

BACHELOR´S THESIS (AMK)

Mechanical and Production Engineering

Naval Architecture

2019

Petri Mehto

OPTIMIZED ANALYSIS METHOD FOR HOISTING DESIGN OF LARGE STEEL STRUCTURES


TURKU AMK
TURKU UNIVERSITY OF
APPLIED SCIENCES

BACHELOR'S THESIS | ABSTRACT

TURKU UNIVERSITY OF APPLIED SCIENCES

Mechanical and Production Engineering | Naval Architecture

2019 | 56 pages, 3 pages in appendices

Petri Mehto

OPTIMIZED ANALYSIS METHOD FOR HOISTING DESIGN OF LARGE STEEL STRUCTURES

This thesis was commissioned by Hull Detail Design department of Meyer Turku Oy. The main objective of this thesis was to create an analysis method to get relatively fast and reliable results from the hoisting model to be used in Hoisting Design. The goals of this thesis were to create a hoisting model and to come up with an approach to analyze the results from the hoisting model. This work was done in the Hull Detail Design department of the company and it was done for the Hoisting Design. This thesis is working as a pilot review of how the analyzing of a model can be made.

First, a review of the theory and standards of the mechanical strength was made. Next, the hoisting model was created twice and the modelling time was measured on the second run. The trolley system was developed as a part of the hoisting model. Finally, the method to analyze a model was introduced.

The program used in this modelling was FEMAP with Nastran NX add-on included. FEMAP was used in modelling and Nastran NX was used in post-processing.

The measured modelling time turned out to be relatively short. Because this thesis was a review of the structural analysis, the general principle is introduced with the exclusion of details on how the automation for the analysis should be made.

KEYWORDS:

Hoisting design, Structural analysis, Finite Element Analysis

OPINNÄYTETYÖ (AMK) | TIIVISTELMÄ

TURUN AMMATTIKORKEAKOULU

Kone- ja tuotantotekniikka | Laiva- ja meritekniikka

2019 | 56 sivua, 3 liitesivua

Petri Mehto

OPTIMOITU ANALYSOINTIMENETELMÄ SUURIEN TERÄSRAKENTEIDEN NOSTOSUUNNITTELUUN

Opinnäytetyö suoritettiin Meyer Turku Oy:n runkosuunnitteluosastolla. Tämän opinnäytetyön päätavoitteena oli luoda analysointimenetelmä, jolla saadaan suhteellisen nopeat ja luotettavat tulokset nostomallista käytettäväksi nostosuunnitteluun. Tavoitteina oli luoda nostomalli ja luoda lähestymistapa nostomallin tulosten analysointiin. Tämä työ tehtiin runkosuunnitteluyksikössä nostosuunnittelun käyttöön. Työ toimi pilottitarkasteluna sille, miten rakenteen analysointi voidaan suorittaa.

Aluksi tehtiin katsaus mekaanisen kestävyuden standardeihin ja teorioihin. Seuraavaksi nostomalli luotiin kahdesti, ja mallinnusaika mitattiin toisella mallinnuskerralla. Osana nostomallia luotiin nosto systeemi. Lopuksi esitettiin periaate, jolla luotua nostomallia voidaan analysoida.

Mallinnuksessa käytettiin FEMAP-ohjelmaa Nastran NX:llä varustettuna. FEMAP-ohjelmaa käytettiin mallinnuksessa, ja jälkikäsitteily suoritettiin Nastran NX:llä.

Mitattu mallintamiseen käytetty aika osoittautui lopulta suhteellisen lyhyeksi. Koska työ oli analysoinnin osalta pilottitarkastelua, ei tarkkaa selvitystä analysoinnin automatisoinnista suoritettu, mutta pääperiaate esitetään.

ASIASANAT:

nostosuunnittelu, rakenteen analysointi, elementtianalyysi

CONTENT

GLOSSARY AND ABBREVIATIONS	8
1 PREFACE	11
1.1 Acknowledgements	11
1.2 Objectives of the thesis	11
1.3 Hull production profile	12
2 HOISTING DESIGNING	13
2.1 Principles for lifting	13
2.1.1 Distribution of load	15
2.1.2 Moment in structure	17
2.2 Trolley system setup	18
2.2.1 Trolley system stability	20
3 FINITE ELEMENT ANALYSIS	22
3.1 Equation for deformation	22
3.2 Constitutive equation	24
3.3 The Equation of Equilibrium	25
3.4 Finite element analysis process in FEMAP	27
3.4.1 Materials in the model	28
3.4.2 FEMAP model	29
3.4.3 Ways of fixing the model	32
3.4.4 Boundary conditions	33
3.4.5 Analyzing results	35
4 WELD ANALYSIS	36
4.1 Lifting lugs	36
4.1.1 Welding design for lifting lugs	37
4.1.2 Weld strength analysis of lifting lugs	38
4.2 Lifted structure	40
5 STRUCTURAL MEMBER ANALYSIS	42
5.1 Mechanical buckling strength of members using Eurocode 3	42
5.1.1 Cross-section verification for plastic resistance	43
5.1.2 Cross-section verification for buckling resistance	43

5.2 Mechanical buckling strength of members using DNV GL rules	45
6 CREATING A HOISTING MODEL	47
6.1 Trolley system in FEMAP	47
6.2 From global model to local model	48
6.3 Checking the local model	49
6.4 The hoisting model	50
6.5 The analyzing method	51
7 CONCLUSIONS	52
REFERENCES	55

APPENDICES

- Annex 1. IACS steel grades
- Annex 2. Admissible stresses in structural member
- Annex 3. Admissible stresses in welds

EQUATIONS

Equation 1. Sling force (Marine Insight 2018)	14
Equation 2. Center of gravity (Räsänen 2000)	15
Equation 3. Discrete mass distribution	15
Equation 4. Example of orthographic projection matrix	16
Equation 5. Orthographic projection (Giovanni 2018).....	16
Equation 6. Body mass force	17
Equation 7. Body moment.....	17
Equation 8. Stability condition of the system	20
Equation 9. Total force of the trolley.....	21
Equation 10. Normal strain definition (Kelly 2015).....	22
Equation 11. Normal strain differentiation	22
Equation 12. Normal strain in all directions	23
Equation 13. Shear strain definition (Kelly 2015)	23
Equation 14. Shear strain differentiation (Kelly 2015)	24
Equation 15. Shear strain for all planes	24
Equation 16. Strain tensor (Lagace 2001).....	24
Equation 17. Constitutive equation (Heil 2010)	24
Equation 18. Stress-strain relation (Heil 2010).....	25
Equation 19. Newton's second law	25
Equation 20. Force equation for stress and movement of body.....	26
Equation 21. Equation of Motion in one-dimension (Kelly 2015)	26
Equation 22. Equation of Motion in 3-dimension (Kelly 2015)	26
Equation 23. Equation of Equilibrium (Kelly 2015)	26
Equation 24. Maximum stress in lifting lug	38
Equation 25. Maximum force in lifting lug.....	38
Equation 26. Admissible stress for welds (SFS-EN 13155+A2 2009).....	41
Equation 27. Maximum width-to-thickness ration (EN 1993-1-1 2004).....	43
Equation 28. Plastic resistance (EN 1993-1-1 2004).....	43
Equation 29. Relative slenderness (EN 1993-1-1 2004).....	43
Equation 30. Slenderness (EN 1993-1-1 2004).....	43
Equation 31. Reduction factor (EN 1993-1-1 2004).....	44
Equation 32. Admissible buckling resistance (EN 1993-1-1 2004)	44
Equation 33. Elastic compressive column buckling stress (DNVGL-CG-0128 2018) ...	45
Equation 34. Minimum critical buckling stress (DNVGL-CG-0128 2018)	46
Equation 35. Buckling utilization factor (DNVGL-CG-0128 2018).....	46

DRAWINGS

Picture 1. Tilt of structure	13
Picture 2. Force subjected to the sling wire	14
Picture 3. Example of using the discrete mass distribution equation.	16
Picture 4. Transversal section of the trolley system setup.....	18
Picture 5. Longitudinal section of the trolley system setup	19
Picture 6. Top view of the trolley system setup	19
Picture 7. Triangle composed by the trolley forces.....	20
Picture 8. Force acting in each cable wire of the trolley	21
Picture 9. Deformation by shear and normal force.	23
Picture 10. One-dimensional body element that is under the influence of body force. .	25
Picture 11. FEA process from creating a model to the results (Saratech 2018).....	27
Picture 12. Example of defining the material properties in FEMAP	29
Picture 13. Local model to which the finite element analysis was done.....	30
Picture 14. Example of missing structures	32
Picture 15. Example of constrains of Main articulation	34
Picture 16. Typical lifting lug (padeye type).....	36
Picture 17. Example of full penetration welding in lifting lug	37
Picture 18. Force and total stress is calculated in the shown point of interest.	39
Picture 19. Total stress in lifting lug.....	39
Picture 20. Area from where the force is calculated	40
Picture 21. Model of hoisting system.....	47
Picture 22. Model Info panel	48
Picture 23. The making of the local model	48
Picture 24. Mesh size decrease using quadrilateral mesh.....	49
Picture 25. Remesh of the beams.....	49
Picture 26. Preliminary hoisting model.	50
Picture 27. Design values which surpass the admissible boundary values	51
Picture 28. Example of unsupported steel structure.	52
Picture 29. Von Mises stress for the unsupported structure	53
Picture 30. Effect of added supports in stresses	54

TABLES

Table 1. Materials used in the global model.....	28
Table 2. Physical quantities to be determined and the units used	28
Table 3. Items that should be noted and checked from the model before analysis	31
Table 4. Standard types of nodal degrees of freedom in FEMAP	34
Table 5. Basis values to be known according to Eurocode 3	42
Table 6. Basis values to be known according to DNV GL	45

GLOSSARY AND ABBREVIATIONS

Glossary

API	Application Programming Interface which is used to make macros in FEMAP
Block	Building unit in which ship is divided. Is typically composed of two or more sections.
Hoisting plan	Detailed plan for lifting the section, block or max block.
Lifting lug	Welded steel part where hoisting cables are docked.
Structural member	Support that is a constituent part of the structure
Trolley	Lifting beam system which is attached to the lifted structure with cables.

Abbreviations

COG	Center of gravity
FEA	Finite element analysis

Physical quantities

A_{cc}/A	Cross-section area
α	Tension or compression factor for welds
α_S	Shear factor for welds
α_{sling}	Sling angle or Angle of Loading
c_{cc}	Design width of a cross-section
d_{eye}	Perpendicular distance of the lifting lug's eye
δ_{ij}	Kronecker delta
E	Young's modulus
ϵ	Sling angle factor
ϵ	Strain tensor
η	Buckling utilization factor
η_{all}	Allowed buckling utilization factor
\bar{F}_j	Combined force in one trolley
$F_{tot, ll}$	Maximum force of lifting lug
F_{sling}	Force induced to sling
f_{end}	End constrain factor
f_y	Admissible stress of material
G	Shear modulus
g	Acceleration due to gravity, value 9.81 m/s^2
γ_{Mi}	Partial factor of resistance
I_i	Second momentum of area around axis i
I	Second momentum of area around the weakest axis
i_j	Radius of gyration in axis j
L_{Ei}	Buckling length in plane $x-i$
L_{ll}	Length of the lifting lug
l_{pill}	Length of the member
l_{weld}	Welding length
λ	Lamé constant
$\bar{\lambda}_j$	Slenderness around axis j
M_j	Combined momentum in trolley
μ	Lamé constant
$N_{c,Rd}$	Plastic resistance
N_{Ed}	Design resistance

(continues)

Physical quantities continues

n_{mc}	Number representing the total number of mounted cables
$n_{s,ll}$	Safety factor of lifting lug stress
ν	Poisson's ration
R_{eH}	Minimum yield stress of the considered member
$R_{eH,S}$	Specified minimum yield stress of the stiffener
$R_{perm,ll}$	Admissible stress of lifting lug
ρ	Mass density in kg/m^3
σ_{av}	Average axial compressive stress
σ_{cr}	Minimum critical buckling stress
σ_{Ec}	Elastic compressive column buckling stress
$\sigma_{i,Ed}$	Design stress of structural member in i -axis
$\sigma_{tot,ll}$	Maximum design stress of lifting lug
$\sigma_{wi,Ed}$	Design stress of weld in i -axis
t_{cc}	Design thickness of a cross-section
t_{ll}	Thickness of lifting lug
τ_{Ed}	Design shear stress of structural member
$\tau_{w,Ed}$	Design shear stress of weld
χ	Reduction factor
\bar{w}	Force in one trolley cable

1 PREFACE

1.1 Acknowledgements

This work has been carried out for Hoisting Design of the Department of Hull Detail Design at Meyer Turku Oy. The Hull Detail Design is acknowledged for its support to provide me with the opportunity to carry out this work using FEMAP.

I want to thank my managers, Head of Hull Detail Design Janne Merinen and Head of Hull Production Henrik Mantere who gave me the opportunity to be a part of the team. I also wish to thank the Hull Project Manager of the Carnival project Tuomo Hyvönen who took on the duty to be my mentor on behalf of Meyer Turku Oy. Special thanks to Tomi Salonen who helped me understand the process of hoisting design. Thanks to Vesa Platan who gave me a lot of information about the hoisting design and the gantry crane structures. Thanks to Matti Rautiainen who helped me create the trolley system and Heikki Lammi who provided me with the global model. Furthermore I wish to thank all my colleagues in Hull Detail Design who helped me with the glossary of ship building and have supported me in my everyday work.

1.2 Objectives of the thesis

To lift large steel structures, hoisting designer needs to know how to place lifting lugs to a structure in a way that girders and bulkheads do not undergo the stress levels and bending in which the steel might break or become deformed. Furthermore the breaking of the structure might cause severe injuries in production sites. The method of getting to the point where the lifting can be done safely and without damaging the lifted structure is the main objective of this thesis. In addition the modelling and analysis of the hoisting model should be made relatively easy.

To get to this several points are to be checked. For example, the center of mass and the mass distribution of the structure have to be known. Also the basic knowledge of the mechanical strength of the structure has to be evaluated. The hoisting model has to be created to get the base data required for analysis.

1.3 Hull production profile

Meyer Turku Oy is a ship building company that is currently specialized in cruise ships and ships of complex structures and systems. The company has undergone some major investments in the past five years and continues to grow as the production rate increases. One part of the company is the Hull Production and the Hull Detail Design is a part of this department. Hull Production produces hulls for the ships. Ship hull is assembled by lifting outfitted steel *blocks*. To transfer these blocks from one production facility to another they need to be transported. One part of this transportation process is to lift the block and attach it to rest of the ship's hull.

