,4
21	-1330,9
22	-859,2
23	-15,4
24	20,2
25	-244,5
26	-510
27	36
28	-4,8
29	-211,7

30	-810
31	-491,5
32	-152,6
33	-35,9
34	44,8
35	-656,7
36	93,3
37	-71,8
38	-65,7
39	-128,2
40	-425,7
41	-372,9
42	-112,3
43	-11,1
44	-134,9
45	-288,6
46	-602,2
47	-47,3
48	42,4
49	51,4
50	-1099,3
51	-1138,2
52	-23,4
53	23,5
54	-135,2
55	-478,4
56	-55,8
57	-912,1
58	-735,1
59	-57,2
60	0
61	-2,4

62	-358,7
63	-15,4
64	-460,3
65	-353,3
66	-8,4
67	0
68	36,9
69	-393,7
70	66,2
71	-76
72	-632,1
73	-515,2
74	-186,2
75	-2,5
76	82,6
77	135,1
78	-646,8
79	-517,2
80	-752,7
82	-658,6
83	-511,8
84	0
85	0
86	123,4
87	49,1
88	25,9
89	-70
90	-134,6
91	-314,5
92	-278,4
93	-166

94	-42
95	75,2
96	-200
97	86,6
98	473,9
99	63,7
100	499,5
101	727,7
102	-424,5
103	74,7
104	244,8
105	-549,7
106	67,4
107	76,9
108	-303,7
109	291,8
110	0
111	0
112	0
113	0
114	-409,8
115	-966,8
116	-1240,7
117	-1172
118	-1113
119	-1005,7
120	-889,9
121	-801,8
122	-73,3
123	-184,2
124	1,5
125	-283,9

126	0
127	0
128	0
129	0
Total Reacti Loads in Direction	-30592,1
Total Action Loa in Z-Direction	-28898
Deviation	5,86%

COMPARISON OF ACTION AND REACTION FORCE FOR HÉAD SEA:

Sum Loads @ HC Support X		Sum Loads @ basement X	
Element No.	[kN]	Element No.	[kN]
21	-31,8	1	255
22	42,6	2	-714,7
23	-387,8	3	-1191,2
24	280,2	4	601,3
25	34,3	remark: to be ignored, trusses are n arranged parallel to the deck	
26	876		
27	-70,1		
28	73,7		
Total	817 kN	-1050 kN	
Total Reaction Loads in X-Direction		817 kN	

Comparison:		
Action Loads on Sub-COG in X-Direction (KN)		868
Own weight of HC panels (abt. 30 tons per panel x and y)		37
Total Action loads (KN)		905
	Deviation (%)	10.78

BASEMENT Z	
Element No.	[kN]
1	14,4
2	-0,4
3	-162,5
4	-698,7
5	-965,4
6	-79
7	-109,9
8	-31,8
9	-745,4
10	-1086,9
11	-9,7
12	-671,4
13	-90,3
14	-409,2
15	-501,2
16	-170,6
17	41,8
18	-225
19	315,1
20	-216,6
21	-1584,8
22	-989,7
23	51,3

24	-52,7
25	-141,8
26	-656,1
27	-137,1
28	12,3
29	-200,5
30	-949,2
31	-575
32	-117,4
33	-25,9
34	-36,2
35	-286,1
36	-12,7
37	-272,3
38	-58
39	-97,9
40	-476,2
41	-436,2
42	-100
43	-3,3
44	-269,6
45	-91,2
46	-530,2
47	218,5
48	-83,5
49	46,9
50	-1299
51	-1293
52	-24,6
53	-11,2
54	-10,7

55	-811,4
56	-81,7
57	-1174,6
58	-854,2
59	-41,2
60	0
61	42,7
62	-404,9
63	-87,4
64	-581,7
65	-415
66	26,5
67	0
68	75,5
69	-22,3
70	-72,2
71	-188,3
72	-901,1
73	-673,8
74	-175,7
75	-0,7
76	-6,4
77	111,3
78	-110,8
79	-204,4
80	-277,2
82	-488,5
83	-154,2
84	0
85	0
86	63,7
87	83,4

88	92,8
89	8,7
90	-40,6
91	-177,2
92	-115,3
93	-35,5
94	13,1
95	104,4
96	-623,8
97	-404,6
98	638,9
99	-229,4
100	1006,9
101	-105,3
102	-734
103	-181,2
104	-400,5
105	-1415,9
106	-123,5
107	-350,6
108	-1270,9
109	-78,3
110	0
111	0
112	0
113	0
114	-300,4
115	-969,5
116	-1396,5
117	-1456
118	-1280,3

119	-1105,7
120	-869,5
121	-203,6
122	-426,9
123	-421,5
124	-860,8
125	76,4
126	0
127	0
128	0
129	0
Total reaction loads in Z-direction	-37257
Action Loads at Sub-COGs in direction	-36412
Deviation (~~%~~)	2,32