



JOKER MÜHENDİSLİK

GEMİ; PROJE,İNŞAA,
DONATIM,ONARIM

”FEM Analysis Report”

DEC.-2024

Aydıntepe Mah. Çoşkun Sk.
No:14 D.1 Tuzla / İSTANBUL
Tel:00-90-532 367 01 99
info@jokermuhendislik.com
hakan@jokermuhendislik.com

Contents

1	Project Scope	3
1.1	Preface	3
1.2	Project Summary	3
1.3	Analysis Type and Assumptions	3
1.4	Units	3
1.5	Licensed Software	3
2	Model Details & Arrangements	4
2.1	3D Views	4
2.2	Arrangements	6
3	Finite Element Model	8
3.1	Mesh Properties	8
3.2	Element Types	10
3.3	Material Information	10
3.3.1	Mechanical Properties	10
3.3.2	Thickness Details	11
3.4	Loads & Boundary Conditions	12
3.4.1	Loads	12
3.4.2	Constraints	14
3.5	Results & Discussions	15
3.5.1	Von Mises Stress Distribution	15
3.5.2	Conclusion	16

1 Project Scope

1.1 Preface

This finite element analysis report was prepared by an engineering company “JOKER Engineering”

1.2 Project Summary

The purpose of this FEM analysis is to check the strength of the deckhouse supports and the existing structure of the vessel after the newly added deckhouse on the main deck.

Although the carried load was a rigid structure, the “compatible load” option was chosen in FEM software instead of the “rigid load” option in order to be a safer solution.

1.3 Analysis Type and Assumptions

Linear structural analysis with small displacement and small strain was considered as a suitable method for the assessment. A linear elastic material model was used to represent the material properties of the structure.

1.4 Units

SI unit system was followed in analysis:

Force	:	Newton
Deflection	:	Millimeter
Pressure and Stress	:	Mega-pascal

1.5 Licenced Software

Rhinoceros is used for modelling 3D geometry.
Rhinoceros version: 7 SR37

Finite element model geometry is prepared by using “MSC APEX” and “MSC Nastran” is used as a finite element solver.
MSC APEX version: 2023.2
MSC NASTRAN version: 2023.2

2 Model Details & Arrangements

2.1 3D Views

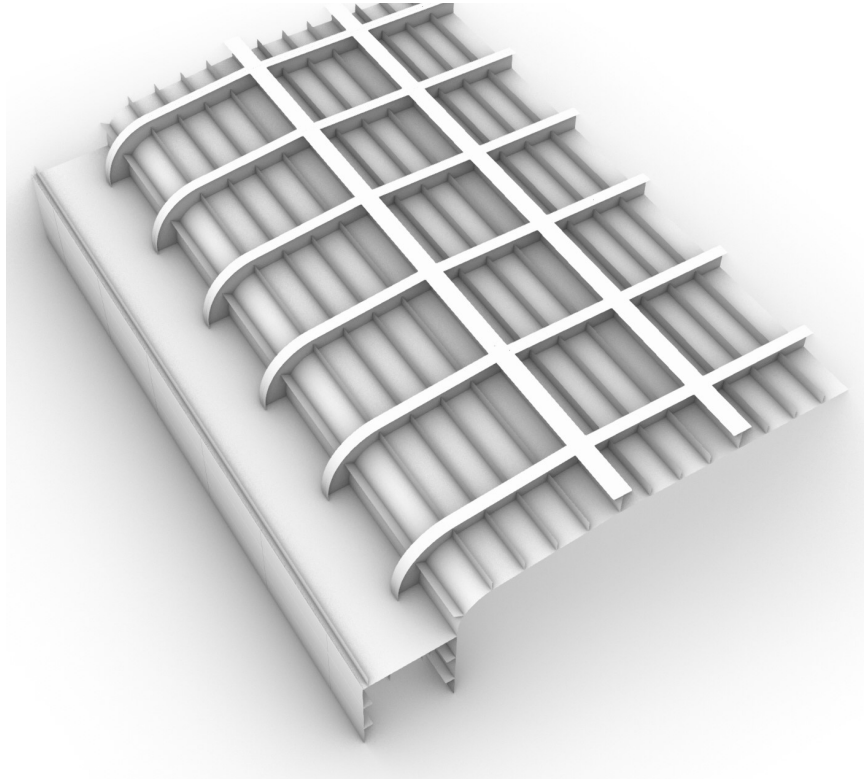


Figure 1: 3D View of existing structure.

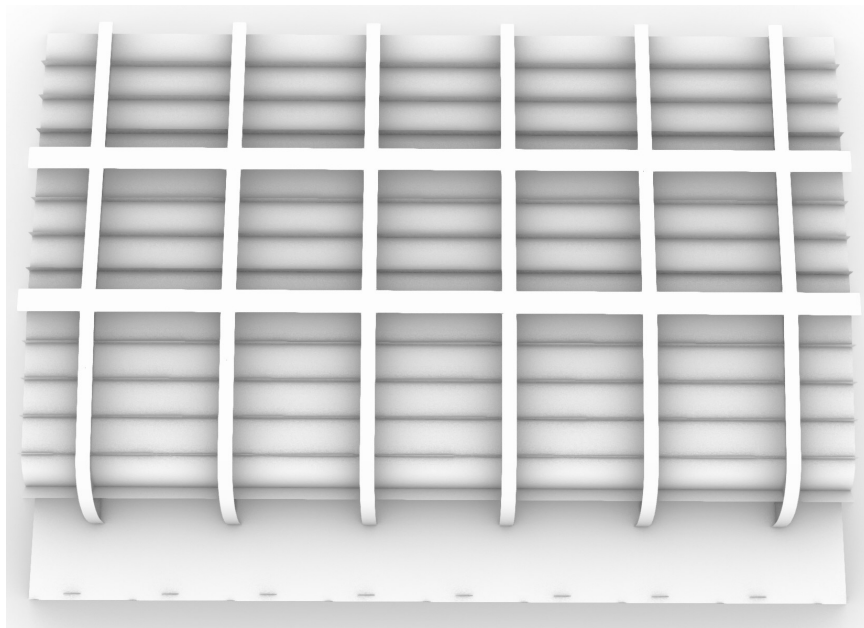


Figure 2: 3D View of existing structure.