Hoisting Design as a part of the Hull Detail Design forms a Hoisting Plan that includes the lifting of the block to the hull. In order to do this the Hoist Designer needs a method to get required information from the CAD model of that block.

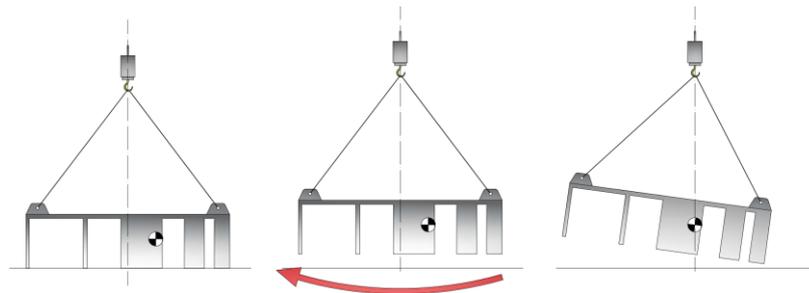
2 HOISTING DESIGNING

The main goal of hoisting design is to transport or lift large structures. Shifting has to be executed so that no significant damage is inflicted to the structure. Risks for health and safety also need to be evaluated.

2.1 Principles for lifting

According to Directive 2006/42/EC the lifting accessories must be designed so that the minimum value for stress safety coefficient of static loads is 1.5. For dynamic stress safety coefficient the minimum is generally 1.1. These values are valid for all machinery and lifting accessories that are operated manually. (Directive 2006/42/EC 2006, Annex I, Section 4.1.2.3.) No lifts should be made if the static or dynamic load of the lifted structure exceeds these values. Furthermore SFS 13155+A2 (2009, 33) standard requires that mechanical bearing parts as well as the lifted structure are designed to withstand two times the static load limit. Because of this lifting is designed with two times the acceleration due to gravity $g = 9.81 \text{ m/s}^2$.

When the lifting weights are estimated the calculations should also include the lifting gear. For example, if the maximum total load for a crane is 1200 tons and the lifting gears (including steel ropes, hooks, beams etc.) weigh 200 tons then the total net weight for the lifted structure is the maximum total load minus the weights of the lifting gears (1000 tons in this case). In addition to knowing the net weight, the distribution of the weight has to be calculated. For this the *center of gravity* (COG) needs to be calculated. If the COG is not below the main hook, the lifted body will start to tilt until the hook and the COG are in same vertical line (Picture 1).



Picture 1. Tilt of structure. If the COG is not under the hook, the lifted load starts to tilt until the COG is in the same vertical line with the hook.

Tilting might cause the welding of lifting lugs or the wire that attaches the hook to the lifting lug to break. It might also lead to the deformation of the lifted body. In worst case scenario the load will fall which might lead to serious injuries. Therefore the lifting is designed in a way that the attachment points for the wires are chosen so that the main hook is in the same vertical line as the COG. If a lifting beam is used instead of a single

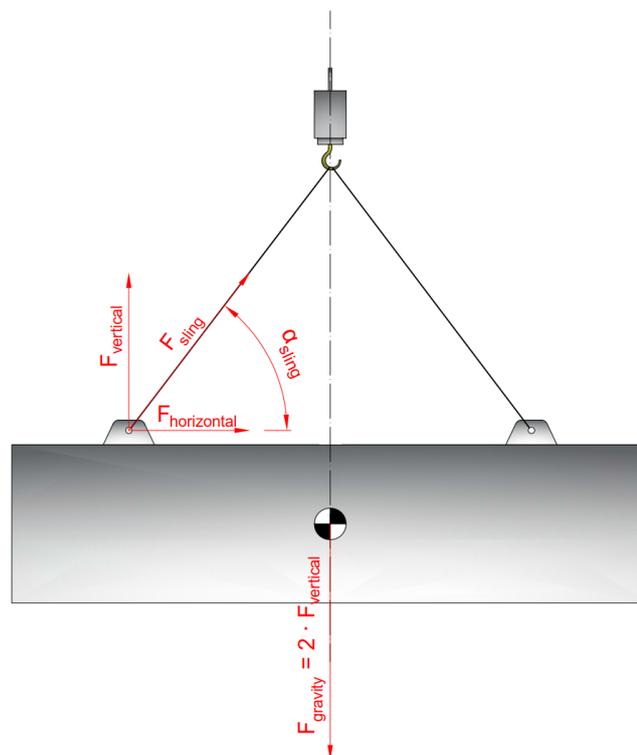
hook, the point that needs to be in the same vertical line with the COG is called the *point of suspension*.

Wires between hooks and lifting lugs are called *sling wires*. When the angle of the sling α_{sling} decreases in respect to the horizontal line, the load the lifting lug must carry increases in accordance to the equation 1 (Marine Insight 2018).

$$F_{sling} = \frac{F_{vertical}}{\sin(\alpha_{sling})}$$

Equation 1. Sling force (Marine Insight 2018)

Where $F_{vertical}$ is the load subjected to the sling when it is in 90° in respect to the horizontal line (Picture 2).



Picture 2. Force subjected to the sling wire

If the angle is too low it can cause the sling wire to be overloaded and it might break. The factor $1/\sin(\alpha_{sling})$ in equation 1 is called the *sling angle factor* and is denoted with ϵ (ASME 2018). It can be seen that with sling angle of 30° the sling angle factor is 2 and the slings loading capacity halves. In reality the lifting lugs are three dimensionally distributed and the weight of the load is usually not uniformly distributed.

2.1.1 Distribution of load

When calculating the COG for large structures we need to know all the masses, m_i , and COGs for single objects and use the equation 2 (Räisänen 2000).

$$COG = \frac{\sum_i^n COG_i m_i}{\sum_i^n m_i}$$

Equation 2. Center of gravity (Räisänen 2000)

Large structure can consist of over 1000 single bodies. Therefore the calculation is usually done with a computer. Determining the COG can be difficult if the masses and COG's for single components are not all known. For a single body with homogenous materials the center of gravity, COG_i , is equal to the body's geometrical centroid.

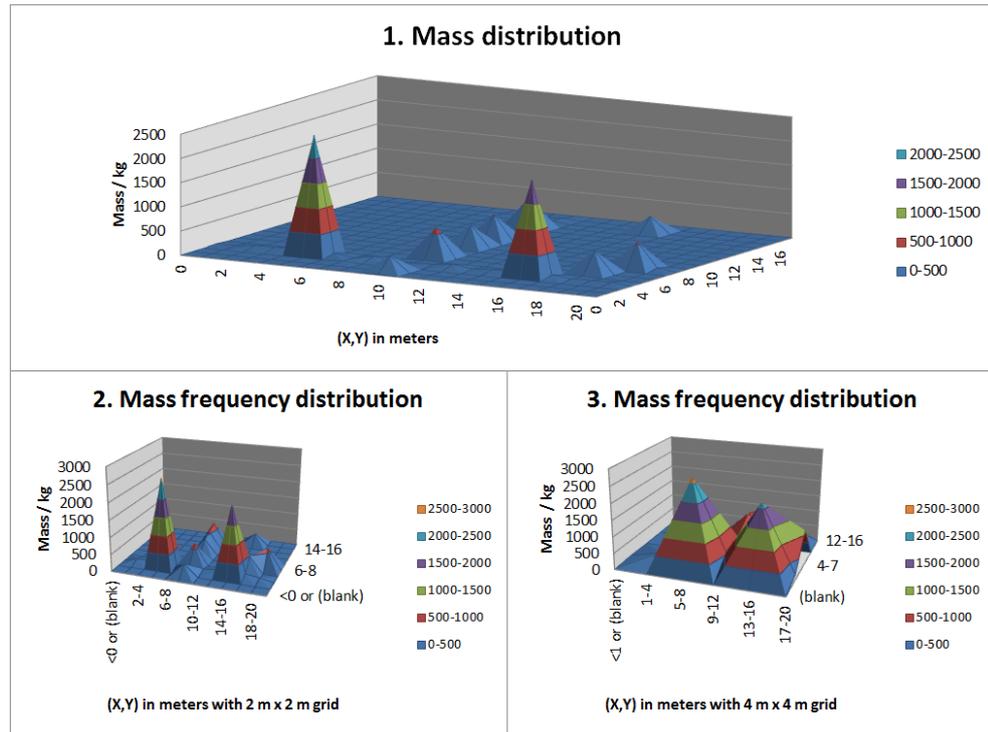
To simplify the problem of mass distribution, all single objects can be approximated as point objects with mass, m_i , and position, \bar{x} . The position \bar{x} in this case is the center of mass of that object. Furthermore the mass of a point object can be seen as a function of the position, for no two or more single objects can be in the same place. This can be written in the following form:

$$f_m: \bar{x} \rightarrow m$$

Equation 3. Discrete mass distribution

Where $\bar{x} \in \mathbb{R}^3$ and $m \in \mathbb{R}$. In this thesis f_m is called a function of *discrete mass distribution*, for it is clearly a distribution function. It tells how mass is distributed according to its' location. It's also a discrete function, for all the masses were approximated as point masses. By taking all vectors composed of coordinates \bar{x} and mass values m a *mass distribution* is acquired. This vector is denoted with $\bar{v} = (\bar{x}, m)$. Using only the result given by equation 3, the analysis of how much mass is distributed for example on a shell area of a block can be difficult. All the positions would have to be checked and only those located in or near the shell added together.

By grouping together masses that are within close proximity with each other, forms the *grid* of coordinates, thus making the readability of the data easier. When a mass is plotted using grid, the function is called *discrete mass frequency distribution*, noted with F_m . The accuracy can now be modified by modifying the density of the grid. Density can't be too low, because the distribution comes closer for being just one peak (Picture 3).



Picture 3. Example of using the discrete mass distribution equation.

To examine the distribution for example on plane, (Y, m) , coordinates $\bar{v} = (\bar{x}, m)$ need to be projected. Projection can be done in many different ways, but for mass distribution the *orthographic projection* is used in this thesis (equation 5), because it transforms not wanted coordinates to zero. This is done using 4×4 *orthographic matrix*, P_o , defined by equation 4 (Giovanni 2018).

$$P_o = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

Equation 4. Example of orthographic projection matrix

Transformation to plane is now simply the product of \bar{v} and P_o . The result of this product is denoted with v_{Ym} where index Ym is the plane of the projection.

$$\overline{v_{Ym}} = P_o \bar{v} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{pmatrix} x \\ y \\ z \\ m \end{pmatrix} = \begin{pmatrix} 0 \\ y \\ 0 \\ m \end{pmatrix}$$

Equation 5. Orthographic projection (Giovanni 2018)

2.1.2 Moment in structure

If the COG and the mass distribution are plotted in a same diagram, it shows how much the structure would bend if it was to hang freely from the COG. For calculating the bending, the bending moment for each point of mass needs to be calculated. In freely hanging balance only gravity affects the structure. Therefore for a single body mass the force, G_m , affecting to it is:

$$G_m = m \cdot g$$

Equation 6. Body mass force

where $g = 9.81 \text{ m/s}^2$ is the standard acceleration due to gravity. Moment M_m in respect to GOC is now:

$$M_m = G_m \cdot (COG - \bar{x})$$

Equation 7. Body moment

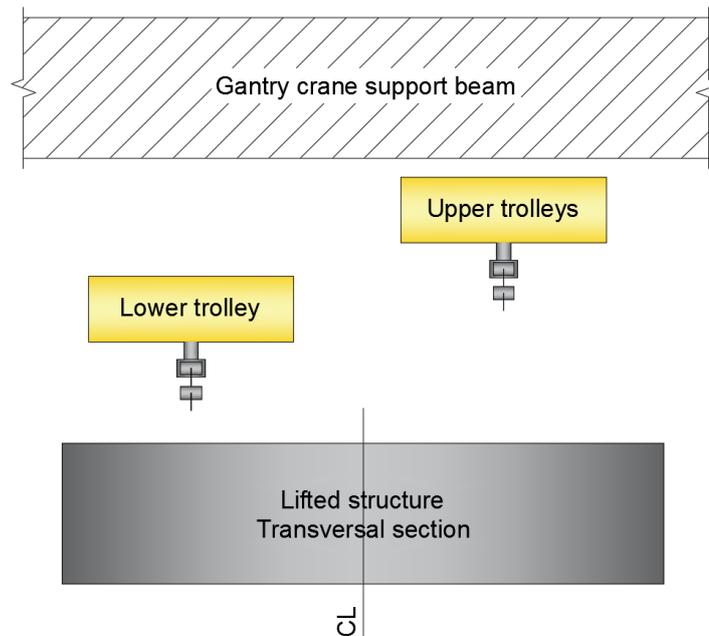
In reality lifting lugs are welded to the structure and through wires the force is conducted to the main lifting beam. Furthermore the lifting lugs are not typically located on the same vertical line as the COG and the point of suspension. Additionally the point of interest is not usually just the moment respect to the COG but globally around the lifting lugs.

Apart from the location of COG and moments in the structure the point of interest is forces and directions of forces. Moments and forces together causes stress to the structure. From equations 6 and 7 it is clear that these moments and forces are the result of gravity. In other words if the stresses in structure are known forces and moments can be solved. Functions that need to be solved depend on the situation and therefore the initial values from which they are solved. These initial values were one of the key interests that were determined using solver NX Nastran in FEMAP (Rautiainen 2019).

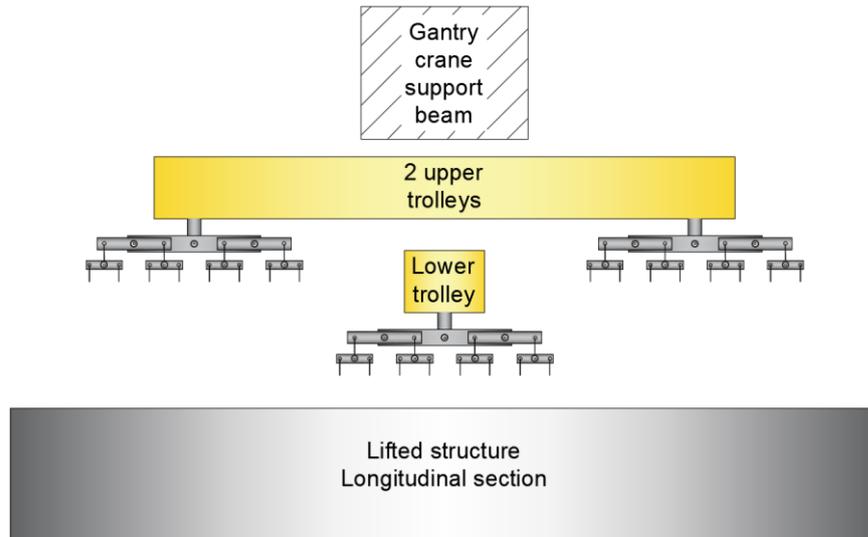
To be able to get reliable initial values the lifting setup should be known and modelled, for it dictates how the structure is attached, how many mount points there are and how these points are distributed. In other words, what kind of a trolley system setup is used?

