Structure design of a sailing yacht hull by rules and direct method

Ivan Klarić

Master Thesis

presented in partial fulfillment

of the requirements for the double degree:

"Advanced Master in Naval Architecture" conferred by University of Liege

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

developed at University of Genoa

in framework of the

"EMSHIP"

Erasmus Mondus Master Course

in "Integrated Advanced Ship Design"

Supervisor: Prof. Dario Boote, University de Genoa

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

Genoa, February 2012

Declaration of Authorship

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

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

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

I have acknowledged all main sources of help.

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

I cede copyright of the thesis in favour of the University of Genoa, University of Liege and Ecole Centrale de Nantes.

Date:
17.1.2012

Signature:
Ivan Klarić

Table of Contents

Table of figures

List of major label

Symbol	Meaning	First use
L_{OA}	length overall	
L_{WL}	length of water line	
L	scantling length	(1.1)
B	breadth	
D	molded depth	
d	draft for scantlings	
Δ	displacement	
t	thickness of plates / stiffeners	(1.2)
s	frame spacing	
c	correction factor	
k	aspect ratio coefficient	
σ_a	design stress	
SM	section modulus	(1.3)
C_F	design head for Internals factor	(1.4)
A	area of profile	(1.5)
b	breadth of profile	(1.5)
h	height of profile	(1.5)
z_g	center of gravity for profile	(1.6)
I	moment of inertia	(1.7)
CG	centre of gravity	(5.1)
δ	deflection	(5.2)
F	force	(5.2)
E	modul of elasticity	(5.2)
σ	stress	(5.3)
M	moment	(5.3)

ABSTRACT

The thesis was done at Perini-Navi shipyard, Viareggio and at University of Genoa, Italy. Thesis is divided in two parts. Fist part is scantling of the main frame for a sailing boat and in second part it has been performed the finite element analysis (FEM) of the structure. Scantling (main dimensions), for the yacht of 58 m in length has been done according to the American Bureau of Shipping (ABS) rules, Guide for Building and Classing Offshore Racing Yachts (ORY). The purpose of the scantling is to become familiar with the structure design of the sailing yacht and also to understand how to apply the rules. Material that shipyard is using for the construction of the sailing yachts is aluminium. Because of aluminium nature and big use in shipbuilding industry, one chapter is dedicated for this material with all properties, alloys, advantages and disadvantages.

The scope of the work was to perform the FEM analysis for the specific parts of ship hull. As shipyard already had FEM analysis of the entire hull, in agreement with shipyard and university, in this thesis were investigated main mast base foundation and connection of the keel with the structure of the yacht.

During the building of the hull, main mast for sails was changed and main mast base foundation that was already built does not satisfy for new loads coming from the main mast. Because of that, it is necessary to verify this structure on new loads. Also, purpose of this first model was to become familiar with the software features. So it was analysed difference between shell and solid elements, influence of mesh on the results, linear and nonlinear analysis. Nonlinear analysis has been done because the structure was exceeding yielding stress of 215 MPa for aluminium. After, when all analysis has been performed, it was necessary to propose the solution to shipyard; how to solve the problem on existing structure and how to adapt the new structure for this kind of loads.

Second model is connection of the keel with the structure of the yacht. Keel is 8 m long and total mass of the keel; aluminium structure with the lead inside is 65 tons, and according to the rules, keel must withstand its own mass acting in centre of gravity. To satisfy the rules criteria and to prove direct method that have been used for the calculation of the keel deflection and stresses, FEM analysis of the keel – structure connection has been done. The analysis expected to show that deflection of the keel is 41 mm and the analysis showed that deflection is 74 mm at centre of gravity and more than 200 mm at the end of the keel. After, it was necessary to verify the stresses and deflection on the hull structure where the keel is connected. Connection between structure and keel is with the pin. The models were made separately, because of the complexity to simulate pin connection and the forces measured by probes on the keel are transmitted to the structure. But analysis showed that deflection is quite small at hull structure and that structure is very well design.

1. INTRODUCTION

A yacht is recreational boat or ship. The first appearances were in 17th century when the Dutch navy built yachts to catch and pursuit pirates around and into the shallow waters of the Netherland. Because of that, the name yacht has the roots in Dutch language and the word "jagen", meaning to hunt or chase. The word Yacht usually refers to a small, fast craft for the purpose of small voyages and short crossing. In meanwhile they were also used for non-military governmental roles such as customs duties and delivering pilots to the ships. The later use was for the private and pleasure purpose, the first yachts were attracted the attention of rich Dutch merchants. By the start of the 17th century yacht came in two broad categories, *speel-jachts* for sport and *oorlog-jachts* for naval duties. [1]

The delight of yachting for pleasure was developing very fast and also the trend cross the channel to the England. One of the first Englishmen who were sailing with yachts was Charles II. During the 17th century, yachting began to flourish across Europe. All of kinds were commissioned as yachts to the rich and powerful, from tiny open boats to small frigates. Yachts were instrumental in discovering new lands and defending vital waterways. They served both as pleasure craft and as working ships, carrying people and messages swiftly and comfortably to the shore.

Originally yachts were built in wood and in construction quite similar to what was customary in the normal shipbuilding of that time. The hull was single (massive) planking connected to closely spaced wooden frames. The frames were connected to wooden floors and those to the bottom planking. In the early days many yachts still had flat or slightly curved bottom. At the upper side the frames were connected to the deck beams on which the deck planking was laid. Longitudinal stringers were mostly absent. Later when yachts got keels the construction changed. The sections became rather more V shaped asking for different construction techniques. The stem beam, the keel beam and the stern beam were introduced, which functioned also as longitudinal stiffeners, to which the frames were connected, which in turn were connected by the floors. The difficulties and weaknesses in the available connecting techniques of that time however posed a serious limit on the achievable overall strength and in more in particular the overall rigidity of the yacht hull structure. All wooden construction was only to return in yacht building after the 1970's, when new and serious bonding techniques became available, such as the epoxy resins, together with new wood laminating techniques. So in the 1930's the new "composite" construction technique came into force, in which the keel, stem, stern, frames, beams and floors were all constructed in steel (and bolted or riveted, later welded together) to which the still wooden hull and deck planking was connected. This was a big improvement and however still rather heavy. Still later the completely steel hull came into play in which now in the composite construction also the wooden hull planking and later also the wooden deck planking was replaced by steel and all were riveted or welded together. This yields a sound and stiff construction for the hull. This construction technique, using either steel or aluminium, lasts till today and is mostly favoured for the bigger yachts or for yachts with high demands on resistance against external local loads, such as yachts designed for use on long ocean voyages or in the arctic regions [2].

The Perini Navi Group is operative in the super yacht sector under two specific brand names: Perini Navi (sail yacht division) and Picchiotti (motor yacht division). The Group was founded in 1983 as Perini Navi and produced sailing yachts. From 1983 until nowadays Perini-Navi built more than 50 sailing yachts and has several others in production. Today the Perini Navi Group is operative in five specific market sectors:

- Large sailing yachts that range from 40 to 60 metres
- Large sailing yachts over 60 meters and custom projects
- A Racing Line of sailing yachts
- A Fast Cruising line of sailing yachts
- Picchiotti motor yachts

The Perini Navi group is composed of four companies bought over time and acquired with the objective of maximizing production capacity and entering new market segments. And companies are located in:

- Perini Navi, Viareggio – Italy
- Cantiere Picchiotti, La Spezia – Italy
- Perini Istanbul, Yildiz – Turkey
- Perini Navi USA, Newport, Rhode Island - USA

All the sail handling systems are developed and constructed in the shipyard, as are the masts and the rigging. Exclusive Perini Navi software is installed on board and monitors their yachts while under sail automatically transmitting data to the construction office.

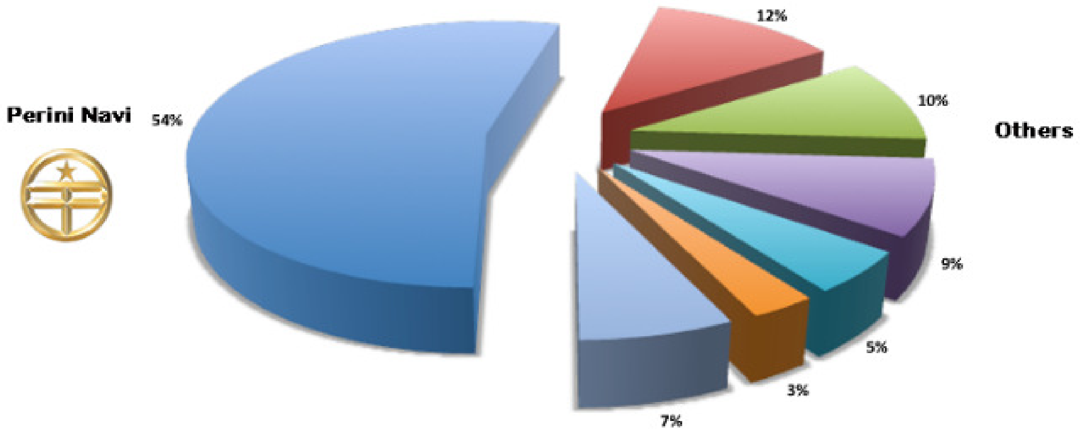


Figure 1. Word market of sailing yacht over 45 m in 2010

The five biggest sailing yachts in the world:

Name	Launch	LOA	Gross tonnage	Shipyard
Eos	2006	92.9 m	1500	Lurssen
Athena	2004	90 m	1123	Royal Huisman
Maltese Falcon	2006	88 m	1110	Perini Navi
Mirabella V	2004	75.2 m	1004	Vosper Thornycroft
Phocea	1976	75.1 m	530	DCAN

Figure 2. Maltese Falcon sailing yacht, built in Perini Navi at 2006

This brief introduction was to obtain basic information about history of yacht and Perini Navi Group. Further in the thesis scantling will be done according to the ABS rules. Scantling will be done for the main frame, of 58 m sailing yacht. From the scantling it is necessary to understand the rules and how to apply them.

Figure 3. 58 m sailing yacht from Perini Navi

After scantling, one chapter will be for a material, the aluminium, because this material is very important material in shipbuilding and it has lot of advantages and properties that will be mention here.

At the end of the thesis finite element analysis will be done. It will not be performed analysis of the entire hull. Because shipyard already has that analysis, so in agreement with university and shipyard it was decided to simulate keel – structure connection. That part of a yacht has a big importance in sailing yacht. Just to imagine the complicity of this model it is enough to say that mass of the keel is 65 tons and the length is 7990 mm, type of the keel is the swing keel and connection with the structure is with the pin. As classification society is demanding this kind of calculation, it is necessary for the shipyard to perform such calculation and present results to the classification society. Model will be done in SolidWorks and FEM analysis will be done in ANSYS Workbench. However, before this project will be start to model and analyse, from great importance is to become familiar with the ANSYS features, to be able to set up the true model of the keel – structure connection. And because of that, first in beginning of the project, one small and simply model will be done. It will be the model of main mast base foundation. That model is chosen, because during the construction of the sailing yacht, mast design was changed and the main mast base foundation didn't satisfy anymore the new loads. So through this quite simple model, it was necessary to learn the software, verify its feature and the structure on new type of loads.

2. SCANTLING

To determining the scantlings there are two standard methods. The first method and by far the oldest is rule-of-thumb and the second is engineering analysis. Rule-of-thumb is the most reliable and formalized scantling rules. These rules are based on engineering analysis cross checked against a database of successful vessels. The results are then condensed and simplified for quick application using easily determined factors. Such rules establish the required construction materials and dimensions based on a few easily obtainable numbers, such length overall, displacement and boat speed. Scantling rules have been one of the principal methods of specifying boat construction for well over a hundred years. They have been used by classification societies, such as the American Bureau of Shipping, Lloyds and others, by many of the finest designers and builders. And because of their ease of use, as compared with detailed engineering analysis, many builders and designers prefer to work with scantling rules. It is important to keep in mind that scantling rules work only for the specific type and size of boat intended by the initial rulemaker.

Scantling will be done according to ABS[1] rules. Guide for Building and Classing Offshore Racing Yachts, 1994. The ABS is a classification society with a mission to promote the security of life, property and the natural environment, primarily through the development and verification of standards for the design, construction and operational maintenance of marine-related facilities. Classification society is non-governmental organization that establishes and maintains technical standards for the construction and operation of ships and offshore structures. The society will validate that construction is according to these standards and carry out regular surveys in service to ensure compliance with the standards.

ABS was first charted in the state of New York in 1862, to certify ships captains. It has been involved in the development and improvement of safety standards. Born out of a need for industry self-regulation, ABS published its first technical standards, Rules for Survey and Classing Wooden Vessels, in 1870. When the era of wooden ships gave way to iron, ABS established standards for these structures, published as Rules for Survey and Classing of Iron Vessels. And after when iron gave way to steel, ABS published Rules for Building and Classing Steel Vessels were established and published in 1890. These Steel Vessel Rules continue to be revised and published annually.

[1] ABS – **A**merican **B**ureau of **S**hipping

ABS today has more than 3,300 employees worldwide and is broadly divided in two groups; Engineering review and Surveying. Engineers work in office buildings worldwide, the headquarters of ABS American Division is located in Houston, Texas, USA, for Europe headquarters is in London and headquarters for Asia Pacific region is located in Singapore.

For the sailing yacht rules provide all the main aspects of sailing yacht design, those are: materials, details and fastenings, plating, internals, rudders and keels. The only areas on which no indications are provided are for the mast and rigging. Where the hull scantlings are concerned, the Rules in section 7 provides formulas and tables for the thickness calculation of plating; aluminium, steel, fibre reinforced plastic (both single skin and sandwich) and wood are considered. The same approach is assumed for the scantling of internal reinforcements. For all those aspects that are not included in the Offshore Racing Yacht Guide, reference should be made to the “Guide for Building and Classing of Motor Pleasure Yachts” (ABS, 2000) for displacement and semi-planing yachts. Designers of large sailing yachts capable of sustaining highspeeds (in the planing regime) are referred to the “Guide for Building and Classing High Speed Craft” (ABS, 2001) for appropriate hull plating and internal structure scantlings.

Sec. 2 – 2.1:

Scantling length is given by the following equation:

$$L = \frac{L_{OA} + L_{WL}}{2} \qquad (1.1)$$

$L_{OA} = 58.6$ m - is the overall length of the hull

$L_{WL} = 50.43$ m

$L = 54.52$ m

Sec. 2 – 2.2:

$B = 11.4$ m - is the greatest molded breadth, excluding appendages

Sec. 2 – 2.5:

$D = 3.85$ m - is the molded depth at side in meters, measured vertically from the bottom of the canoe hull at centerline to the top of main weather deck at side.

Sec. 2 – 2.7:

$d = 2.15$ m - draft for scantlings, is the maximum distance in meters or feet measured vertically from the bottom of the canoe hull at its lowest point at centerline to the maximum estimated displacement waterline.

Sec. 2 – 2.15.1:

Steel and aluminum, boat is build from aluminum and more about that material will be in next chapter. Here just some main characteristic of used material are shown.

Alloy:	5083
Temper:	H321
Thickness:	Up to the 38 mm
Minimum ultimate tensile strength, welded condition:	275 N/mm^2
Minimum yield strength, unwelded condition:	215 N/mm^2

Main dimension of the sailing yacht:

$L_{OA} = 58.6$ m

$B = 11.4$ m

$D = 3.85$ m

$T = 2.15$ m

$v = 15.5$ kn

$\Delta = 540$ t

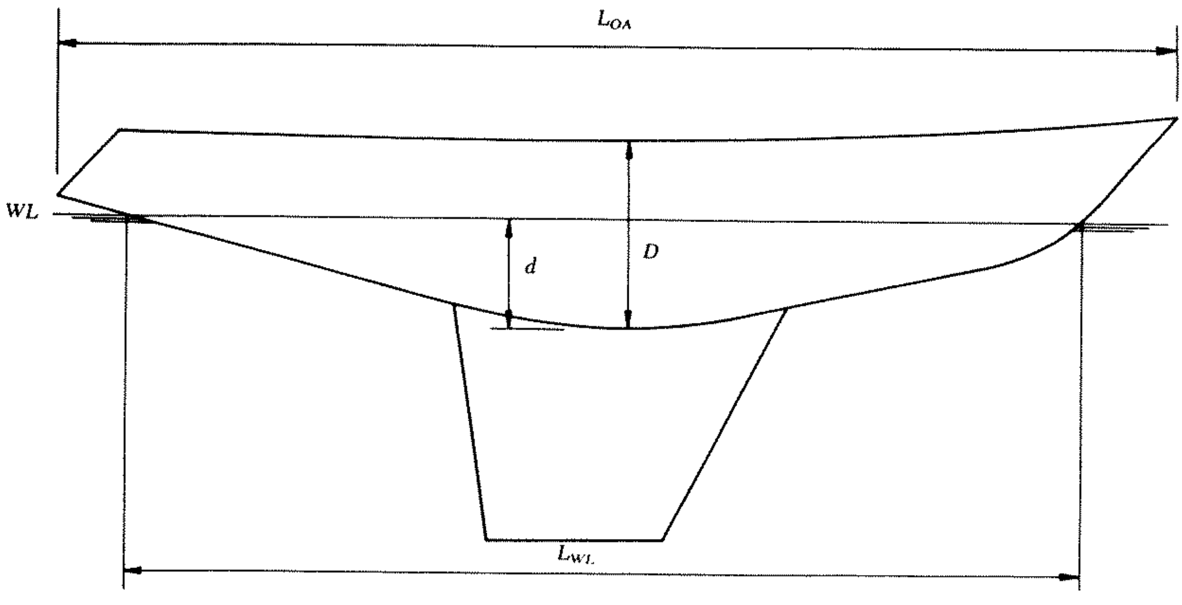


Figure 4. Sailing yacht profile at centreline [3]

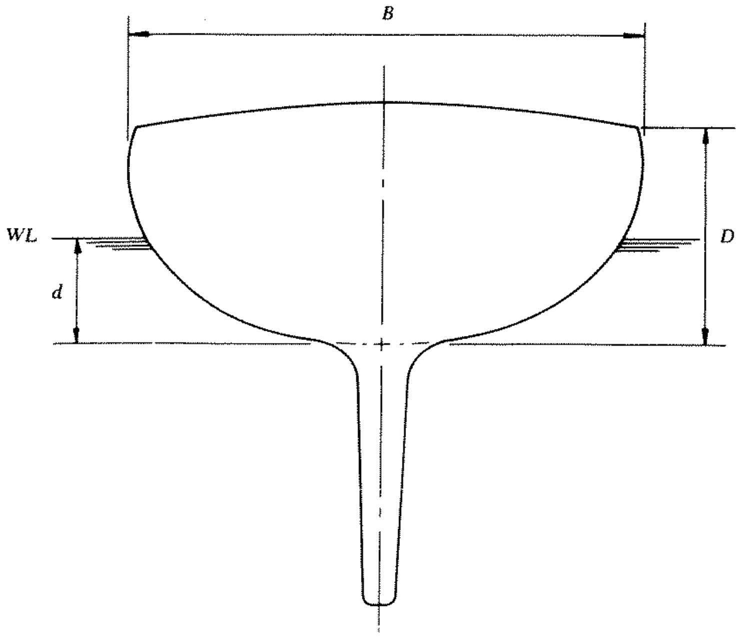


Figure 5. Transverse section of sailing yacht [3]

Sec. 8 – 8.1.1:

For aluminum or steel structural arrangement in general the hull is to be longitudinally framed with the deck and shell longitudinal supported by transverse web rings, transverse bulkheads or combination of both. Provided they are in turn effectively supported and of adequate strength the vertical boundaries of cabin houses and cockpits may be considered to support plating and internals. Transversely framed hulls in association, as necessary, with longitudinal girders, transverse webs and transverse bulkheads, will also be considered. Web frames or transverse bulkheads are to be fitted in way of masts and elsewhere, as necessary where the mast is deck stepped, special consideration will be given to the deck internal structure under the mast. Transverse web rings, transverse bulkheads or deep brackets are to be provided as necessary, in way of the chain plates. Transverse structural bulkheads with large openings are to have scantlings not less than required for internals in the same location. Care is to be taken to ensure structural continuity and hard spots are to be avoided.

Scantling will be done only for the main section of the boat. Main goal of the scantling is to obtain all necessary dimensions of the main frame and to show how to use the rules to perform such calculation. To do that, main frame will be divided in following sections; deck, side, bottom and inner bottom. For the first section the most detail scantling will be done with all explanation and for the rest of them, the same rules and the way will be done, just with changing of the coefficients. Everything what is changing from the first section it will be remarked.

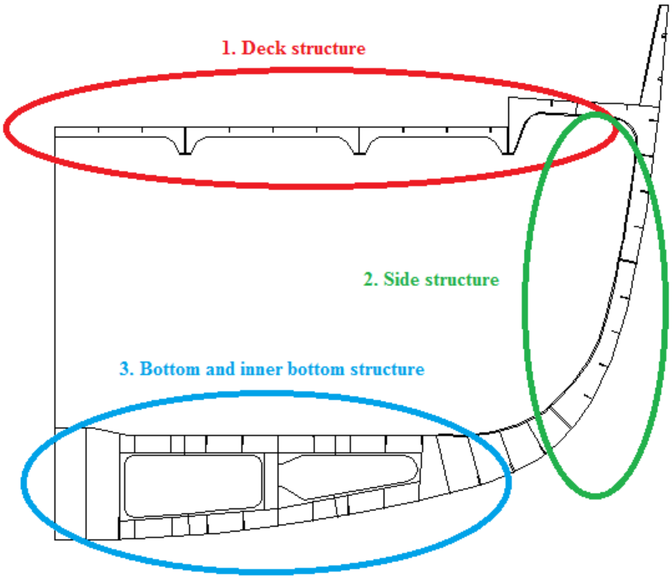


Figure 6. Sections of the main frame at which the scantling will be performed

2.1 Arbitrary Plate Scantling

To understand better what will be done in next parts, let's perform one simple scantling for the arbitrary chosen plate with next dimensions:

Dimension of the plate: l = 10000 mm
tr = 5000 mm

Figure 7. Arbitrary chosen plate

First step is to calculate thickness of the plate and after primary and secondary stiffeners; the thickness is calculated according to the next formula:

$$t = sc\sqrt{\frac{pk}{\sigma_a}} \tag{1.2}$$

Where frame spacing will be 500 mm in transversal direction on Figure 7 and other parameters and formulas will be calculated and explained in next 2.2 chapter, for this arbitrary plate is just important to understand the procedure how to perform the scantling. Position of the plate is chosen to be at h = 5 m.

$t = 8.05$ mm

And after all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 5$ mm

<u>Chosen thickness of the plate will be 9 mm.</u>

Calculation for the section modulus of the stiffeners is according to the next formula:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k \tag{1.3}$$

It is important to understand that procedure is starting from the smallest secondary longitudinal and the smallest plate of 500 x 625 mm dimensions. The secondary longitudinals are transferring the stresses to the secondary transversals. So the length of them will be between two transversals. Secondary transversal are transferring the stresses to primary longitudinals and primary longitudinals to primary transversals. It is necessary to follow this order to be sure in spacing and length of each member, because section modulus is proportional with spacing and with length on square.

First iteration:

Let us suppose the plate will contain secondary longitudinal and transversals and edges will be supported by primary longitudinal and transversals.

Table 1. Spacing and length of the stiffeners for the first iteration

Stiffener	spacing [mm]	length [mm]	number of elements
Secondary longitudinals	500	625	9
Secondary transversals	625	5000	15
Primary longitudinals	5000	10000	2
Primary transversals	10000	5000	2

From calculation for section modulus next is obtained:

Table 2. Section modulus and profiles of stiffeners for the first iteration

Stiffener	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Secondary longitudinals	FB	90 x 8		6,47	10,8
Secondary transversals	T profile	500 x 12	120 x 12	517,97	671
Primary longitudinals		Too big dimensions		16575	
Primary transversals					

From obtained results, primary longitudinal will be too big dimensions compared with the plate, so it is necessary to reduce the spacing.

Second iteration:

Now we reduce the spacing of primary longitudinal and by that the length of the secondary and primary transversals is also reduced.

Table 3. Spacing and length of the stiffeners for the second iteration

Stiffener	spacing [mm]	length [mm]	number of elements
Secondary longitudinals	500	625	8
Secondary transversals	625	2500	15
Primary longitudinals	2500	10000	3
Primary transversals	10000	2500	2

Now next profiles are obtained:

Table 4. Section modulus and profiles of stiffeners for the second iteration

Stiffener	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Secondary longitudinals	FB	90 x 8		6,47	10,8
Secondary transversals	T profile	250 x 10	100 x 12	129,49	161
Primary longitudinals		Too big dimensions		8287,5	
Primary transversals					

Still the section modulus is big so now it is necessary to reduce the length of the primary longitudinals by adding more primary transversals.

Third iteration:

For the third iteration spacing of the primary transversal is reduced and with that length if the primary longitudinal is reduced too.

Table 5. Spacing and length of the stiffeners for the third iteration

Stiffener	spacing [mm]	length [mm]	number of elements
Secondary longitudinals	500	625	8
Secondary transversals	625	2500	12
Primary longitudinals	2500	2500	3
Primary transversals	2500	2500	5

The profiles are:

Table 6. Section modulus and profiles of stiffeners for the third iteration

Stiffener	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Secondary longitudinals	FB	90 x 8		6,47	10,8
Secondary transversals	T profile	250 x 10	100 x 12	129,49	161
Primary longitudinals	T profile	450 x12	120 x 12	517,97	671
Primary transversals	T profile	450 x 12	120 x 12	517,97	671

Finally the stiffeners of arbitrary chosen plate, is possible to see on next figure:

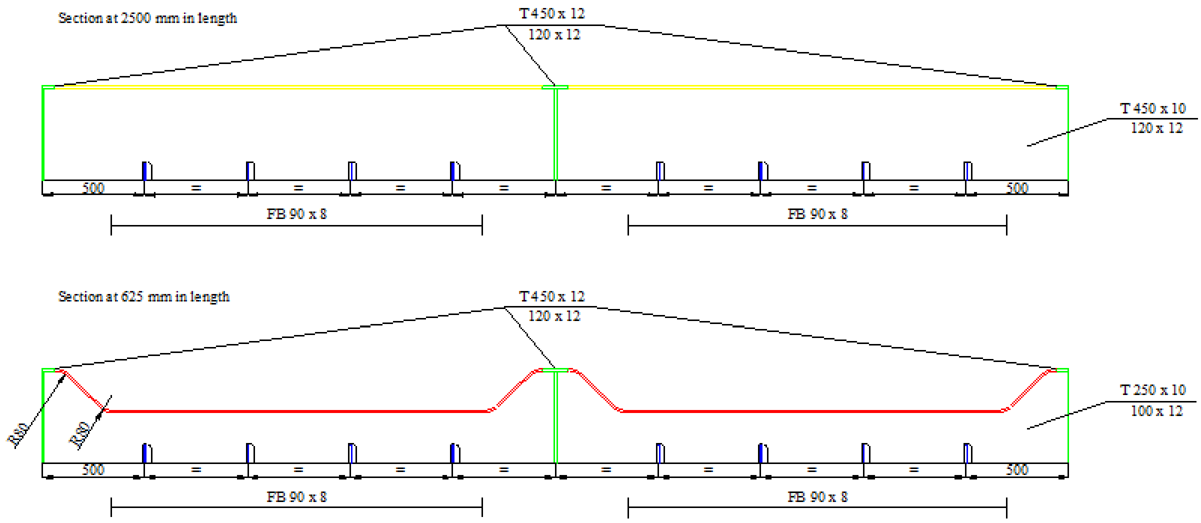


Figure 8. Profiles and dimensions for the arbitrary chosen plate

Primary longitudinal	- green
Secondary longitudinal	- blue
Primary transversals	- yellow
Secondary transversals	- red

The same principle will be applied for dimensioning of the main frame of the yacht.

2.2 Deck Structure Scantling

Sec. 7 – 7.1:

The thickness of the shell, deck and bulkhead plating is to be not less than given by the following equation:

$$t = sc\sqrt{\frac{pk}{\sigma_a}}$$

Where:

$s = 400$ mm - the spacing of the deck longitudinal

c - the correction factor for curved plating, it is not to be taken less than 0.7, however deck plate is not curved panel so factor will be 1.

$p = 0.01Fh$

h - the design head, depending about position of the plate, for deck plates:

$h = 0.04L + 1.83 \rightarrow h = 4.01$ m

F - the design head reduction factor, for deck plates:

$F = 1.102 - 0.0004s \rightarrow F = 0.942$

$p = 0.0377$

k - the coefficient varying with the plate panel aspect ratio, according to the equation:

$$k = \frac{0.5}{\left(1 + 0.623\left(s/l\right)^6\right)}$$

$l = 690$ mm - distance between two frames, this distance was chosen in agreement with the coordinator from Perini Navi

$k = 0.488$

σ_a - the design stress, for shell and deck it should be 0.6 of minimum ultimate tensile strength. For aluminum, the minimum ultimate tensile strength is for the welded condition, $\sigma_a = 0.6 \cdot 275 = 165$ N/mm^2

Thickness of the deck plate is:

$t = 4.22$ mm

After all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 4$ mm

Chosen thickness of the deck plate will be: $t = 5$ mm

Position of deck will be at 5800 mm from the center line.

Now it will be calculated section modulus of the stiffeners and according to section modulus stiffener will be chosen. There are two approaches to do this calculation. First, it is possible to start from dimensioning of the primary longitudinals, than transversals and for the last secondary longitudinals, or it is possible to go reverse, from the secondary longitudinals to the primary longitudinals. The scantling here will be done according to the second possibility; first secondary longitudinal will be dimensioned (same way to perform calculation as for the arbitrary plate). This way is chosen because of the technology for welding aluminium. The smallest thickness that is possible to weld is 4.75 mm. If scantling is done on the way that we first dimensioning the primary longitudinal and after come to secondary, obtained thickness could be less than 4.75 mm and then it is necessary to repeat all the procedure to obtain good dimension of the secondary longitudinal.

Sec. 8 – 8.1.3:

The section modulus of each floor, girder, stringer, longitudinal, frame, beam and stiffening member, in association with plating to which it is attached, is to be not less than given by the following equation:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k$$

For the deck longitudinals:

$C = 817$

$s = 0.4$ m - the spacing of the deck longitudinals

$l = 0.69$ m - length between support points

h- F x design head for the shell plating given in chapter 7.1 for the mid length location of the internal. For obtain values of F it is necessary to do to calculate:

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12 \qquad (1.4)$$

For given C_F and from next table it is possible to obtain value of F

Table 7. Values of F necessary for calculate section modulus

C_F	1,0 and greater	0,90	0,80	0,70	0,60	0,50	0,40	0,30	0,20	0,10	0,05	0 or negative
F	0,25	0,28	0,32	0,36	0,42	0,49	0,57	0,67	0,77	0,88	0,94	1,00

$F = 0.88$ and $h = 4.01$ m

$h = 3.53$ m

σ_a - the design for internals it should be 0.5 of minimum ultimate tensile strength. For aluminum, the minimum ultimate tensile strength is for the as-welded condition, $\sigma_a = 0.5 \cdot 275 = 137.5$ N/mm^2

$SM_k = 0$ -the required increase of section modulus, here is equal to zero because on the deck there are no influences of ballast keel, only bottom structure will be influenced.