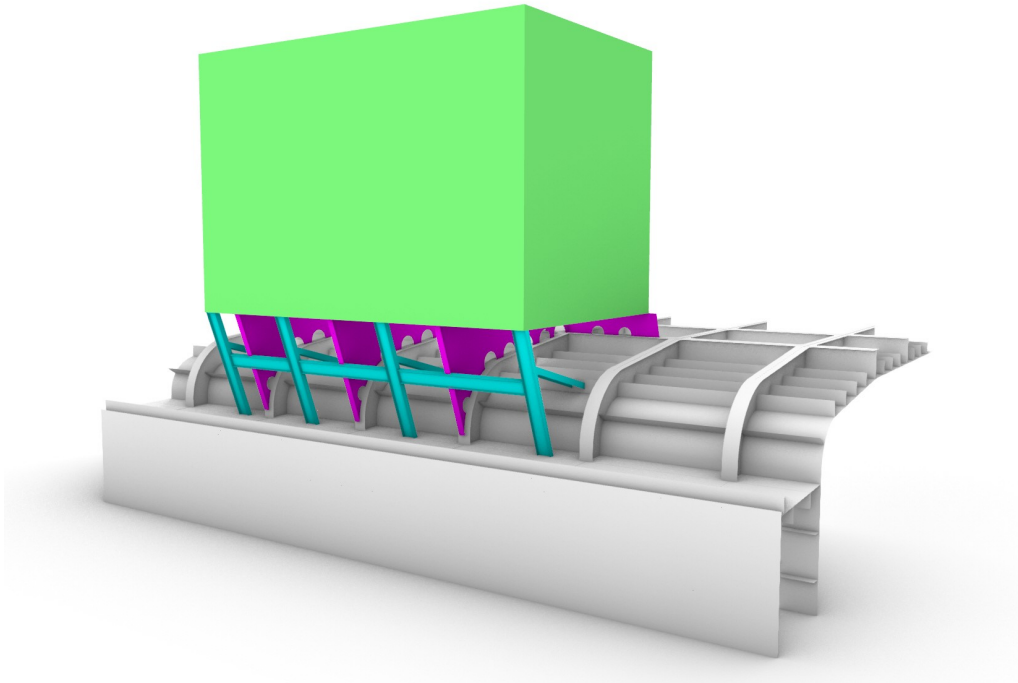


Figure 3: 3D View of deckhouse arrangement.

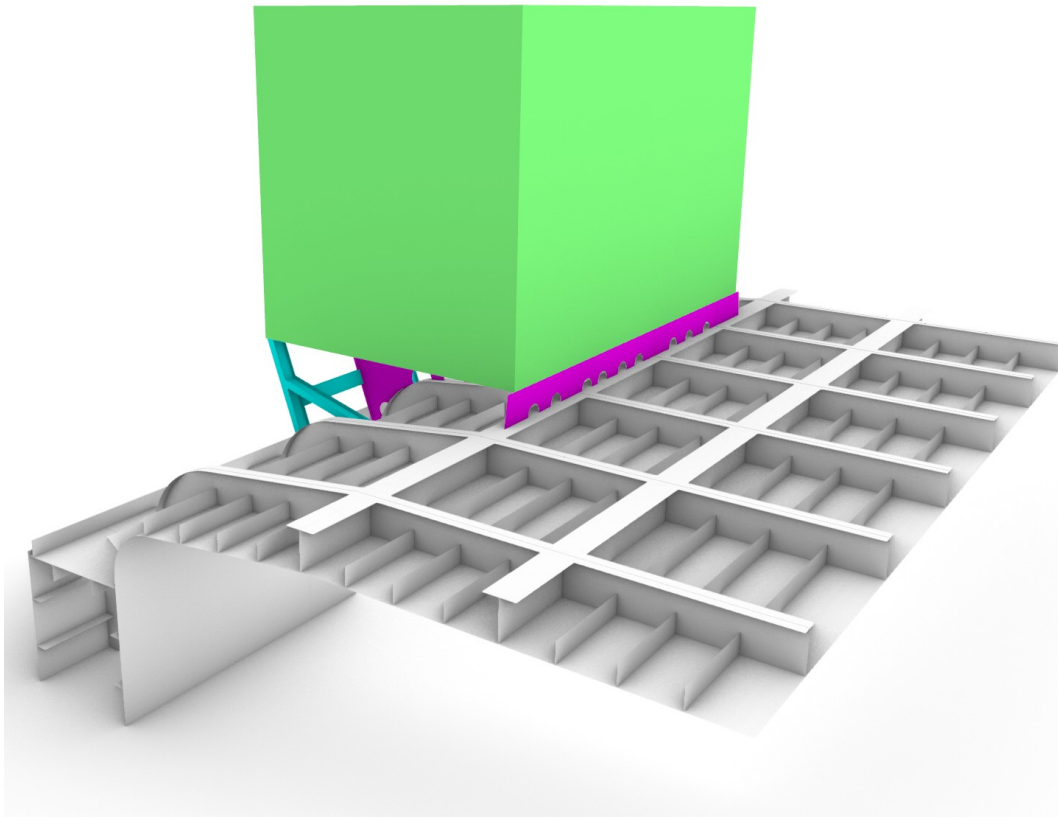


Figure 4: 3D View of deckhouse arrangement.