2.2 Trolley system setup

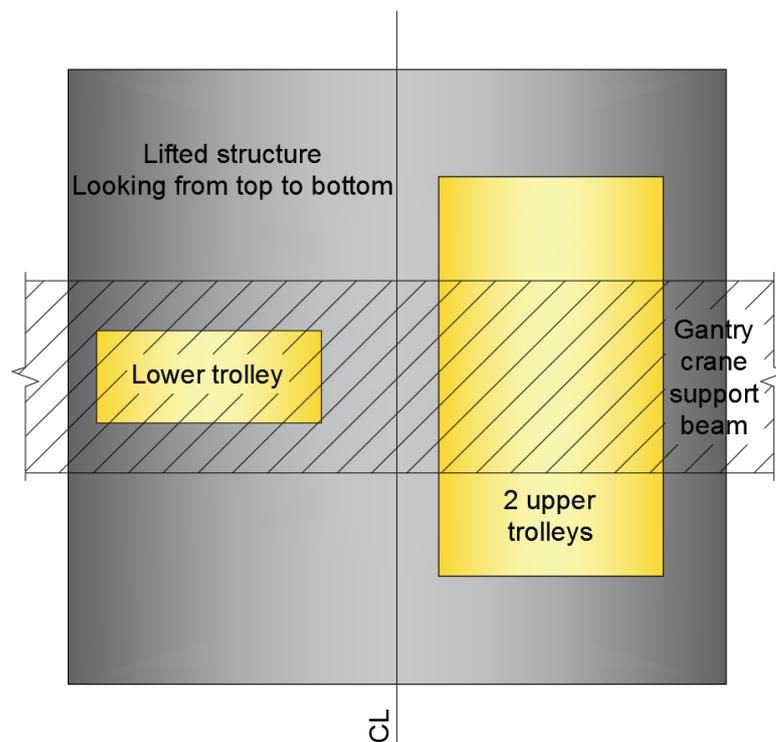
Setup was consisted of three trolleys with eight attaching point each, totaling a maximum of 24 attaching points. Each of these trolleys with necessary dimensions was modelled to FEMAP. In pictures 4, 5 and 6 the principle of the trolley system is shown. Pictures are shown in ship coordinate system. Upper trolleys were forced to move in same line with each other while the lower trolley was able to move.



Picture 4. Transversal section of the trolley system setup.



Picture 5. Longitudinal section of the trolley system setup. Upper trolleys move in same line with each other. This is due to the carriage that unites them.



Picture 6. Top view of the trolley system setup. The carriages that move the trolleys are attached to the gantry crane main support beam.

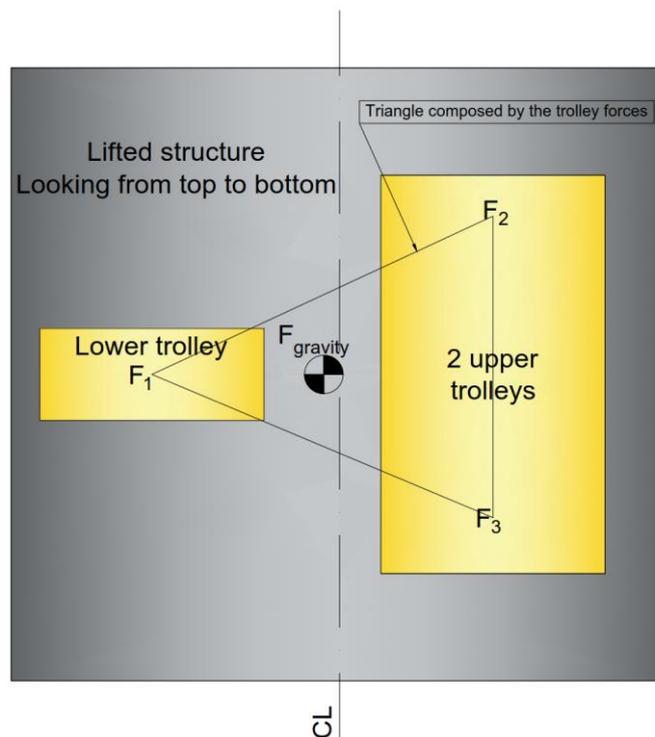
2.2.1 Trolley system stability

Analogous reasoning can be made to the demand between the trolley system and the structure's COG in comparison to picture 1 of section 2.1 for the simple one hook system. Boundary condition in this is also the demand that the lifted structure is not allowed to tilt. To simplify this problem the trolley system can be handled as a four force system with one force acting down (gravity) and three acting up (trolleys). This system is stable when equation 8 is valid.

$$\begin{aligned} \sum M_i &= M_{gravity} + \sum M_j = 0 \\ \sum \vec{F}_i &= \vec{F}_{gravity} + \sum \vec{F}_j = 0 \end{aligned}$$

Equation 8. Stability condition of the system

In the equation above $j \in \{1,2,3\}$ represents the forces in trolleys. This equation constrains the forces so that the center of gravity will set somewhere inside the triangle composed of said forces \vec{F}_j (Picture 7).



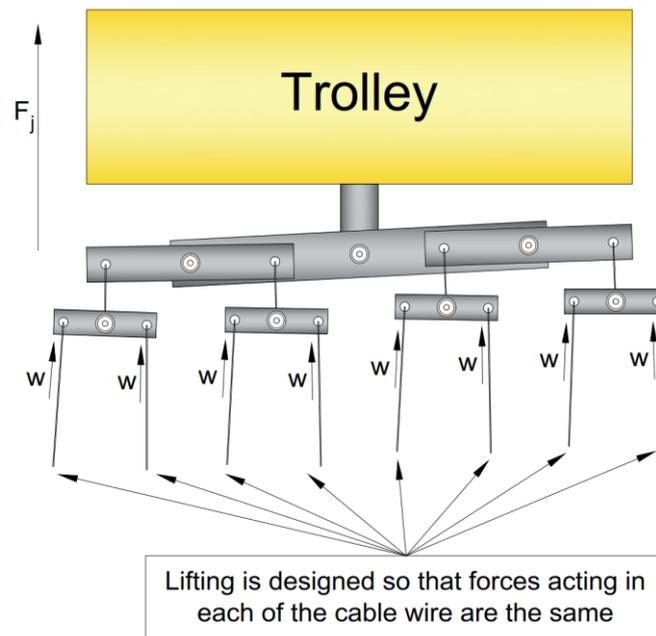
Picture 7. Triangle composed by the trolley forces. The center of gravity will land somewhere inside this triangle

For a trolley the mounting point forces need to be the same. In other words:

$$\bar{F}_j = n_{mc}\bar{w}$$

Equation 9. Total force of the trolley

In the equation above n_{mc} is the total number of mounted cables and \bar{w} is the force acting in one cable with the assumption that friction is negligible. With this assumption $|\bar{w}|$ is same for all cables (Picture 8). This assumption leads to the total moment around the main articulation (Picture 15) to be zero when the trolley is settled down to balance.



Picture 8. Force acting in each cable wire of the trolley

The trolley used in design is allowed to tilt from the main articulation (also called the center of rotation). The maximum horizontal tilt the trolley must withstand is indicated by the manufacturer and can vary between different cranes. Lifting beams intended only for horizontal use must withstand a tolerance of 6° from the horizontal. This is also a requirement for devices that hold the moving parts in position. These devices need to be designed so that they work even if the maximum tilt angle indicated by the manufacturer is surpassed the required 6° . (SFS-EN 13155+A2 2009, 43.)

3 FINITE ELEMENT ANALYSIS

To understand structural effects of hoisting, values for stress, strain and deformation can be evaluated with computational model.

For this the differential equations for these quantities will need be solved (Saratech 2018). Formulation of these equations is presented in sections 3.1 - 3.3. FEMAP in this work was used to solve these equations. The *Equation for Deformation* presented in section 3.1 tells the stress-strain relations. The *Constitutive Equation* presented in section 3.2 is used to determine the stresses in structure as a function of the body's deformation. Finally the *Equation of Equilibrium* presented in section 3.3 gives the stress gradients. In practice the last equation shows how rapidly the stresses are changing. In section 3.4 FEA analysis process is explained and different element types used in model are introduced.

3.1 Equation for deformation

Considering the cuboid with one corner in coordinates (x, y, z) undergoes the displacement $u_i(x, y, z)$ where i represents the coordinate axis and with edge lengths of dx , dy and dz . Edges also undergo deformation. The displacement of edges are denoted for x-axis with $u_x(x + dx, y, z)$, for y-axis with $u_y(x, y + dy, z)$ and with $u_z(x, y, z + dz)$ for z-axis. Let's consider the normal strain ϵ_i in just one direction, for example in x-direction denoted with ϵ_x . Equation 10 is then valid (Kelly 2015).

$$\epsilon_x = \frac{u_x(x + dx, y, z) - u(x, y, z)}{dx}$$

Equation 10. Normal strain definition (Kelly 2015)

The right hand side of the equation above is by definition the partial derivative of u_x in respect to x .

$$\epsilon_x = \frac{\partial u_x}{\partial x}$$

Equation 11. Normal strain differentiation

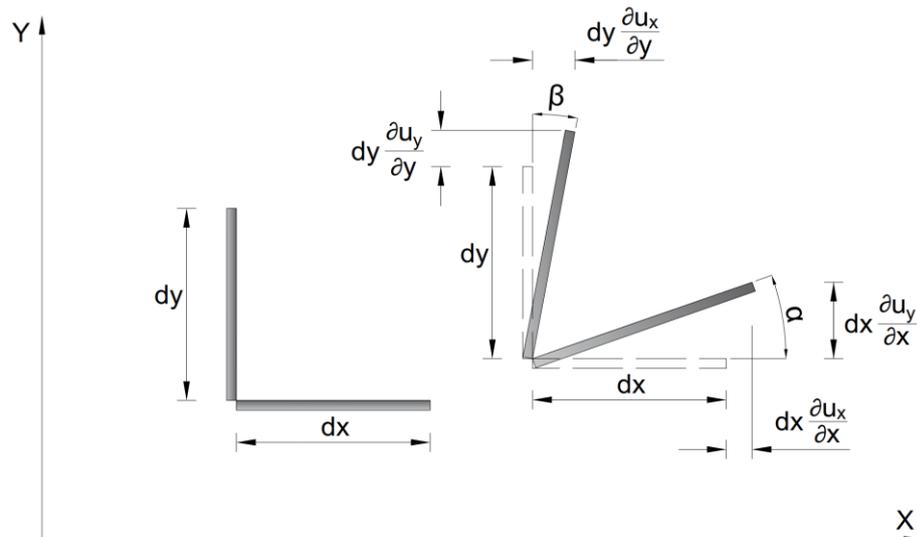
For other directions the strain becomes:

$$\begin{aligned}\epsilon_x &= \frac{\partial u_x}{\partial x} \equiv \epsilon_{xx} \\ \epsilon_y &= \frac{\partial u_y}{\partial y} \equiv \epsilon_{yy} \\ \epsilon_z &= \frac{\partial u_z}{\partial z} \equiv \epsilon_{zz}\end{aligned}$$

Equation 12. Normal strain in all directions

Equations in 12 are written with double indexes to notify that equations are meant for normal strain. In addition to the equations in 12 also the shear strains are needed. This is done by taking the cross terms into account.

Consider the 2-dimension case of two lines (picture 9). One line is in the x-direction and one in the y-direction with lengths dx and dy . As they undergo deformation caused by shear and normal force, two angles denoted with α and β are generated between the non-deformed and deformed state. (Kelly 2015.)



Picture 9. Deformation by shear and normal force.

Shear strain γ_{xy} can be defined using the angles α and β as the total angle of change:

$$\gamma_{xy} = \alpha + \beta \approx \tan \alpha + \tan \beta$$

Equation 13. Shear strain definition (Kelly 2015)

Equation 13 is valid for small angles. When inspecting only a small displacement, combining picture 9 with equation 13, the shear strain γ_{xy} can be written in form:

$$\gamma_{xy} = \frac{\partial u_y}{\partial x} + \frac{\partial u_x}{\partial y}$$

Equation 14. Shear strain differentiation (Kelly 2015)

This can be generalized to consider also (X,Z)- and (Y,Z)-planes.

$$\begin{aligned}\gamma_{xy} = \gamma_{yx} &= \frac{\partial u_y}{\partial x} + \frac{\partial u_x}{\partial y} \\ \gamma_{xz} = \gamma_{zx} &= \frac{\partial u_z}{\partial x} + \frac{\partial u_x}{\partial z} \\ \gamma_{yz} = \gamma_{zy} &= \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial y}\end{aligned}$$

Equation 15. Shear strain for all planes

Normal and shear strains shown in equations 12 and 15 are called *engineering strains* (Lagace 2001). If written in form of *strain tensor*, number 1/2 is added to shear strains. Normal strains are same for engineering strains and for strain tensor. Strain tensor is shown in equation 16. This is the first of three equations that is solved in FEMAP (Sarattech 2018).

$$\epsilon = \begin{bmatrix} \epsilon_{xx} & \epsilon_{xy} & \epsilon_{xz} \\ \epsilon_{yx} & \epsilon_{yy} & \epsilon_{yz} \\ \epsilon_{zx} & \epsilon_{zy} & \epsilon_{zz} \end{bmatrix} = \begin{bmatrix} \epsilon_{xx} & \frac{1}{2}\gamma_{xy} & \frac{1}{2}\gamma_{xz} \\ \frac{1}{2}\gamma_{yx} & \epsilon_{yy} & \frac{1}{2}\gamma_{yz} \\ \frac{1}{2}\gamma_{zx} & \frac{1}{2}\gamma_{zy} & \frac{1}{2}\epsilon_{zz} \end{bmatrix}$$

Equation 16. Strain tensor (Lagace 2001)

3.2 Constitutive equation

For solid materials that are isotropic and homogenous the stress-strain relation can be written in the form of equation 17 (Heil 2010).

$$\sigma_{ij} = 2\mu\epsilon_{ij} + \lambda\epsilon_{kk}\delta_{ij}$$

Equation 17. Constitutive equation (Heil 2010)

Where μ and λ are *Lamé constants*. For solid materials the constant μ is the shear modulus denoted with G . Symbol δ_{ij} is Kronecker delta (Wolfram Research 2019). Equation 17 is called *Constitutive equation*.