$SM = 4$ cm^3

Chosen profile for the deck longitudinal will be:

Flat bar profile 70 x 6 mm $\qquad SM = 4.9$ cm^3

For the deck transversal:

$C = 817$ - for the transverse frames

$s = 0.69$ m - the spacing of the transverse frames

$l = 1.6$ m -length between support points

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.38$$

$F = 0.57$

$h = 2.29$ m

$\sigma_a = 137.5$ N/mm^2.

$SM = 24.03$ cm^3

Chosen profile for the deck longitudinal will be:

Flat bar profile 120 x 12 mm $SM = 28.8$ cm^3

For the deck primary longitudinals:

$C = 817$

$s = 1.6$ m - the spacing of the transverse frames

$l = 2.76$ m -length between support points

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.71$$

$F = 0.36$

$h = 1.44$ m

$\sigma_a = 137.5$ N/mm^2.

$SM = 104.28$ cm^3

Chosen profile for the deck longitudinals will be:

T profile: web 250 x 8 mm

flange 100 x 12 mm $SM = 133.82$ cm^3

For the deck main frames:

$C = 817$

$s = 2.76$ m - the spacing of the transverse frames

$l = 1.6$ m -length between support points

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.38$$

$F = 0.57$

$h = 2.29$ m

$\sigma_a = 137.5$ N/mm^2.

$SM = 96.14$ cm^3

Chosen profile for the deck longitudinal will be:

T profile: web 250 x 8 mm

flange 60 x 8 mm $SM = 112.43 \text{ cm}^3$

All section modulus of the profiles are calculated in Microsoft office – excel 2007. And how it is done it is possible to see from next figures.

Flat bar profile calculation of section modulus:

The web depth to thickness ratio is not exceed 59
Flange with to thickness ratio not more than 12
Flat bar stiffeners the depth to thickness ratio is in general not to exceed 12.

Flat bar profile calculation SM:

$$SM = \frac{b \cdot h^2}{6} \text{ cm}^3$$

b =	6	mm
h =	70	mm
Varifacation <12:	11,67	
SMprofile =	4900,00	mm^3
SMprofile =	4,90	cm^3

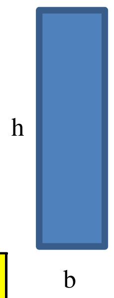
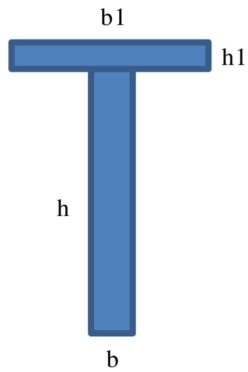


Figure 9. Section modulus for flat bar profile

T profile calculation of section modulus:



Web:		
b =	8	mm
h =	250	mm
Flange:		
h1 =	12	mm
b1 =	100	mm
Area:		
A =	2000,00	mm^2
A1 =	1200,00	mm^2
Σ A + A1 =	3200,00	mm^2
Location of the center of gravity for each element:		
z =	125	mm
z1 =	256	mm
Area * center of gravity:		
A * z =	250000	mm^3
A1 * z1 =	307200	mm^3
Σ A*z + A1*z1 =	557200	mm^3
REAL CENTER OF GRAVITY:		
zg =	174,13	mm

I for the square $bh^3/12$:		
I =	10416667	mm^4
I1 =	14400	mm^4
$Ai * (Zi - Zg)^2$:		
$A * (z - zg)^2$ =	4,83E+06	mm^3
$A1 * (z1 - zg)^2$ =	8,04E+06	mm^3
Moment of inertia:		
I =	1,52E+07	mm^4
I1 =	8,06E+06	mm^4
Σ I + I1 =	2,33E+07	mm^4
Section modulus of T profile:		
SM =	1,34E+05	mm^3
SM =	133,82	cm^3
Verifecation for web < 59:	31,25	
Verifecation for flange < 12:	8,33	

Figure 10. Section modulus for T profile

Formulas that have been used:

$$A = b \cdot h \tag{1.5}$$

$$z_g = \frac{\sum A_i \cdot z_i}{\sum A_i} \tag{1.6}$$

$$I = \frac{b_i \cdot h_i^3}{12} + A_i \cdot \left(z_i - z_g\right)^2 \tag{1.7}$$

$$SM = \frac{I}{z_g} \tag{1.8}$$

According to the ABS – Motor Pleasure Yacht 2000 *Ch7A.1.2:*

Openings in webs, girders and other structural internal members are to be arranged clear of concentrated loads or areas of high stresses. Slots in transverses and girders for longitudinals or beams in these areas are to be fitted with filler plates.

Access and lighting holes are to be arranged clear of areas of load concentration of high stresses with suitably radiuses corners. The depths of holes are generally not to exceed 0.5 times the depth of the member.

Frames in the deck will be lighter with the lighting holes in the flange of transversals. Height of the lighting holes will be 125 mm and that is 0.5 times from longitudinal that is 250 mm high. Dimension will be 300x125 and 200x125, depends about position.

Table 8. Deck profiles

Direction	Type	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Longitudinal	Primary	T	250 x 8	100 x 12	104,28	133,82
Longitudinal	Secondary	FB	70 x 6		4,00	4,90
Transverse	Main frame	T	250 x 8	60 x 8	96,14	112,43
Transverse	Frame	FB	120 x 12		24,03	28,80

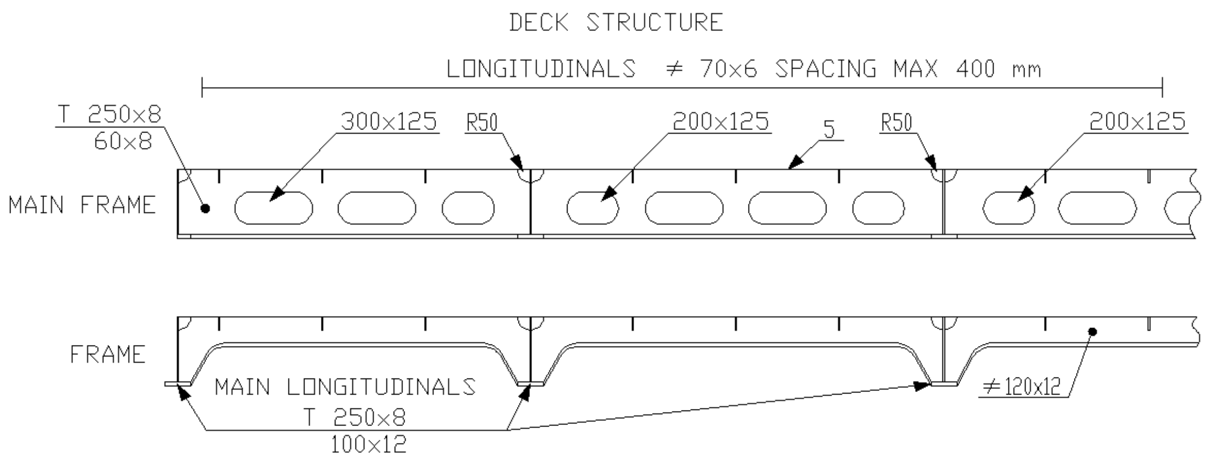


Figure 11. Deck structure

2.3 Side Structure Scantling

Sec. 7 – 7.1:

$$t = sc\sqrt{\frac{pk}{\sigma_a}}$$

Where:

$s = 330$ mm - distance between two side longitudinals

$c = 0.7$

$p = 0.01Fh$

h - the design head, depending about position of the plate, for side plates:

Basic head:

$h_1 = 3.0d + 0.14L + 1.62 \rightarrow h_1 = 15.07$ m

Design head:

$h = 0.8h_1 \rightarrow h = 12.56$ m

F - the design head reduction factor, for side plates:

$F = 3.08$ - not less than 0.8D

$p = 0.387$

k - the coefficient varying with the plate panel aspect ratio, according to the equation:

$$k = \frac{0.5}{\left(1 + 0.623\left(s/l\right)^6\right)}$$

$l = 690$ mm - distance between two frames

$k = 0.496$

$\sigma_a = 165$ N/mm^2

Thickness of the side plate is:

$t = 7.82$ mm

After all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 3.3$ mm

Chosen thickness of the side plate will be: $t = 8$ mm

Sec. 8 – 8.1.3:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k$$

For the shell longitudinals:

$C = 817$

$s = 0.33$ m - the spacing of the side longitudinals

$l = 0.69$ m - length between support points

h- F x design head for the shell plating given in chapter 7.1 for the mid length location of the internal. For obtain values of F it is necessary to do to calculate:

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12$$

$F = 0.88$ - from the Table 1.

Design head from chapter 7.1: $h = 12.56$ m

$h = 11.05$ m

$\sigma_a = 137.5$ N/mm^2

$SM_k = 0$ -the required increase of section modulus

$SM = 10.3$ cm^3

Chosen profile for the side longitudinal will be:

Flat bar profile 90 x 8 mm $SM = 10.8$ cm^3

For the side transversal:

$C = 817$

$s = 0.69$ m - the spacing of the transverse frames

$l = 1.32$ m - length between support points

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.3$$

$F = 0.67$

$h = 8.41$ m

$\sigma_a = 137.5$ N/mm^2.

$SM = 60.07$ cm^3

Chosen profile for the side transversal will be:

T profile: web 160 x 9 mm

flange 90 x 12 mm $SM = 65.46$ cm^3

For the side primary longitudinal:

$C = 817$

$s = 1.32$ m - the spacing of the primary longitudinal

$l = 0.69$ m -length between support points

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12$$

$F = 0.88$

$h = 11.05$ m

$\sigma_a = 137.5$ N/mm^2.

$SM = 41.26$ cm^3

Chosen profile for the primary side longitudinal will be:

T profile: web 160 x 7 mm

flange 80 x 12 mm $SM = 52$ cm^3

Table 9. Side profiles

Direction	Type	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Longitudinal	Primary	T	160 x 7	80 x 12	60,07	65,46
Longitudinal	Secondary	FB	90 x 8		10,30	10,80
Transverse	Main frame	T	160 x 9	90 x 12	41,26	52,00

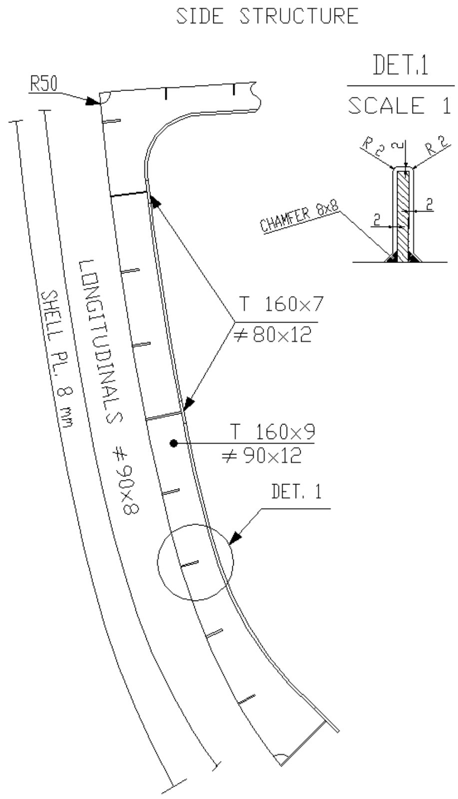


Figure 12. Side structure

2.4 Bottom Structure Scantling

Sec. 7 – 7.1:

$$t = sc\sqrt{\frac{pk}{\sigma_a}}$$

Where:

$s = 330$ mm - distance between two bottom longitudinals

$c = 0.7$

$p = 0.01Fh$

h - the design head, depending about position of the plate, for bottom plates:

Basic head:

$h_1 = 3.0d + 0.14L + 1.62 \rightarrow h_1 = 15.07$ m

Design head:

$h = 1.8h \rightarrow h = 27.126$ m - according to the Figure 13.

F - the design head reduction factor, for bottom plates:

$F = 3.85$ - not less than D

$p = 1.044$

k - the coefficient varying with the plate panel aspect ratio, according to the equation:

$$k = \frac{0.5}{\left(1 + 0.623\left(s/l\right)^6\right)}$$

$l = 690$ mm - distance between two frames

$k = 0.496$

$\sigma_a = 165$ N/mm^2

Thickness of the bottom plate is:

$t = 12.94$ mm

After all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 3.3$ mm

The bottom shell thickness is to be increased for the extent shown in next figures:

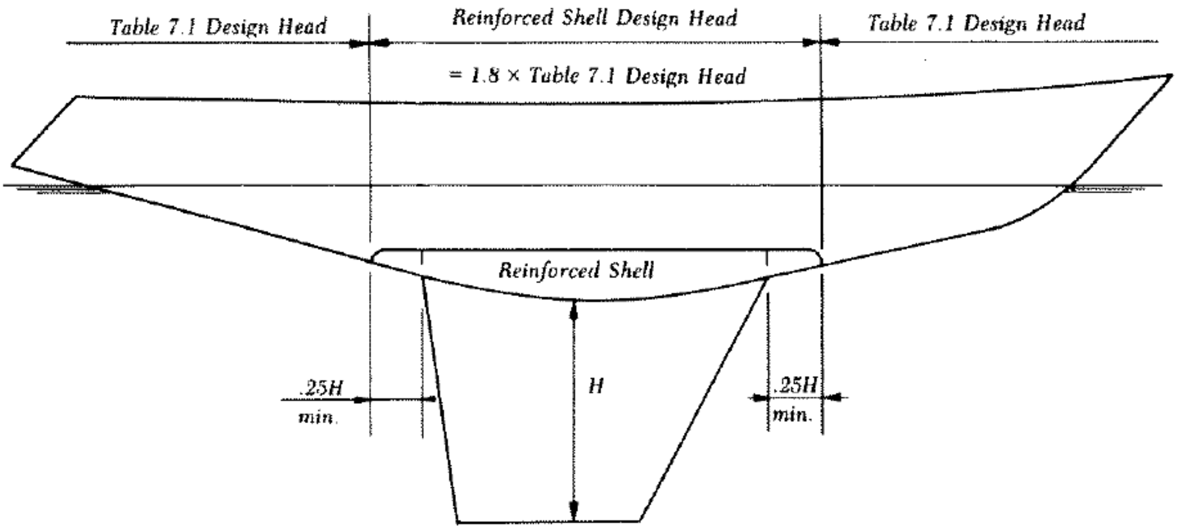


Figure 13. Profile at centerline

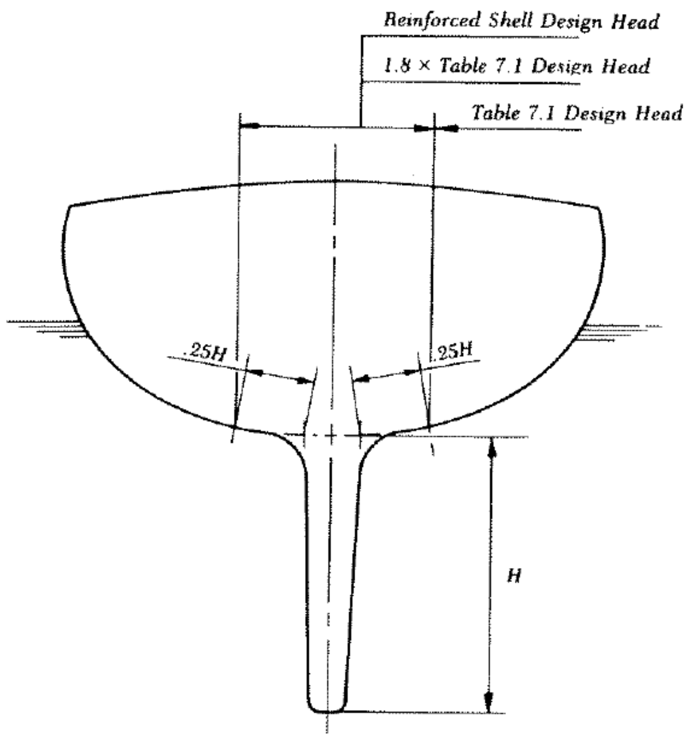


Figure 14. Transverse section

Chosen thickness of the bottom plate will be: $t = 14$ mm

Sec. 8 – 8.1.3:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k$$

For the shell longitudinals:

$C = 817$

$s = 0.33$ m - the spacing of the bottom longitudinals

$l = 0.69$ m - length between support points

h - F x design head for the shell plating given in chapter 7.1 for the mid length location of the internal. For obtain values of F it is necessary to do to calculate:

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12$$

$F = 0.88$ - from the Table 1.

Design head from chapter 7.1: $h = 12.56$ m

$h = 1.2h = 15.07$ m

$\sigma_a = 137.5$ N/mm^2

$$SM_k = \frac{NW_k Y_k}{n\sigma_a}$$ - for the floors and frames in way of the ballast keel

$N = 0.5$

$W_k = 52189$ N - weight of the ballast keel

$Y_k = 3.756$ m - vertical distance from mid-depth of floor at centerline to center of gravity of ballast keel in m

$n = 26$ - number of floors in way of keel, not less than 3

$SM_k = 12.3$ cm^3

$SM = 22.6$ cm^3

Chosen profile for the bottom longitudinal will be:

Flat bar profile 120 x 12 mm $SM = 28.8$ cm^3

Inner bottom:

Position of inner bottom will be 2950 mm from the center line.

Sec. 7 – 7.1:

$$t = sc\sqrt{\frac{pk}{\sigma_a}}$$

Where:

$s = 330$ mm - distance between two inner bottom longitudinals

$c = 1$

$p = 0.01Fh$

h - the design head, depending about position of the plate, for inner bottom plates:

Basic head:

$h_1 = 3.0d + 0.14L + 1.62 \rightarrow h_1 = 15.07$ m

Design head:

$h = 1.2h \rightarrow h = 18.08$ m

F - the design head reduction factor, for inner bottom plates:

$F = 3.85$ - not less than D

$p = 0.696$

k - the coefficient varying with the plate panel aspect ratio, according to the equation:

$$k = \frac{0.5}{\left(1 + 0.623\left(s/l\right)^6\right)}$$

$l = 690$ mm - distance between two frames

$k = 0.496$

$\sigma_a = 165$ N/mm^2

Thickness of the inner bottom plate is:

$t = 10.57$ mm

After all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 3.3$ mm

Chosen thickness of the inner bottom plate will be: $t = 12$ mm

Sec. 8 – 8.1.3:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k$$

For the shell longitudinals:

$C = 817$

$s = 0.33$ m - the spacing of the inner bottom longitudinals

$l = 0.69$ m - length between support points

h - F x design head for the shell plating given in chapter 7.1 for the mid length location of the internal. For obtain values of F it is necessary to do to calculate:

$C_F = \dfrac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12$

$F = 0.88$ - from the Table 1.

Design head from chapter 7.1: $h = 12.56$ m

$h = 11.05$ m

$\sigma_a = 137.5$ N/mm^2

$SM_k = 0$ - no influence of the ballast keel

$SM = 10.31$ cm^3

Chosen profile for the bottom longitudinal will be:

Flat bar profile 70 x 6 mm $SM = 11.67$ cm^3

Usually where main frame is positioned, in the bottom it is possible to do bulkhead between two bottom tanks. So the scantling will be done for calculation of the bulkhead in the bottom with stiffeners of the bulkhead and also when the tanks are located in bottom. Scantling of opening and stiffeners will be done.

Thickness of the bulkhead in the bottom structure:

Sec. 7 – 7.1:

$t = sc\sqrt{\dfrac{pk}{\sigma_a}}$

Where:

$s = 330$ mm - distance between two stiffeners of bulkhead

$c = 1$

$p = 0.01Fh$

h - the design head, depending about position of the plate, for bulkhead:

Basic head:

$h_1 = 3.0d + 0.14L + 1.62 \rightarrow h_1 = 15.07$ m

Design head:

$h = 0.8h \rightarrow h = 12.56$ m

F - the design head reduction factor, for side plates:

$F = 3.85$ - not less than D

$p = 0.484$

k - the coefficient varying with the plate panel aspect ratio, according to the equation:

$$k = \frac{0.5}{\left(1 + 0.623\left(s/l\right)^6\right)}$$

$l = 800$ mm - distance between bottom and inner bottom longitudinals

$k = 0.498$

$\sigma_a = 165$ N/mm^2

Thickness of the bulkhead is:

$t = 8.94$ mm

After all other requirements are met the thickness for aluminum in general, is to be not less than *s/100* or *2.5* mm whichever is greater. According to this:

$t = 3.3$ mm

Chosen thickness of the bulkhead in the bottom plate will be: $t = 10$ mm

Bulkhead stiffeners:

Sec. 8 – 8.1.3:

$$SM = \frac{Chsl^2}{\sigma_a} + SM_k$$

For the shell longitudinals:

$C = 817$

$s = 0.33$ m - the spacing of the inner bottom longitudinals

$l = 0.69$ m - length between support points

h - F x design head for the shell plating given in chapter 7.1 for the mid length location of the internal. For obtain values of *F* it is necessary to do to calculate:

$$C_F = \frac{l - 0.254}{0.0542L + 0.559} \rightarrow C_F = 0.12$$

$F = 0.88$ - from the Table 1.

Design head from chapter 7.1: $h = 12.56$ m

$h = 11.05$ m

$\sigma_a = 137.5$ N/mm^2

$SM_k = 0$ - no influence of the ballast keel

$SM = 10.31$ cm^3

Chosen profile for the bulkhead stiffener will be:

Flat bar profile 70 x 6 mm $SM = 11.67\ \text{cm}^3$

Table 10. Bottom profiles

Direction	Place	Profile	Longitudina	Stiffeners	$SM_{required}$	$SM_{profile}$
Longitudinal	Bottom	FB	120 x 12		22,60	28,80
Longitudinal	Iner bottom	FB	70 x 6		10,31	11,67
Longitudinal	Bottom bulkhead	FB		70 x 6	10,31	11,67

Where openings in the bottom are, thickness of the profile will remain equal as thickness of the bulkhead, only it is necessary to add stiffeners at edges to reduce the stress concentrations.

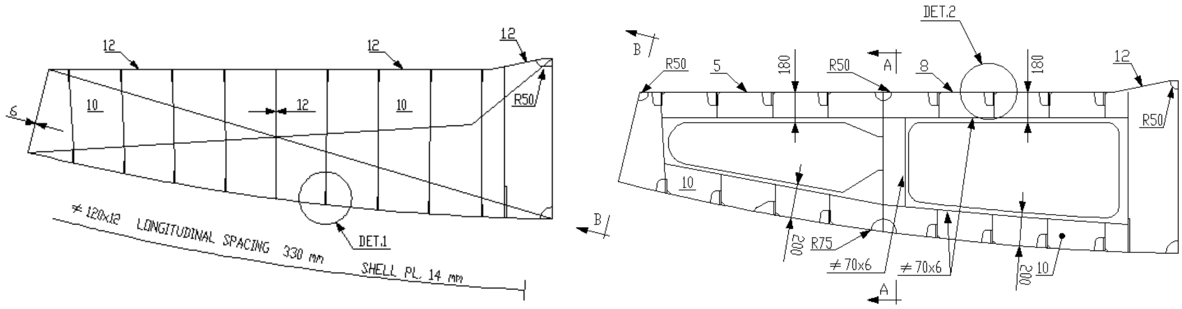


Figure 15. Bottom structure

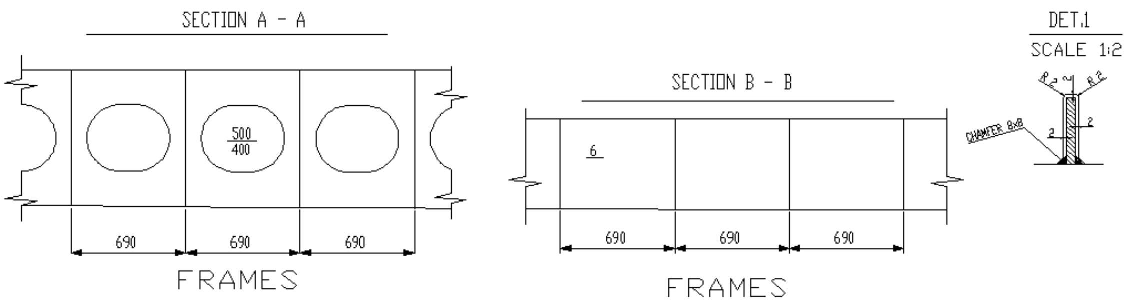


Figure 16. Details of the bottom structure

Remarks: Dimensions of the keel structure are taken from the Perini Navi and finally scantling of the main frame section is possible to see on the next Figure 17.

Detail drawing of the scantling is possible to find at last page of the thesis on A3 format.

Table 11. All sections of the main frame with corresponding dimensions

Deck structure (t = 5 mm)						
Direction	Type	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Longitudinal	Primary	T	250 x 8	100 x 12	104,28	133,82
Longitudinal	Secondary	FB	70 x 6		4,00	4,90
Transverse	Main frame	T	250 x 8	60 x 8	96,14	112,43
Transverse	Frame	FB	120 x 12		24,03	28,80
Side structure (t = 8 mm)						
Direction	Type	Profile	Web [mm]	Flange [mm]	$SM_{required}$	$SM_{profile}$
Longitudinal	Primary	T	160 x 7	80 x 12	60,07	65,46
Longitudinal	Secondary	FB	90 x 8		10,30	10,80
Transverse	Main frame	T	160 x 9	90 x 12	41,26	52,00
Bottom (t = 14 mm) and inner bottom (t = 12 mm)						
Direction	Place	Profile	Longitudinals	Stiffeners	$SM_{required}$	$SM_{profile}$
Longitudinal	Bottom	FB	120 x 12		22,60	28,80
Longitudinal	Iner bottom	FB	70 x 6		10,31	11,67
Longitudinal	Bottom bulkhead	FB		70 x 6	10,31	11,67

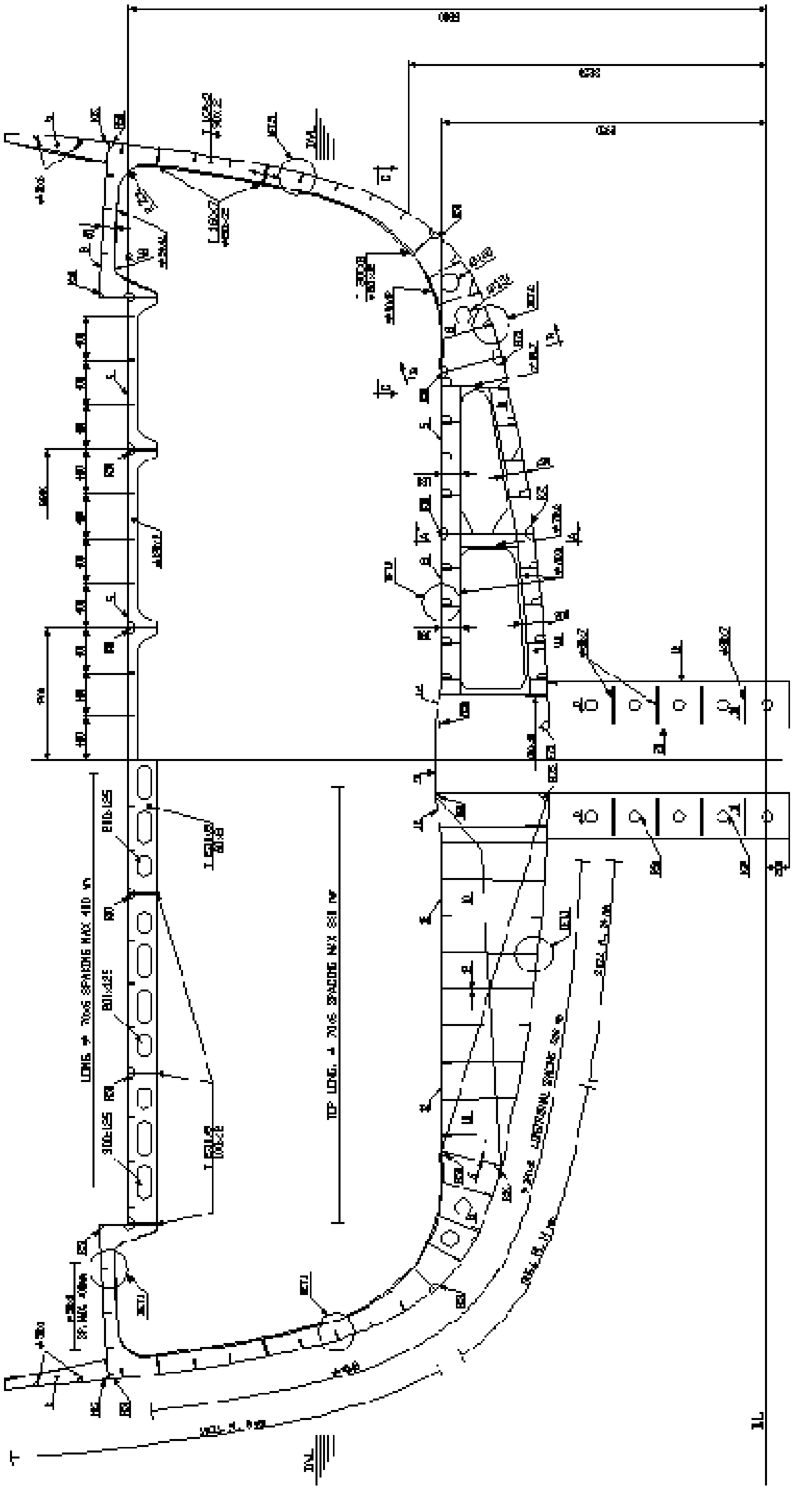


Figure 17. Main frame of the sailing yacht

3. ALUMINUM

Development of Metal Boats

Although older than fiberglass, metal boats are quite a modern development compared to wood construction. The first known all-metal boat was a riveted-iron barge built in 1787 by J. Wilkinson. Although this barge was successful, when Richard Trivithick and Robert Dickenson later proposed all-iron ships in 1809, they were met with incredulity and mirth. After all, everyone knew that iron was heavier than water and would sink. Nine years later, in 1818, first all metal commercial self-propelled boat, the *Vulcan*, was constructed at Faskine, near Glasgow. And she stayed in service until 1875. Metal hulls were now fully accepted, particularly for craft over 60 m, because they were nailed, screwed and bolted, then wooden structures could ever hope to be.

Perhaps surprisingly, experiments with aluminum vessels started not long after steel. The firs all aluminum boat is believed to be a leeboard sailboat built in 1890. Probably the first all aluminum powerboat was the *Mignon*, constructed in Switzerland in 1892, it was driven by 2-hp (1.5 kW) naphtha engine. In 1894, several 5.48 m surfboats were built of aluminum in the United States for a polar expedition. Weighing in at 170 kg compared to 773 kg for the equivalent screw fastened wood, they were first aluminum boats built in North America.

3.1. Types of Aluminum

Production of aluminum at the present time the ore, bauxite, is mined containing roughly 56 per cent aluminum. The actual extraction of the aluminum from the ore is a complicated and costly process involving two distinct stages. Firstly the bauxite is purified to obtain pure aluminum oxide known as alumina; the alumina is then reduced to a metallic aluminum. The metal is cast in pig or ingot forms and alloys are added where required before the metal is cast into billets or slabs for subsequent rolling, extrusion, or other forming operation. Sectional material is mostly produced by the extrusion process. This involves forcing a billet of the hot material through a die of the desired shape. More intricate shapes are produced by this method than are possible with steel where the sections are rolled. However, the range of thickness of section may be limited since each thickness requires a different die.