2.2 Arrangements

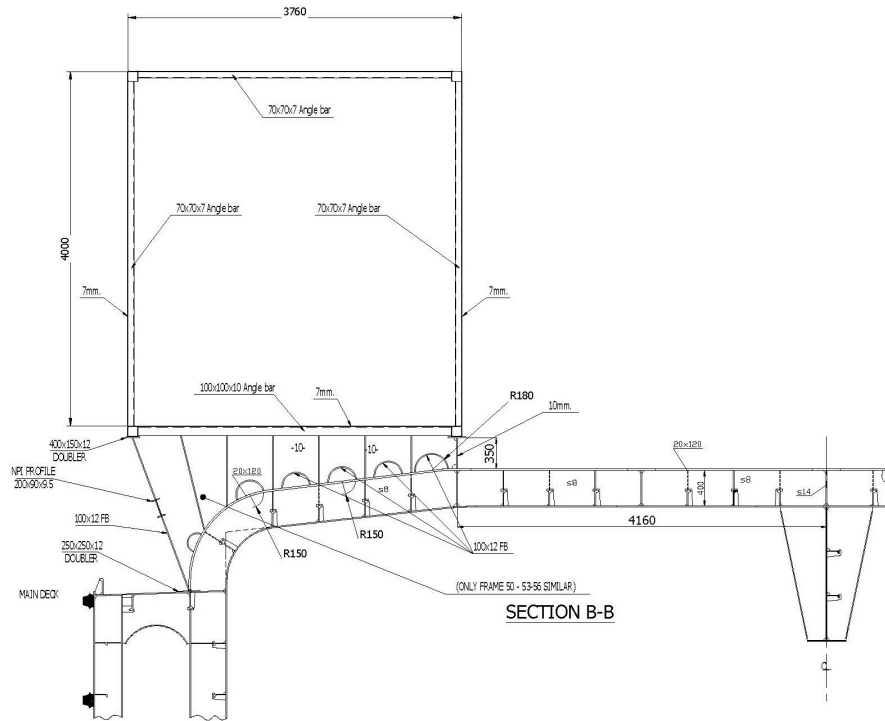


Figure 5: Web Frame-support

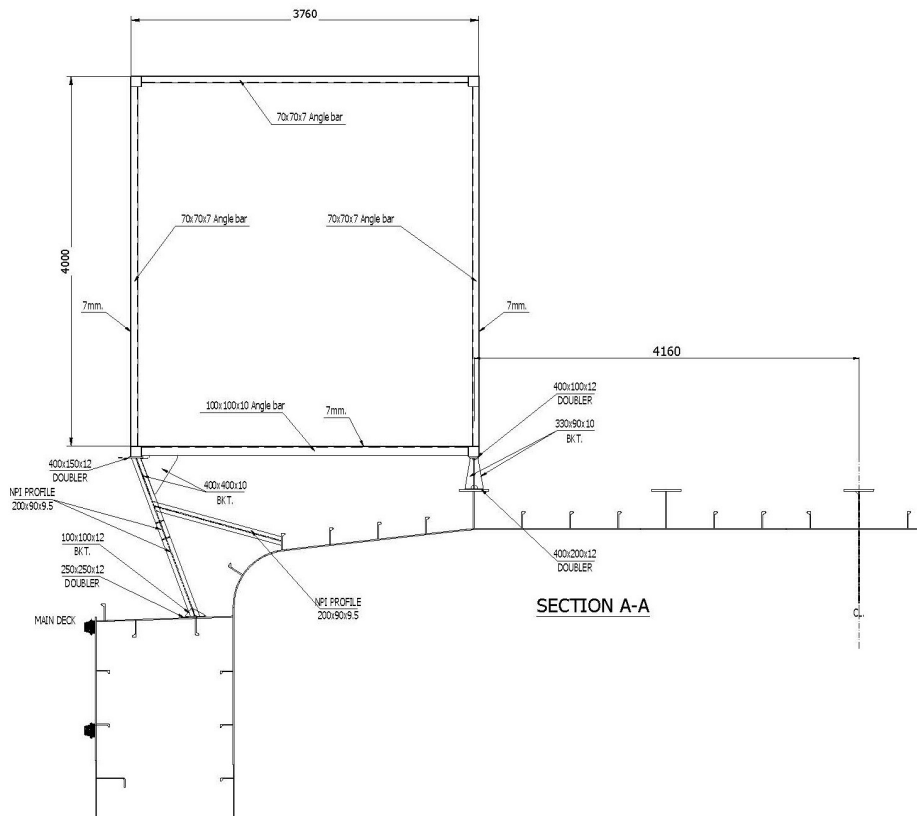


Figure 6: Ordinary support

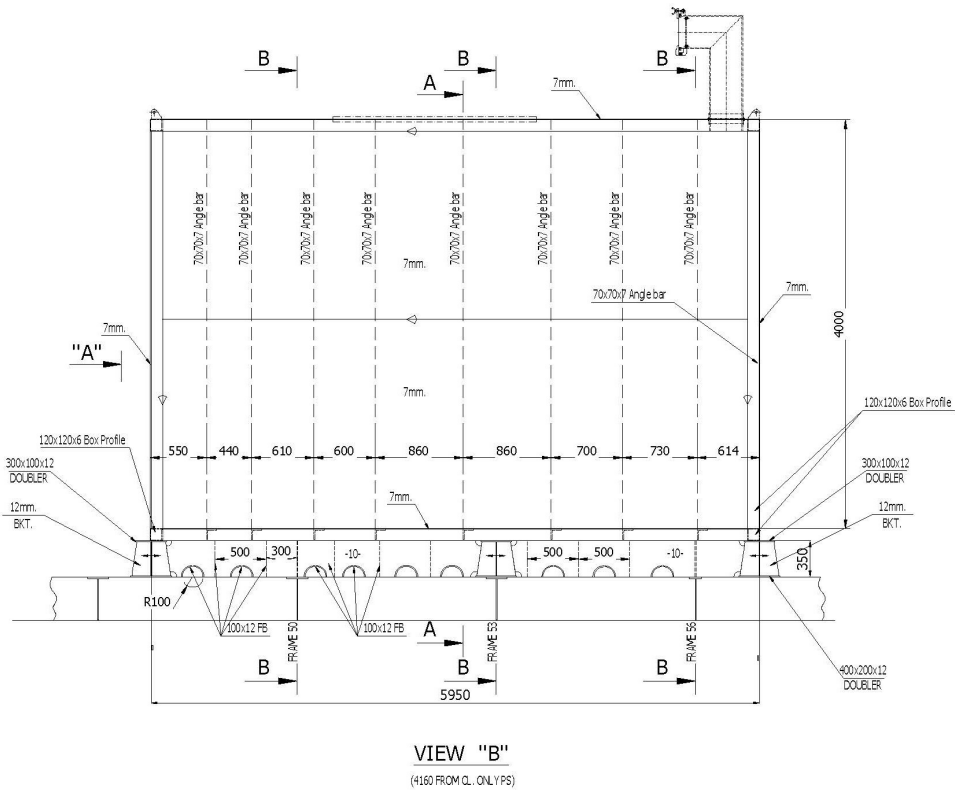


Figure 7: Side View.