Using elastic constants E and ν where E being the Young's modulus and ν being the Poisson's ration, the equation 17 can be rewritten in a new form (Heil 2010).

$$\sigma_{ij} = \frac{E}{(1 + \nu)} \epsilon_{ij} + \frac{E\nu}{(1 + \nu)(1 - 2\nu)} \epsilon_{kk} \delta_{ij}$$

Equation 18. Stress-strain relation (Heil 2010)

The constitutive equations shown in equations 17 and 18 are just two examples of constitutive equations. In case of solid materials with linear elastic properties constitutive equation is reduced to Hooke's law.

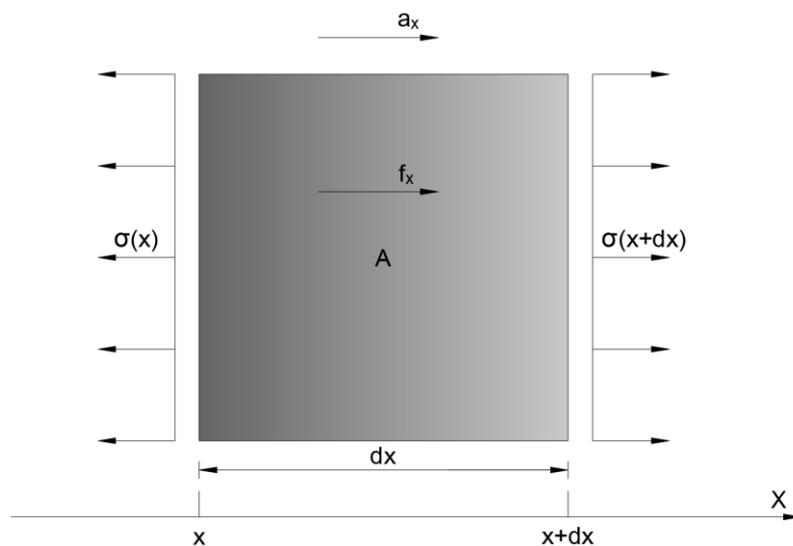
3.3 The Equation of Equilibrium

To get the force in any portion of the given object the Newtonian force equation needs to be constructed and solved. General Newton's equation for force F_i and acceleration a_i is:

$$\sum_i^n \bar{F}_i = m\bar{a}_i$$

Equation 19. Newton's second law

Consider the equation above in a simple one-dimensional case with body's length of dx and area of A . Stresses are noted with $\sigma(x)$ and $\sigma(x + dx)$. In addition the average force per unit volume is f_x . At this point there are no restrictions for the body's movements, so said body can be in an accelerating motion. Consider this average acceleration be a_x (Picture 10).



Picture 10. One-dimensional body element that is under the influence of body force.

Volume V of the object in picture 10 is $A dx$. If the body's density is ρ_{body} , then the equation 19 can be written in form:

$$\sigma(x + dx)A - \sigma(x)A + f_x A dx = \rho_{body} A dx a_x$$

Equation 20. Force equation for stress and movement of body

By dividing with dx and substituting the area A , it can be seen that the left hand side is the *stress gradient* $d\sigma/dx$ plus the force f_x . The right hand side represents the body's movement. This is the *Equation of Motion in one-dimension*. (Kelly 2015.)

$$\frac{d\sigma}{dx} + f_x = \rho_{body} a_x$$

Equation 21. Equation of Motion in one-dimension (Kelly 2015)

By converting this to a 3-dimensional case the general *Equation of Motion in 3-dimension* can be written. (Kelly 2015; Saratech 2018)

$$\frac{\partial \sigma_{ix}}{\partial x} + \frac{\partial \sigma_{iy}}{\partial y} + \frac{\partial \sigma_{iz}}{\partial z} + f_i = \rho_{body} a_i$$

Equation 22. Equation of Motion in 3-dimension (Kelly 2015)

In above equation $i \in \{x, y, z\}$. If the movement of body is restricted so that it is not moving or the acceleration is negligible, that is $a_i = 0$ for all $i \in \{x, y, z\}$, then the equation 22 is reduced to *Equation of Equilibrium* shown below.

$$\frac{\partial \sigma_{ix}}{\partial x} + \frac{\partial \sigma_{iy}}{\partial y} + \frac{\partial \sigma_{iz}}{\partial z} + f_i = 0$$

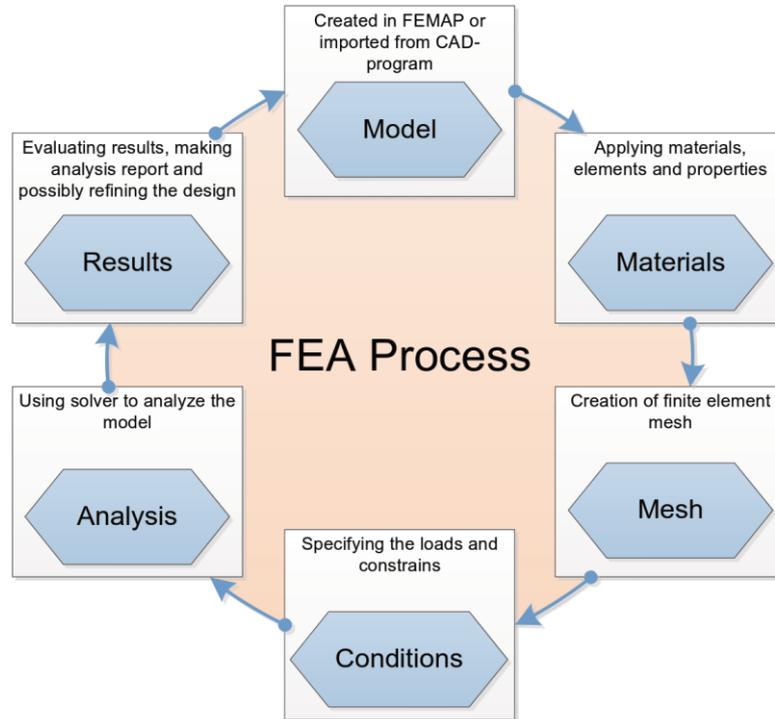
Equation 23. Equation of Equilibrium (Kelly 2015)

The force f_i in hoisting is usually the combined force of gravitation and non-structural loads.

The equations 16, 17 and 23 will need to be solved when analyzing the deformation and stresses of the body (Saratech 2018). In FEMAP this can be done by using the solver NX Nastran. It can be seen from these equations that different material properties (density and elastic constants) affect the behavior of the body under load. Property values can be given to FEMAP in different ways depending on the element/property type that is used. To clarify the basic process cycle which includes the selection of materials and element types, FEMAP's FEA process needs to be understood.

3.4 Finite element analysis process in FEMAP

In sections 3.1 - 3.3 the physical background of solving the model in given conditions was reviewed. To get to the point where this can be done the model needs to be created in FEMAP (or imported from other CAD-program), materials need to be chosen and boundary conditions need to be determined. For the analysis results to be realistic, it is crucial that the boundary conditions (loads and constrains) are correct. Simple FEA process according to Saratech (2018) is illustrated in picture 11.



Picture 11. FEA process from creating a model to the results (Saratech 2018)

The process in this thesis follows the picture above closely with the difference that the model which was used was mostly complete and the materials were already applied to the model. Model was also already meshed using large mesh size by the original creators of the model.

It should be noted that different material choices greatly affect how the model behaves. For this the materials used in the model of this thesis are reviewed in section 3.4.1.

3.4.1 Materials in the model

The two different material types which were used in the model are listed in table 1. The materials used were assumed to be linearly elastic, and therefore for instance the temperature dependency of the properties of those materials was not acknowledged.

Table 1. Materials used in the global model

Material name in FEMAP	Properties	Used for
Isotropic	Uniform properties in all directions	Profiles
2D Orthotropic	Anisotropic properties in two different directions	Plates

For *isotropic* and *orthotropic materials* FEMAP allows to specify many different properties. Material properties that are important when examining the load effect on steel structures are Young's modulus E , Shear modulus G , Poisson's ration ν and mass density ρ . In addition to these the limit normal and shear stresses should be determined. Units that were used are listed in table 2.

Table 2. Physical quantities to be determined and the units used. Name in FEMAP is also listed if it differs from the standard name.

Physical quantity	Name in FEMAP if different	Marking	Unit
Young's modulus		E	Pa = kg/ms ²
Shear modulus		G	Pa = kg/ms ²
Poisson ration		ν	Unitless
Density		ρ	kg/m ³
Yield strength, for normal stress	Tension	$R_{e,normal}$	N/mm ²
Yield strength, for shear stress	Shear	$R_{e,shear}$	N/mm ²

Ultimate yield strength is not determined because allowed stresses in structure are all in elastic domain. For limit stresses units that were used are N/mm². These differ from other units and are determined for checking purposes only. All the calculations done by FEMAP are based on first four properties listed in table 2 so limit stresses are not affecting the results.

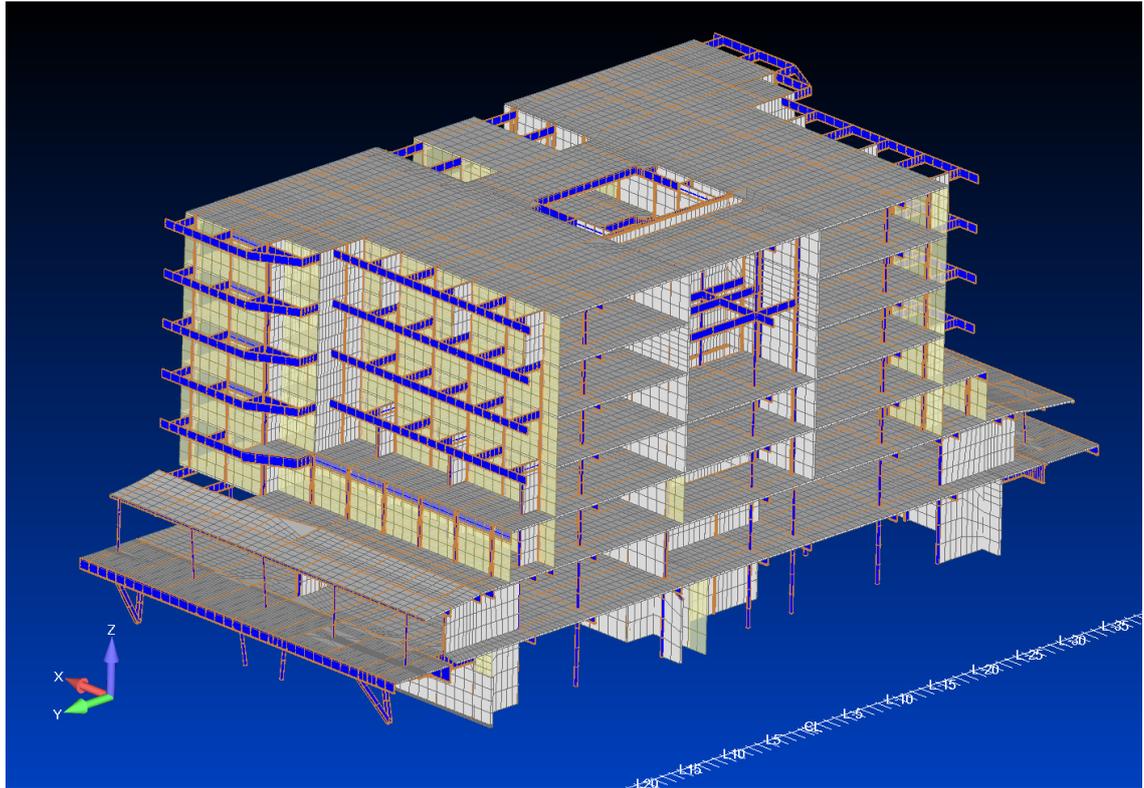
For isotropic materials an example of defining the material properties is illustrated in picture 12. For orthotropic materials the principle is the same, but properties are given for both material directions.

Property	Value
Young's Modulus, E	2,06E+11
Shear Modulus, G	7,9231E+10
Poisson's Ratio, nu	0,3
Expansion Coeff, a	0,
Conductivity, k	0,
Specific Heat, Cp	0,
Heat Generation Factor	0,
Limit Stress - Tension	235,
Limit Stress - Compression	0,
Limit Stress - Shear	0,
Mass Density	9000,
Damping, 2C/Co	0,
Reference Temp	0,

Picture 12. Example of defining the material properties in FEMAP

3.4.2 FEMAP model

The model used was part of the *global model* of a ship. Global model is a complete ship models with errors in model are due to scalability etc. It is used to calculate the quantities like hogging, sagging, global forces, buckling and vibration. Therefore it can be utilized to model the loads taking place during production. Respectively a *local model* refers to the one section of the global model with. The local model used in this work is shown in picture 13. *Hoisting model* refers to the model with the trolley system reviewed in section 2.2 and essential boundary conditions taken into account. Hoisting model is reviewed in section 6. Global model was used because it was already modelled. It should be noted that global model (and local model respectively) is very coarse for it has a very large mesh size and it doesn't give detailed information of the structures. For example large holes are shown but smaller are not and some structures might be incorrect.



Picture 13. Local model to which the finite element analysis was done

In table 3 some items that should be noted and possibly fixed before the model can be used as a basis for analysis are listed. Item in the table and the ways of fixing them are reviewed in section 3.4.3. The reason that the global model lacks some of these items is that the model is built for other purpose than hoisting design. For this reason demand of analyzing the FEA results increases (Rautiainen 2019).