Pure aluminum has a low tensile strength and is of little use for structural purposes; therefore the pure metal is alloyed with small percentages of other materials to give greater tensile strength. There is a wide assortment of aluminum alloys and only true marine aluminum alloys will stand up to corrosion in salt water. Aluminum is also isotropic material. That is meaning that the strength is uniform in all direction. No matter which way you oriented the plate, it has the same strength up or down, fore-n-aft, diagonally, and even through the thickness of the plate [4].

There are 6 families of aluminum alloys. In marine applications belong to one of two families, 5000 series – aluminum and magnesium and 6000 series – aluminum, magnesium and silicon. Aluminum alloy we can sort in two main categories:

- Work-hardening alloys, whose characteristics are determined by rolling or extrusion operations and intermediate annealing. These alloys belong to the 1000, 3000 and 5000 series.
- Heat treatable alloys whose characteristics are determined by solution heat treatment, quenching and ageing. These alloys belong to the 2000, 6000 and 7000 series.

The standardized metallurgical conditions of extruded and rolled aluminum alloys are symbolized by a letter followed by a number of digits:

- The letter indicates the basic condition: O, F, H, T
- The first digit denotes the method of manufacture or type of heat treatment
- The following digits indicate the degree of purity or the treatment methods

Table Families of aluminum alloys

Alloy element	Series
None	1000
Copper	2000
Manganese	3000
Magnesium	5000
Magnesium and silicon	6000
Zinc and magnesium	7000

Table Aluminum Boatbuilding Alloy Physical Properties

Alloy	Temper	Form	UTS, psi x 1.000 (mPa)	Yield, psi x 1.000 (mPa)	Elongation
5083	H111	extrusions	40 (276)	24 (165)	16%
	H321	sheet&plate	44 (303)	31 (214)	16%
	H323	sheet	45 (310)	34 (234)	16%
	H324	sheet	50 (345)	39 (269)	16%
	H321	extrusions	41 (283)	29 (200)	16%
5086	H111	extrusions	35 (241)	21 (145)	16%
	H112	plate	44 (241)	16 (193)	14%
	H32	sheet&plate	40 (276)	28 (234)	12%
	H34	drqwn tube	44 (303)	34 (131)	10%
5054	H111	extrusions	33 (227)	19 (131)	14%
	H112	extrusions	36 (214)	12 (83)	18%
	H32	sheet&plate	39 (248)	26 (179)	10%
	H34	sheet&plate	42 (269)	29 (227)	10%
5056	H111	extrusions	42 (289)	26 (179)	18%
	H112	extrusions	41 (283)	19 (131)	22%
	H321	sheet&plate	46 (317)	33 (227)	16%
	H323	plate	48 (331)	36 (248)	16%
	H324	sheet	53 (365)	41 (283)	16%
6061	T6	sheet&plate	42 (289)	35 (241)	17%
	T6	extrusions	38 (262)	35 (241)	17%
	T6	rod&bar	42 (289)	35 (241)	17%
	T6	drawn tube	42 (289)	35 (241)	17%
	T6	pipe	42 (289)	35 (241)	17%

Note : Modulus of elasticity E = 68.981 mPa

UTS = Ultimate Tensile Strength

Elongation is a percent of a 50 mm sample.

Material of the boat will be from group 5083 and that group has the highest strength non-heat-treatable alloy in commercial use. It has good formability and welds ability and retains excellent tensile strength in the weld zone by virtue of its as-rolled properties. It is used most often in structures requiring high weld efficiency and maximum weld strength. 5083 also has excellent resistance to corrosion. And from that group it is chosen the H321 type of aluminum. H321 have next chemical properties.

Table Aluminum 5083 – H321 Chemical properties

Component	Wt. %	Component	Wt. %	Component	Wt. %
Al	92.4 - 95.6	Mg	4 - 4.9	Si	Max 0.4
Cr	0.05 – 0.25	Mn	0.4 - 1	Ti	Max 0.15
Cu	Max 0.1	Other, each	Max 0.05	Zn	Max 0.25
Fe	Max 0.4	Other, each	Max 0.15		

Physical properties:

Density is $\rho = 2.66\ \frac{\text{kg}}{\text{m}^3}$

Melting range $570 - 640\ ^{\text{o}}\text{C}$

Table Aluminum 5083 – H321 Mechanical properties

Hardness, Brinell	85	Hardness, Knoop	109
Hardness, Rockwell A	36.5	Hardness, Rockwell B	53
Hardness, Vickers	96	Ultimate Tensile Strength	317 MPa
Tensile Yield Strength	228 MPa	Elongation at Break	16%
Modulus of Elasticity	70.3 GPa	Compressive Modulus	71.7 GPa
Poissn's Ratio	0,33	Fatigue Strength	159 MPa
Shear Modulus	26.4 GPa	Shear Strength	190 MPa

3.2. Advantages of Aluminum:

1. Light aluminum. Aluminum is lighter than steel for the same strength. For example aluminum plate should be between 1.25 to 1.5 times thicker than steel for the same strength. Aluminum weighs 2.691 kg/m^3 versus steel 7.849 kg/m^3 and that is only 34 percent of steel weight. Even taking the larger thickness's multiplier of 1.5, this means that aluminum is about half the weight for the same strength.

2. More stable and faster aluminum boats. The light weight obtainable from aluminum construction lowers the centre of gravity of a boat, making it more stable and thus more seaworthy. Less weight also means you can go faster with the same power or sail area, or have a higher ballast ratio in a sailboat for more sail area and improved performance. Alternately you can use less power for the same speed or get greater range with the same tankage.

3. Aluminum superstructure on steel hulls. It is so difficult to make steel light enough and most of vessels must use wood, FRP[2], or aluminum superstructures rather than

[2] FRP – **F**iber-**R**einforced **P**olymer

steel. If they don't, it is nearly impossible to keep the boat's center of gravity low enough for adequate stability.

4. Light weight equals labor – saving. The lighter weight of the components makes building in aluminum less labor-intensive than steel. For example steel plate of 37 m^2 would weight 1360 kg and would require careful handling and heavy gear. But for the same dimension, aluminum plate would weigh just 640 kg.

5. Easier to work in aluminum. Aluminum is softer and easier to bend, cut, and form. It cuts about three times faster than steel. Aluminum bends so easily, that round bilge hulls are little problem. It can be cut with ordinary woodworking equipment, and quickly and easily drilled, sanded and filed to exact dimension.

6. No compromise on the hull shape. The freedom to build inexpensively nearly any hull shape in aluminum means that the hull form doesn't have to be compromised with developable surfaces or chines when they are not wanted. This leads to more efficient hydrodynamics for better performance and increased seakindliness.

7. Faster welding. Welding aluminum is roughly three times faster than welding steel. Even allowing for somewhat heavier welds (i.e. more passes) for the comparably thicker aluminum plate of the same strength, the total hours in welding aluminum should be about half that for a similar steel hull.

8. No rust, lower maintenance. Aluminum doesn't rust at all. It can corrode when it is in contact with dissimilar metals or from stray electrical currents. Aluminum is so corrosion resistant and totally rust free that it doesn't even have to be painted above the waterline or on the inside.

9**. No added plate thickness to allow for corrosion.** Rusting wastes away the steel plates and the plates are getting thinner with age and that is not case with aluminum. Steel will lose about 0.1 mm of thickness every year.

10. Aluminum is nonsparking and nonmagnetic. Being nonsparking makes aluminum safer both in the building shop and in operation. Fires can't be ignited by the friction spark of some heavy object falling or scraping against the aluminum structure. The lack of magnetic interfaces is a great plus for navigation and electronics.

11. Aluminum is less sensitive to stress risers. Aluminum's plastic deformation offers another benefit as well. Steel is particularly sensitive to sharp corners in construction. This is called notch sensitivity, the notches or sharp corners are called stress risers.

12. No attack by bacteria. Aluminum hulls are not subject to attack by sulfate reducing bacteria. These little known bacteria can accumulate in bilges, ballast tanks and fuel tanks, usually in commercial vessels and can eat their way through 8 mm of steel plate in a year. They can cause catastrophic corrosion quickly if not detected.

13. Reducing labor costs compensate for increased material costs. The much greater ease of working with and forming aluminum substantially reduces labor costs. Usually the labor cost reduction combined with the lower total weight of metal purchased can be enough to offset the substantially higher cost of the aluminum itself.

3.3. Disadvantages of Aluminum

No material is perfect and in spite of the many pluses listed previously, aluminum does have some drawback compared to steel.

1. Aluminum is considerably more expensive

2. Less abrasion resistant. Aluminum hulls are less resistant to abrasion. Steel is the best material when abrasion is in the question. And because of that steel is the best material for build tugs, canal boats, barges and dredges.

3. Aluminum can melt and burn in a fire. For a structural metal, aluminum has a low melting point, at approximately 592 ^oC. It can even burn in an exceptionally intense fire. Steel with melting point of approximately 1427 ^oC is the only truly fireproof boatbuilding material. But again comparing with a wood aluminum is more fire resistant than wood.

4. Welding equipment is more expensive. The gas shielded welding equipment required for aluminum is more expensive than the stick/electrode welding used for steel. And the best for welding aluminum is in the enclosed building. If not, breezes will blow the gas shield away from the arc, causing defective welds.

5. Qualified workers and equipment are harder to find.

6. Aluminum alloy is harder to locate. Aluminum alloy are frequently tough to find in many regions outside of North America and Europe. Even where aluminum suppliers and manufactures are common, it can be difficult to purchase the sizes and quantities of material necessary for small boat project.

4. FINITE ELEMENTH METHOD

A ship structure is, in general, so huge and statically indeterminate that 3-D analysis is not easy. For the strength analysis of the structure methods can be categorized as:

Analysis using simple beam theory

Analysis of frame structure: Slope deflection method

Analysis of three dimensional structure: Finite element method

The finite element method (FEM) is an essential and powerful tool for solving structural problems not only in the field of shipbuilding but also in the design of the most industrial products and even in non-structural fields. FEM can be used for a wide variety of problems in linear or nonlinear solid mechanics, dynamics, and ships structural stability problems, in accordance with the development of computer technology and its popularization.

The conventional method in solving stress and deformation problems is an analytical one using theories of beams, columns and plates, etc. Hence its application is restricted to most simple structures and loads. On the other hand FEM:

- Divides a structure into small elements
- Assumes each element to be a mathematical model
- Assembles the elements and solves the overall

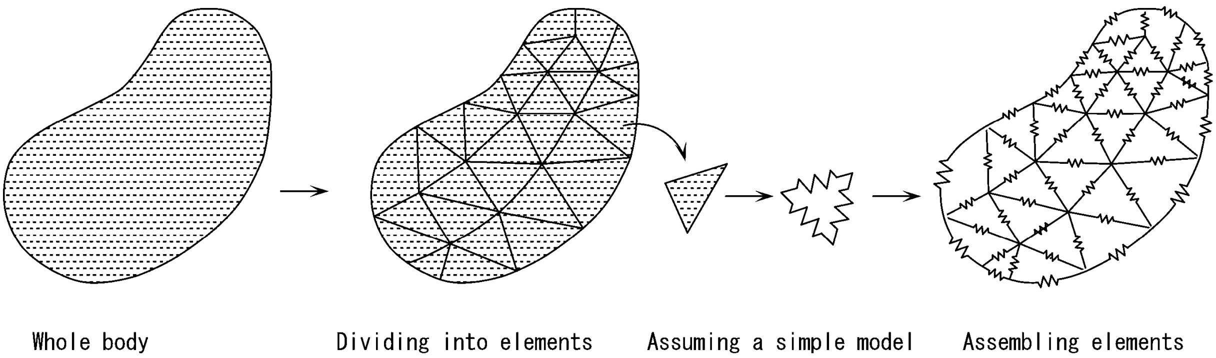


Figure 18. Methodology of FEM [5]

Typical elements of FEM are:

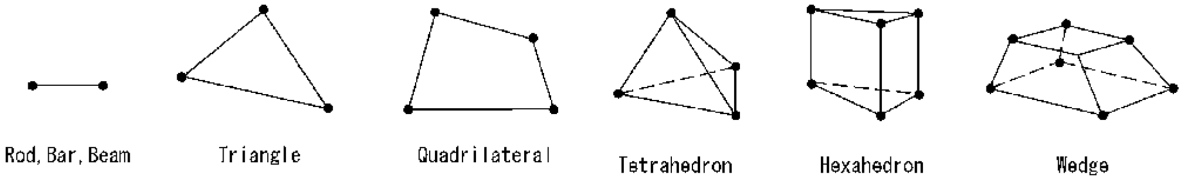


Figure 19. Elements types in FEM [5]

Characteristics of FEM are as follows:

- It doesn't give exact solution but solves approximately, because structures are modeled as a combination of simple elements and/or loads.
- It is a kind of numerical experiment without experimental devices, models, or instruments. Hence it is economical and time-savings.
- It can solve actual structural problems by using some models, although their shapes and loads are complex. It is even used for non-structural problems.
- It is used for a wide variety of steel, nonferrous materials and complex materials.
- It relies on computer technology for both hardware and software
- It is applicable as a black box tool; even engineers do not have to know the theory, because there are many general purpose FEM programs which are easy to operate.

How to approach to the FEM analysis is the best to understand from the next figure.

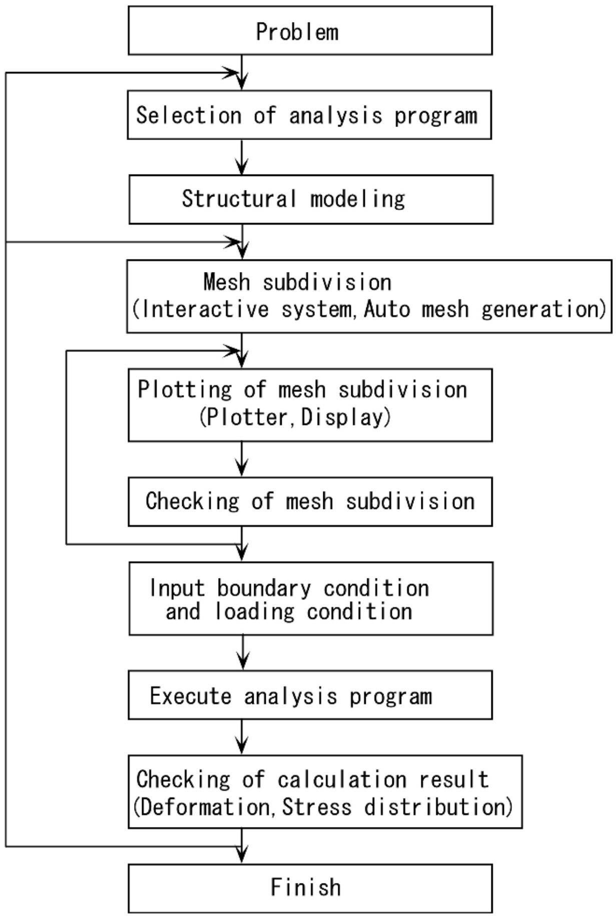


Figure 20. Approach to FEM analysis [5]

So the first step is to choose appropriate analysis program for the specified problem, then the modeling is done by determining the appropriate size of the structure. If model is symmetric it is possible to reduce the size of the model by choosing suitable boundary conditions and loading conditions. After, it is necessary to prepare the geometrical data of the finite elements with visual checking of the validity of the input data through the computer display. The loading dates as well as the boundary conditions of the structure are then added. Than execute the program for analysis, calculated results must be assessed to check whether could be some error in the input date in view of the calculated deformation, stress, etc. If there was mistake in the input or misunderstanding of the problem the procedure should be repeated from the beginning [5].

5. FINITE ELEMENTH METHOD ANALYSIS

For the analysis of the structure next software's have been used; models have been done in SolidWorks 2010 and ANSYS Workbench version 13.0, multi-purpose analysis software, was chosen to perform the modeling of the models.

SolidWorks is a 3D mechanical CAD (computer-aided design) program that runs on Microsoft Windows. It was founded in 1993 with headquarters in Waltham, Massachusetts, USA. That is a parasolid-based solid modeler, and utilizes a parametric feature-based approach to create models and assemblies.

ANSYS is an engineering simulation software (computer-aided engineering, or CAE), with headquarters in Cononsburg, Pennsylvania, USA. It has the ability to handle a very well as perform all the analysis that was needed for the project. It can vary mesh densities throughout model to allow for run-optimization, create high-precision results based on the stress fields that developed and the model could handle arbitrary geometries. FEM analysis will not be performed for the all hull structure; instead it will be done just for some parts of the structure. All parts that will be analyzed in this chapter are parts of the 58 m sailing yacht, 2193 series, built in Perini Navi shipyard.

Figure 21. Sailing yacht 2193 series [6]

From the next Figure 22 it is possible to see the parts that had been analyzing in Perini Navi. Those are:

- Main mast base foundation
- Keel – Structure connection

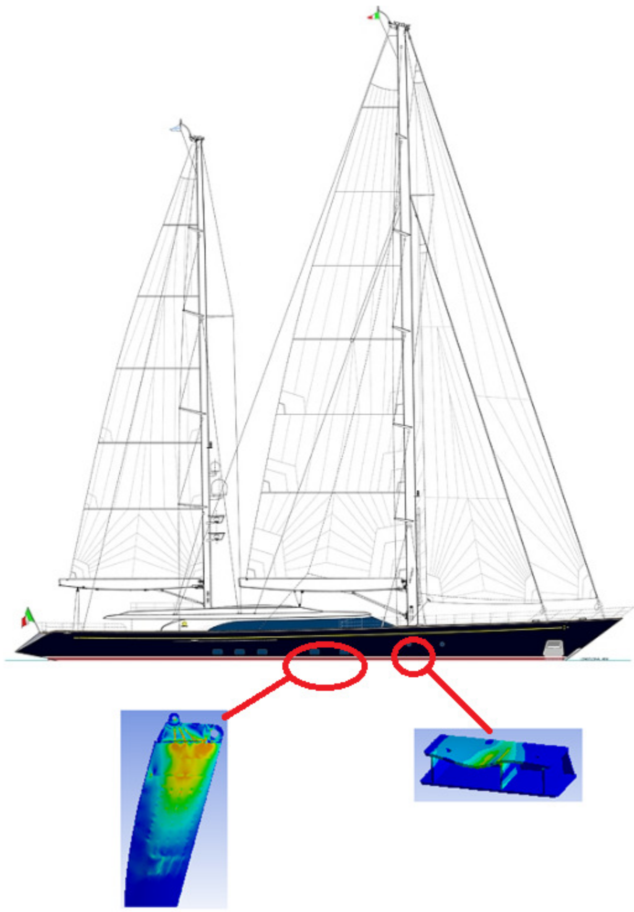


Figure 22. Parts of ship hull structure on which it were performed FEM analysis

5.1. Main Mast Base Foundation

Main mast base foundation was first project that was done in Perini Navi shipyard. This project was chosen to be the first because of the construction simplicity and in beginning it is necessary to become familiar with the software, to verify the entire program features that are necessary to simulate reality and to obtain good results, however it was needed to set up the parameters to perform the calculations. From the shipyard's point of view, it was interesting to perform such analysis because at the ship shown on Figure 21, main mast base foundation was already built and in meanwhile the mast design was changed, because of that, the new loads coming from the mast was changed. And the existing main mast base foundation was not design for that new mast, new loads. So it was necessary to obtain what are the stresses in existing main mast foundation coming from the new mast and to solve a problem, if problem does exist. At Figure 23 are shown elements of the main mast base foundation structure. It is compose of one bottom and top plate, one longitudinal and six brackets. Because of the construction and load symmetry, model will be cut at longitudinal and only half of the model will be analyzed.

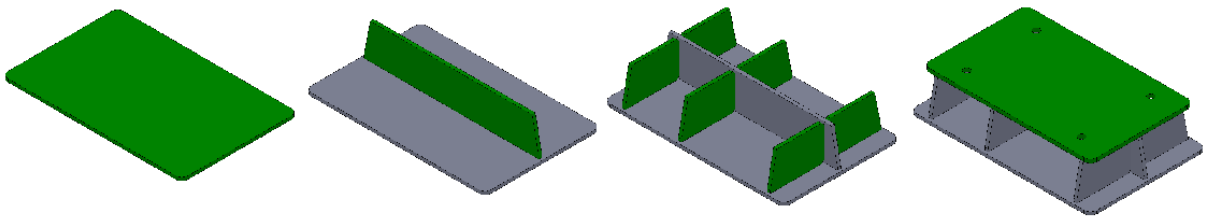


Figure 23. Parts of main mast base construction

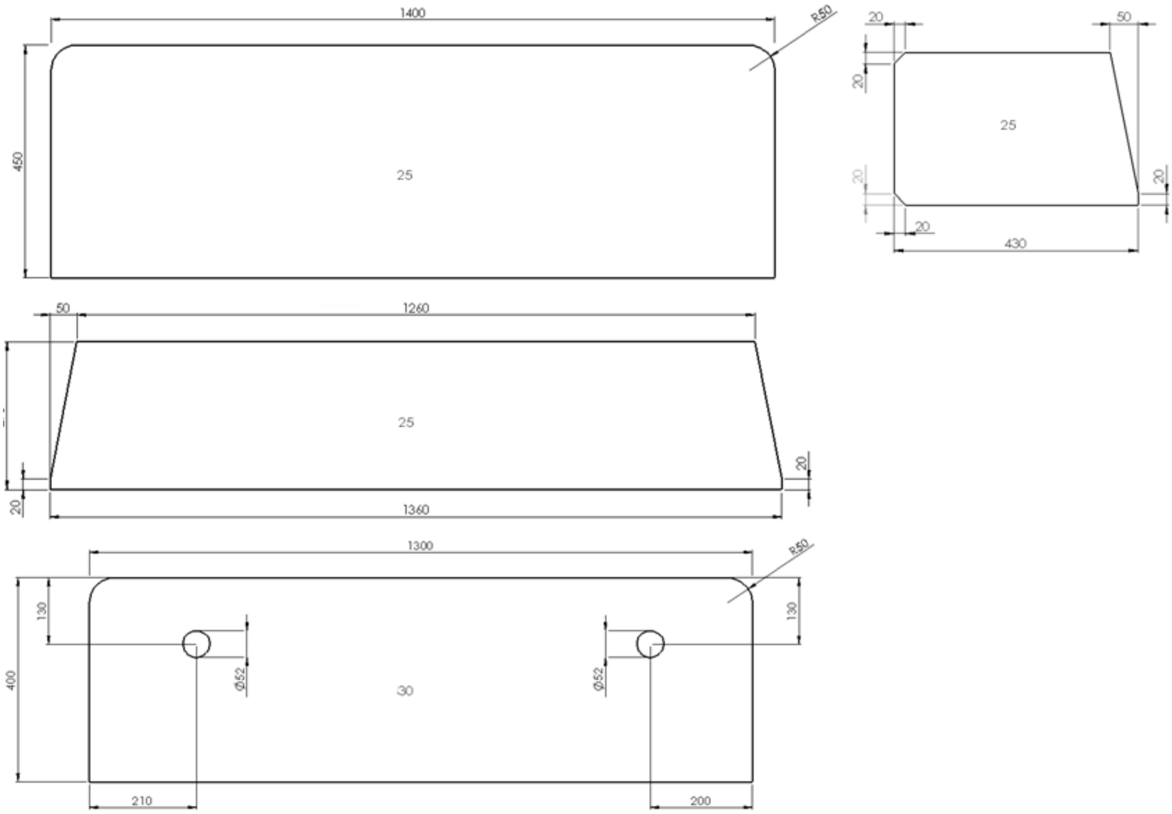


Figure 24. Dimension of the main mast base foundation

Foundation is subjected to two different load cases. First and main load is coming from the main mast on the structure and its value is 3167 kN. Second load case is coming from the jacks acting on main mast base, there are two jacks and force on each jack is 1100 kN (Figure 25).

Two load cases:

- Main load – from mast 3167 kN
- Load on jack – 1100 kN

Boundary condition is set on the bottom plate and it will be fixed support on the bottom plate and symmetry is set around longitudinal stiffener or around z axis.

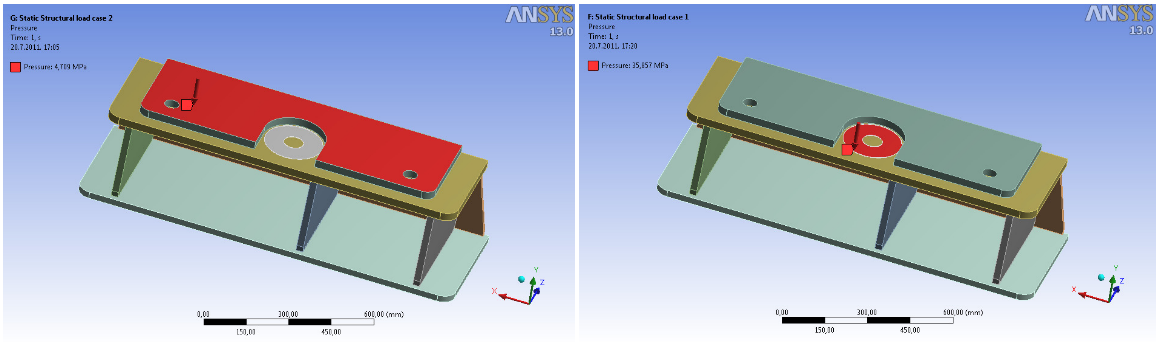


Figure 25. Visualization of the first and second load case of main mast base foundation

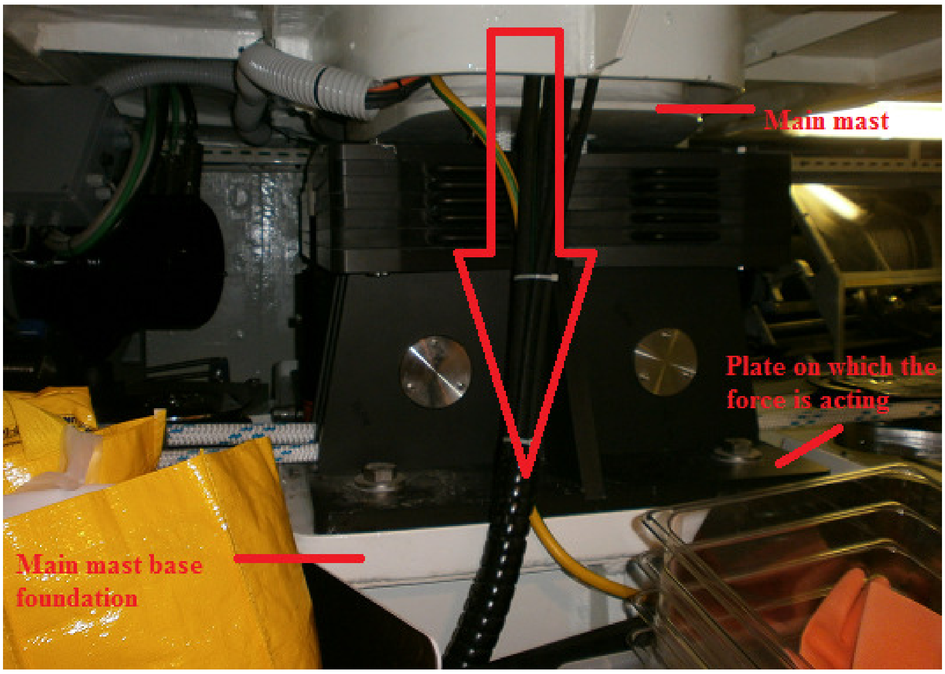


Figure 26. Connection of the mast with the structure for the first load case

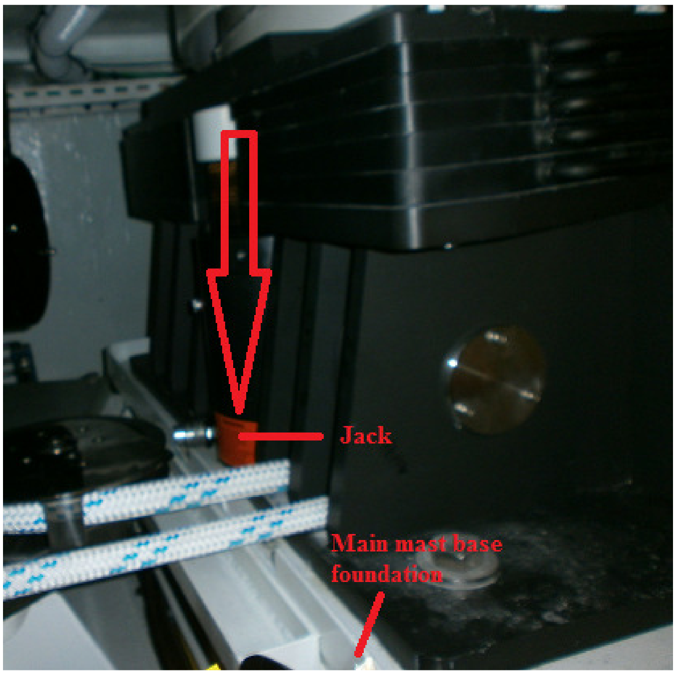


Figure 27. Connection of the mast with the structure for the second load case

5.1.1. Comparison between first and second load case

In this part of the project it is necessary to check structure for both load cases. Main goal is to find the worst load case for the structure and after, when all features are defined and good mesh is selected; final model will be subjected to the worst load case to verify the results. In beginning model will be made with solid elements and the mesh is possible to see from next Figure 28.

Any finite element program, including ANSYS, works by taking a large object and dividing into many, smaller components. This process is called meshing, and the density of the mesh is the number of elements, or smaller objects, created by meshing. The mesh properties impacts both the accuracy and computation time of the model. The mesh density can vary through-out the model and it is good practice to have a high density mesh in locations were stresses are changing rapidly over small areas, and a low density mesh where stresses are changing more gradually.

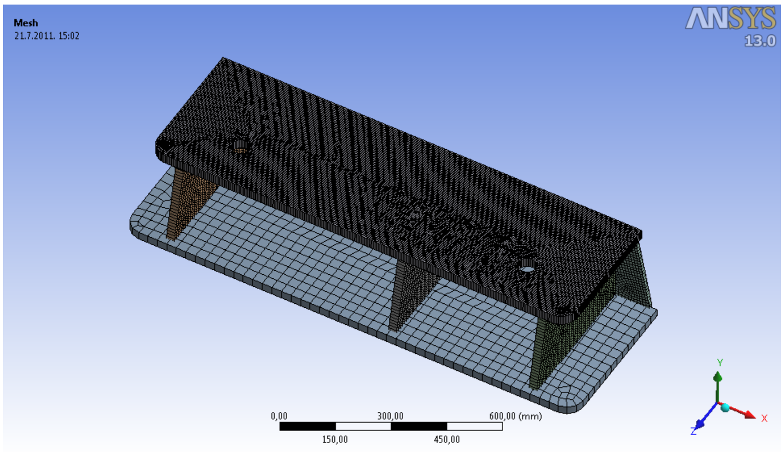


Figure 28. Final mesh of the model

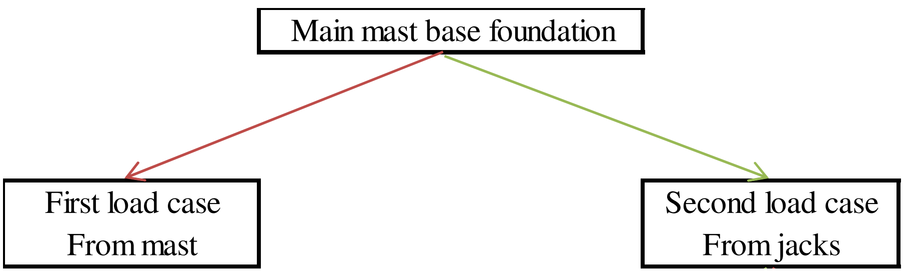


Figure 29. First step of calculations

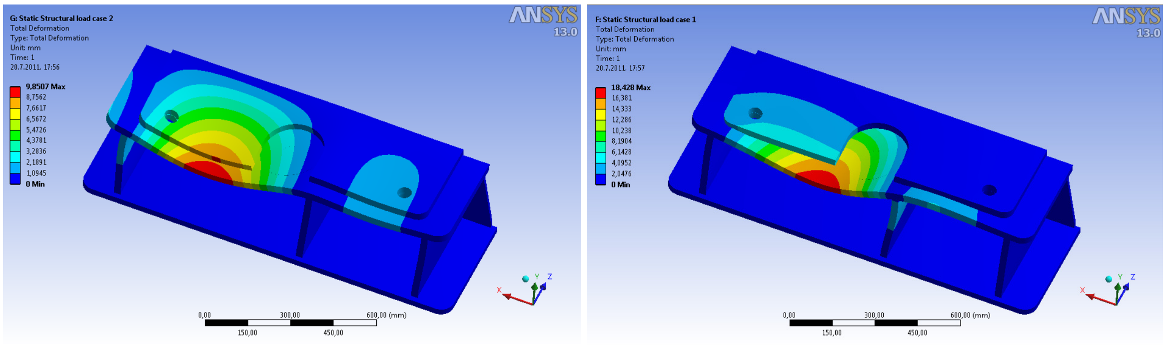


Figure 30. Total deformation for the first and second load case

From Figure 30 it is possible to see that deformation is almost twice bigger when second load case is acting on structure. For second load case total deformation is 18.42 mm and for the first load case 9.85 mm.