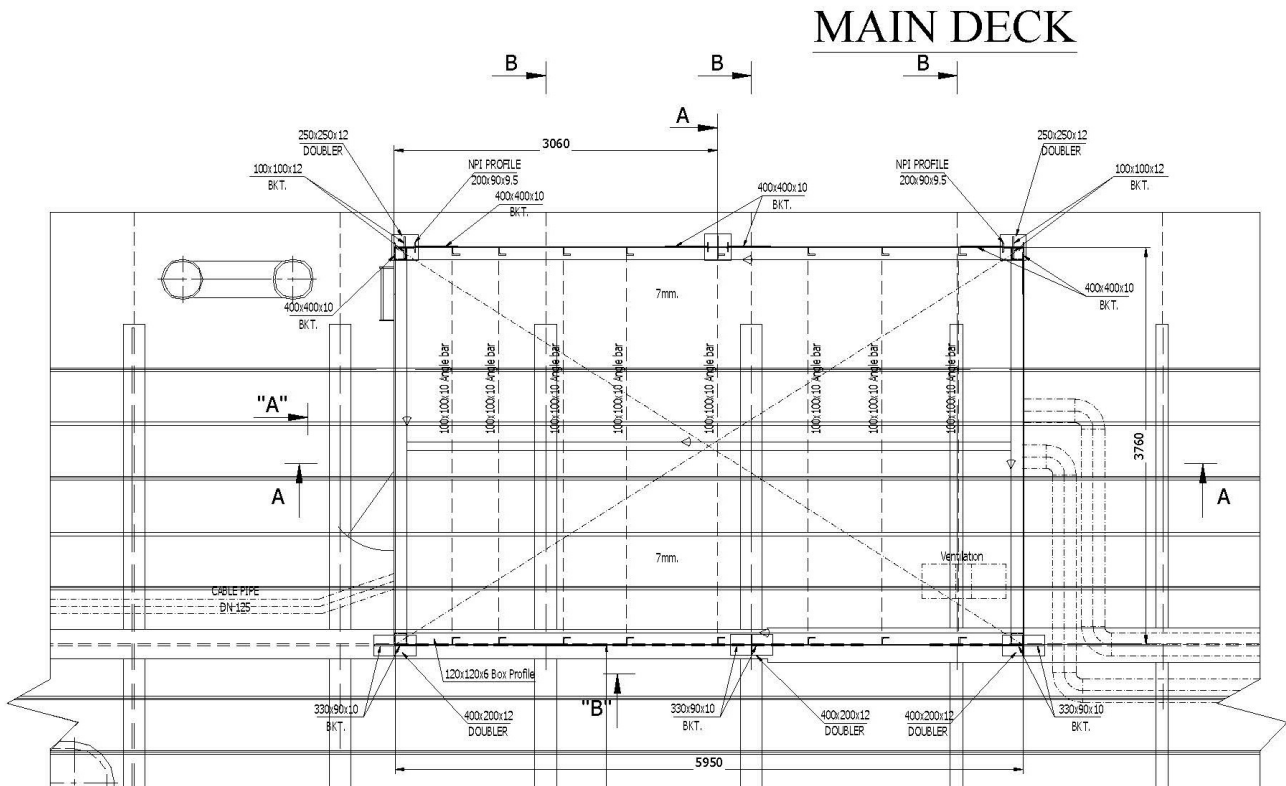


Figure 8: Top View.

3 Finite Element Model

3.1 Mesh Properties

The finite element model details are listed and mesh models can be seen in Figures 9 to 11. The mesh size for the entire model is selected as 50mm. Shell element types are preferred for FEM model.

FEM:

Number of Elements	Number of Nodes
109146	109151

Elements by topology	Number of Nodes
Total Nodes	109151
Total Elements	109146
Beam Elements	0
Shell Elements	109146
Tetrahedral Elements	0
Hexahedral Elements	0

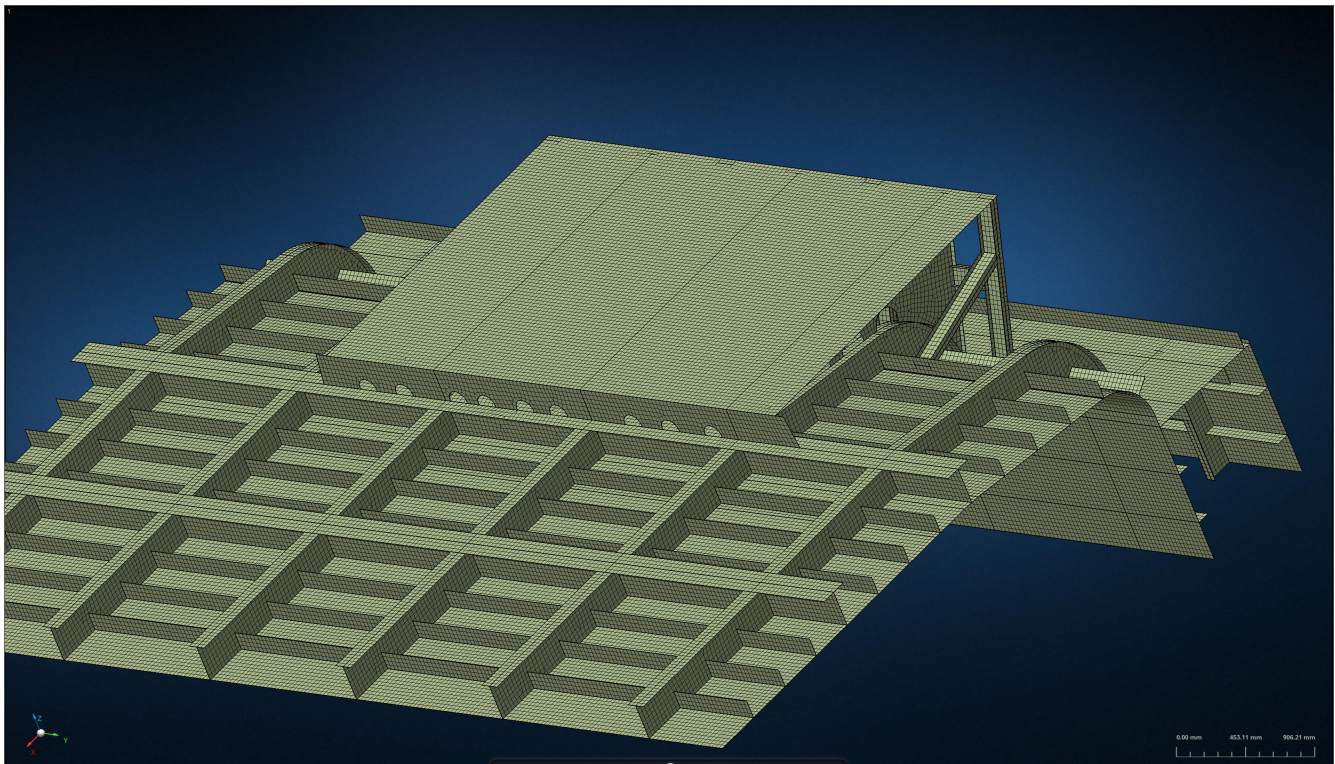


Figure 9: FEM Model.