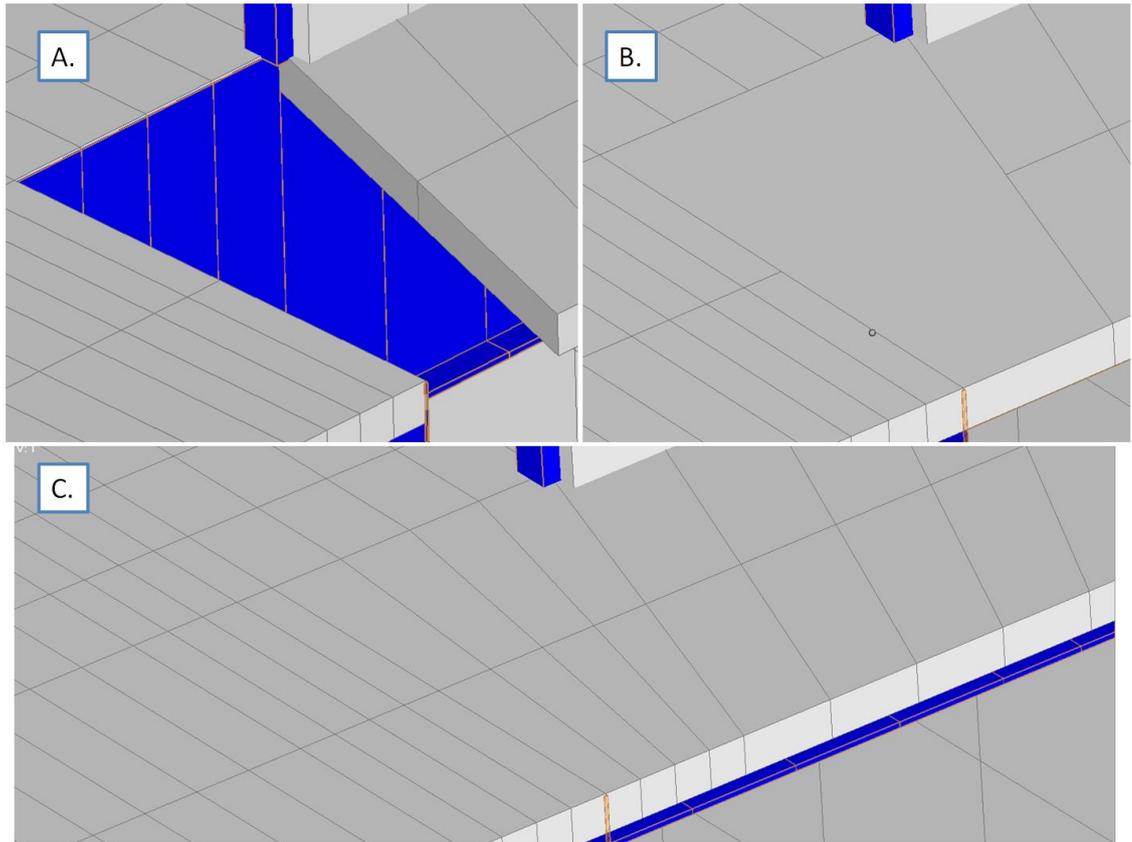
Table 3. Items that should be noted and checked from the model before analysis

Item	Description	Fixing proposal
Missing structures	Plates, beams and pillars that are not in model but are in actual structure	Structures to be added to the model. Copy element command should be used when possible.
Additional structures	Structures that are in FEMAP model but are not included in hoisted structure	These structures should be removed.
Missing openings	Most holes are missing from the model. For example doors and windows in bulkheads, shell and holes in beams are not in the model.	Openings should be modelled locally or by taking it in consideration with mass density.
Wrong mass/density	Some element might have wrong mass/density value for hoisting design purpose.	Material properties should be fixed so this is correct.
Mesh size	Global model and therefore local model has very large mesh size.	Mesh size should be reduced so that model can be made to correct size.

3.4.3 Ways of fixing the model

1. Missing structures

All the missing elements were added. In most cases the simplest way to do this is to use *copy element command*. After elements are added the mesh size should be made equivalent to adjacent structures. Picture 14 illustrates an example of missing structure being plate.



Picture 14. Example of missing structures. A. represent the missing plate. In B. the plate is added and in C. the mesh size is reduced so that the result is equivalent to the adjacent structures.

2. Additional structures

The local model should represent the actual production model as accurately as possible. In a case of additional structures in the model, these structures should be removed.

3. Missing opening

If it can be seen before the analysis that stress peaks will emerge around the opening, it should be added. While most of the openings were missing from the local model, these were taken into consideration by mass density and by using 2D orthotropic materials. When analysis result pinpoints high stress peaks in a structure where there should be an opening, a more accurate model from that area with same forces can be made.

4. Wrong mass/density

Mass density should include the weight of the mass, weldments and possible simplifications of the model. If the model for example doesn't contain openings, these openings should be taken into consideration in mass density. One way of doing this is by comparing the area with and without openings.

5. Mesh size

General mesh size was chosen to be 0.64 m between nodes. This size was chosen because it allowed the removing of excessive elements that were not a part of the original structure. Size of the mesh was not constant throughout the model for it had already being pre-meshed by its original creators.

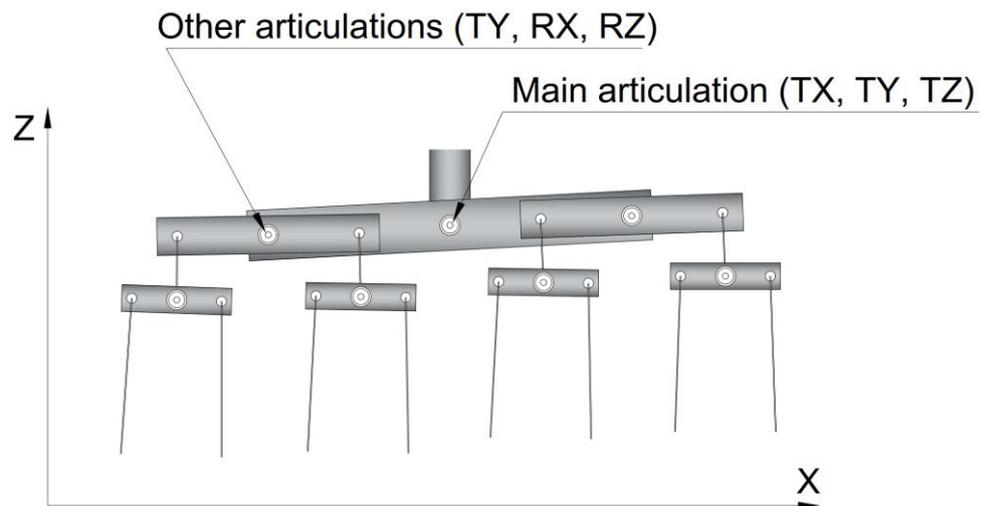
3.4.4 Boundary conditions

Standard types to set constrains are *Fixed*, *Pinned* and *No Rotation*. These constrain are usually tied to the global coordination system. Fixed constrain restricts the whole structure movement, while Pinned allows the structure to rotate. No Rotation is opposite to Pinned. It allows the translation of the structure but rotation is restricted. In addition to standard types, constrains can be given for arbitrary coordinates. For translation these are marked with *TX*, *TY* and *TZ*. Respectively for rotation these are marked with *RX*, *RY* and *RZ*. In total there are six nodal degrees of freedom and any combination can be selected. If no constrains are selected, the body is said to be *Free*.

Table 4. Standard types of nodal degrees of freedom in FEMAP, where X indicate constrained degree of freedom.

Type	TX	TY	TZ	RX	RY	RZ
Free						
Fixed	X	X	X	X	X	X
Pinned	X	X	X			
No Rotations				X	X	X

There are crucial boundary conditions for the trolley system and for the local model. The trolley system should be allowed to rotate, but no translations are allowed. This is due to the fact that the trolley is attached to the gantry crane main supporting beam. It can be seen from table 4 that no translation happens when the trolley's boundary condition is Pinned. The Pinned condition is actually valid only for the main articulation of the trolley. Other articulations constraint depends solely on how they are allowed to move and in what orientation the trolleys are modelled (Picture 15). In the picture below is one example of articulation constraints for trolleys.



Picture 15. Example of constraints of Main articulation (center of rotation) and other articulations. Constraint of other articulation is called Y Symmetric.

In addition to the trolleys the local model should be constrained. Constraint should not allow the model to move in X or Y-direction but allows it to move in Z-direction. Rotation is usually allowed. In order to do this, three nodal points of the structure are constrained: one in X direction and two in Y direction. These nodes should be chosen so that minimum amount of stress is focused on them (Rautiainen 2019).

3.4.5 Analyzing results

When using materials listed in table 1 NASTRAN NX allows the examining of over 90 different output vectors including *Translation*, *Applied forces*, *Constraint forces*, *Moments*, *Stresses* and *Bending moments*. What output vector should be viewed typically depends on the viewed structure and also the roughness of the model. For the local model mostly output vectors of forces, moments and translations were used. It should be noted that output vectors usually serve as initial values for the manual calculations for structures that are welded with *under matching welds*. When analyzing structures with *matching welds*, can the output vectors sometimes be used as a final result.

In structures like in picture 13 *fillet welds*, *butt joints*, *full penetration*, *partial penetration welds* and *intermitted welds* are used. From these five weld types, the fillet weld is most common. Fillet welds are used on T joints. These are typically bulkheads to deck joints, girders and transversals to deck joints. Butt welds are used on internal seams of plates. For more critical structures joining is done with a full penetration or a partial penetration weld. From these weld types the fillet and intermitted welds can typically be under matching.

This splits the analysis into two different categories depending on what is examined. One category is the durability of the basic material and the other is the durability of the welds. For basic materials this can be read straight from the output vectors while durability of welds needs to be calculated based on the output vectors.

Under matching welds are typically the weakest points in structure. Keeping that in mind calculations are carried out for fillet welds in chapter 4. If the analysis yields results of too high stresses or translations, additional supporting structures are to be placed.

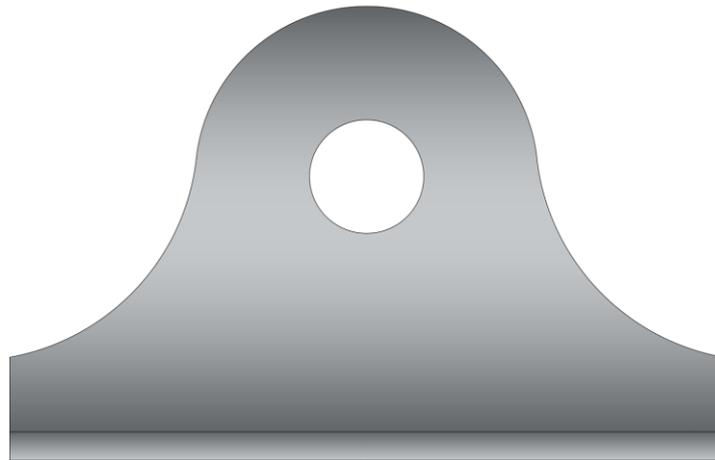
4 WELD ANALYSIS

In Meyer Turku, structural steel material qualities used in steel production are based on ISO 3834 and fulfilled in IACS regulations and EN grades (SFS-EN ISO 3834 2006). Material qualities used in this work are shown in Annex 1. Aluminum is also used in some projects for special places in a superstructure. Hoisting design is typically done so that all structures are assumed with steel quality of A or A36. The material of which the structures are made of is called the *basic material*. Due to material thickness in new ships is decreasing, A36 quality is replacing the normally used A quality.

Before the lifting can take place the lifting lugs are welded. In lifts safety factor, $n_{s,ll}$, for the lifting lug's weldings is typically 4. Therefore 1 cm² welding area corresponds to approximately 1 ton of weight. Using this useful rule of thumb it is easy to determine the lifting lug needed. (Platan 2018.)

4.1 Lifting lugs

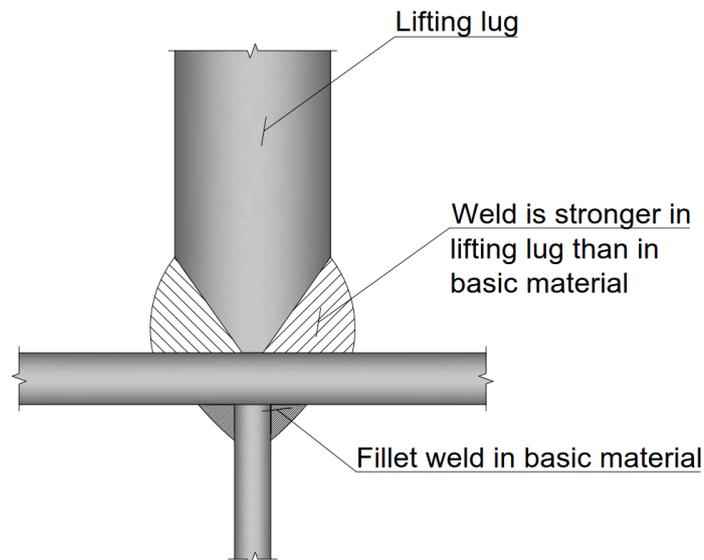
Lifting lugs are welded to be a part of the structure. After the lift is completed, lifting lugs are removed. Various types are used, but most crucial feature is the welding length, l_{weld} , of the lug. It dictates the lifting weight the lug can carry. In picture 16 the typical lifting lug is shown.



Picture 16. Typical lifting lug (padeye type)

4.1.1 Welding design for lifting lugs

The best way to ensure that the welding of a lifting lug is stronger than of the basic material is to use full penetration weld (Picture 17).



Picture 17. Example of full penetration welding in lifting lug

Other advantage of a full penetration weld is that the welds in the basic material and in the lifting lug can be aligned.

The structure behind the plate of which the lifting lug is installed is typically a girder, a web or a bulkhead with a-measure being typically less than 10 mm. This leads to the effective weld area of the lifting lug in case of full penetration weld being larger than that of the background plate's. Therefore the background plates' thickness can be used to determine the amount of weight the lifting lug can carry. In case of a larger web thickness or with bulkheads with large thickness the weld size of bottom plate is increased. In addition the weld length of the bottom plate is longer than in lifting lug. The weld ratio needs to be correct to ensure the durability of the lifted structure.

If the structural design is done before the hoisting design, additional support (brackets, thickening plates etc.) can be added below the attachment point of the lifting lug to make the structure stronger. In an ideal case the demand of hoisting should be taken into consideration at the structural design point. This can be difficult if the schedule interval between the structural design and hoisting design is large.

4.1.2 Weld strength analysis of lifting lugs

Modelling lifting lugs is not a mandatory part of the local model. If the modelling is not done the necessity to analyze the lifting lug is increased. From this analysis, the kind of lifting lug to be used can be determined more accurately. Initial values in this case are the angle (α_{sling}) and force (F_{sling}) in the sling. Additionally the effective weld measure s that in case of partial penetration weld is the thickness t_{ll} of the lifting lug is needed and also the length of the weld l_{weld} that can be approximated with the length of the lifting lug L_{ll} . By assuming that the force is distributed evenly to the lifting lug, the maximum stress can be calculated.