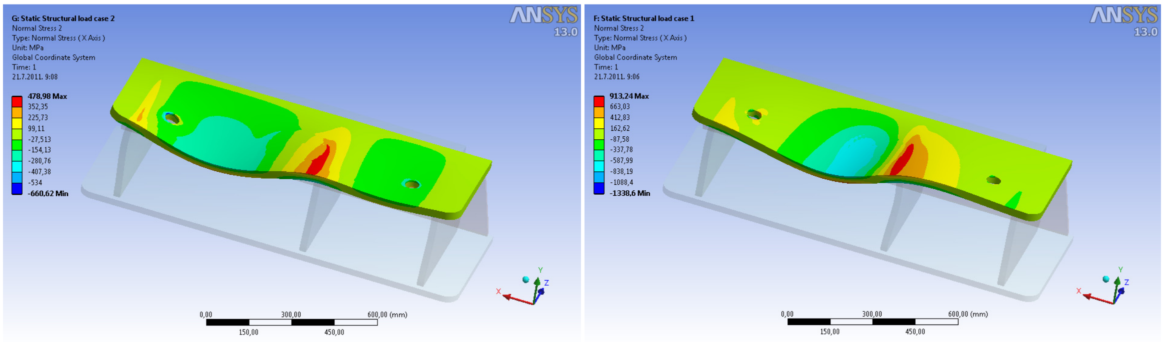


Figure 31. Normal stress in the top plate for the second load case

Normal stress is twice bigger in top plate for second load case and that load case will be applied for the next research of the structure.

Stresses and strains have six components in ANSYS, x, y, z, xy, yz and xz. For stresses and strains, components can be required under Normal (x, y, z) and Shear (xy, yz, xz).

Comparison:	First load case	Second load case
Total deformation	9.85 mm	18.42 mm
Normal stress	479 MPa	913 MPa

5.1.2. Comparison between shell and solid element

Model will be modelled and verified for two types of elements:

- Shell element

Model is made with element shell 181. Shell 181 is suitable for analyzing thin to moderately thick shell structure. It is four node element with six degrees of freedom at each node. Shell 181 is well suited for linear, large rotation and large strain nonlinear applications.

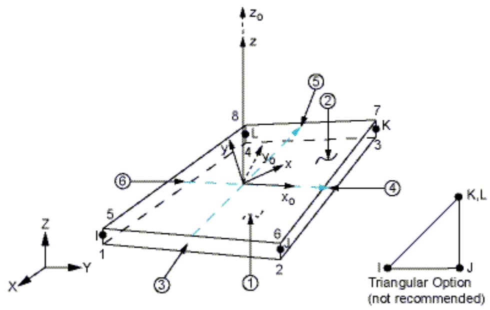


Figure 32. Shell 181 geometry

- Solid element

  Solid 186 elements have compatible displacement shapes and are well suited to model curved boundaries. The element is a high order 3-D 20 nodes solid element that exhibits quadratic displacement behaviour. The element is defined by 20 nodes having three degrees of freedom per node; translation in the nodal x, y and z directions. The element may have any spatial orientation. Solid 95 has plasticity, creep, stress stiffening, large deflection and large strain capabilities.

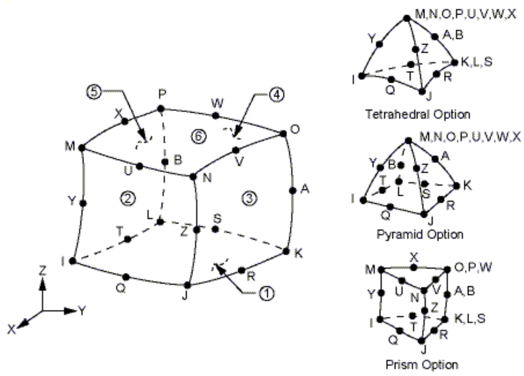


Figure 33. Solid 186 geometry

Connections of elements – wherever the model had a break in either the geometry or the mesh density, or both, it was necessary to add a connection. If neither of these boundaries existed the elements generated could simply have their nodes tied to each other: causing them to distort together, as they shared the same location. When a break in the geometry or mesh density occurred, the nodes no longer lined up with each other, and could not be automatically paired. Even when the locations where identical, if the nodes represented elements from different mesh sizes or object an additional connection was still needed to link the nodes together structurally.

In second step it is necessary to verify the results of shell and solid elements to the second load case of main mast base foundation.

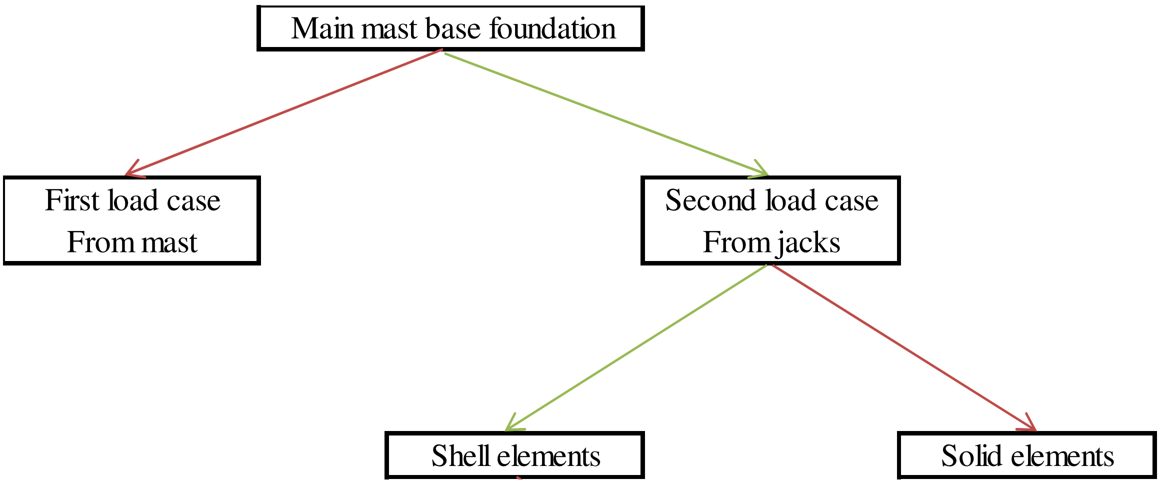


Figure 34. Second step of calculations

This kind of calculation is necessary to perform, because if results are similar it is better to work with shell elements. With shell element is much faster to perform analysis, because model will be modelled with smaller number of elements and that is leading to the faster calculation or faster CPU time.

Next results will be compared:

- Von – Mises stress
- Normal stress
- Shear stress
- Total deformation
- Error – because of too seldom mesh

One remark about figures of the results, it is possible to notice that figure for shell and solid are looking quite the same, from the thickness of element point of view. Actually graphical presentation for the results of the models is always with true thickness of the model and never just as a surface. But that models are really from surfaces and solids, that will be proven at the next Figure 35 and Figure 36.

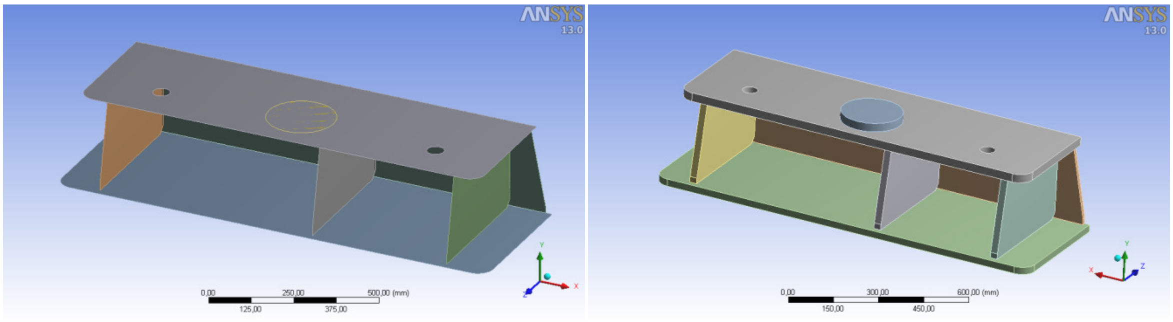


Figure 35. Shell - left, solid - right

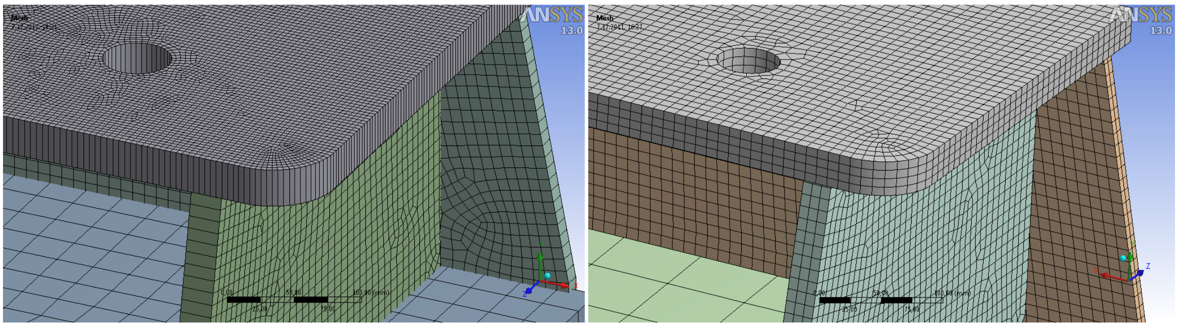


Figure 36. Mesh of shell – left, solid – right

At Figure 36 it is possible to notice that for shell elements there are just one element in the thickness and for the solid, the number of elements in the thickness depends about the element size.

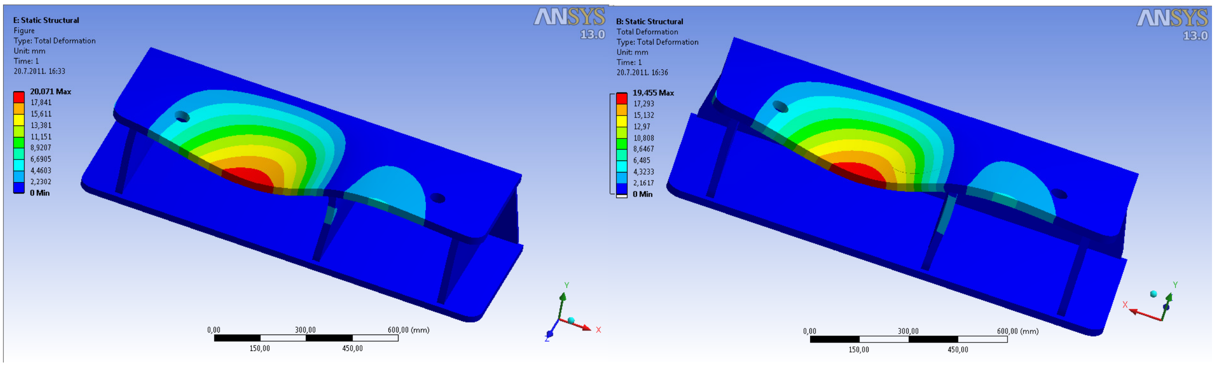


Figure 37. Total deformation, shell – left, solid – right

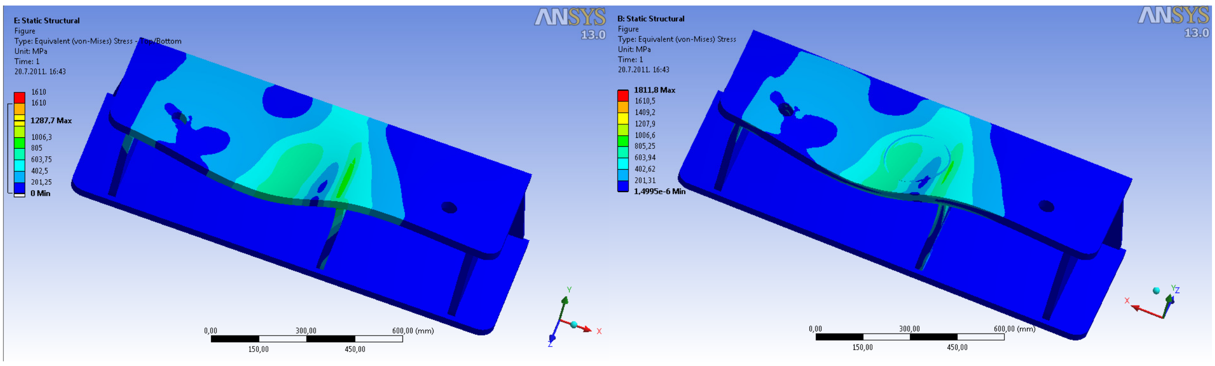


Figure 38. Von-Mises, shell – left, solid – right

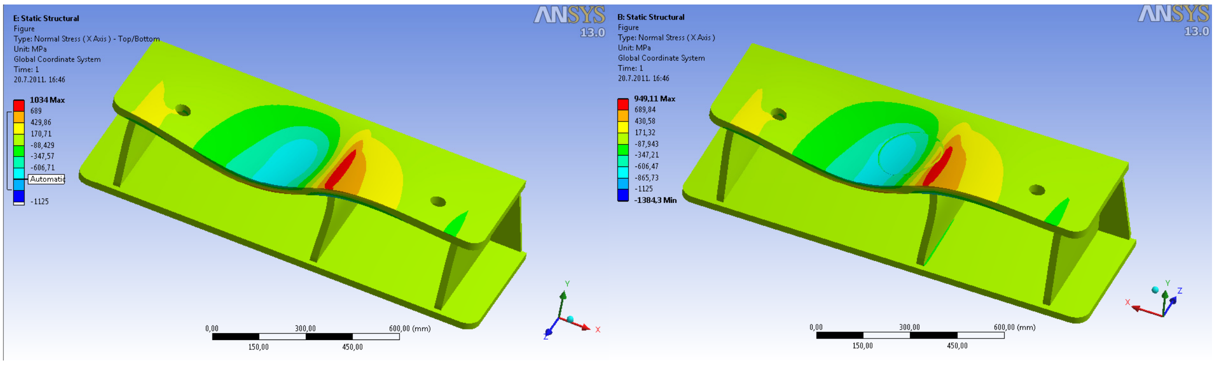


Figure 39. Normal stress, shell – left, solid – right

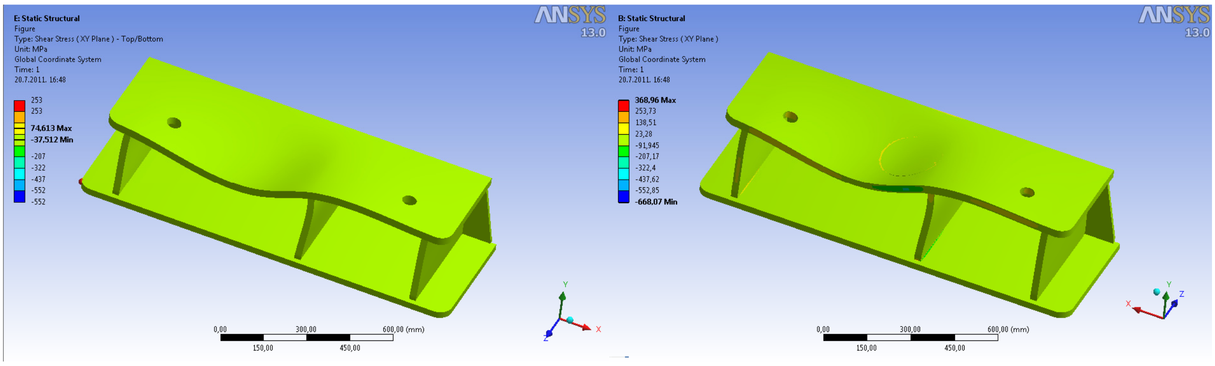


Figure 40. Shear stress, shell – left, solid – right

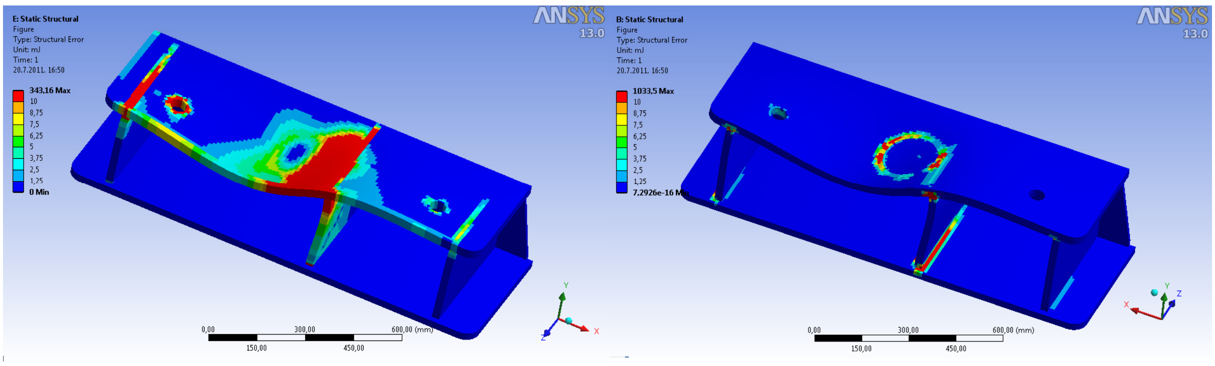


Figure 41. Total error, shell – left, solid – right

Comparison:	Solid element	Shell element
Total deformation	19.45 mm	20.07 mm
Von-Mises	1812 MPa	1610 MPa
Normal stress (tension)	949 MPa	1034 MPa
Normal stress (compression)	1384 MPa	1125 MPa
Shear stress	368 MPa	253 MPa

The most interesting area is top plate; therefore the main focus and observation of results will be on the top plate.

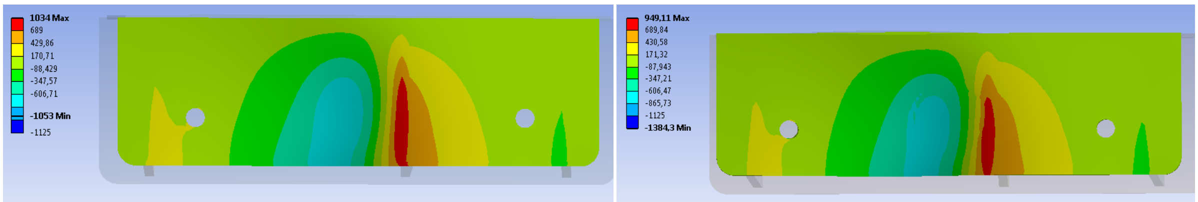


Figure 42. Top view, shell – left, solid - right

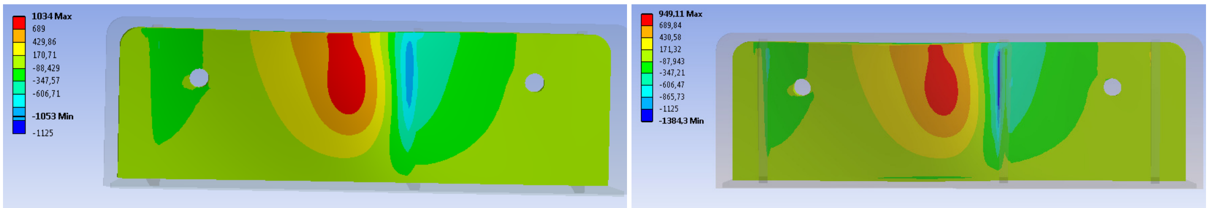


Figure 43. Model with shell elements – bottom view

From all these photos above, it is possible to conclude that both of elements types, shell or solid are giving similar results. According with this comparison, shell elements will be applied for the final model calculation.

From Figure 41 is also possible to notice that shell elements are making bigger errors then solid elements. So with shell elements it will be necessary to work with denser mesh, however even with denser mesh, number of elements will be less than with the solids and calculations will be faster. Also at this structure with this load case the shear stress isn't big and in next models it will be take in consideration only the Von-Mises stress and normal stress, and still every time it will be verified shear stress too and if it will be necessary to mention it in some case, it will be done too.

5.1.3. Linear analysis

A linear material model assumes stress to be proportional to strain. That means it assumed that the higher the load applied, the higher the stresses and deformation will be, proportional to the changes in the load. It also assumes that no permanent deformations will result, and that once the load has been removed the model will always return to its original shape.

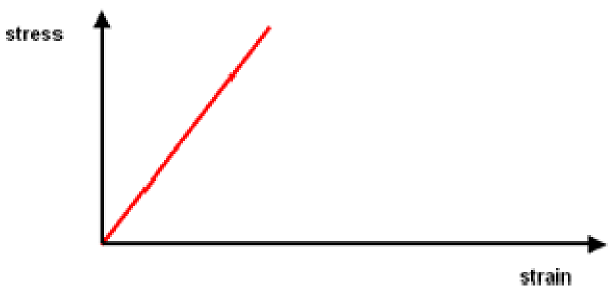


Figure 44. Stress – strain relation for linear analysis

From all results that are obtained before, model will be made from shell elements and it will be subjected to second load case.

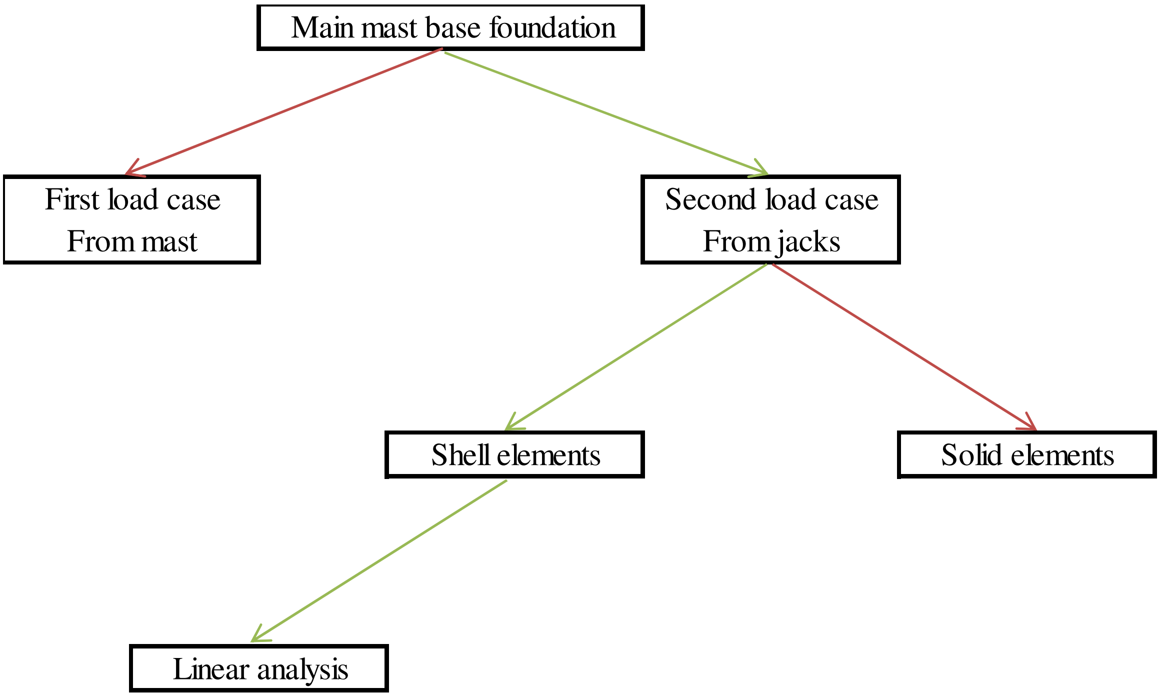


Figure 45. Third step of calculations

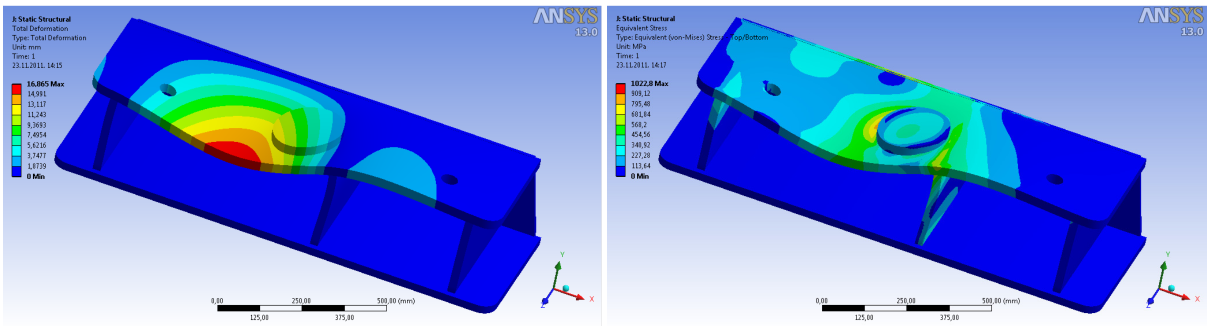


Figure 46. Total deformation and Von-Mises, linear analysis

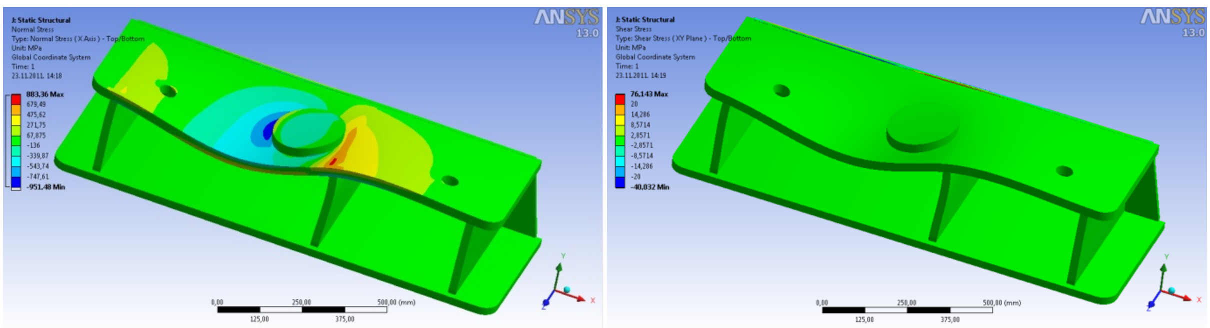


Figure 47. Normal and shear stress, linear analysis

As it possible to notice the stresses are bigger than yielding of material Von-Mises stress is 1000 MPa and Normal stress is 750 MPa. To verify the construction it is necessary to perform nonlinear analysis of the structure.

Nonlinear analysis

Nonlinear material has proportional relation between stress and strain until material doesn't reach yielding. Than material is losing elasticity and it is becoming plastic material.

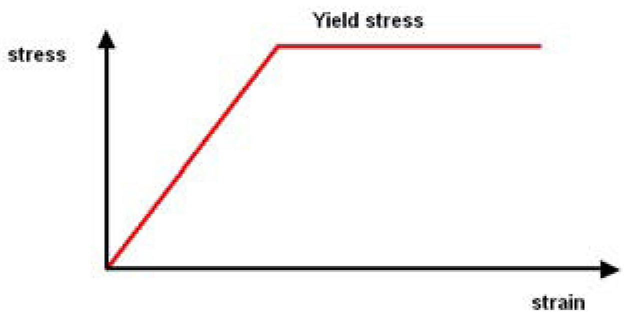


Figure 48 Stress – strain relation for nonlinear analysis

Everything is remaining the same as it was on the model for linear analysis, only in ANSYS Workbench it will be selected the nonlinear material. For obtain the final results, it is necessary to increase load in steps. The chosen number of steps is 10, that is meaning that force will start with 10% of its value and by every step it will be increased by next 10%, until it reach the maximum value. And in the next figures are shown results of nonlinear analysis.