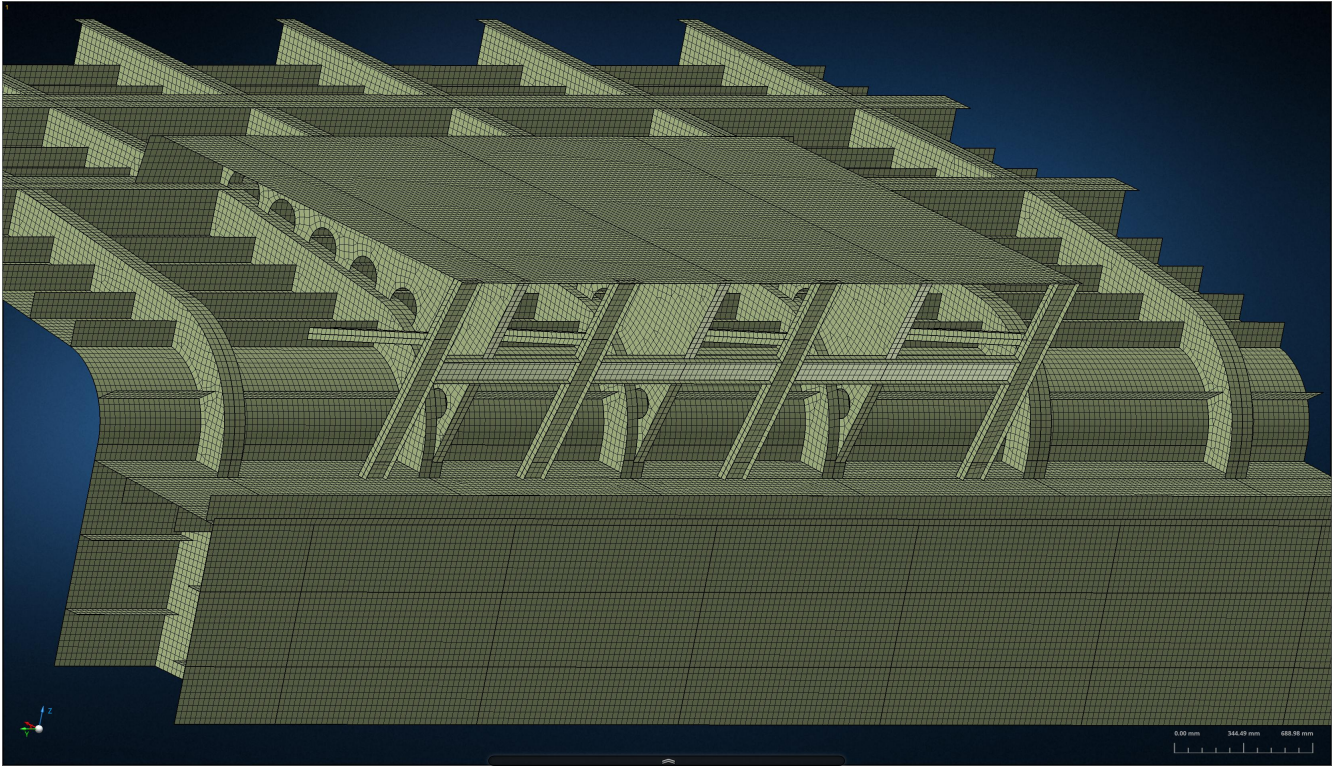


Figure 10: FEM Model.