Because of the safety factor $n_{s,ll} = 4$ permitted maximum permitted stress $R_{perm,ll}$ is 89 MPa. Permitted stress is calculated from table of Annex I for A36 material. The point of interest this case is the maximum pulling force in one of the lifting lugs corners (Pictures 18 and 19). For lifting lug with thickness of t_{ll} the maximum stress $\sigma_{tot,ll}$ and force $F_{tot,ll}$ at the point of interest are presented in equations 24 and 25. These equations are valid when the cross section of the attachment surface is rectangular. Equation for the stress is:

$$\sigma_{tot,ll} = \frac{F_{sling}}{t_{ll}L_{ll}} \left(\sin \alpha_{sling} + \frac{6 \cos \alpha_{sling} \cdot d_{eye}}{L_{ll}} \right)$$

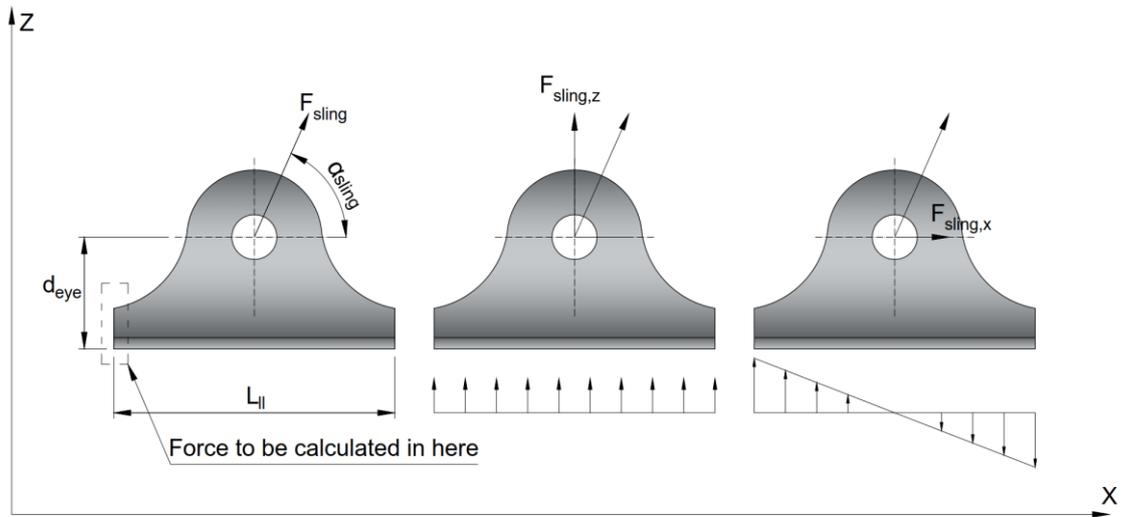
Equation 24. Maximum stress in lifting lug

where d_{eye} is the perpendicular distance of the lifting lug's eye (picture 18). The equation above is derived from the definitions of the normal stress and shear stress. Force equation follows from the equation 24.

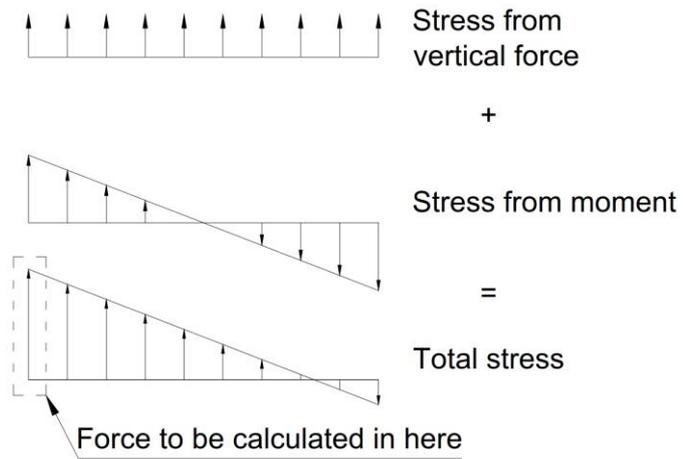
$$F_{tot,ll} = \sigma_{tot,ll} \cdot t_{ll} \cdot \Delta l_{ll} = \frac{F_{sling} \cdot \Delta l_{ll}}{L_{ll}} \left(\sin \alpha_{sling} + \frac{6 \cos \alpha_{sling} \cdot d_{eye}}{L_{ll}} \right)$$

Equation 25. Maximum force in lifting lug

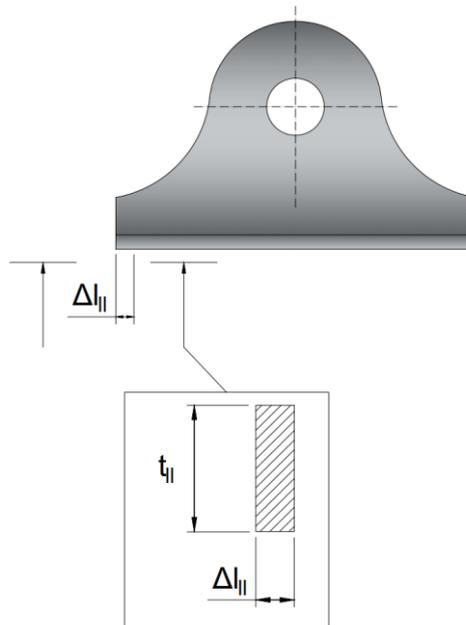
In equation 25 value $t_{ll} \cdot \Delta l_{ll}$ represents a small area in where the force is calculated (Picture 20).



Picture 18. Force and total stress is calculated in the shown point of interest.



Picture 19. Total stress in lifting lug



Picture 20. Area from where the force is calculated

If the total stress $\sigma_{tot,ll} \leq R_{perm,ll}$ then also the force is in the permitted limits of the lifting lug. Equations 24 and 25 give the information if the lifting lug can carry the permitted load and can be used to evaluate the size of the lifting lug to be installed. For a lifted structure to withstand the lift, the welds in it must also be evaluated.

4.2 Lifted structure

The evaluation of every weld can be done by direct calculation using basic analysis formulation shown in sections 3.1, 3.2 and 3.3. This may turn out to be difficult and time consuming when the system is large and complex because every weld should be modelled and evaluated one by one. A way to bypass this a bit is to make calculations for every typical structure that occur. This means categorizing every structure. This categorizing will however take time if no typical standard structures are known. Standards from different classification societies and directives provide a broad outlook to how the categorizing should be done.

For example Eurocode 3 gives direct instructions to cross-section classification limited by its local buckling resistance (EN 1993-1-1 2004, section 5.5.2).

After the categorizing is done one way or the other, then for example local forces and moments can be used to evaluate the durability of the welds in different categories. In SFS-EN 13155+A2 (2009, 71) this is done by calculating the equation A2.1 shown in Annex 2 in case of the welds. In case of welds the equation A2.1 can be written in way shown in equation 26.

$$\left(\frac{\gamma}{\alpha f_y}\right)^2 \cdot (\sigma_{wx,Ed}^2 + \sigma_{wy,Ed}^2 - \sigma_{wx,Ed}\sigma_{wy,Ed}) + \left(\frac{\gamma\tau_{w,Ed}}{\alpha_s f_y}\right)^2 \leq 1$$

Equation 26. Admissible stress for welds (SFS-EN 13155+A2 2009)

Here $\gamma = 1.1$, f_y is according to table A2.1 in Annex 2, $\sigma_{wi,Ed}$ are similar tension and compression stresses $\sigma_{i,Ed}$ shown in equation A2.1 in Annex 2 but in this case evaluated for the welds. Coefficients α and α_s represent different weld categories according to SFS-EN 13155+A2 (2009, 71) and EN 25817 (1993). These coefficients are shown in tables A3.1 and A3.2 in Annex 3.

If equation 26 is satisfied, then according to SFS-EN 13155+A2 (2009, 69-71) the welds mechanical strength is sufficient. It should be noted that the equation is valid for welds that undergo compression/tension and bending. For example in a case of only compression or tension the equation is simplified similarly as equation A2.2 in Annex 2. In addition to weld analysis, the structural member's mechanical strength should also be analyzed.

5 STRUCTURAL MEMBER ANALYSIS

Like in weld analysis the analysis for the structural member depends on internal forces and the combinations of them. By categorizing the structures and calculating the demanded quantities directly by using basic formulation shown in sections 3.1, 3.2 and 3.3 or using methods provided by classification societies or different standards, this can be achieved. In the limits of this thesis the review to structural member analysis is only discussed for buckling when the structure only undergoes compression due to axial force. To get the whole idea of mechanical strength in the structure, other cases should be reviewed when doing the analysis as well.

The inspection of mechanical buckling strength of the structural members in section 5.1 is done by using the Eurocode 3 standard EN 1993-1-1 for classes 1, 2 and 3. For ship building the inspection is done in section 5.2 using DNV GL rules. All the formulation applies only to structural steel (S 235, S355 etc.). Using the table A1.1 in Annex 1 this can be expanded to apply to structural ship building steels A and A36 as well. In calculations in sections 5.1 and 5.2, it is assumed that no lateral torsion occurs.

5.1 Mechanical buckling strength of members using Eurocode 3

In a case of axial force affecting the compression the design force N_{Ed} can be evaluated from the FEMAP. Below in table 5 are some basis values that should be known.

Table 5. Basis values to be known according to Eurocode 3 or to be evaluated before the calculations. All the quantities are shown in basic units.

Quantity	Units	Description	Observations EN 1993-1-1
A_{cc}	m ²	Cross-section area	For class 1, 2 and 3. For class 4 the effective cross-section area is used
f_y	N/m ²	Admissible stress according to table A2.1 in Annex 2	
c_{cc}	m	Width of a cross-section for compression parts	
t_{cc}	m	Thickness of a cross-section for compression parts	
I_y and I_z	m ⁴	Second momentum of area around two axis of the cross-section	
i_y and i_z	m	Radius of gyration in two different axis of the cross-section	
L_{Ey} and L_{Ez}	m	Buckling lengths in planes x-y and x-z	
E	N/m ²	Young's modulus of the material	

5.1.1 Cross-section verification for plastic resistance

To get the class of the cross-section the maximum width-to-thickness ration has to be evaluated using the equation 27. (EN 1993-1-1 2004, 42-44)

$$\frac{c_{cc}}{t_{cc}} \leq \acute{\epsilon}$$

Equation 27. Maximum width-to-thickness ration (EN 1993-1-1 2004)

In the equation above $\acute{\epsilon}$ is admissible stress related coefficient in accordance with EN 1993-1-1 (2004, 42-44).

After the class of the cross-section has been found, the plastic resistance $N_{c,Rd}$ needs to be verified. This can be done for classes 1, 2 and 3 using equation 28 (EN 1993-1-1 2004, 49).

$$N_{c,Rd} = \frac{A_{cc}f_y}{\gamma_{M0}}$$

Equation 28. Plastic resistance (EN 1993-1-1 2004)

Here $\gamma_{M0} = 1.00$ is partial factor for resistance of cross-section that doesn't depend on the class of the cross-section (EN 1993-1-1 2004, 45). In case of welding the factor γ_{M0} was 1.10 (Equation 26). For a structure to meet plastic resistance terms the value received from the equation 28 needs to be larger than the design force N_{Ed} (EN 1993-1-1 2004, 49).

5.1.2 Cross-section verification for buckling resistance

First the slenderness $\bar{\lambda}$ of the structural member is determined. For this relative slenderness value λ_1 is calculated from equation 29 (EN 1993-1-1 2004, 59).

$$\lambda_1 = \pi \sqrt{\frac{E}{f_y}}$$

Equation 29. Relative slenderness (EN 1993-1-1 2004)

Slenderness $\bar{\lambda}_j$ can now be calculated from equation 30 (EN 1993-1-1 2004, 59).

$$\bar{\lambda}_j = \frac{L_{Ej}}{i_j \cdot \lambda_1}$$

Equation 30. Slenderness (EN 1993-1-1 2004)

In equation 30, $j \in \{y, z\}$, L_{Ej} is a buckling length and i_j is a radius of gyration. From the buckling curve the reduction factor χ can be read or it can be calculated using equation 31. The buckling curve to be used depends on cross-section limits. (EN 1993-1-1 2004, 57-59.)

$$\chi = \frac{1}{\Phi + \sqrt{\Phi^2 - \bar{\lambda}_j^2}}$$

Equation 31. Reduction factor (EN 1993-1-1 2004)

In equation 31 $\Phi = 0.5 \left(1 + \alpha(\bar{\lambda}_j - 0.2) + \bar{\lambda}_j^2 \right)$ and α is called the *imperfection factor* and depends on what buckling curve is used. Because the slenderness was calculated in both around the y- and z-axis the minimum reduction factor is chosen. Also the reduction factor is always smaller or equal to value 1.0 (EN 1993-1-1 2004, 59).

Finally in the equation 32 the admissible buckling resistance $N_{b,Rd}$ is calculated. If the $N_{b,Rd}$ is larger than design force N_{Ed} then no buckling should occur. (EN 1993-1-1 2004, 56-57.)

$$N_{b,Rd} = \frac{\chi A f_y}{\gamma_{M1}}$$

Equation 32. Admissible buckling resistance (EN 1993-1-1 2004)

In the equation above partial factor γ_{M1} has value 1.00 when loading is static (EN 1993-1-1 2004, 45).

5.2 Mechanical buckling strength of members using DNV GL rules

In DNV GL guideline DNVGL-CG-0128 (2018) the buckling is reviewed using minimum critical buckling stress σ_{cr} rather than admissible buckling resistance $N_{b,Rd}$. By converting the critical buckling stress to force using area these can be made comparable. However the way the minimum buckling stress is interpreted differs from section 5.1. In table 6 the quantities that are needed before the calculations to take place are marked.

Table 6. Basis values to be known according to DNV GL or to be evaluated before the calculations. Units are according to DNVGL-CG-0128.

Quantity	Units	Description	Observations DNVGL-CG-0128
A	cm^2	Cross-section area of the member	For class 1, 2 and 3
σ_{av}	N/mm^2	Average axial compressive stress in member	
$R_{eH,S}$	N/mm^2	Specified minimum yield stress of the stiffener	
I	cm^4	Second momentum of area around the weakest axis	
l_{pill}	m	Length of the member	Length is measured according to section 4.2 of standard DNVGL-CG-0128
E	N/mm^2	Young's modulus of the material	
R_{eH}	N/mm^2	Minimum yield stress of the considered member	

To be able to calculate the minimum elastic buckling stress σ_{cr} the elastic compressive column buckling stress σ_{EC} is needed. This can be calculated from equation 33 below (DNVGL-CG-0128 2018, 48).

$$\sigma_{EC} = \pi^2 E f_{end} \frac{I}{Al_{pill}^2} \cdot 10^{-4}$$

Equation 33. Elastic compressive column buckling stress (DNVGL-CG-0128 2018)

In the equation above f_{end} is the end constrain factor. Values of f_{end} depend on the buckled shape of the member. The minimum critical buckling stress σ_{cr} can now be calculated from equation 34. (DNVGL-CG-0128 2018, 48-49.)

$$\begin{aligned} \sigma_{cr} &= \sigma_{EC} && \text{for } \sigma_{EC} \leq 0.5R_{eH} \\ \sigma_{cr} &= \left(1 - \frac{R_{eH-S}}{4\sigma_{EC}}\right) R_{eH} && \text{for } \sigma_{EC} > 0.5R_{eH} \end{aligned}$$

Equation 34. Minimum critical buckling stress (DNVGL-CG-0128 2018)

To know if the compressed member will buckle the buckling utilization factor η is determined. This can be done using equation 35. (DNVGL-CG-0128 2018, 48)

$$\eta = \frac{\sigma_{av}}{\sigma_{cr}}$$

Equation 35. Buckling utilization factor (DNVGL-CG-0128 2018)

Factor η needs to be lower or equal to allowable buckling utilization factor η_{all} defined in DNV GL rules. For compression $\eta_{all} \in \{0.65, 0.75\}$. (DNVGL-RU_SHIP-Pt3Ch8. 2017, 8-10.)