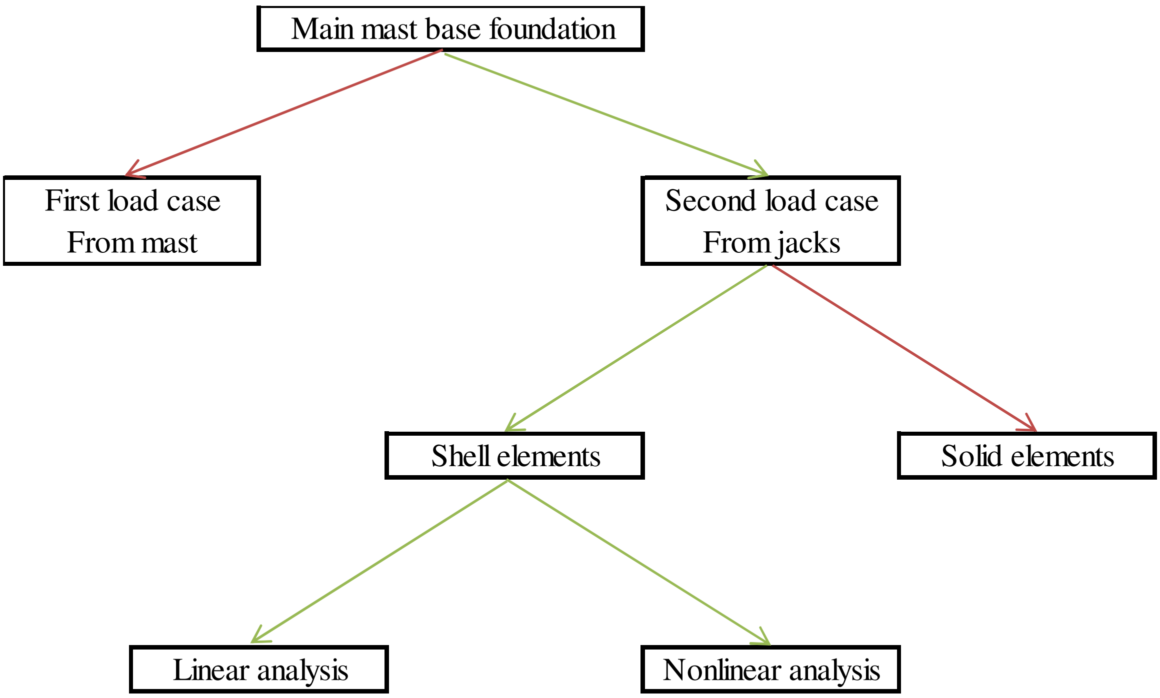


Figure 49. Fourth step of calculations

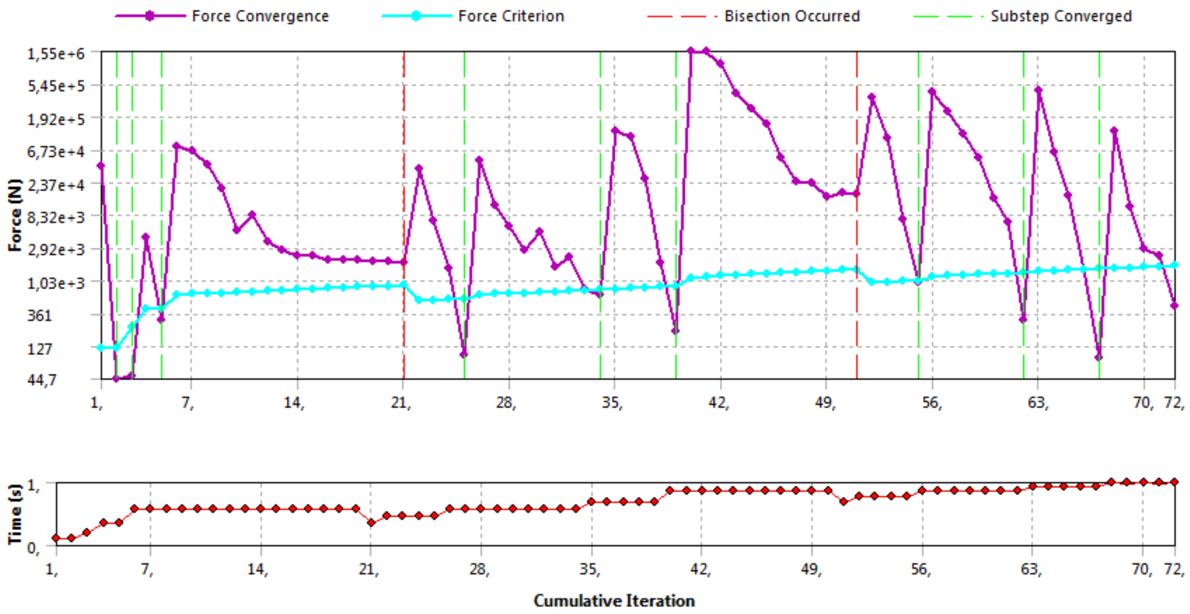


Figure 50. Nonlinear analysis, convergence of the results

Graph of convergence doesn't look like typical graphs of convergence. However convergence is reached. How does ANSYS work in this situation, it start with 10% of the force, and when the force convergence (pink line) go under the force criterion (light blue line) that means that convergence for that 10% of the force is reached and now force will be increased with next 10% (green line). It is possible to notice that on the graph, there are nine green lines and that is exactly 100% of the force, if it was started with 10% and each step was 10% more. Also on the graph there are two places where bisection was occurred (red line), that meaning the force couldn't reach the convergence and on the place like that, value of the force will go back 10% to the previous value and step of increasing for next convergence will be 5% instead of the 10%.

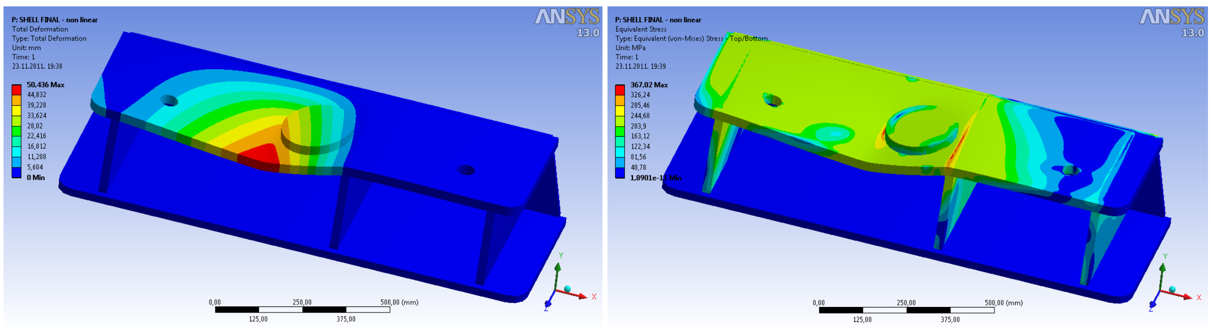


Figure 51. Total deformation and Von-Mises stress, nonlinear analysis

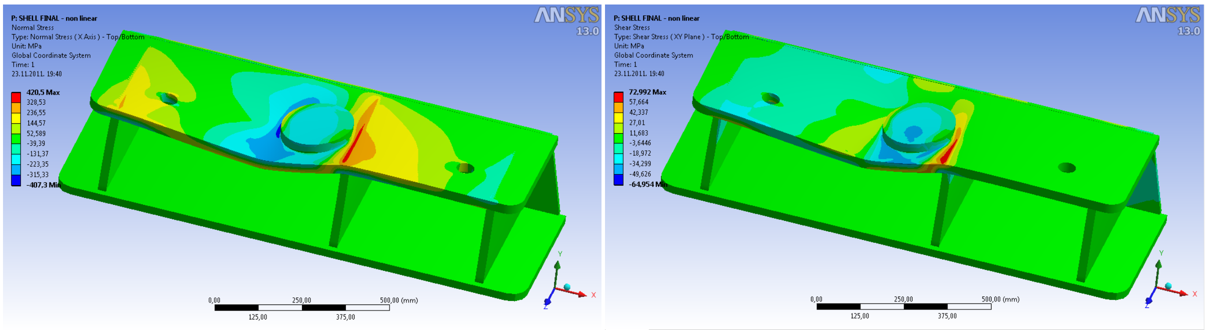


Figure 52. Normal and shear stress, nonlinear analysis

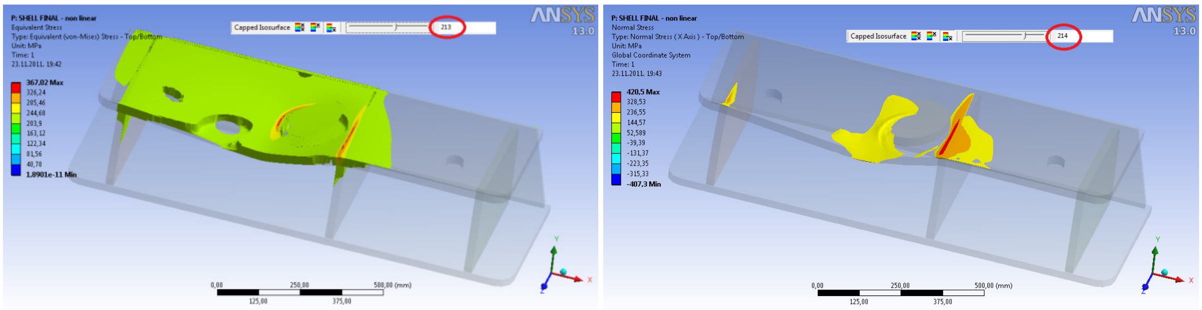


Figure 53. Places the where stresses are bigger than yielding (215 MPa), Von-Mises – left, Normal stress - right

From results it is for notice that deformations are much bigger than in linear analysis and that is prove that nonlinear material was used. Also the value of stresses is reduced and however still stresses are bigger than yielding. Von-Mises stress is 368 MPa and Normal stress is 420 MPa. It is possible to conclude that this structure is not good for this kind of load and it is necessary to do some refit of the structure.

Comparison:	Linear analysis	Nonlinear analysis
Total deformation	16.86 mm	50.43 mm
Von-Mises stress	1022 MPa	368 MPa
Normal stress	883 MPa	420 MPa
Shear stress	76 MPa	72 MPa

5.1.4. Solution of the problem, for existing structure

For existing structure additional bracket will be added under the most critical area, where the load acts on the structure. Different position of the bracket will be investigated.

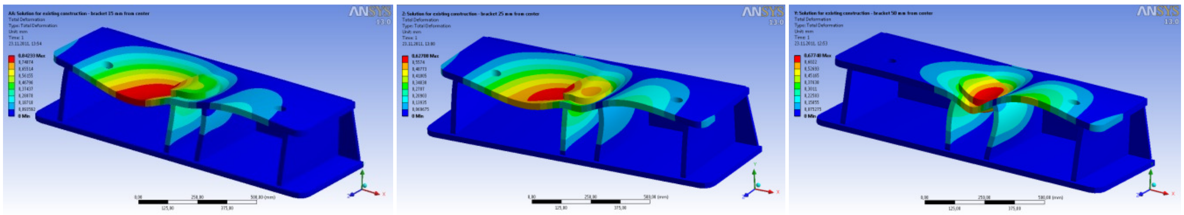


Figure 54. Total deformation of the structure with additional bracket in different positions

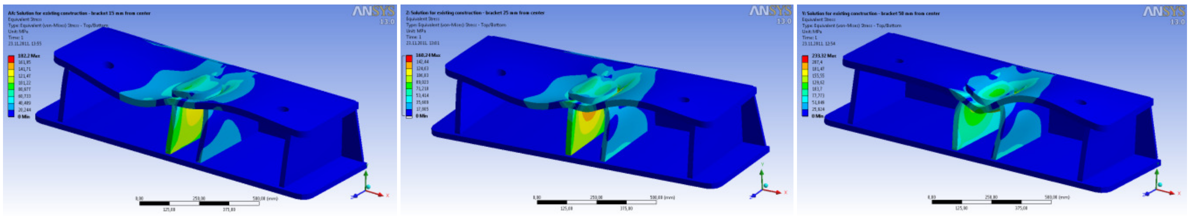


Figure 55. Von-Mises stress of the structure with additional bracket in different positions

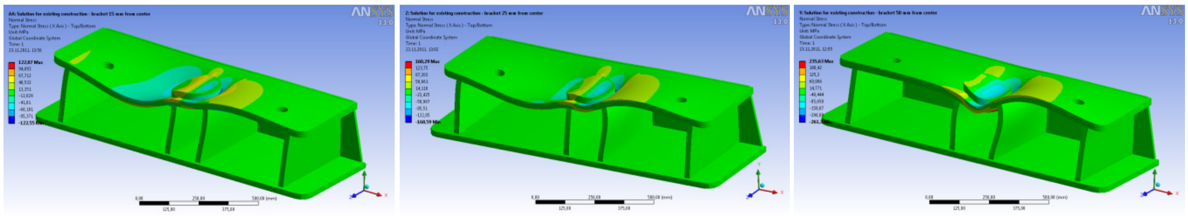


Figure 56. Normal stress of the structure with additional bracket in different positions

Exact position of additional bracket is checked with few models and the best position is selected. Notice that bracket will be moved from middle to the side where are the biggest deformations.

Position of the bracket	Deformation	Von-Mises	Normal stress
In the middle	1,401 mm	218 MPa	139 MPa
15 mm from the middle	0,842 mm	182 MPa	122 MPa
20 mm from the middle	0,725 mm	169 MPa	142 MPa
25 mm from the middle	0,627 mm	160 MPa	160 MPa
30 mm from the middle	0,550 mm	169 MPa	177 MPa
50 mm from the middle	0,677 mm	233 MPa	235 MPa

All of these models were made from shell elements, and influence of bottom plate was neglected. Fixed constraint was applied on the bottom plate and that constraint forbids the movement of mesh in any direction. Now for the final model it is necessary to check influence of additional piece bracket and construction below bottom plate. On this model additional piece will have next dimension: 200x200x275. This kind of piece will be verified for the situation when it is not possible to weld the same bracket on the structure that is already on the boat. So this piece will be inserted in the structure. Connection in the ANSYS for this bracket will be "no separation". This contact setting is similar to the bonded case. It only applies to regions of faces. Separation of faces in contact is not allowed, and small amounts of frictionless sliding can occur along contact faces.

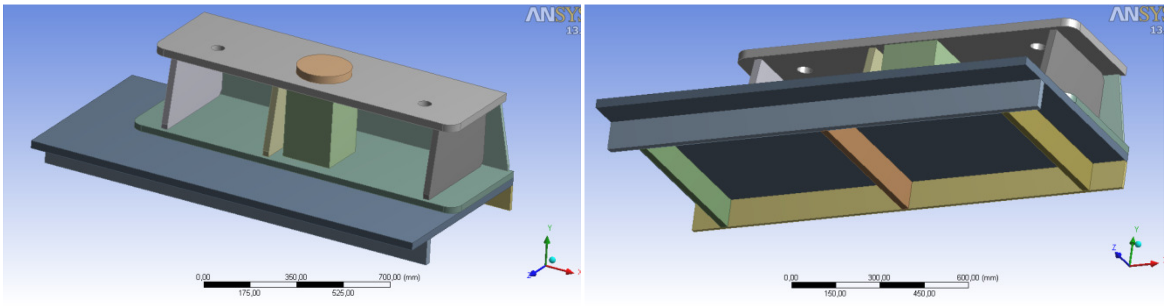


Figure 57. Connection of main mast base foundation with ship structure

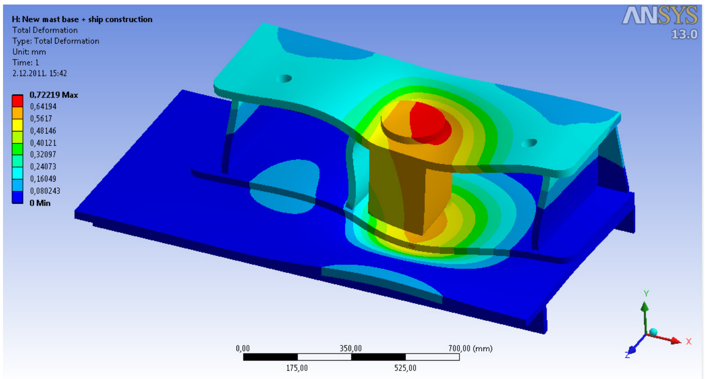


Figure 58. Total deformation of the ship structure

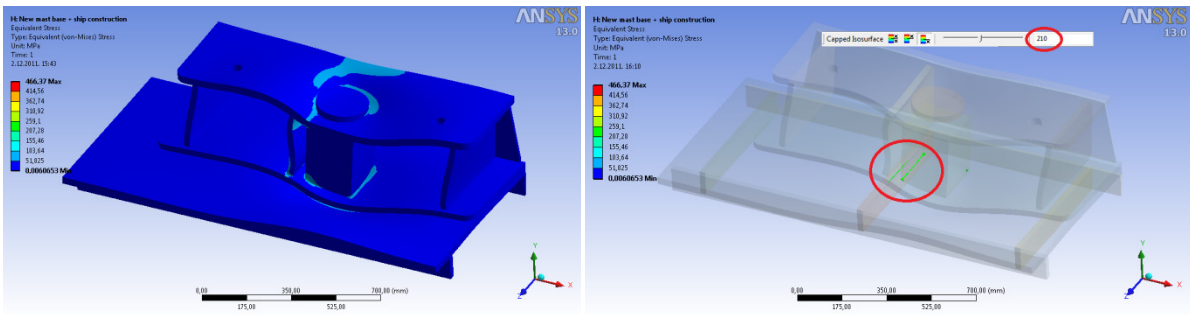


Figure 59. Von-Mises stress of the ship structure

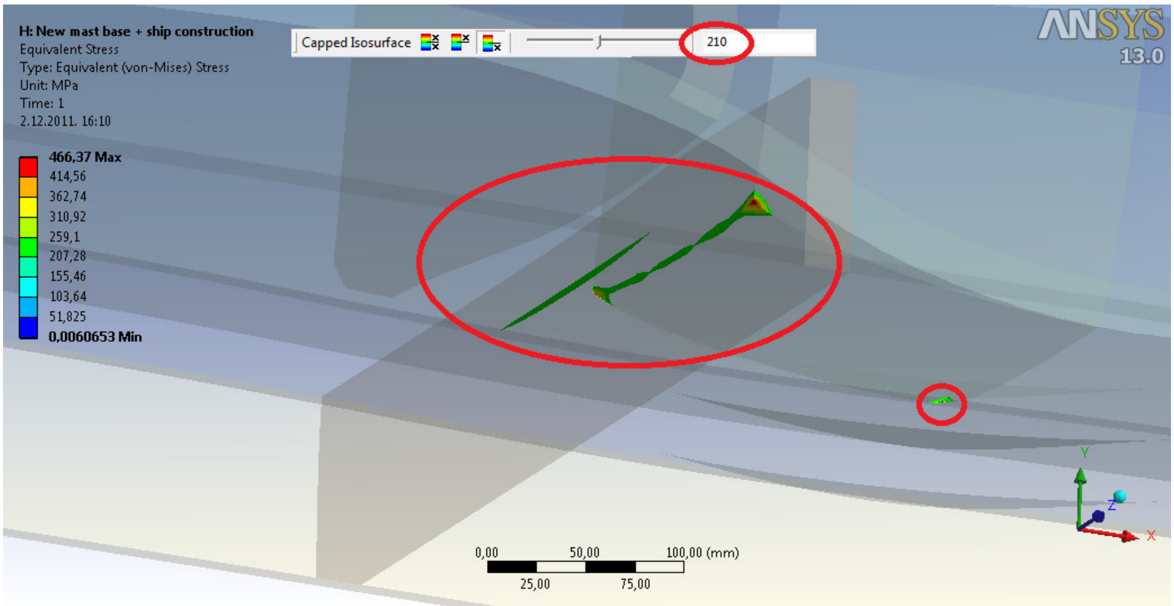


Figure 60. Places where Von-Mises stress is bigger than 210 MPa

It is for notice that stress is the biggest at the edges. That is happening because; on those places the mesh cannot reach the convergence because of the sharp edge. So the result will be acceptable and the structure does not exceeding yielding.

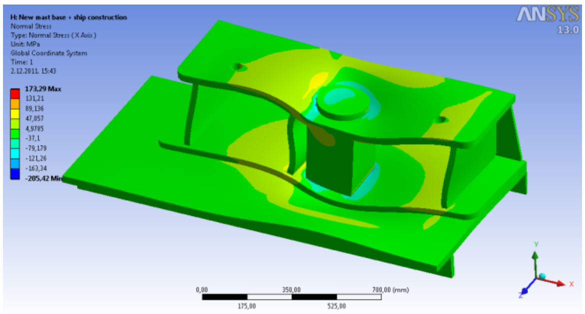


Figure 61. Normal stress of the ship structure

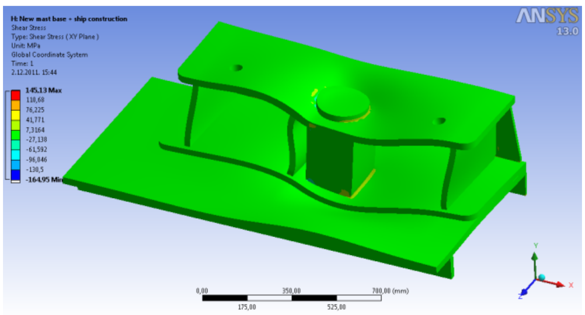


Figure 62. Shear stress of the ship structure

From all the figures above it was shown that mast base foundation with additional piece is applying and transmitting stresses very well to surrounding structure.

5.1.5 Solution for the new construction

In this case, middle bracket will be moved to the centre of plate and it will be verified does it is possible to withstand to the load acting from the jack.

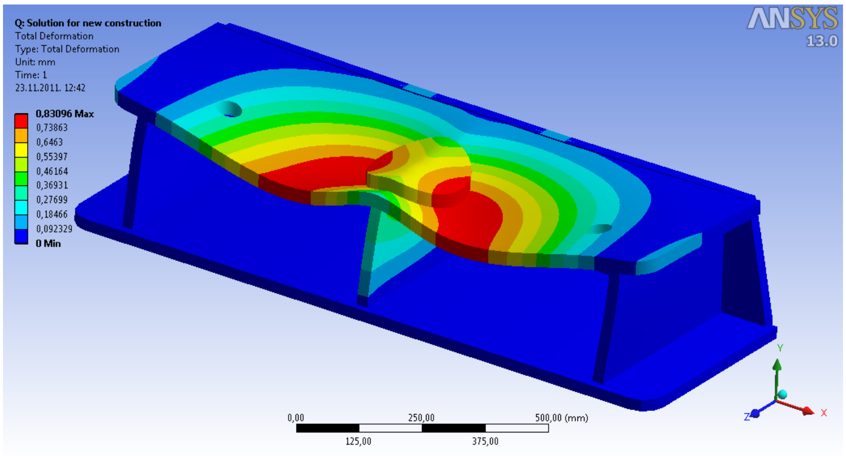


Figure 63. Total deformation, new construction

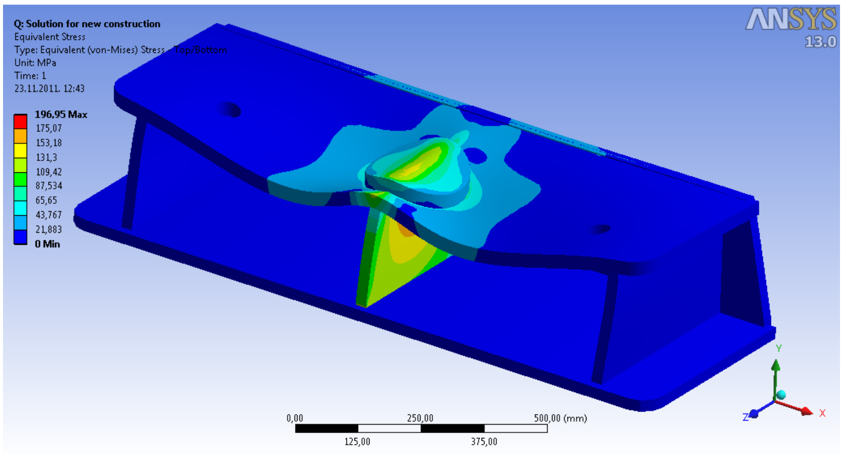


Figure 64. Von-Mises stress, new construction

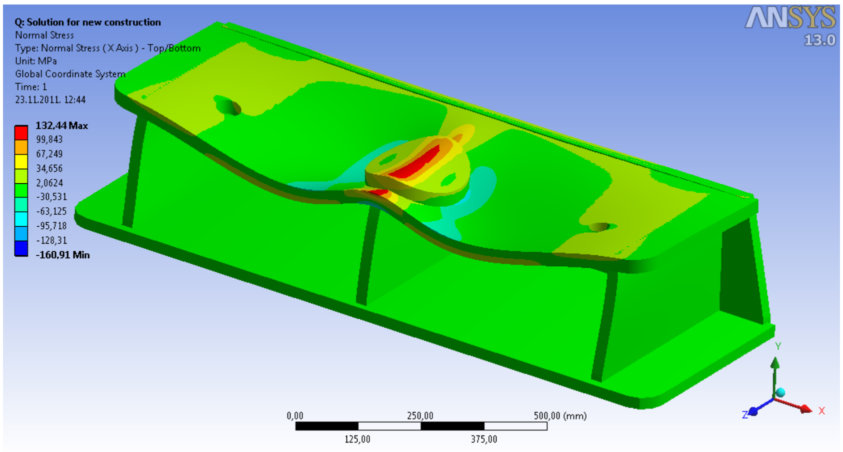


Figure 65. Normal stresses, new construction

New construction:

Total deformation	0.831 mm
Von-Mises stress	197 MPa
Normal stress	132 MPa
Shear stress	8 MPa

According to results that are obtain by ANSYS it is possible to conclude that for the new construction will be enough to put bracket in the middle of construction. The biggest concentration of stresses is on connection of bracket with a plate, however area around does not exceeding plastic deformation.

5.1.6. *Influence of the mesh on the results*

At end of this first model, it will be taken into account short observation to better understand meaning of the mesh density, what influence mesh has on the results, number of elements, CPU time and similar.

Table 5.Influence of the mesh on the result and CPU time

Mesh [mm]	50	40	30	20	10
Von-Mises [MPa]	170,56	175,06	171,76	173,52	194,38
Normal stress [Mpa]	91,76	82,02	117,63	133,36	123,42
Shear stress [Mpa]	3,58	4,56	5,02	6,28	6,93
Total deformation [mm]	0,804	0,762	0,811	0,83	0,816
Error	259	99,21	76,52	30,84	16,97
Number of elements	956	1383	2328	4958	11560
CPU time [s]	1,2	2,23	4,49	9,56	18,08

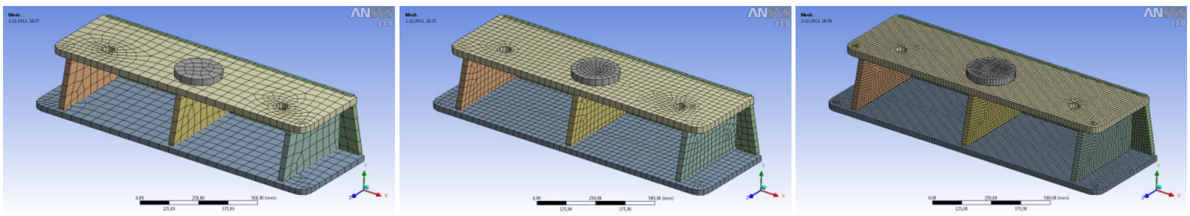


Figure 66. Different mesh sizes, 50 mm, 30 mm and 10 mm elements

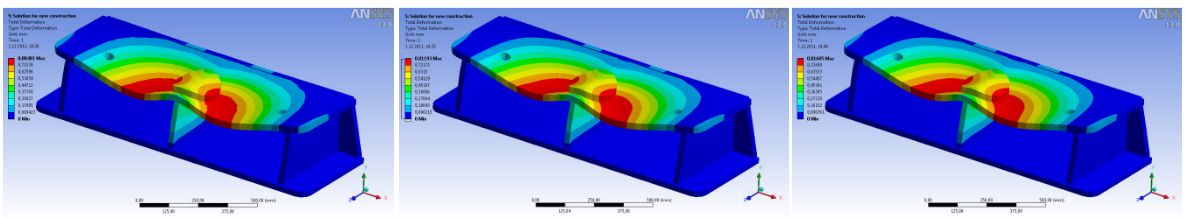


Figure 67. Total deformation for the 50 mm, 30 mm and 10 mm element sizes

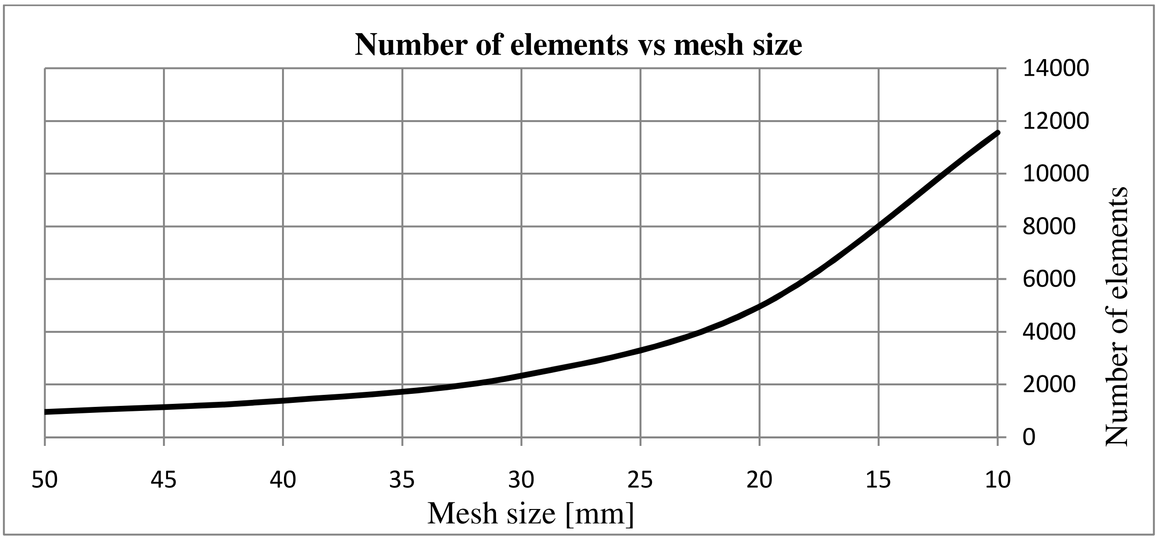


Figure 68. Number of elements vs mesh size

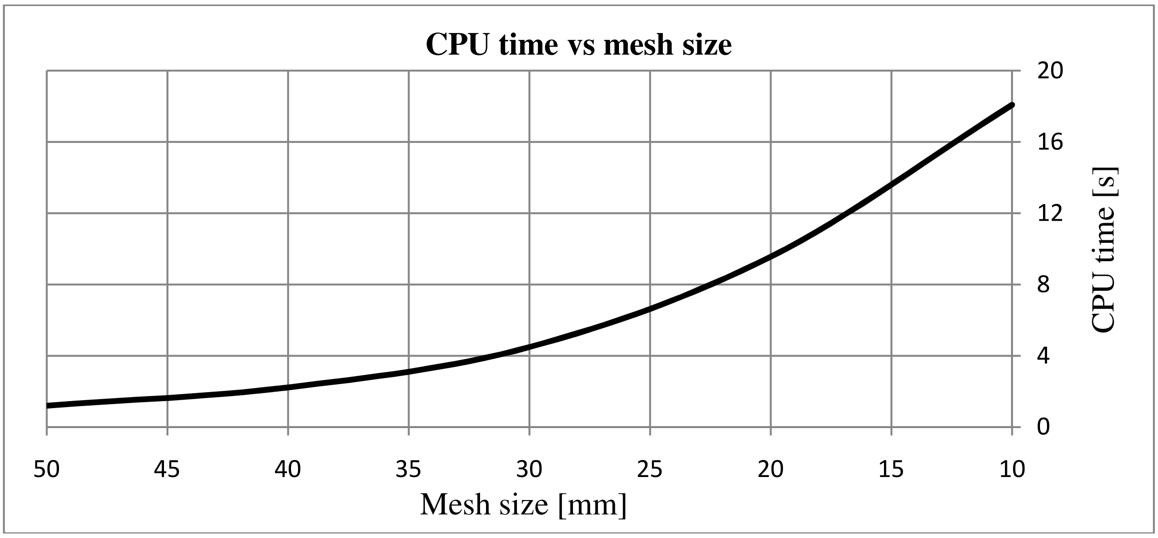


Figure 69. CPU time vs mesh size

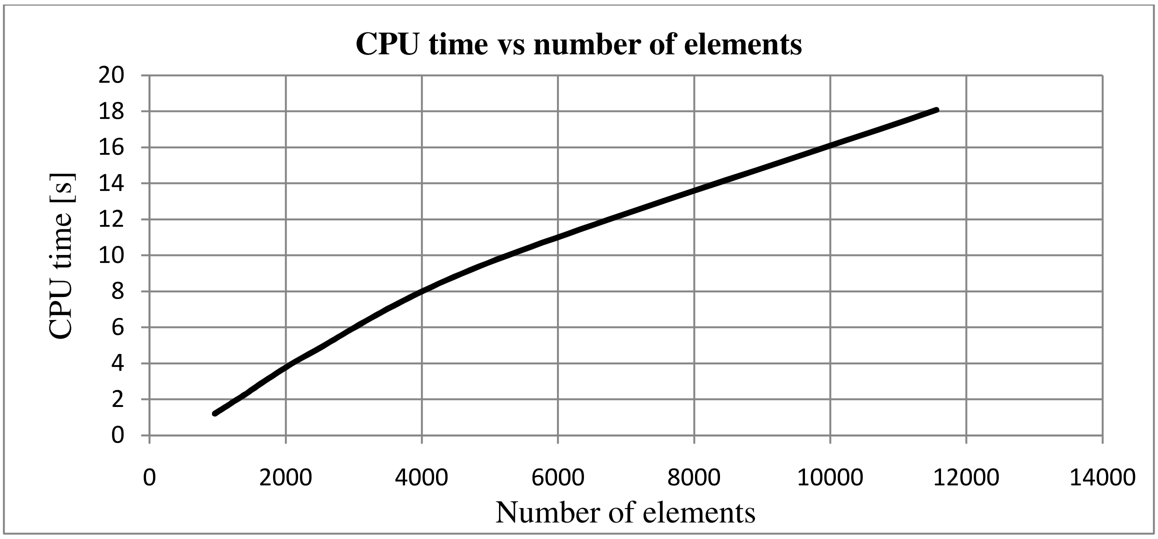


Figure 70. CPU time vs number of elements

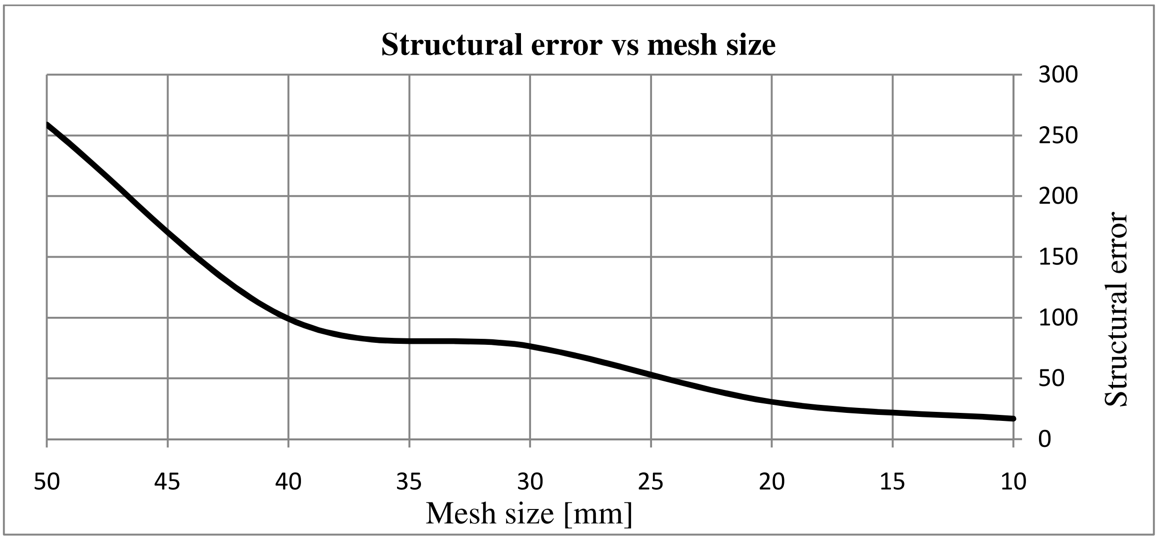


Figure 71. Structural error vs mesh size

From obtained results it is possible to notice that values of stresses are changing a lot for different densities of the mesh. Because of that, it is very difficult to be sure what is a good mesh and what density of mesh should be defined and to be able to do that it is necessary to have lot of experience in this kind of analysis. One of possibilities to be sure in quality of the mesh is to measure structural error in ANSYS Workbench and reduce it to the minimum. Errors are difference between the exact solution of the model equations and the numerical equation. The relative solution error can be formally defined as:

$$E_S = \frac{\left\| f_{exact} - f_{numeric} \right\|}{\left\| f_{exact} \right\|}$$

The goal of numeric simulation is to reduce this error below an acceptable limit. Usually in this thesis that limit was 10, in the ANSYS Workbench error was tried to reduce the error on value which is less than 10. However, the seldom mesh is reason for this errors. In region of high error it is necessary to refined mesh in order to get more accurate answer.