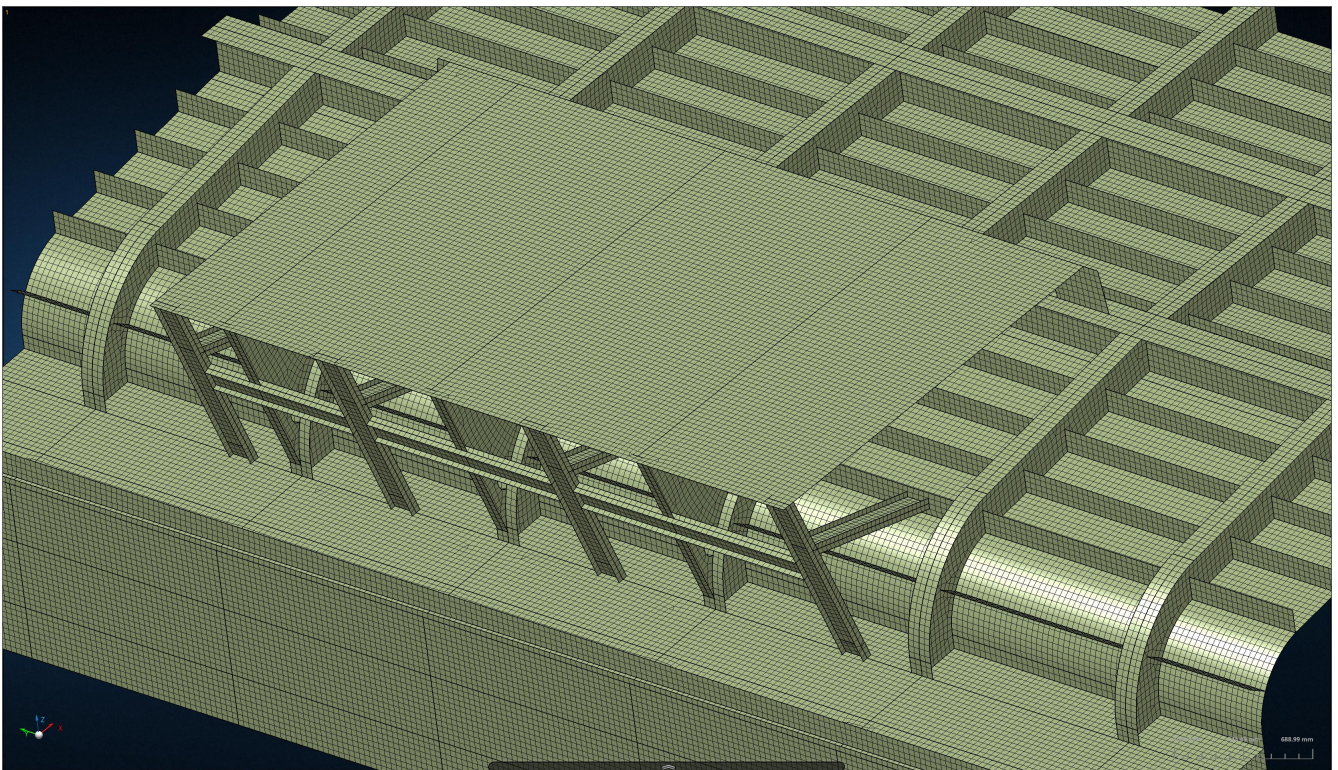


Figure 11: FEM Model.

3.2 Element Types

Using shell elements was found suitable to represent the deformation behavior of 8mm to 20mm steel plates. “CQUAD4” plate element of NASTRAN software was chosen and its geometry and coordinate system can be found in Figure 12 below.

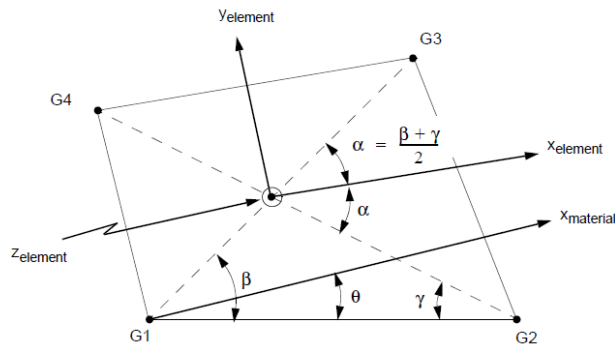


Figure 12: CQUAD4 Element geometry and coordinate systems.

3.3 Material Information

3.3.1 Mechanical Properties

Linear elastic properties of the material is presented in Table 1.

Material	: Steel	Steel
Type	: A-Gr	AH-32
Young's modulus	: 205 GPa	205 GPa
Poisson ratio	: 0.27	0.27
Yield Stress	: 235 MPa	315 MPa
Permissible Stress	: 120 MPa	154 MPa

Table 1: Mechanical properties.

3.3.2 Thickness Details

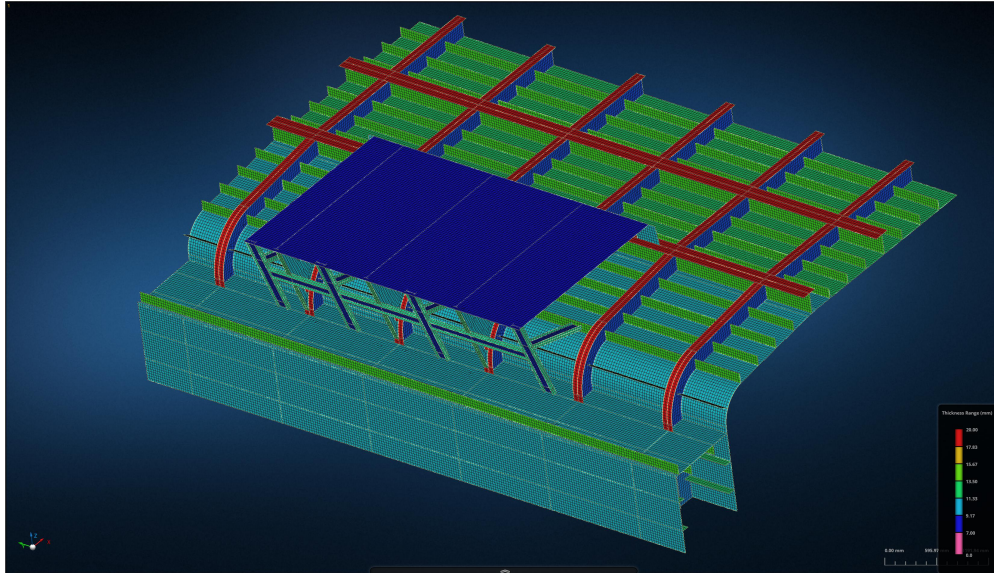


Figure 13: 3D View of thickness range.

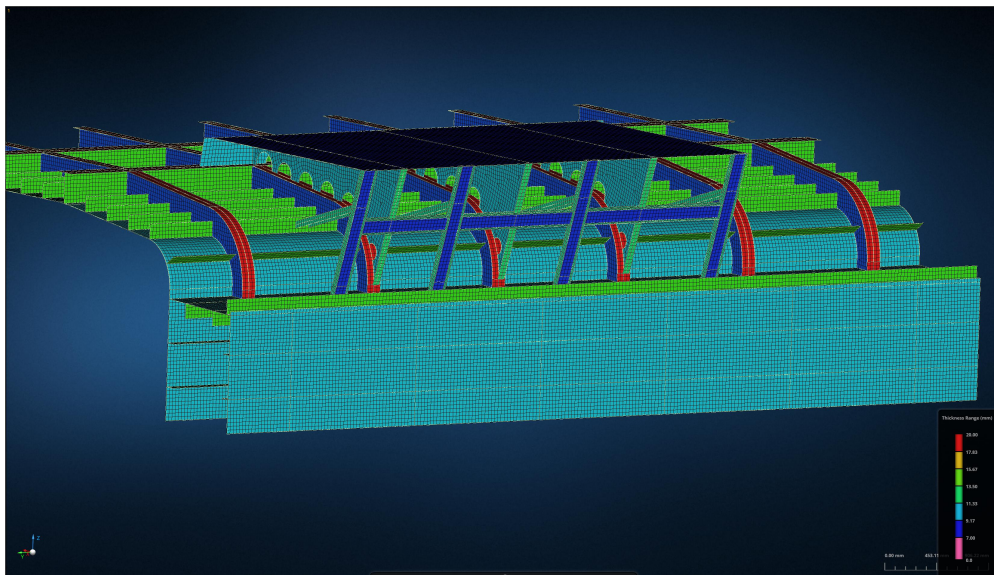


Figure 14: 3D View of thickness range.