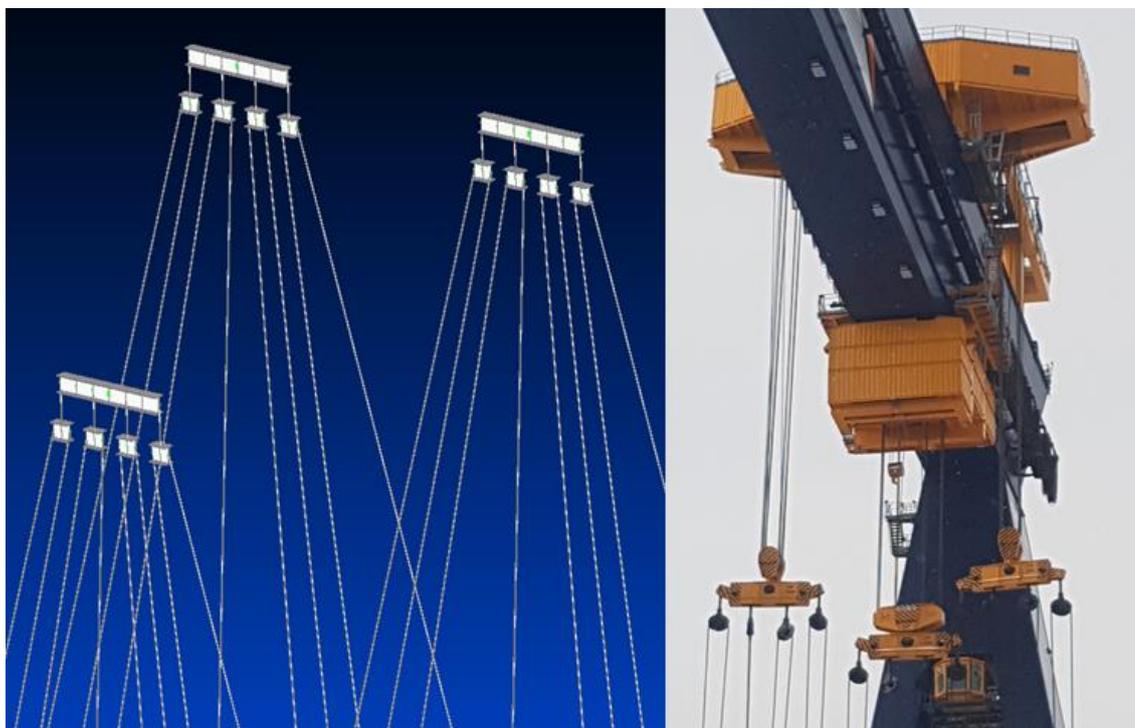
6 CREATING A HOISTING MODEL

When creating a hoisting model few things are needed to be taken into consideration. Before the local model is created, the required trolley system needs to be modelled. Basic principles for this are shown in section 6.1. In sections 6.2 and 6.3 the local model is created from the global model. In section 6.4 the trolley system and the local model are combined and the needed boundary conditions are added. Finally the analysis method of the created model is introduced.

6.1 Trolley system in FEMAP

The hoisting system or the trolley system can be modelled in several ways. To keep the model simple only the trolley's lifting beams, articulations and cable wires were modelled. This made keeping the boundary conditions for the system at minimum possible.

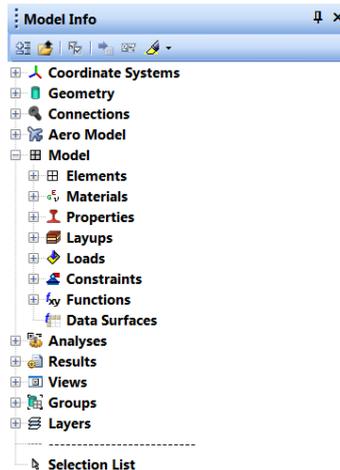
Beams were modelled using HEA profile while the cables were created using tubular profile. In picture 21 the created model of hoisting system and gantry crane in Meyer Turku are shown.



Picture 21. Model of hoisting system. On left side is the modelled hoisting system. On right hand side is the gantry crane in Meyer Turku shipyard.

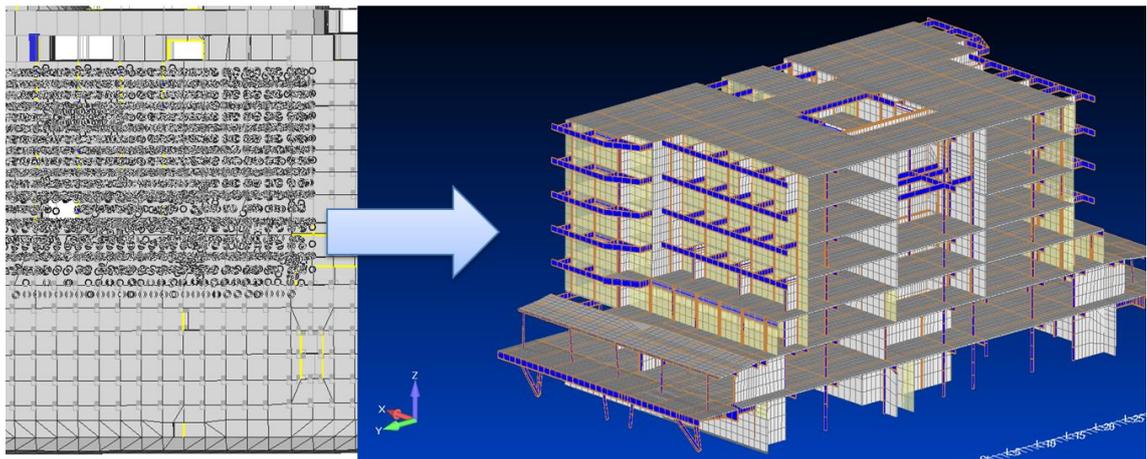
6.2 From global model to local model

The local model needs to be separated from the global model. It is important that all the required elements and properties in the model are included. To ensure this all the needed structures are chosen to be visible. This was done using **Model Info** panel shown in picture 22.



Picture 22. Model Info panel. It can be used to select desired elements, properties, materials etc. to be visible.

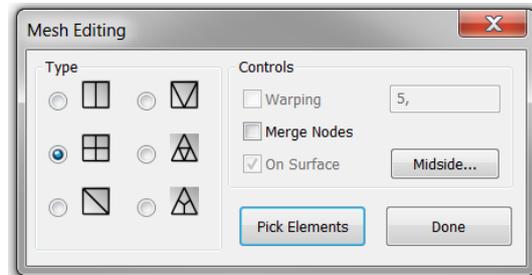
Using the **Select Panel** and selecting the desired selection of the model and copying the selected section to a new project the local model is created (picture 23).



Picture 23. The making of the local model

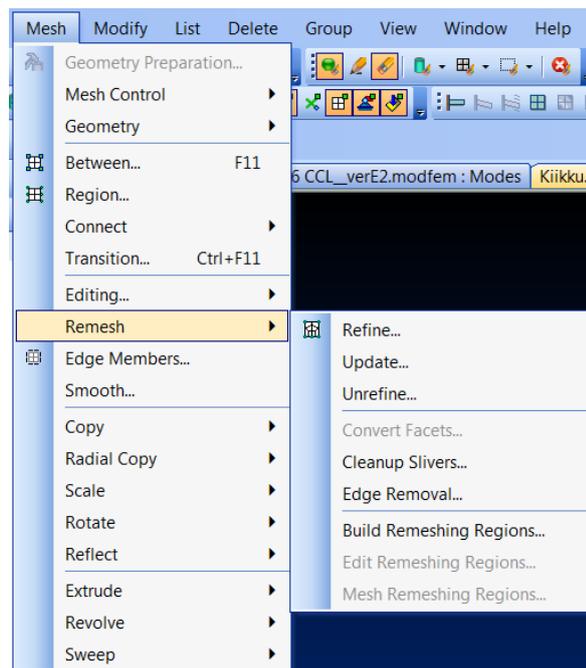
6.3 Checking the local model

After the local model is created the correctness of the model was checked as shown in sections 3.4.2 and 3.4.3. The mesh size was decreased **using the Mesh Editing** tool using quadrilateral cell shape shown in picture 24.



Picture 24. Mesh size decrease using quadrilateral mesh

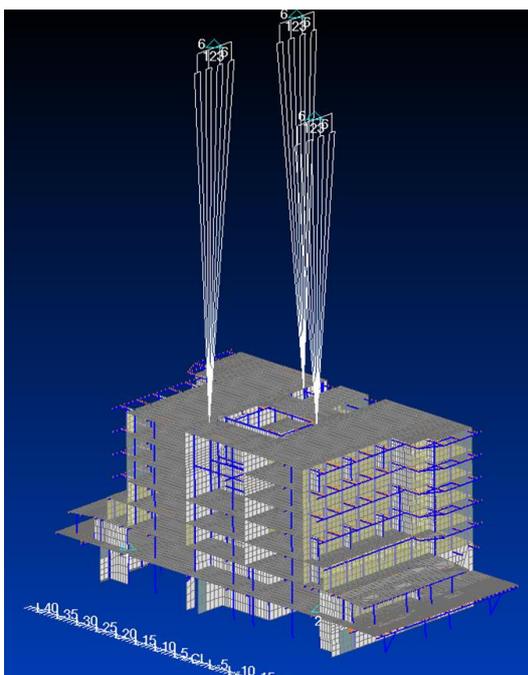
Mesh decreasing was done twice to get the mesh size to be equal to the frame length. The Mesh Editing tool didn't work for beams, so the mesh size decrease for them was done using **Refine** tool under **Remesh** shown in picture 25.



Picture 25. Remesh of the beams

6.4 The hoisting model

After the local model was created the trolley system was added to it. To get the section to move only in vertical direction the boundary conditions were added as shown in section 3.4.4. In picture 26 the preliminary hoisting model is shown. Preliminary hoisting model is a term, which refers to a model without additional supports or a specified cable locations.



Picture 26. Preliminary hoisting model.

To get the required gravity ($2 \cdot 9.81 \text{ m/s}^2$) the acceleration was chosen by creating the **Body Load** with the acceleration being 19.62. The trolley system was centered around the COG so that no large tilt will happen. After the cables were moved to the correct locations and additional supports were added, the hoisting model was completed.

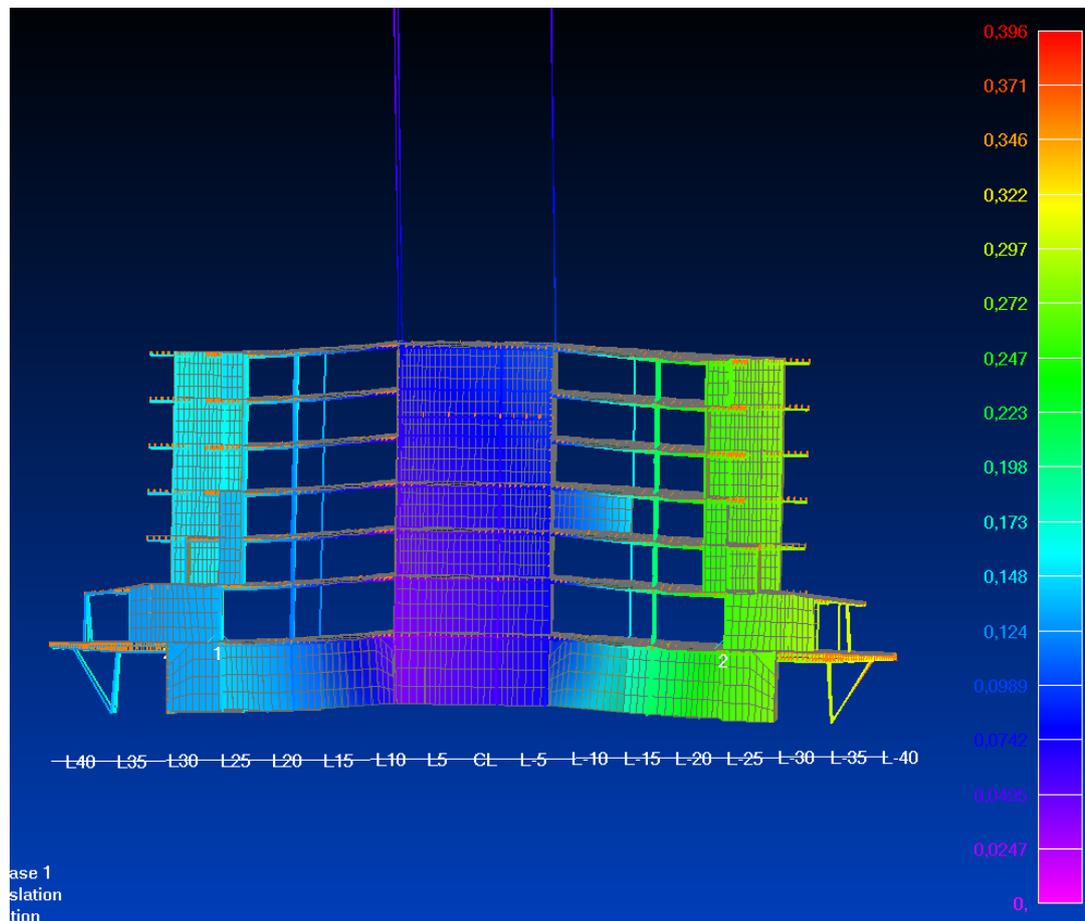
The modelling from the global model to the preliminary hoisting model was done twice. The first time took a lot of effort, because there was no known method of how to create a trolley system. The Second time was done faster taking about four hours. This time included the modelling of the trolley system. For an experienced FEMAP user this could be done even faster. Time required depends on the complexity of the section and the correctness of the global model.

After the hoisting model was created the structure was analyzed using the sufficient standard and basic mechanical strength calculations for welds and structural members. This was done as shown in sections 4 and 5. In these sections only few examples of the required calculations were given, but the principle is that the admissible design loads or stresses are needed to complete the analysis.