In the result for stresses it is possible to notice that results can be changed up to 12% for Von-Mises stress and 31% for Normal stress. So it is necessary to reduce structural error as more as it possible to obtain correct and reliable results. And from Figure 71 it is shown how much the structural error is decreasing with the more dense mesh.

From the graphs in figures above it is also possible to notice properties of the mesh for FEM analysis. The number of elements is grows exponentially if more dense mesh is applied, also with more dense mesh exponentially CPU time is growing. And CPU time and number of elements are growing linearly comparing one to each other. From all of these parameters it is possible to conclude that meshing of the model is most important parameter of good FEM analysis. Also it is very important from the time point of view too, because as it was already mention that CPU time is growing exponentially with more dense mesh. To obtain good mesh and still not so time consummating to perform numerical analysis is very challenging very important task of each engineers.

Short summary of this project; in the project it was performed static finite element analysis of the main mast base foundation. It was necessary to check the structure on two load cases, main load from the mast and secondary load case from the jacks. The main goal was to see where the critical areas are for that loads on the existing structure, solve the problem if it exist and offer the solution for the new structure.

The most of models were done by shell elements because of the good results that are obtained compared by solid elements and also they need less CPU time to perform same simulation. For complicated models like connection of main mast base foundation with ship structure, solid elements were used.

With linear and nonlinear analysis it was shown that existing construction doesn't satisfy for this kind of loads, the some parts of structure exceed the elastic area and go to the plastic area. Solution for existing structure is to add a piece of aluminium and put it under the jacks to hold the deformation and translate the stresses to the lower part of the structure. For the new structure it is necessary to move the bracket in the middle and that construction is able to resist on that loads.

5.2. Keel – Hull Structure Connection

In this project it will be performed static finite element method analysis of the keel – hull structure connection. The type of the keel is swing keel. Swing keel is keel that can be pulled up (rotational movement) in the hull when yacht is cruising with the engines to reduce the resistance of the yacht or when yacht is coming to the port to reduce the draft, however when is using the sails keel is needed to make contra moment for the sails. This keel is rotating around fixed point, and there are some solutions where it is possible just to pull up and then to push back vertically the keel [7]. How the loads should be considered, the first step is to consult the classification society rules. Keel and hull structure will be verified according to the American Bureau of Shipping – Guide for Building and Classing Offshore Racing Yachts.

According to the rules:

9.13.1 Continuity

Where fitted, floors within ballast keels and in spacer structure between the ballast keel and the underside of the hull are to be in line with the floors in the hull. Internal load carrying members within the ballast keel are to be aligned and connected with floors in adjacent structure.

9.13.3 Structure

The shear and primary stresses at any location of the keel structure under the following assumed load are not to exceed the respective allowable stresses given below.

Assumed Load:

Acting Transversely – weight of the keel below the section of the keel under consideration acting at its centre of gravity

Allowable stress:	Shear stress	Primary stress
All materials:	$0.5\tau_y$	$0.5\sigma_y$

Where:

σ_y = Minimum tensile strength of the material, it is not to be taken as greater than 70% of the ultimate tensile strength of the material.

τ_y = Minimum shear yield strength of the material, it is not to be taken as greater the 40% of the ultimate tensile strength of the material.

From the rules only one load case that will be applied on the structure is weight of the keel, or its own gravity.

Material of the keel and structure is aluminium magnesium alloy 5083 H321. And main characteristics of this alloy are:

$\rho = 2660\ \frac{\text{kg}}{\text{m}^3}$ - density

Tensile Yield Strenght - 215 MPa

Compressive Yield Strenght - 215 MPa

Tensile Ultimate Strenght - 305 MPa

Young's modulus - 71000 MPa

Poisson's ration - 0.33

When keel is built, it is necessary to fulfil the keel with material to obtain bigger mass of the keel. In Perini Navi shipyard material that is used for fulfil the keel is lead. Lead is a bluish white lustrous metal. It is very soft, highly malleable, ductile and a relatively poor conductor of electricity. It is very resistant to corrosion, but tarnishes upon exposure to air. Lead has very big density, it can give big mass for small area and also it have almost twice time less melting point of aluminium so it is possible to put the lead inside of the keel in liquid phase. The lead has next main characteristics:

Chemical symbol	Pb
Density	11.34 t/m^3 at 20 ^{o}C
Melting point	327 ^{o}C (melting point of Aluminium is 660.37 ^{o}C)

5.2.1. Model of the keel – structure connection

Model will be divided in two parts. One part is a structure of the keel and second one is structure of the hull. Connection between them is with the pin. Model was separated because to simulate pin connection is very difficult. That is meaning, that two analysis should be performed. In first keel will be submitted to its own gravity and in second the forces will be measured at places where keel is connected with the structure and those forces will be transmitted to the structure of the yacht. Models are possible to see on the Figure 73 and Figure 75. Models are modelled by surfaces because finite element method will be performed by shell elements. There are two reasons to do that by shell elements. First is to safe CPU time and to perform analysis much faster, as we obtain from the previous model. And second one is that model has very big dimensions and if it is meshed by solid element it is necessary to mesh with bigger number of elements to obtain good results. Why is so complicated to do that in solids, because, to obtain good results when plate is deforming, it is necessary to put minimum two or three solid element in thickness to obtain good values of compression and tension of the plate, however for shell it is enough only one element for good results. As ANSYS is limited to 300 000 elements, with solids it is not possible to mesh the structure to obtain accurate results until that number of elements. Mesh of the keel structure with shell elements already has 127 000 elements.

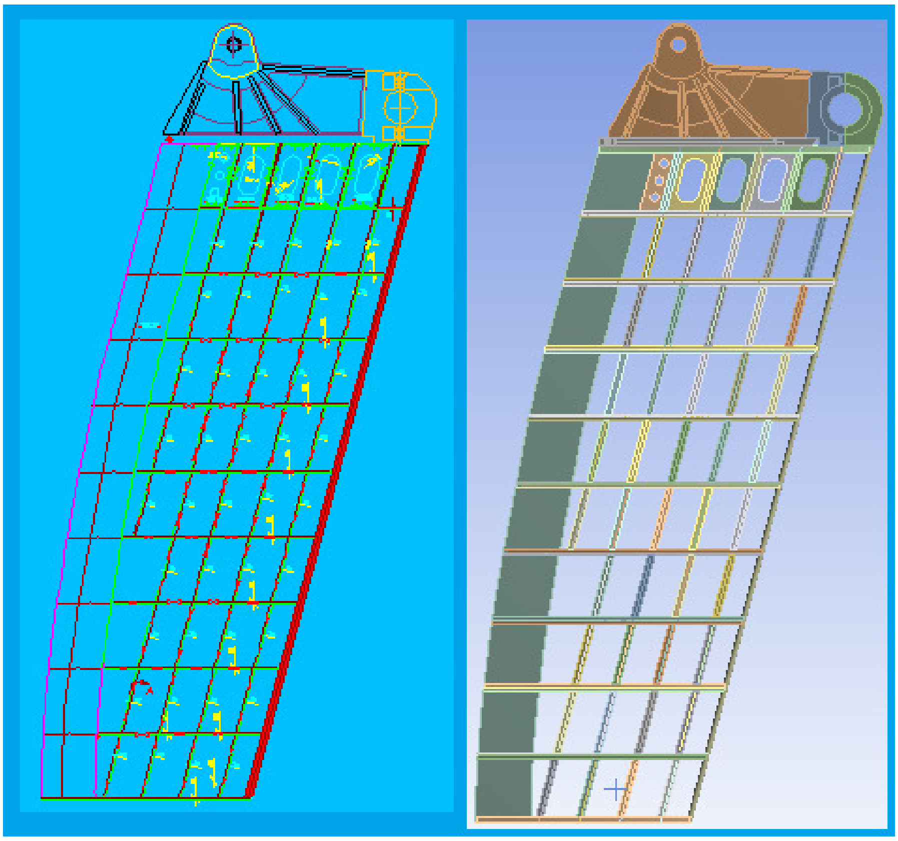


Figure 72. 2-D drawing of the keel from Perini-Navi (left), 3-D model of the keel made in SolidWorks (right)

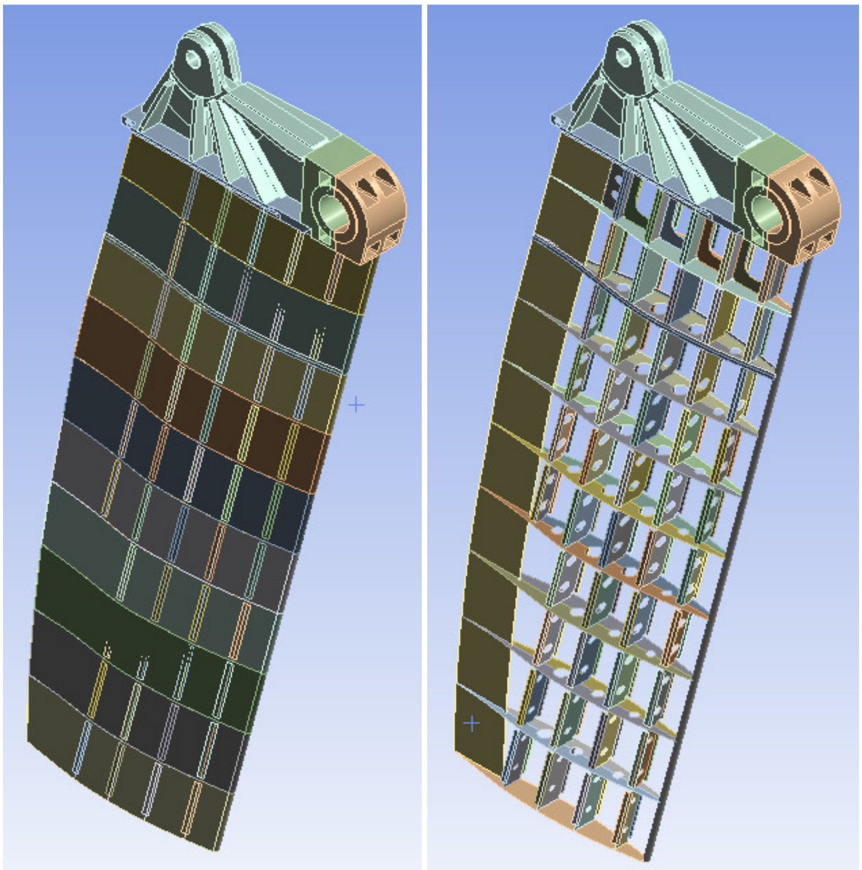


Figure 73. 3-D model of the keel structure

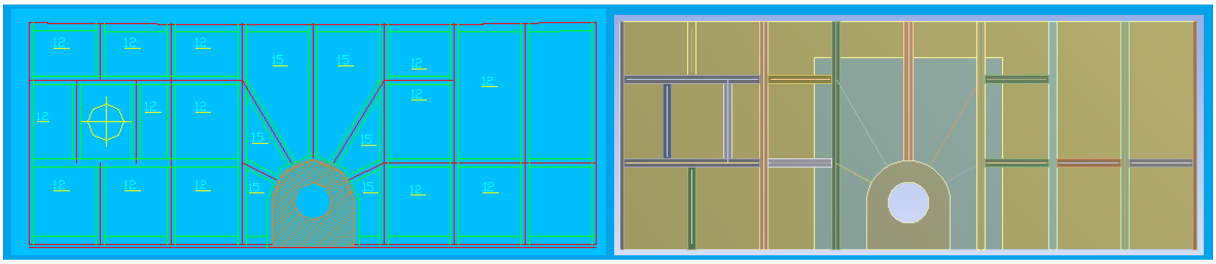


Figure 74. 2-D drawing of the structure from Perini-Navi (left), 3-D model of the structure made in ANSYS (right)

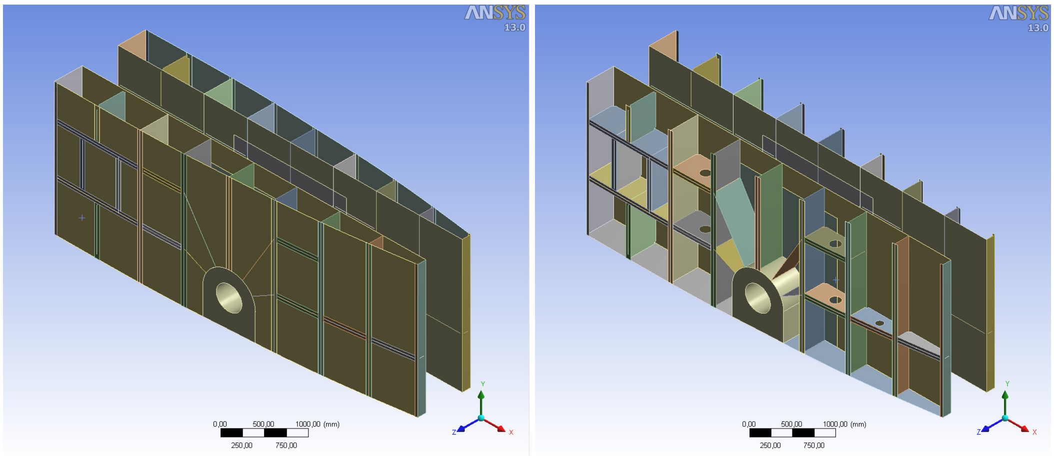


Figure 75. Model of the ship structure

5.2.2. Loads and boundary conditions

Keel structure will be submitted to its own gravity. Keel is 7990 mm long and it is necessary to obtain force of 65 tons in the centre of gravity; 65 tons are coming from the weight of aluminium (around 7 tons) and the lead that is inside of the structure. The exact value of the mass is obtained from Perini Navi drawings and it can be verified from next Figure 76. In the model, lead was not modelled and to simulate 65 tons of weight, density of aluminium was changed to obtain correct value of the structure mass.

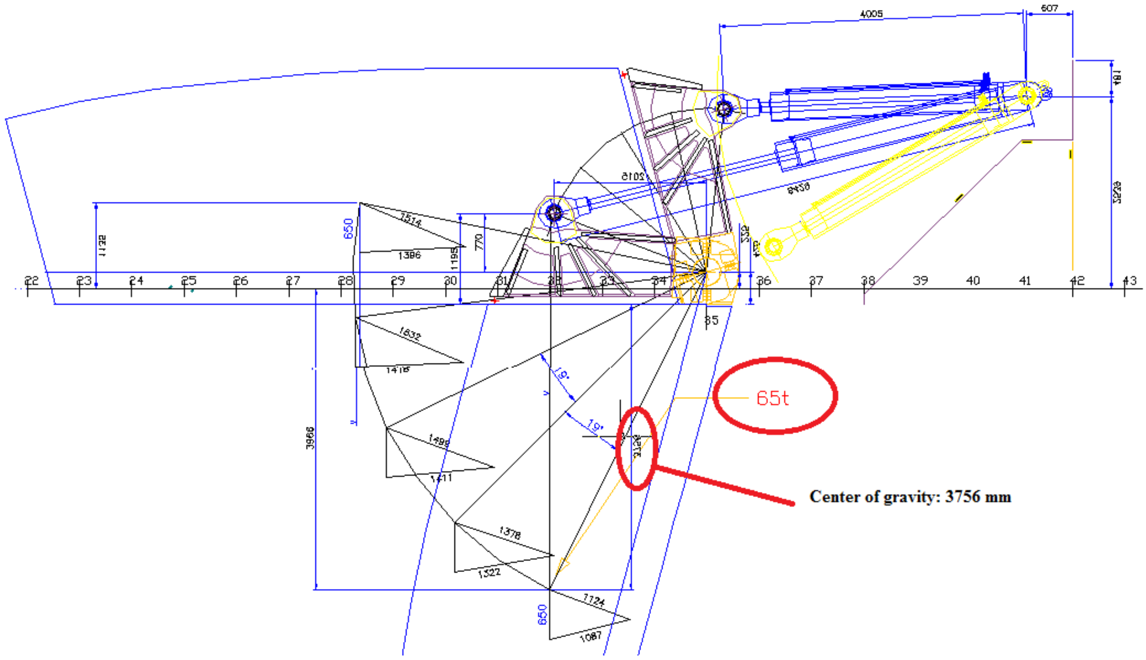


Figure 76. Total mass that acts in cetner of gravity

The structure will be restrained and constrain the degrees of freedom to make the model behave in a realistic manner. As it mentioned before, there are six degrees of the freedom to consider, three translations and three rotations. Boundary conditions are; three displacement support, two in z direction (direction in which the centre of gravity is acting), one in y direction and one compressions only support. First and second displacement supports are coming from bearing head (A – z direction and D – y direction, on the Figure 78) which is placed around pin and his purpose is also to protect vertically movement of the keel. Third displacement support is coming from the plate (B on the Figure 78) and it also restricts the movement of the keel. And compressions only support is used to simulate connection with the pin (C on the Figure 78).

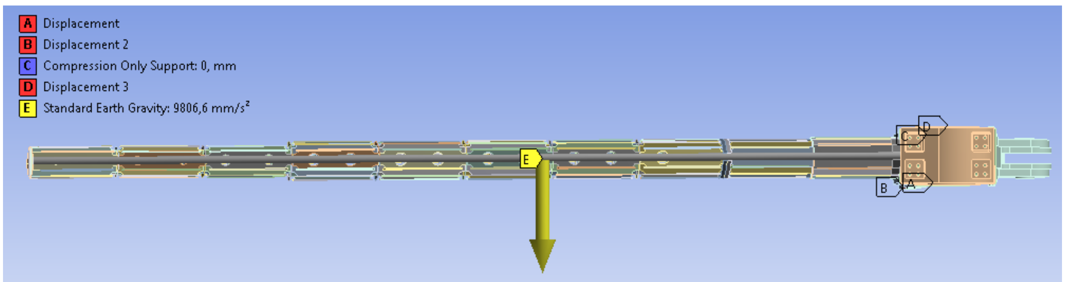


Figure 77. Boundary conditions and centre of gravity

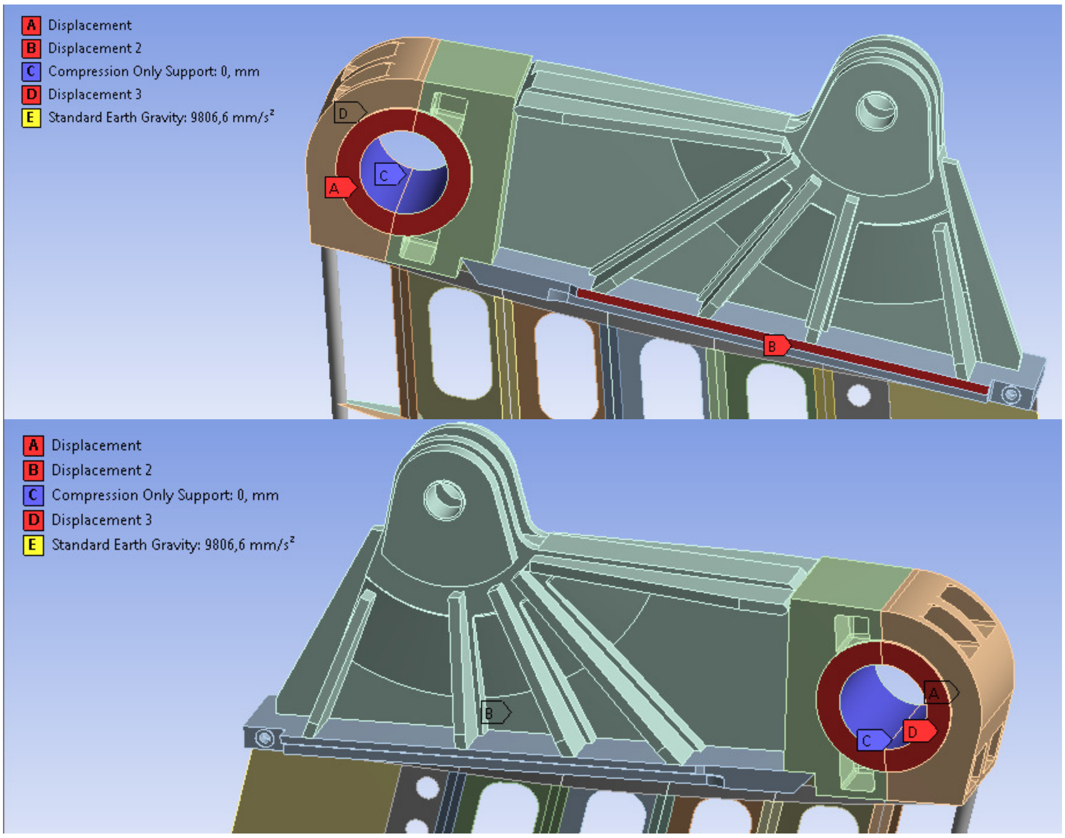


Figure 78. Boundary conditions, red – displacement supports, blue – compression only support (view from both sides)

Displacement support:

- Applies known displacement on vertex, edge, or surface
- Allows for imposed translational displacement in x, y and z (in user-defined Coordinate System)
- By entering “0” in software, means that the direction is constrained leaving the direction blank, means the direction is free.

Compression only support:

- Applies a constraint in the normal compressive direction only
- Can be used on a cylindrical surface to model a pun, bolt, etc.
- Requires an iterative (nonlinear) solution

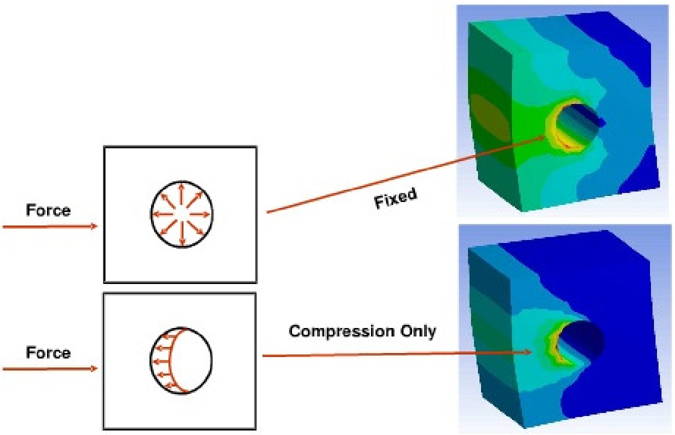


Figure 79. Difference between fixed and compression only support

Standard Earth Gravity is using inertial effects, by accelerating a structure in the direction opposite of gravity (the natural phenomenon). That is accelerating a structure vertically upwards (+) at 9.80665 m/s^2, applies a force on the structure in the opposite direction (-) inducing gravity (pushing the structure back towards earth). For this model it was used (-) symbol for simulate gravity if the earth.

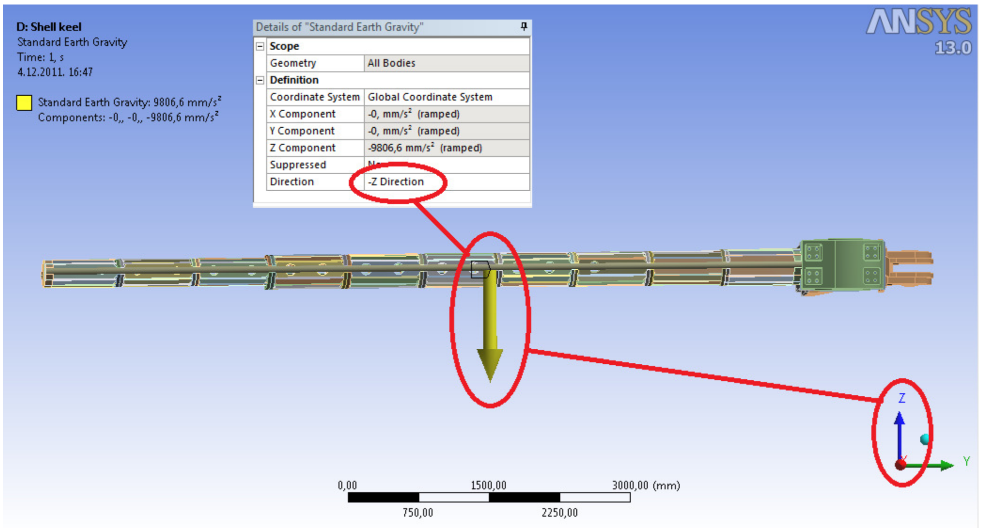


Figure 80. Standard Earth Gravity, properties of ANSYS

When multiple parts are presented, it is necessary to define the relationship between parts. Contact region define, how parts interact with each other. Without contact or spot welds, parts will not be in interaction:

- In structural analyses, contact and spot welds prevent parts from penetrating through each other and provide a means of load transfer between parts.
- Multi body parts do not require contact or spot welds

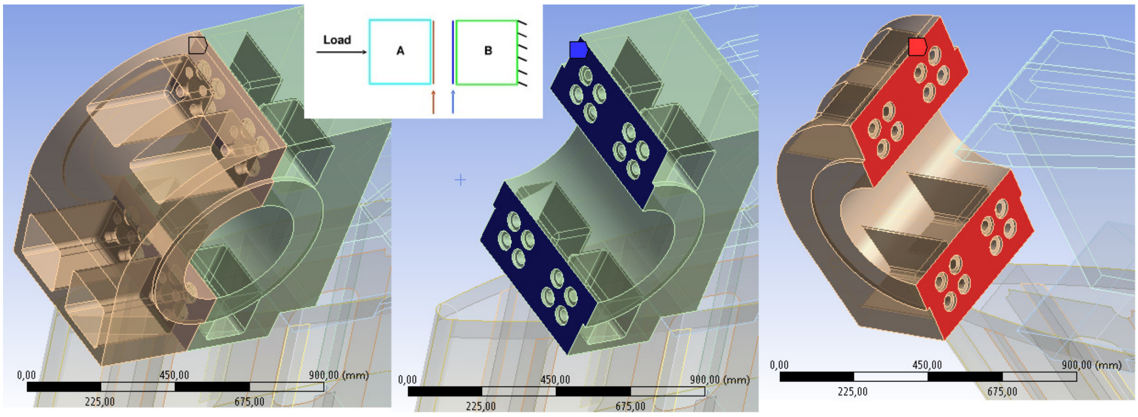


Figure 81. Example of the bonded connection in ANSYS

All connections of the keel between surfaces are bonded connection. Bonded connection is the default configuration for contact regions. If contact regions are bonded, then no sliding or separation between faces or edges is allowed. Think of the region as glued. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. If contact is determined on the mathematical model, any gaps will be closed and any initial penetration will be ignored.

All connections are set up in ANSYS, even if assembly was done in SolidWorks. ANSYS can recognize connection of the solids, but for the surfaces it has a problems, so to be sure in the structure connection, all the connections has been done one by one in ANSYS and every time connection was verified to view the model deflection. This was showing does every of the nodes are properly connected and also does model is behaving as expected. This was very important step in keel modelling, because model from SolidWorks is looking like connected model, however if they don't share common node, then they are not mathematically connected and model will not be solved correctly.

After analysis is performed it is necessary to set up the probes in each support. All probes are measuring force reaction.

Force reaction for the bearing head (in direction of centre of gravity):

$z_{axis} = 733.8 \text{ kN}$

Force reaction for the bearing head (opposite direction of centre of gravity):

$y_{axis} = 2615 \text{ kN}$

Force reaction for the plate:

$z_{axis} = 1359$ kN

Force reaction for the pin:

$x_{axis} = 9.71$ kN

$y_{axis} = -2572$ kN

$z_{axis} = -3.1$ kN

$\text{Total} = 2572$ kN

This force reaction will be input parameters for the yacht structure. So there it will be two different load cases, for two sides of structure. At one side load will be force reaction of the pin + force reaction of the bearing load at opposite direction from the centre of gravity and on other side it will be force reaction of the pin + force reaction of the bearing load at direction of the centre of gravity.

Sum of those vectors is:

Force reaction of the pin + Force reaction of the bearing load in direction of the centre of gravity:

$x_{axis} = 9.71$ kN

$y_{axis} = -2572$ kN

$z_{axis} = 730.7$ kN

$\text{Total} = 2673$ kN

Force reaction of the pin + Force reaction of the bearing load in direction of the centre of gravity:

$x_{axis} = 9.71$ kN

$y_{axis} = 43$ kN

$z_{axis} = -3.1$ kN

$\text{Total} = 44.2$ kN

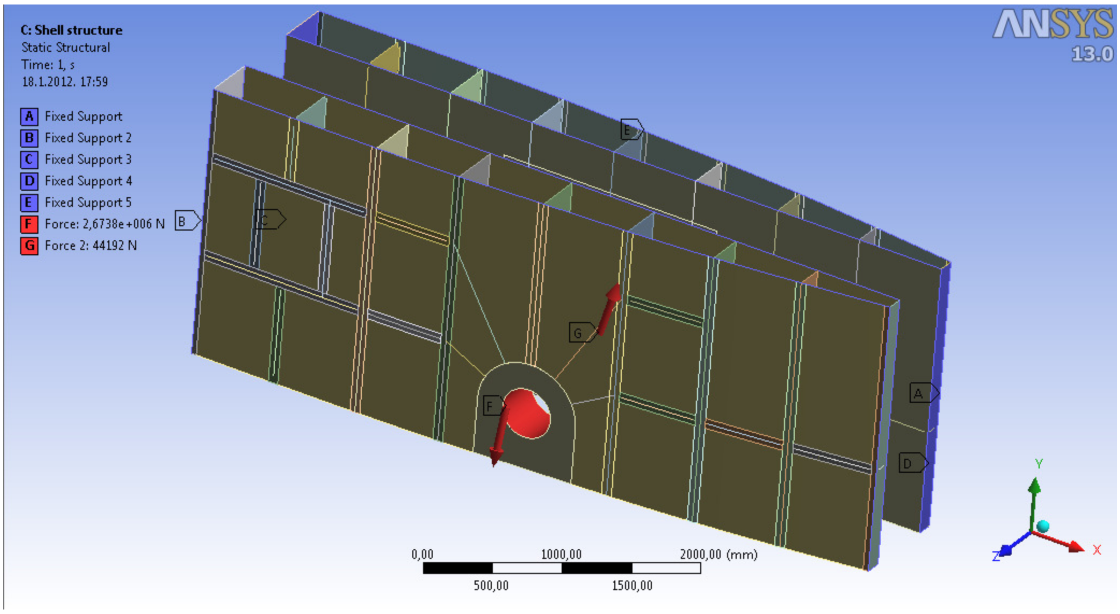


Figure 82. Loads and boundary condition for the structure

On the Figure 82 are shown boundary conditions and loads of yacht structure. Boundary conditions are fixed supports at the edges of the model and also on the top of the model where structure is connected with the hull of the yacht.

5.2.3. Total deformation of the keel by direct method

Total deformation will be calculated with simple calculation for the profile section of the keel. Total deformation will be measured at position where the force is acting, at centre of gravity. According to the Figure 76 and drawings that are obtained from Perini Navi, position of the centre of gravity is at 3756 mm from the connection of the keel with the plate.

Position of the keel centre of gravity according to the SolidWorks is at 3652 mm. However as swing keel head is added as assembly with the keel, that position will be moved and real position where the Earth Gravity will act in ANSYS can be calculated as:

$$CG = \frac{\sum \mathrm{CG_{keel}} \cdot \mathrm{mass_{keel}} + \mathrm{CG_{head}} \cdot \mathrm{mass_{head}}}{\sum \mathrm{mass_{keel}} + \mathrm{mass_{head}}} \tag{5.1}$$

Where is:

$\mathrm{CG_{keel}} = 4338$ mm - measured from the bottom of the keel (7990 mm - 3652 mm)

Remark: 7990 mm is height of the keel

$CG_{head} = 8425$ mm - measured from the bottom of the keel

$\mathrm{mass_{keel}} = 65$ t

$\mathrm{mass_{head}} = 1.76$ t

$CG = 4446$ mm - from the bottom

$CG = 3544$ mm - from the connection of the keel and swing keel head

Real centre of gravity is at 3756 mm from the connection and one that is obtained with software and calculation is 3544 mm. So the difference between them is 212 mm, on 7990 mm long keel, and that is acceptable. From modelling point of view it was very difficult to model shape of the keel 100% equal as it in reality and small difference between those two values was expected.

The section that was modelled is shown in next Figure 83:

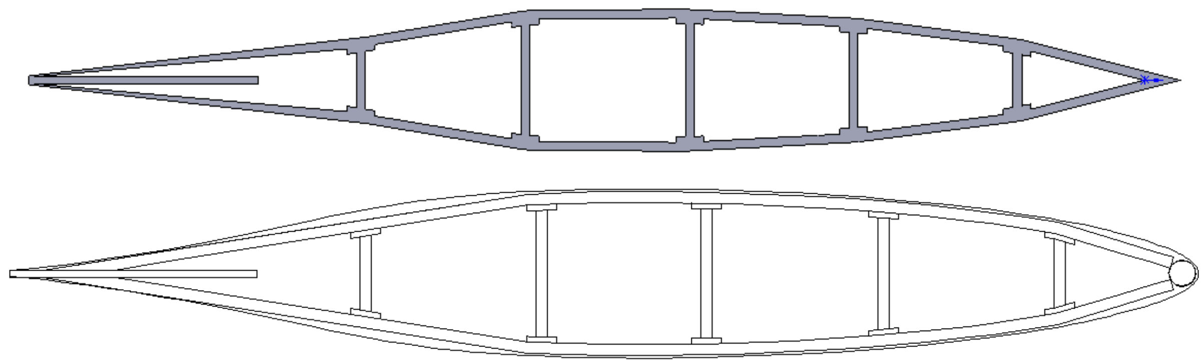


Figure 83. Section of the keel from SolidWorks (up) and Perini Navi (down)

Section was done at SolidWorks and from the same software next informations about section are obtained:

$I = 3220628979 \text{ mm}^4$

$A = 204731 \text{ mm}^2$

According to the beam theory:

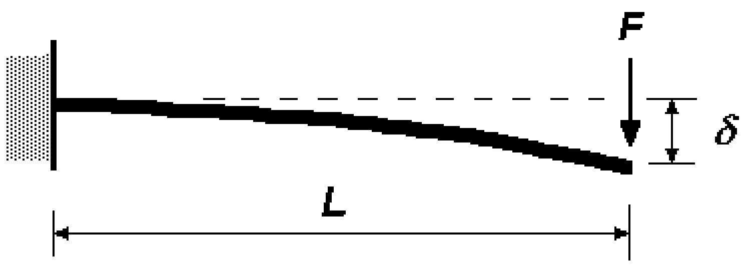


Figure 84. Console with force acting at end of the console

$$\delta = \frac{FL^3}{3EI} \text{ where is:} \qquad (5.2)$$

$M = 65000$ kg

$F = 637650$ N

$E = 71000\ \frac{\text{N}}{\text{mm}^2}$

$\delta = 41.38$ mm

Stresses along the beam are calculated as:

$$\sigma = \frac{M \cdot z}{I} \tag{5.3}$$

$M = F \cdot x$

$z = 195$ mm

Table 6. Stresses of the keel according to the direct method

x [m]	3544	3344	3144	2944	2744	2544
M [Nmm]	2259831600	2132301600	2004771600	1877241600	1749711600	1622181600
σ [N/mm^2]	136,83	129,10	121,38	113,66	105,94	98,22

x [m]	2344	2144	1944	1744	1544	1344
M [Nmm]	1494651600	1367121600	1239591600	1112061600	984531600	857001600
σ [N/mm^2]	90,50	82,78	75,05	67,33	59,61	51,89

x [m]	1144	944	744	544	344	144
M [Nmm]	729471600	601941600	474411600	346881600	219351600	91821600
σ [N/mm^2]	44,17	36,45	28,72	21,00	13,28	5,56

Where the biggest value of x is at place where console is connected with the wall and the lowest value is close to the acting of the force. Stress at connection of the keel with the swing keel head is 137 N/mm^2 or MPa.

5.2.4. Total deformation of the keel and the structure

To be sure in FEM analysis, does it gives good results and that are boundary condition very well chosen for this model, it is necessary to compare the results between direct method and numerical simulation. From direct method values of the stresses and total deformation at the centre of gravity are obtained, those two values will be compared.

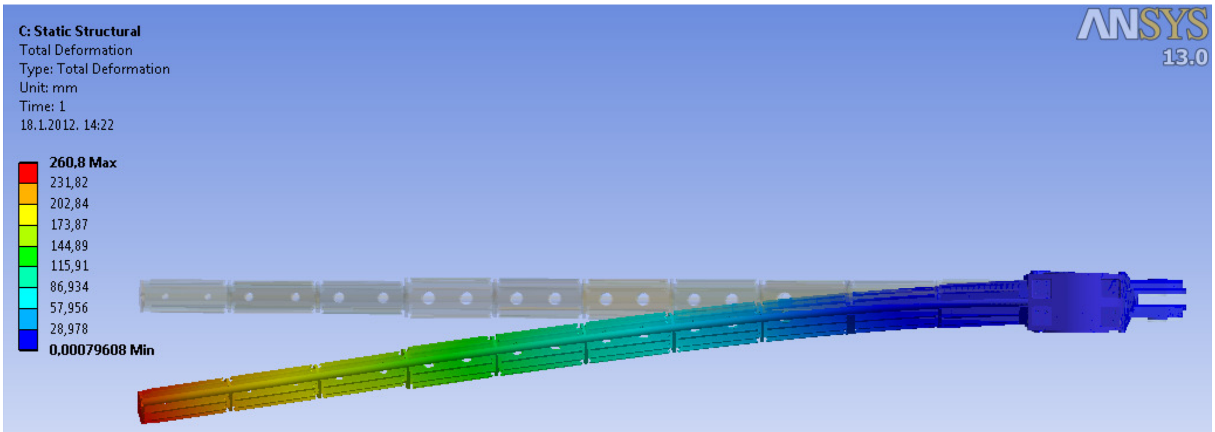


Figure 85. Total deformation of the keel structure

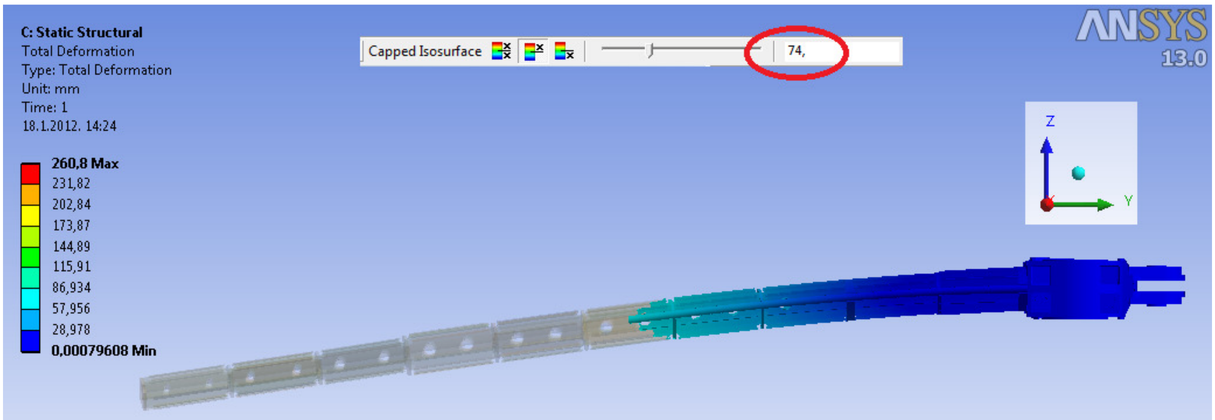


Figure 86. Deformation at position of the centre of gravity

Deformation in the point for centre of gravity where total mass of the keel is acting, according to direct method is 41.38 mm. With FEM analysis it was obtained 74 mm. Direct method is beam theory for the console and in the edge console is restricting movement of the beam in two direction (vertically and horizontally). Boundary conditions of the FEM analysis are chosen like at console and movement in z and y directions were restricted. It was expected that FEM analysis will give bigger deflection because boundary condition are not applying at all head of the keel, but just in some parts, see Figure 78 and it was necessary to obtain magnitude of the deflection value, to be similar.

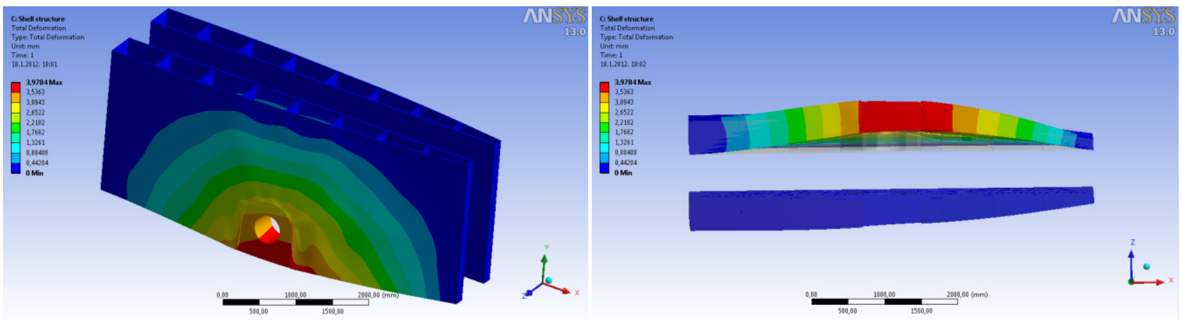


Figure 87. Total deformation of the yacht structure

For the structure maximum deflection is less than 4 mm. That is mainly because the structure is very well and rough dimensioned. Just to obtain feeling about that, it is enough to say that thickness of the plates is 50 mm, where is connection of the pin with the structure.

Deflection of the keel is high and for the structure is low, and compared with stresses rules are satisfied and the deflections are acceptable. The values are shown on next Figure 88.

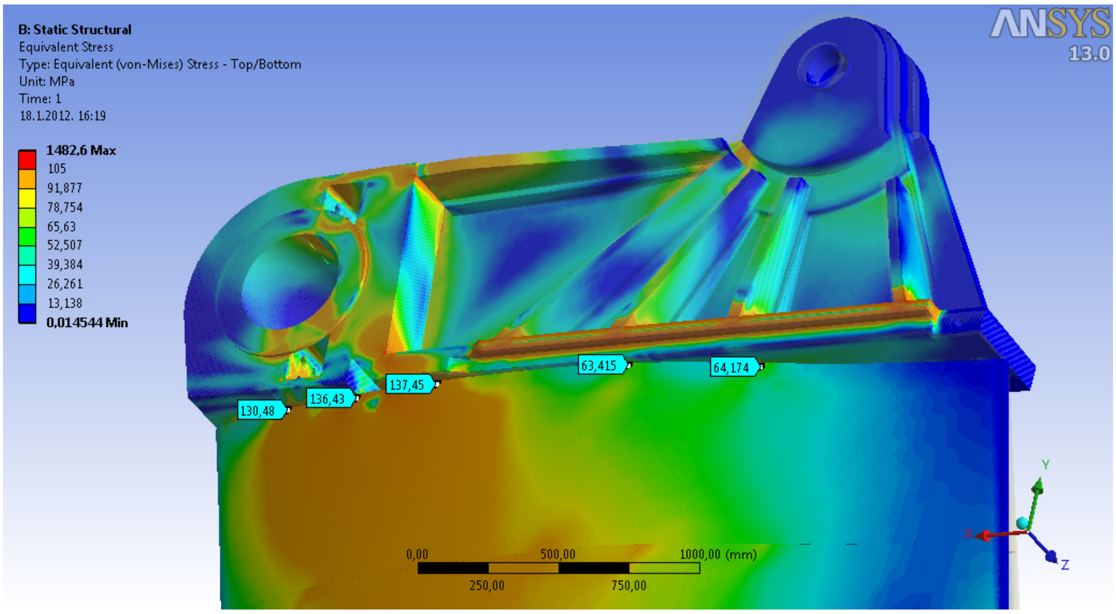


Figure 88. Values of stress at connection of the keel structure with keel head

With direct calculation obtained stress was 137 MPa and from the figure above it is possible to see values of the stresses from the FEM analysis. Stresses are similar to direct calculation and results are good. Some area are bigger than 105 MPa what was the demanding from the classification society, and results will be acceptable because in reality this structure is welded one to each other and in this model at this position are sharp edge and that is leading again to the increasing of the stresses.

To be sure in obtained results it is necessary to verify structural error of the keel and the structure, Structural error should be less than 10. In next Figure 89 and Figure 90, structural error is shown. The biggest errors are happening were boundary conditions and the sharp edges are.

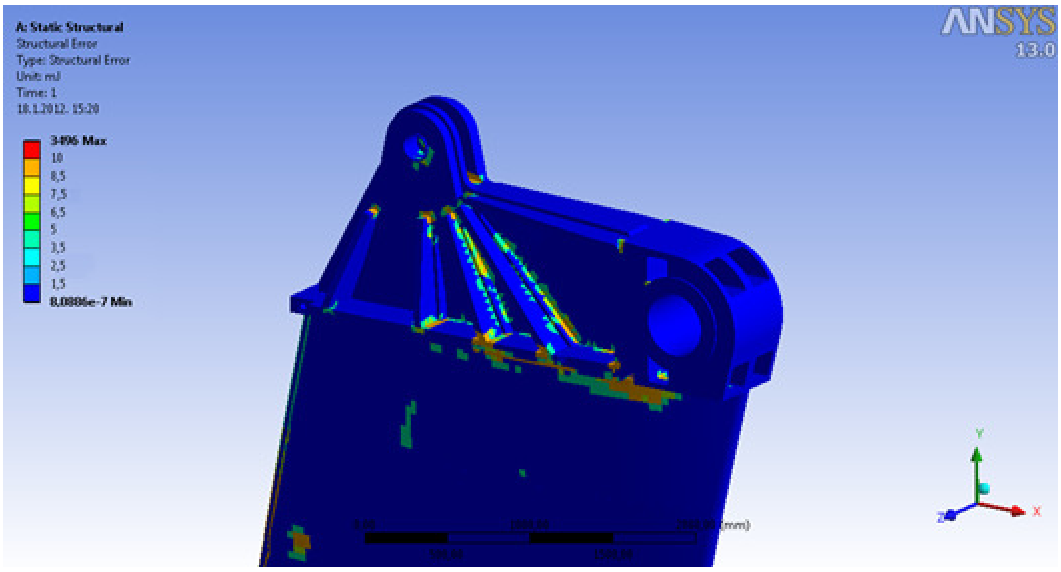


Figure 89. Structural error of the keel

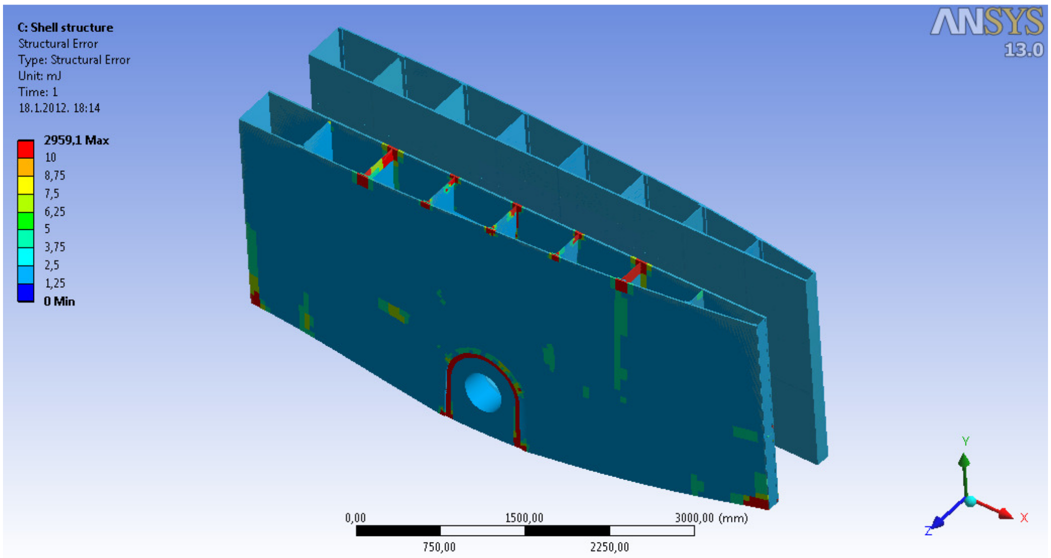


Figure 90. Structural error of the yacht structure

After the all model is properly set up, it is possible to proceed for the further analysis of the results.

5.2.5. Von-Mises stress

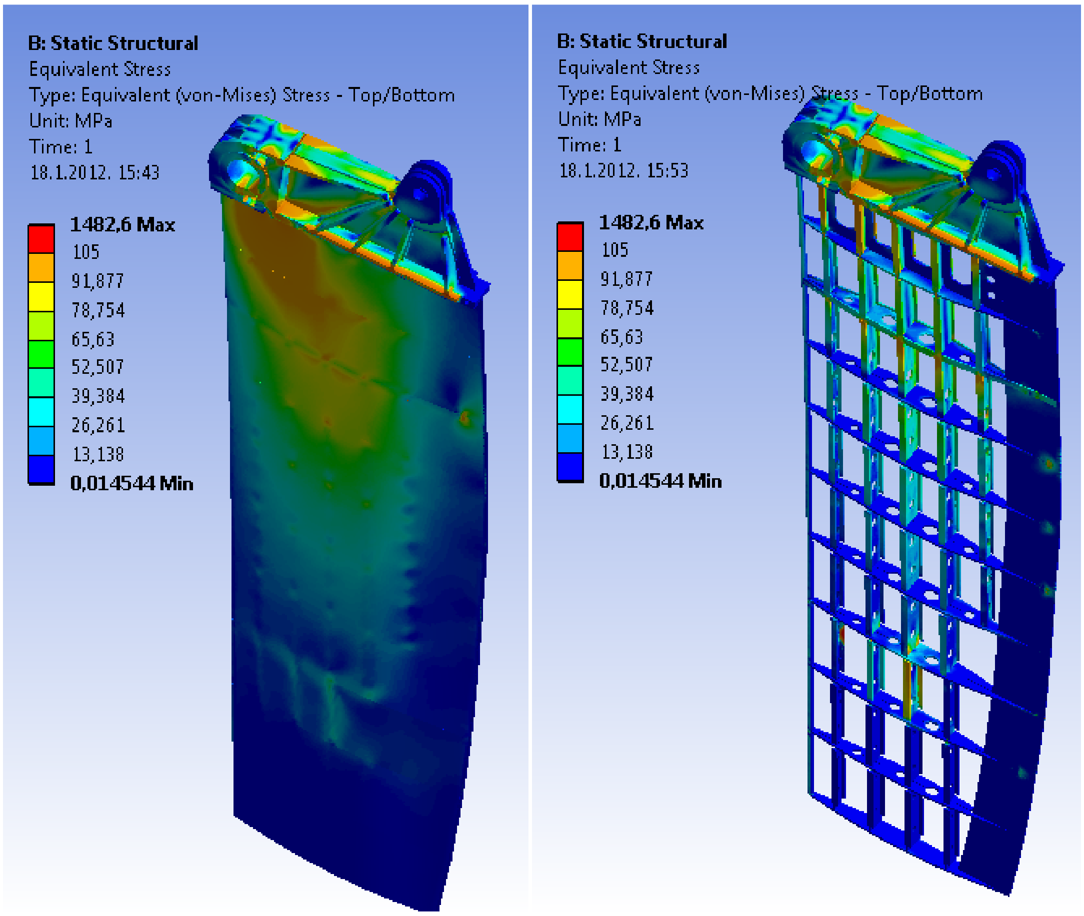


Figure 91. Von-Mises stress of the keel structure

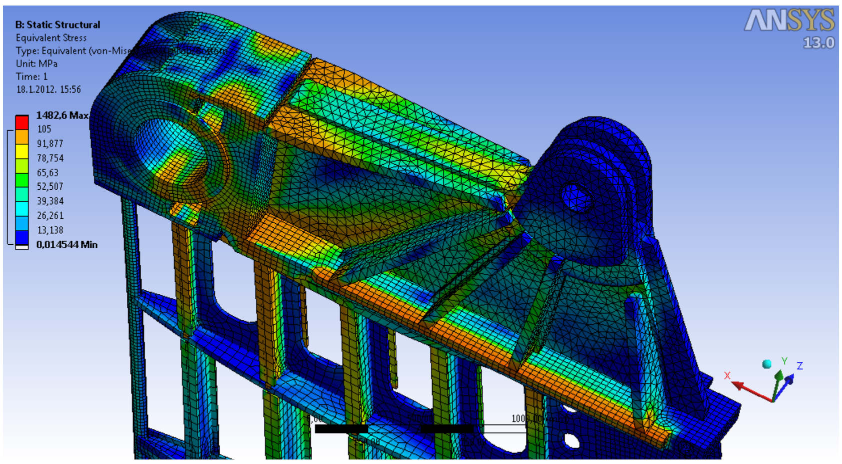


Figure 92. View of the connection swing keel head with structural blade for Von-Mises stress

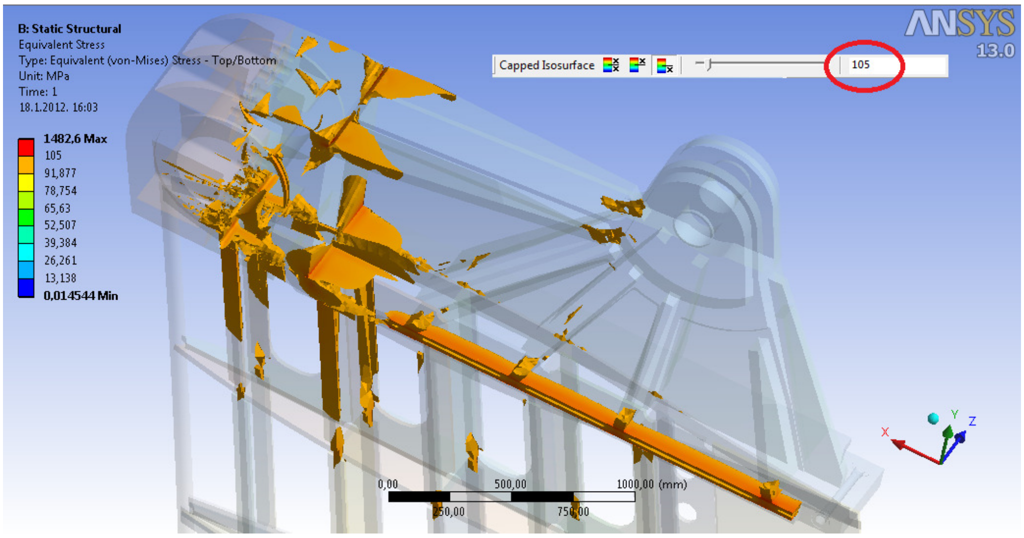


Figure 93. Places where Von-Mises stresses are bigger than 105 MPa

As it shown on results above, the maximum stress is 1400 MPa. And why those results are acceptable, it is because of the positions where maximum stresses are appeared and that position is in areas where boundary conditions are. Those areas will not be taken into a consideration because of boundary conditions. On the Figure 93 is shown where the stresses are bigger than 105 MPa what is required by the rules. And that places are at position of boundary conditions.

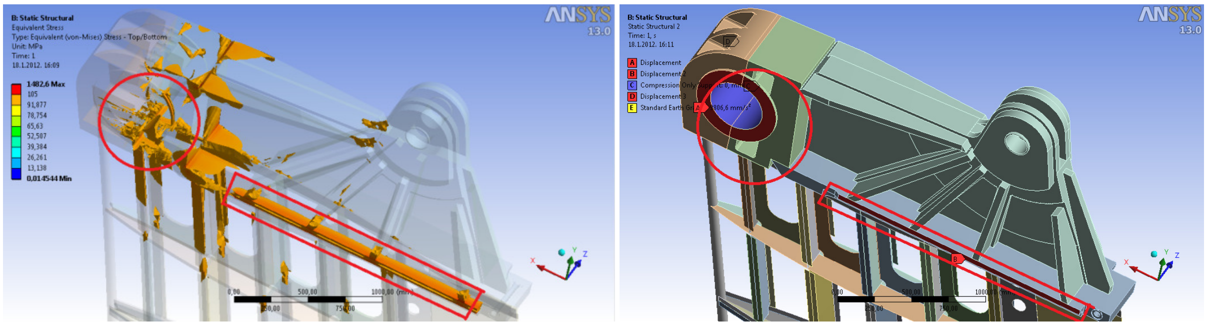


Figure 94. Comparison of the results with boundary conditions

Defining boundary conditions is one of the most important steps in FEM analysis. For local analysis models, the boundary conditions imposed by the surrounding structures should be based on the deformation or forces calculated from the global model. The boundary conditions, for a model have no other purpose than to restrict the rigid body motion [8].