7 CONCLUSIONS

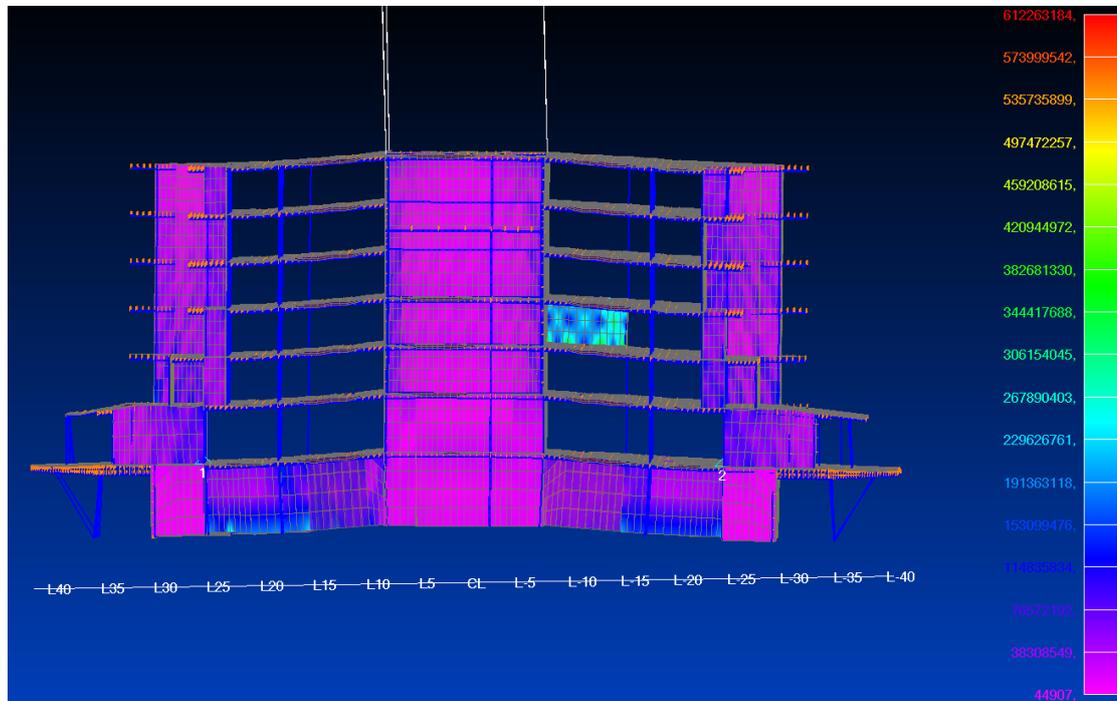
The goal of this work was to create an analysis method to get quick and reliable results from the rough global model of the ship for the hoisting design. While the global model is very rough reliable results can be received if the analysis part is done correctly. A difficult part of the modelling will be getting the structures to be correct. The comparison to more accurate finite element model should also be done. Comparison to a more accurate FEM model was not done, for it was not ready at the time of this thesis.

At first no additional support were added and no correct cable placing was used in the model. From picture 28 it can be seen that the missing support and wrong positioning of the cables made the structure to translate too much with the maximum translation being 396 mm. This occurs in the shell area of the structure which is understandable for the structure is only supported from the middle by the hoisting system with no additional support in the shell area.



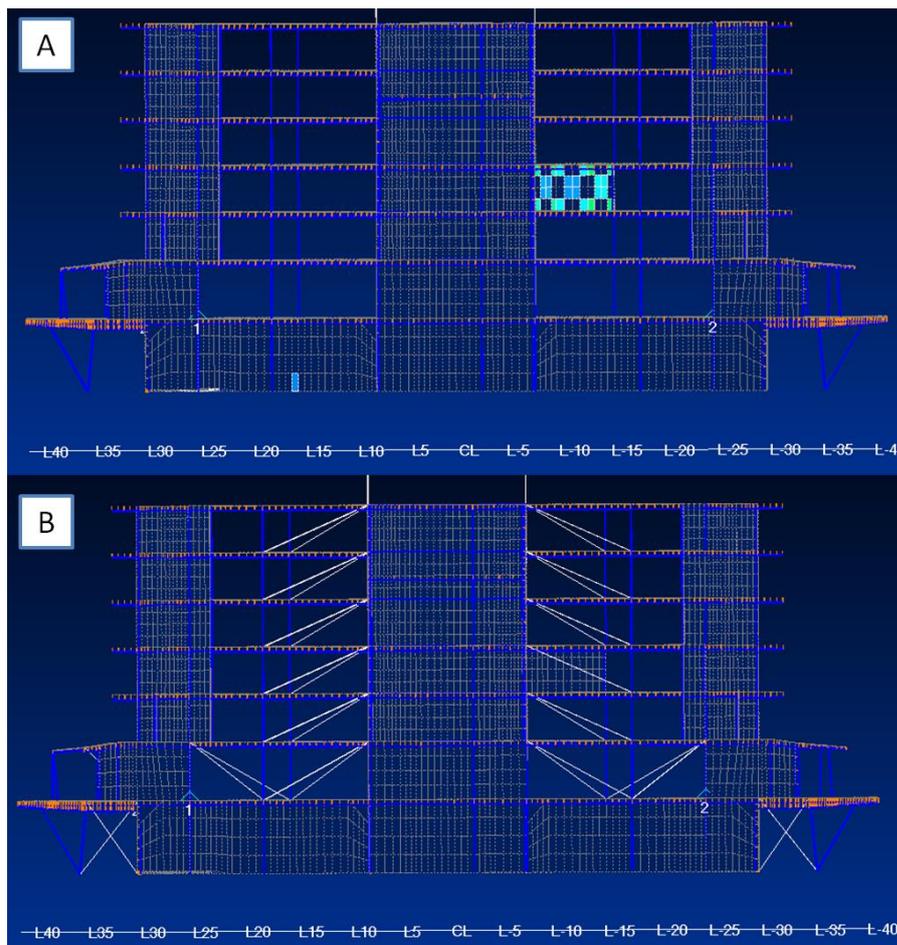
Picture 28. Example of unsupported steel structure to undergo translation in hoisting with acceleration $2 \cdot 9.81 \text{ m/s}^2$. Deformation is scaled 5 times the actual deformation.

The effect of incorrect positioning of the cables and unsupported structure is also shown when evaluating the Von Mises stress of the structure. Largest value of the **Lam Ply1 VonMises** goes to be 612.3 MPa while the average is under 235 MPa . Most stress occurs in the structures that undergoes bending. These structures are typically positioned transversally as can be seen from picture 29.



Picture 29. Von Mises stress for the unsupported structure. Highest value was measured to be $612,3 \text{ MPa}$ while most of the structures were under 235 MPa .

With boundary value of 235 MPa the elements with stress above it can be seen in picture 30. Critical places were mounting points of lifting cables, one transversal bulkhead and lower structures. With added supports there were significantly less elements in which the boundary values were surpassed. Most critical places with no significant change in stress were the mounting points of cables. Even these points were still below the Yield strength of A36 which indicates that in these points should be strengthened with A36 steel.



Picture 30. Effect of added supports in stresses. Above picture (A) represents situation with no additional supports. In below (B) the supports are added.

The analysis part of the model is possible but requires a lot of calculations. If these calculations can be automated in some way or the other the global model can be made as a sufficient basis for the hoisting design. This automation could be an API in FEMAP or Excel with the required formulas created.

Furthermore the situation always needs to be evaluated and basic values like geometrical properties of the members and stress types are needed. After the situation is known literature offers a wide range of approaches to get the results needed. It should also be noted that the viewpoint depends on the purpose of the structure (Ship, Off-shore, Land structure etc.) as were seen in section 5. In lifting the Eurocode 3 and SFS-EN 13155+A2 are commonly used for they demands more from the mechanical durability of the structures and they demand the necessary safety factors for lifted structures. In ship's structures these rules are typically too strict for they are made for onshore steel structures. To comparing the analysis results between different standards and regulations further review should be made. Additionally, programming these rules to be part of a program (eq. FEMAP) or creating a separate table (eq. Excel) with necessary formulas should be considered to automate the analysis.

REFERENCES

Books and publications:

Räisänen, P. 2002. Laivanrakennuksen käsikirja. 2. Painos. Turun Ammattikorkeakoulu

Interviews:

Platan Vesa. Lead Hoisting designer. Interview 1.8.2018. Interviewed by Mehto, P. No denominations. Meyer Turku Oy

Rautiainen Matti. Classification designer. Interview 31.1.2019. Interviewed by Mehto, P. No denominations. Meyer Turku Oy

Laws and directives

Directive 2006/42/EC. 17.3.2006. The European Parliament and of the Council.

Standards

ASME 2018. B30.9 – Slings

DNVGL-CG-0128 2018. Glass Guideline – Buckling

DNVGL-RU_SHIP-Pt3Ch8 2015. Rules for classification, Ships, Part 3 Hull, Chapter 8 Buckling

EN 1993-1-1 2004. Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for building

IACS 2012. Recommendation No.47 Shipbuilding and Repair Quality Standard.

SFS-EN 13155+A2 2009. Cranes. Safety. Non-fixed load lifting attachments

SFS-EN 25817 1993. Arc-welded joints in steel. Guidance on quality levels for imperfections

SFS-EN ISO 3834 2006. Quality requirements for fusion welding of metallic materials.

Internet:

ASTM International 2018. Referred 23.7.2018.

<https://www.astm.org/Standard/standards-and-publications.html>

Giovanni, M 2018. Lectures of Computer Graphics. 3D Viewing. Brandeis University. Referred 24.11.2018. http://www.cs.brandeis.edu/~cs155/Lecture_08_6.pdf

Heil, M 2010. Lectures of Elasticity. Chapter 4 – Elasticity & constitutive equations. The University of Manchester. Referred 6.1.2019.

<https://personalpages.manchester.ac.uk/staff/matthias.heil/Lectures/Elasticity/Material/Chapter4.pdf>

Kelly, P 2015. Solid Mechanics Part II: Differential Equations for Solid Mechanics. The University Of Auckland. Referred 6.1.2019.

http://homepages.engineering.auckland.ac.nz/~pkel015/SolidMechanicsBooks/Part_II/01_DifferentialEquilibriumAndCompatibility/01_DifferentialEquations_Complete.pdf

Lagace, P 2001. Unit 3 – (Review of) Language of Stress/Strain Analysis. Referred 6.1.2019.

<https://ocw.mit.edu/courses/aeronautics-and-astronautics/16-20-structural-mechanics-fall-2002/lecture-notes/unit3.pdf>

Marine Insight 2018. Referred 17.11.2018.

<https://www.marineinsight.com/offshore/offshore-lifting-operations-and-engineering-analysis/>

Saratech 2018. Introduction to FEMAP. Referred 15.12.2018.

https://community.plm.automation.siemens.com/siemensplm/attachments/siemensplm/Simcenter_event_tkb/8/1/FS16_Saratech_00_IntroFemap.pdf

Simões, R 2014. Belgium. Eurocodes, Background and Applications – Design of Members. Referred 20.2.2019.

https://eurocodes.jrc.ec.europa.eu/doc/2014_07_WS_Steel/presentations/05_Eurocodes_Steel_Workshop_SIMOES.pdf

Wolfram Research 2019. Introduction to Kronecker Delta. Referred 1.6.2019.

<http://functions.wolfram.com/introductions/PDF/KroneckerDelta.pdf>

Annex 1. IACS steel grades

Table A1.1. Table of IACS steel grades. Here $R_{eH,min}$ is the minimum Yield stress in N/mm^2 . Quantity R_m refers to Tensile strength in N/mm^2 and A_5 is the minimum elongation in %. (IACS 2012, 49)

Steel grades				Compared with EN grades
Grade	$R_{eH,min}$	R_m	A_5	
				EN 10025-93 EN 10113-93
A	235	400-502	22	S235JRG2
A36	355	490-620	21	S355N/M

Annex 2. Admissible stresses in structural member

Table A2.1. Admissible stresses in structural member. In table below are listed the admissible normal and shear stresses in structural members according to SFS 13155+ A2 (2009, 69).

Steel	Thickness in mm	Admissible stress in tension/compression f_y in MPa	Admissible stress in shear $f_y/\sqrt{3}$ in MPa
S 235	$t \leq 16$	235	135
	$16 < t \leq 40$	225	130
S 275	$t \leq 16$	275	160
	$16 < t \leq 40$	265	153
S 355	$t \leq 16$	355	205
	$16 < t \leq 40$	345	200

If both normal and shear stress in the structure then equation A2.1 shall be verified. (SFS-EN 13155+A2 2009, 69)

$$\left(\frac{\sigma_{x,Ed}}{f_y}\right)^2 + \left(\frac{\sigma_{y,Ed}}{f_y}\right)^2 - \frac{\sigma_{x,Ed}\sigma_{y,Ed}}{f_y^2} + 3\left(\frac{\tau_{Ed}}{f_y}\right)^2 \leq 1 \quad (\text{A2.1})$$

In above equation $\sigma_{x,Ed}$ is the longitudinal stress value, $\sigma_{y,Ed}$ is the transverse stress value and τ_{Ed} is the shear stress value (EN 1993-1-1 2004, 45).

Longitudinal stress values $\sigma_{x,Ed}$ are due to compression or tension while the transverse stress values $\sigma_{y,Ed}$ are due to bending. Shear stress values τ_{Ed} are due to bending and torsion. In case of pure compression/tension, above equation goes to form illustrated in equation A2.2 (Simões 2014).

$$\frac{\sigma_{x,Ed}}{f_y} \leq 1 \quad (\text{A2.2})$$

Annex 3. Admissible stresses in welds

Table A3.1. Admissible stresses in welds. In table below are listed the index values to get the admissible normal and shear stresses in welds according to SFS 13155+A2 (2009, 71).

Type of weld	Kind of stressing	Quality level	α tension	α compression	α_s shear
Weld with full penetration or backwelded - Butt welded - Double semi V-weld, semi V-weld for T-joint	Stress across the weld direction	B, C	1	1	$1/\sqrt{2}$
		D	0.9	1	$1/\sqrt{2}$
Weld with full penetration or backwelded - Weld with small web - Double semi Y-weld, semi Y-weld for T-joint	Stress across the weld direction	B, C	0.9	1	$1/\sqrt{2}$
		D	0.8	0.9	$1/\sqrt{2}$
Weld with full penetration - Double semi Y-weld, semi Y-weld for T-joint - Double fillet weld - Fillet weld	Stress across the weld direction	B, C, D	0.7	0.8	$1/\sqrt{2}$
All types of welds	Stress in the weld direction	B, C, D	1	1	$1/\sqrt{3}$

In table A3.1 shown Quality levels B, C and D are according to EN 25817 (1993). Quality level meanings are listed in table A3.2

Table A3.2. Quality levels of welds. The demand for weld increases from D to B. (SFS-EN 25817 1993)

Quality level	Meaning
D	Quality is average
C	Quality is good
B	Quality is demanding