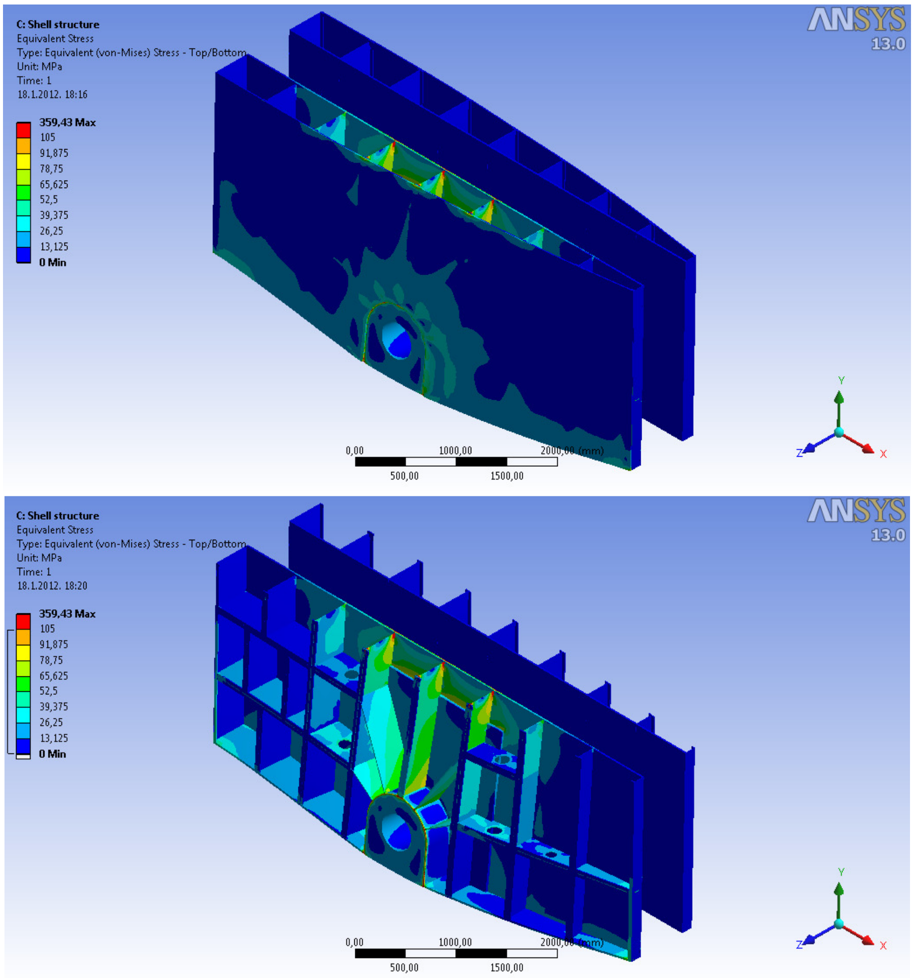


Figure 95. Von-Mises stress for the yacht structure

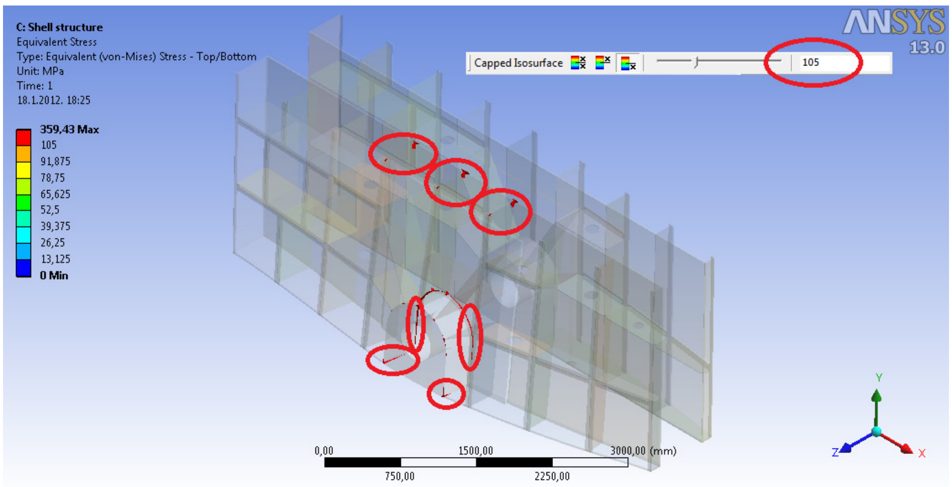


Figure 96. Places where stresses are bigger than 105 MPa on the yacht structure

Von-Mises stress for the yacht structure is 350 MPa, higher then yielding but it happening at position of sharp edges and on the places where boundary conditions for this model are and they will not be taken into a consideration.

5.2.6. Normal stress

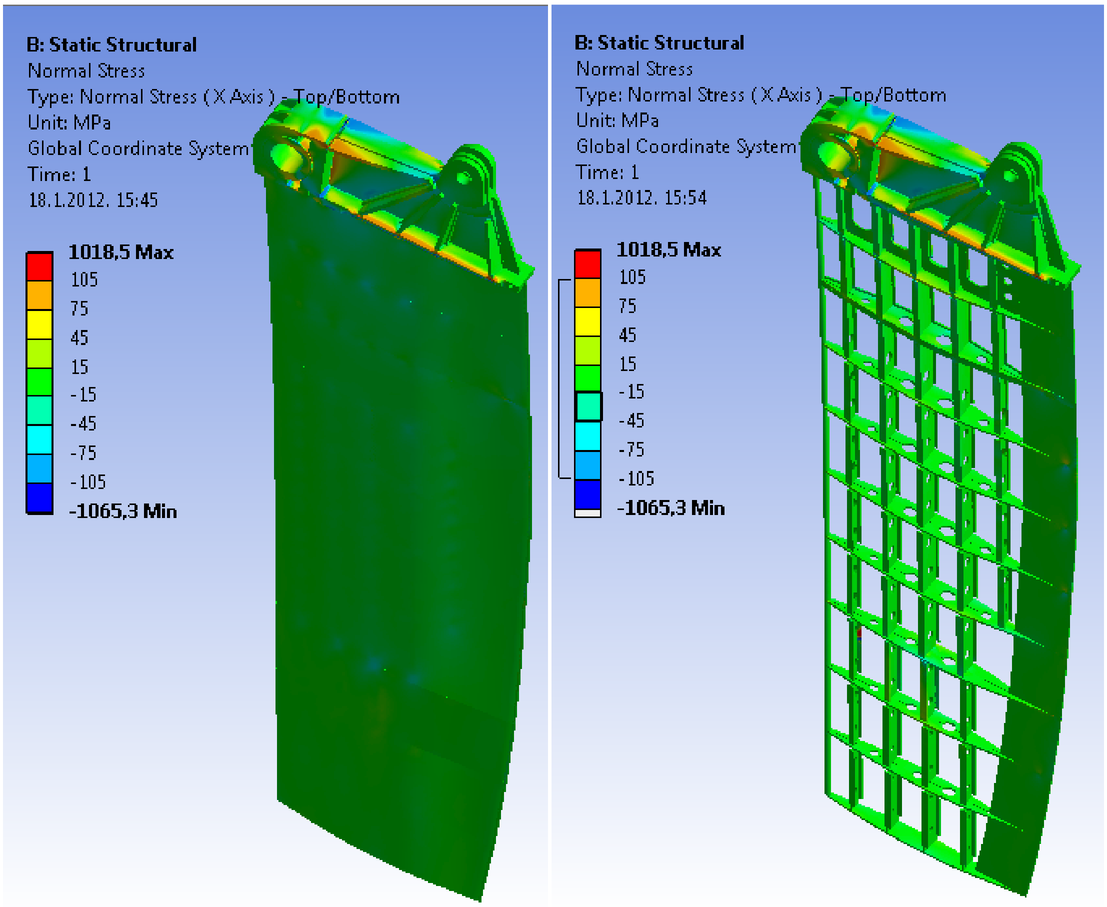


Figure 97. Normal stress of the keel structure

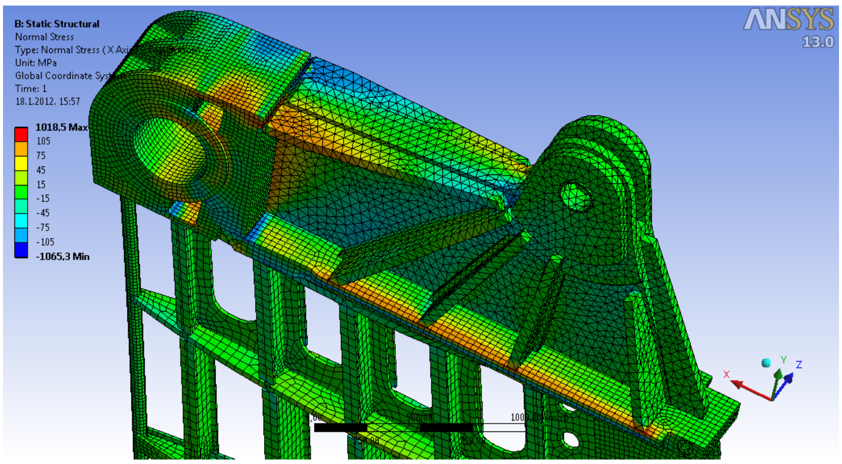


Figure 98. View of the connection swing keel head with structural blade for Normal stress

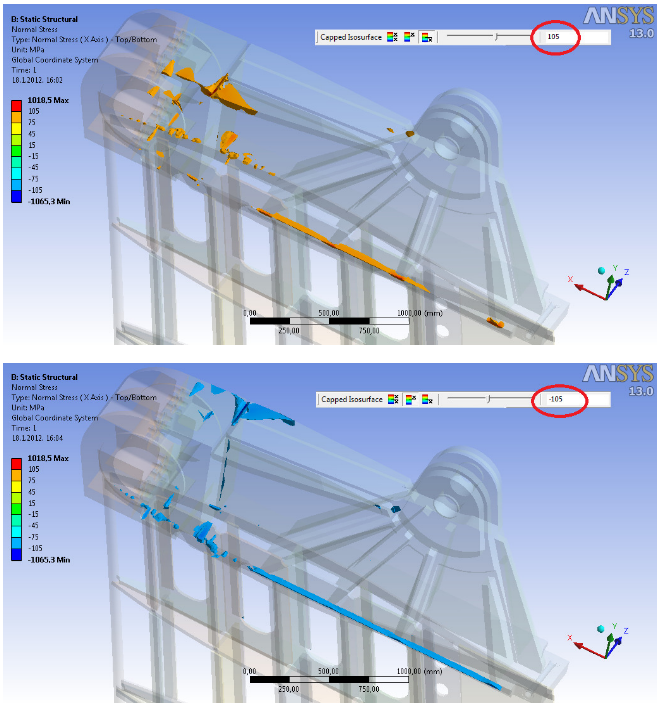


Figure 99. Places where Normal stresses are bigger than 105 MPa

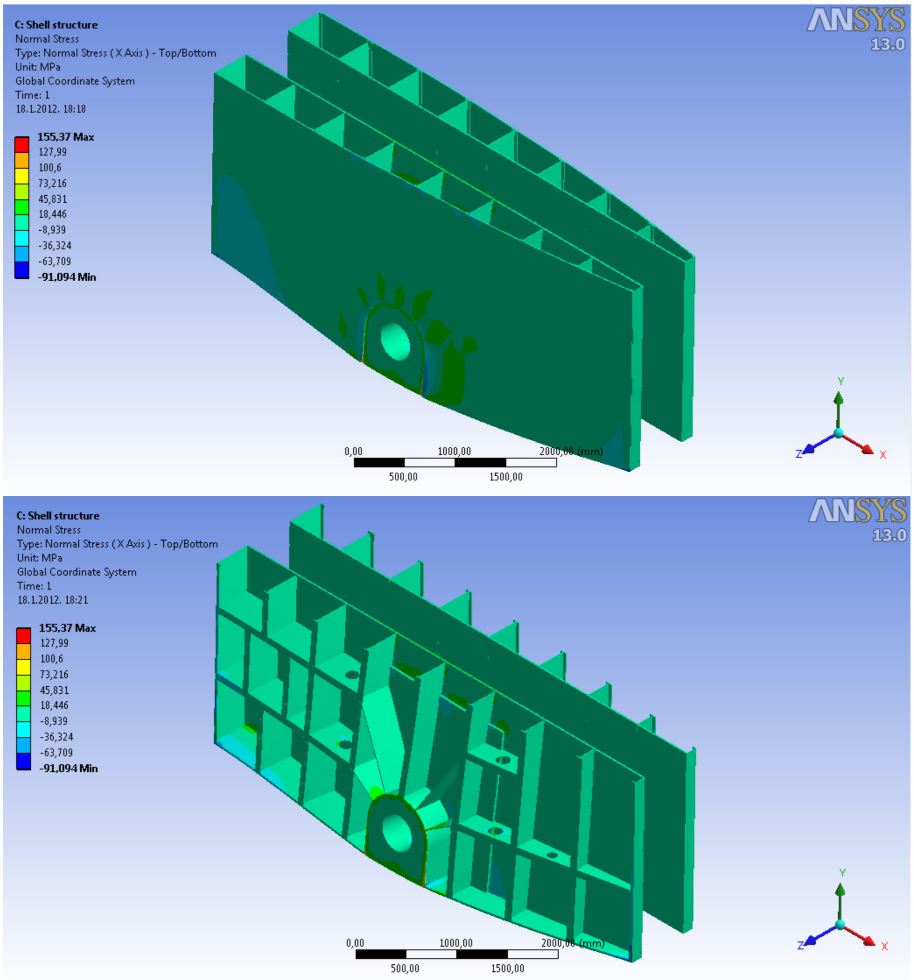


Figure 100. Normal stress for the yacht structure

Maximum normal stresses for the keel structure are 1000 MPa in compression and 1000 MPa in tension. But again maximum stresses are happening where the boundary conditions are, Figure 99 and it will be neglected. From that figure it is possible to see that at other places, stresses are less than 105 MPa. Also for the yacht structure, the biggest normal stress is 156 MPa.

5.2.7. *Shear stress*

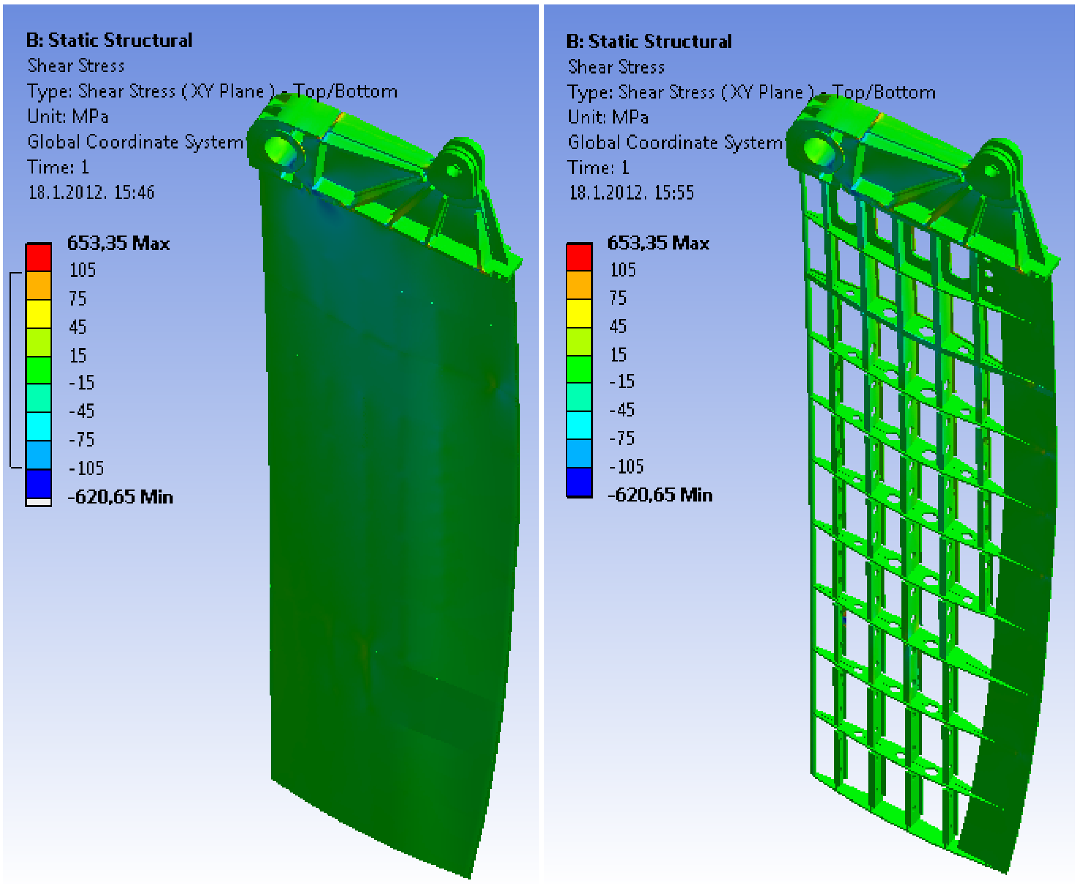


Figure 101. Shear stress of the keel structure

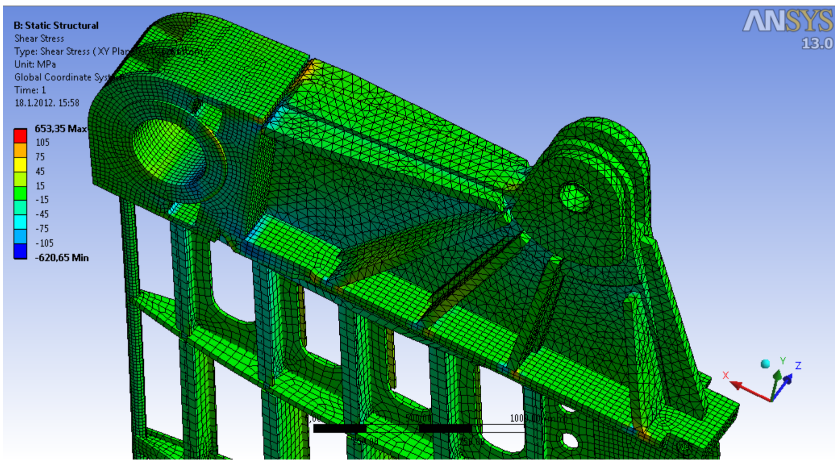


Figure 102. View of the connection swing keel head with structural blade for shear stress

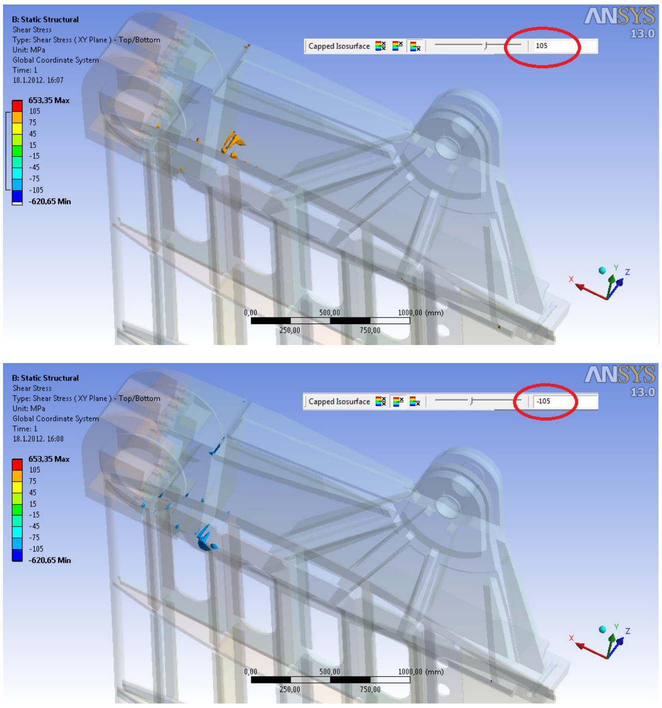


Figure 103. Places where Shear stresses are bigger than 105 MPa

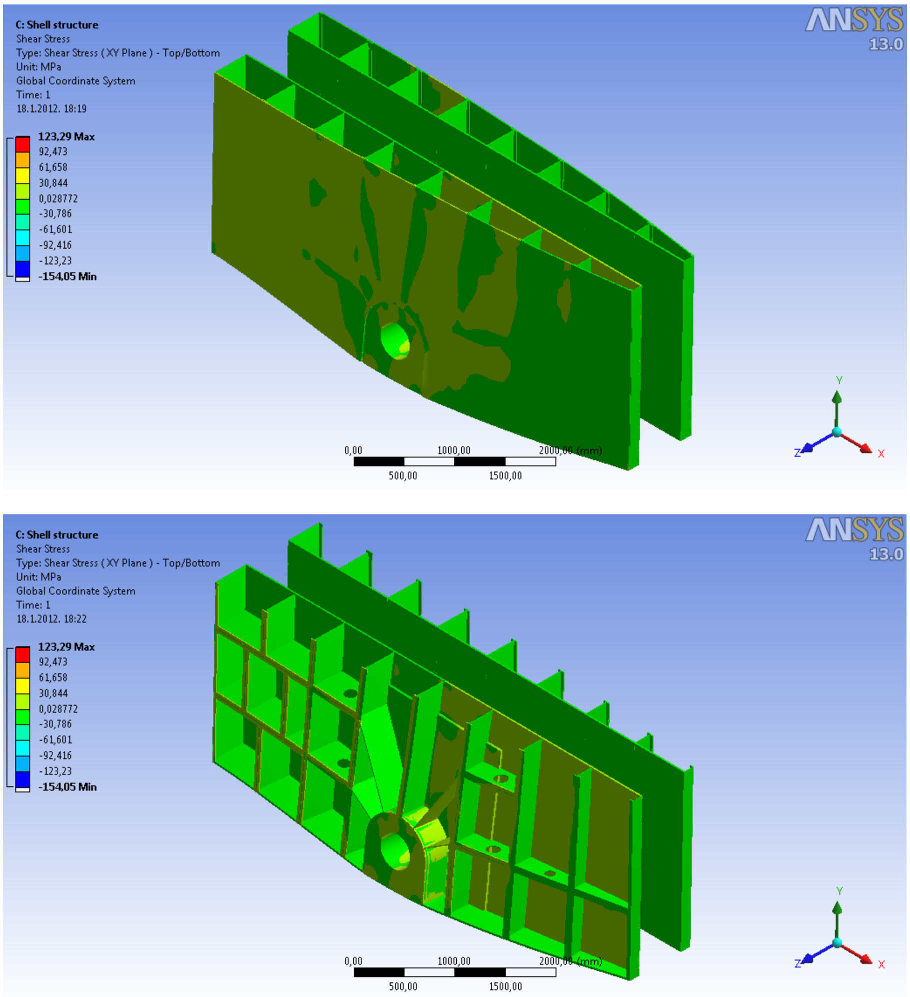


Figure 104. Shear stress for the yacht structure

Maximum shear stress for the keel structure is 654 MPa. And again maximum stresses are happening where the boundary conditions are, Figure 103. From that figure it is possible to see that at other places, stresses are less than 105 MPa. For the yacht structure, the biggest shear stress is 155 MPa.

The keel – structure connection is very well design by Perini Navi shipyard and according to analysis that has been performed, the structure satisfy requirements from the rules. Model for FEM analysis with the chosen boundary conditions simulate reality and results of the numerical analysis have been verified with very simple beam theory calculation for the one section of the keel structure and according to it; the results of numerical analysis are similar with theory.

CONCLUSION

The aluminum is very appreciable material in shipbuilding, because of many advantages and weight reduction of the construction compared with the steel. In the yacht world, the shipyards that are producing the yachts, the aluminum is most common material. However, compared aluminum and steel with the price, steel is much cheaper and that is a reason why the worldwide shipyards, that are producing commercial boats are still using steel as material to build them. From the rules according to which the scantling was performed, the same rules can be applied are for the aluminum and steel. Only the difference will be in ultimate tensile strength and ultimate yield stress. Both of those values are lower for aluminum and because of that, for the same structure, aluminum structure will be thicker usually 1.5 times than structure from the steel. However, density of aluminum is three times lower compared with the steel and because of that structure that has been built from aluminum is thicker but in total approximately two times lighter than steel construction. And one of the most important properties of aluminum is corrosion resistance compared with steel.

The finite element method is an essential and powerful tool for solving structural problems not only in the field of shipbuilding but also in the design of the most industrial products and even in non-structural fields. Software's are developing very fast and programs are becoming more users friendly. ANSYS Workbench has very powerful tool for the contacts and it is very easy to mesh the model with it. Because of the good features for the contacts, it is not necessary to spend additional time to connect nodes of two different parts. Considering the elements which ANSYS Workbench is using for desire calculation, it is always using the best element from its database, for mesh the model for selected kind of calculation (static, CFD, buckling, vibration, thermal etc.) and if it is necessary to select another element, user should use the ANSYS. The type of elements that are chosen should provide satisfactory representation of the deflection and stress distribution within the structure. However through this thesis basic features of the finite element method are analyzed, important parameter of the analysis is the mesh, it has great influence on the results and also on the time to perform the analysis. Shell elements have great advantage compared with solid elements. Considering the time, it is always better to use the linear analysis when it is possible. Boundary conditions should be chosen to make the model behave in a realistic manner. Results for the stresses are influenced a lot with the edges and boundary conditions so in that places it is necessary to care special attention how to comment and take that results into a consideration. The most important for the results analysis is experience of the engineer. During structural dimensioning engineer should evaluate the results using different method to let him to have an idea of the results. If strange results are obtained, engineer should check something; he shouldn't believe immediately the results coming from computer because computer is an instrument for our work and it can't substitute our knowledge and experience.

This thesis set up foundation of the finite element analysis with all explanations that are necessary to understand FEM and this thesis can be very useful for the beginners in this field. ANSYS Workbench in combination with SolidWorks, are very good tools for modelling and performing this kind of numerical analysis. And results from numerical analysis are having good coincidence with direct method calculation.

ACKNOWLEDGEMENTS

First I would like thank to prof. Philippe Rigo, coordinator of this Erasmus Mondus program, for all what he done and what he still is doing for us.

To my coordinator in Italy, prof. Dario Boote, thank you for your time, support and guidelines during the last semester in Italy.

It is a pleasure to thank those who made this thesis possible, especially to Franco Torre, for his help and given possibilities to perform this thesis in Perini Navi shipyard, and especially to my supervisor Stefano Baici, who encouraged, guided and supported me from the initial to the final level of developing and understanding of this thesis.

A special thank is addressed to Audrey Mélotte, with her help all the bureaucracy of the program seemed to be much easier.

I would like to thank to all professors and all universities, for the time and knowledge that was given to us.

I am indebted to my many colleagues to support me during those three semesters and I am happy that I found so many new friends during this program.

And for the end, I am indebted to thank to my family, they are my biggest support.

REFERENCES

[1] – Gardiner, R & Lavery, B – *The Line of Battle: The Sailing Warship 1650-1840,* 1992 (2004 edition), Conway, ISBN 0-85177-954-9

[2] – *Sailing yacht design* – 17th International Ship and Offshore Structures Congress, August 2009, Seoul, Korea

[3] – American Bureau of Shipping – *Guide for Building and Classing Offshore Racing Yachts,* 1994

[4] – Dave Garr – *Boat Strength for Builders, Designers and Owners,* 2000 International marine, USA, ISBN 0-07-023159-1

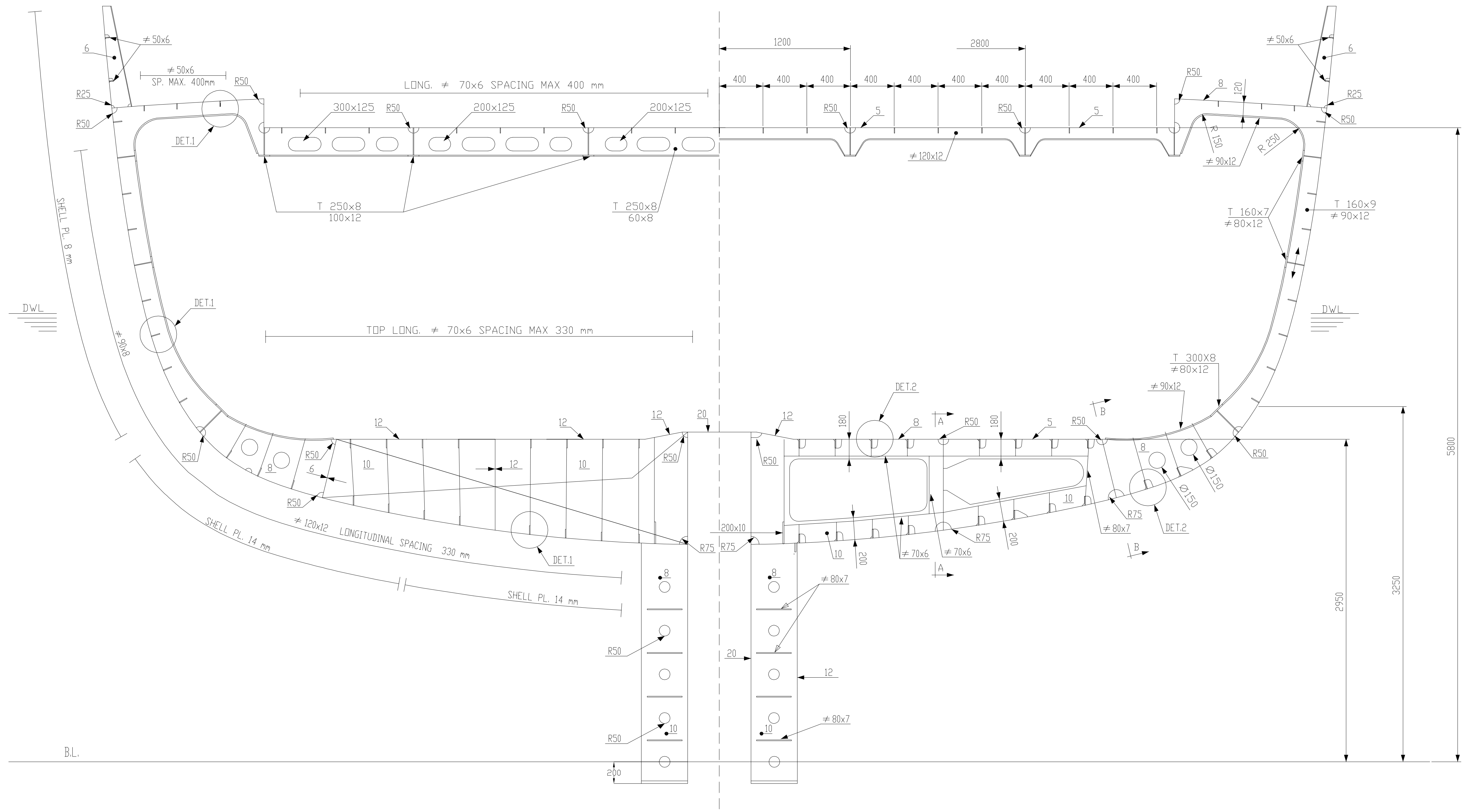
[5] – Y. Okumoto, Y. Takeda, M. Mano and T. Okada – *Design of Ship Hull Structures,* 2009, ISBN 978-3-540-88444-6

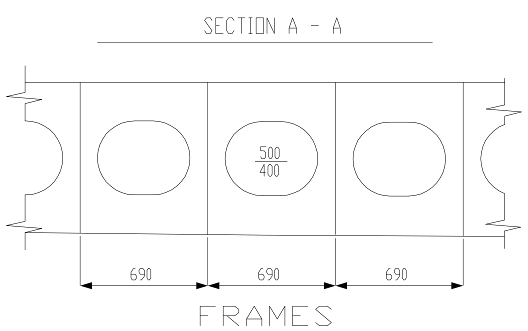
[6] – Perini – Navi shipyard – www.perininavi.it

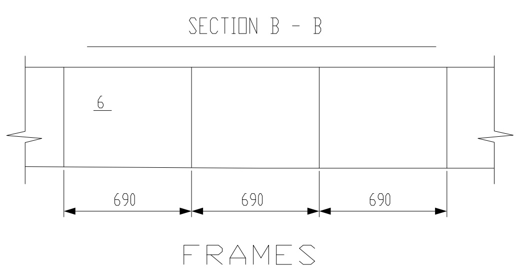
[7] – Lars Larsson, Rolf E. Eliasson – *Principles of Yacht Design* – 2000 London, ISBN 0-7136-5181-4

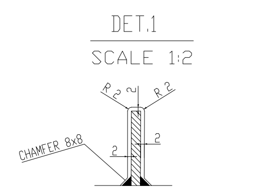
[8] – Yong Bai – *Marine Structural Design* – 2003 Oxford ISBN 0-08-043921-7

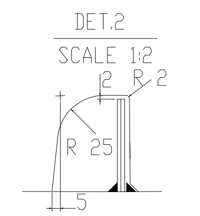
APPENDIX











TITLE		DRAWING N°	
SCANTLING PLAN MIDSHIP SECTION		FILE NAME	REV.
SCALE 1:40	DWN Ivan Klarić		
DATE 17/01/2012	APPVL		