Figure 15: Thickness scale.

3.4 Loads & Boundary Conditions

3.4.1 Loads

Total weight of deckhouse is 13.3 tonnes.

Center of Gravity of the deckhouse;

X	: 31.95 m from AP
Y	: 4.60 m
Z	: 9.30 m from BL

Accelerations;

a_L	: 1.80 m/s ²
a_T	: 5.85 m/s ²
a_V	: 5.95 m/s ²

Applied Force Components;

F_X	: 23400 N
F_Y	: 77805 N
F_Z	: 209608 N

Applied loads can be seen in Figure. 16 and Figure. 18

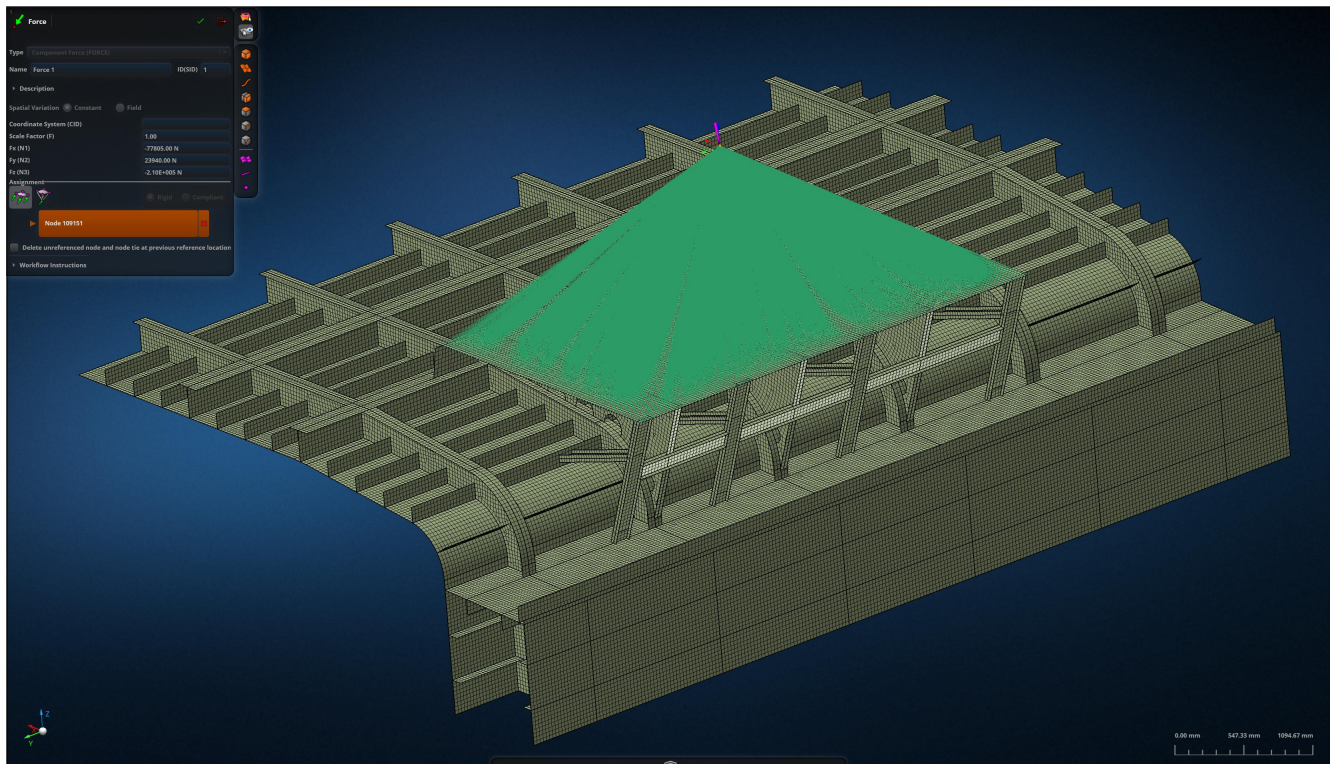


Figure 16: Loads

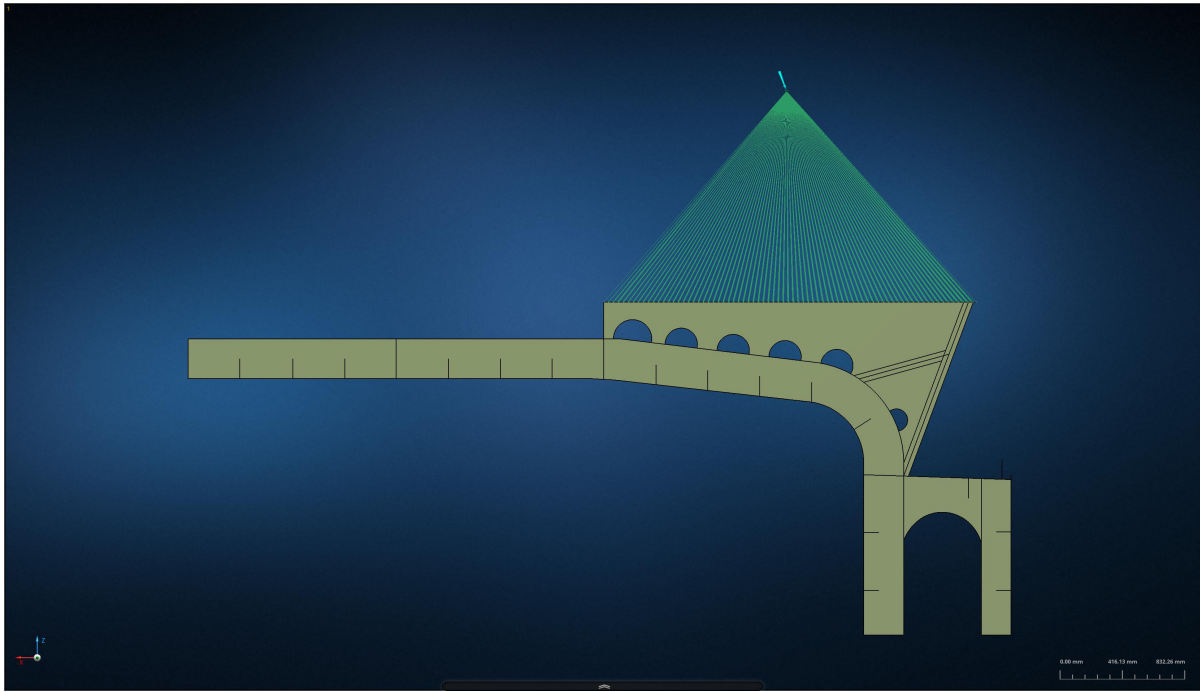


Figure 17: Loads

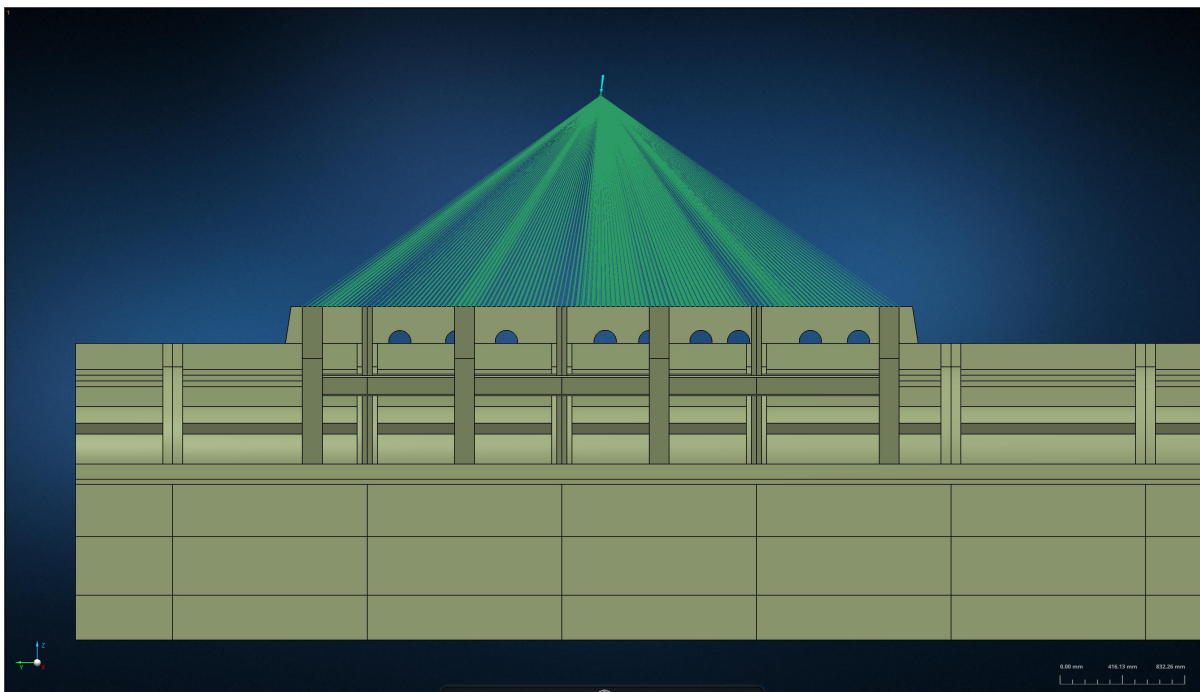


Figure 18: Loads

3.4.2 Constrains

The structure is clamped by center of the trunk deck, and also ends of trunk deck and longitudinal bulkheads. Boundary conditions can be seen in Figure 19 and Figure 20

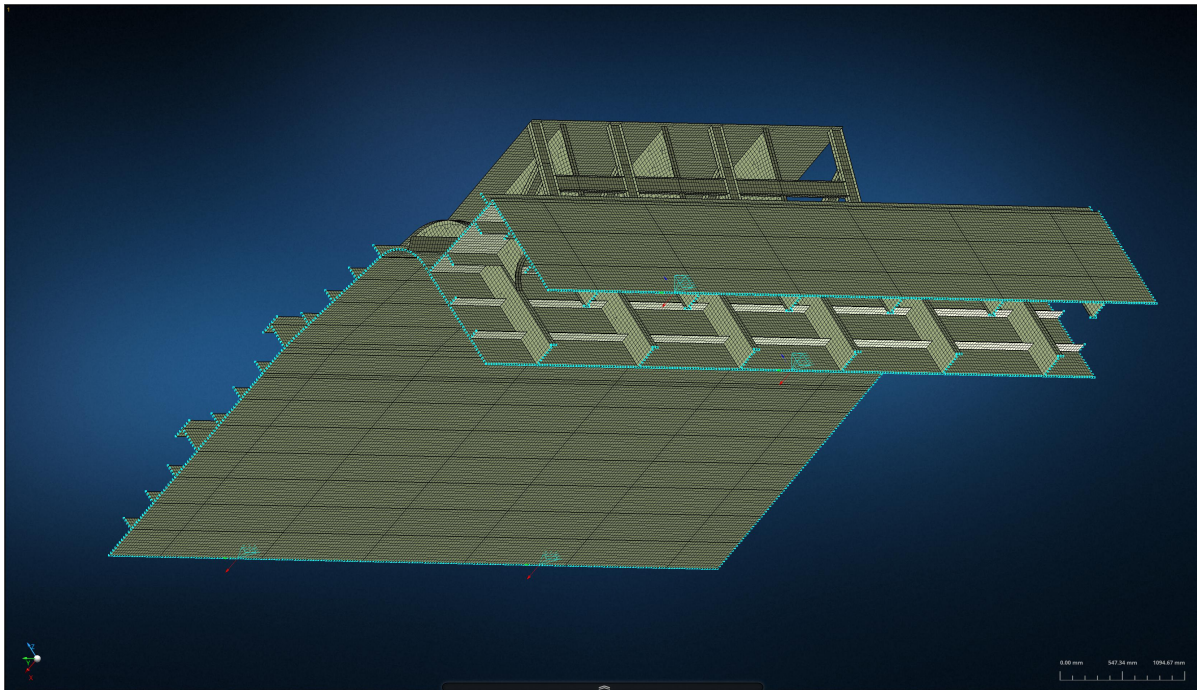


Figure 19: Constrains

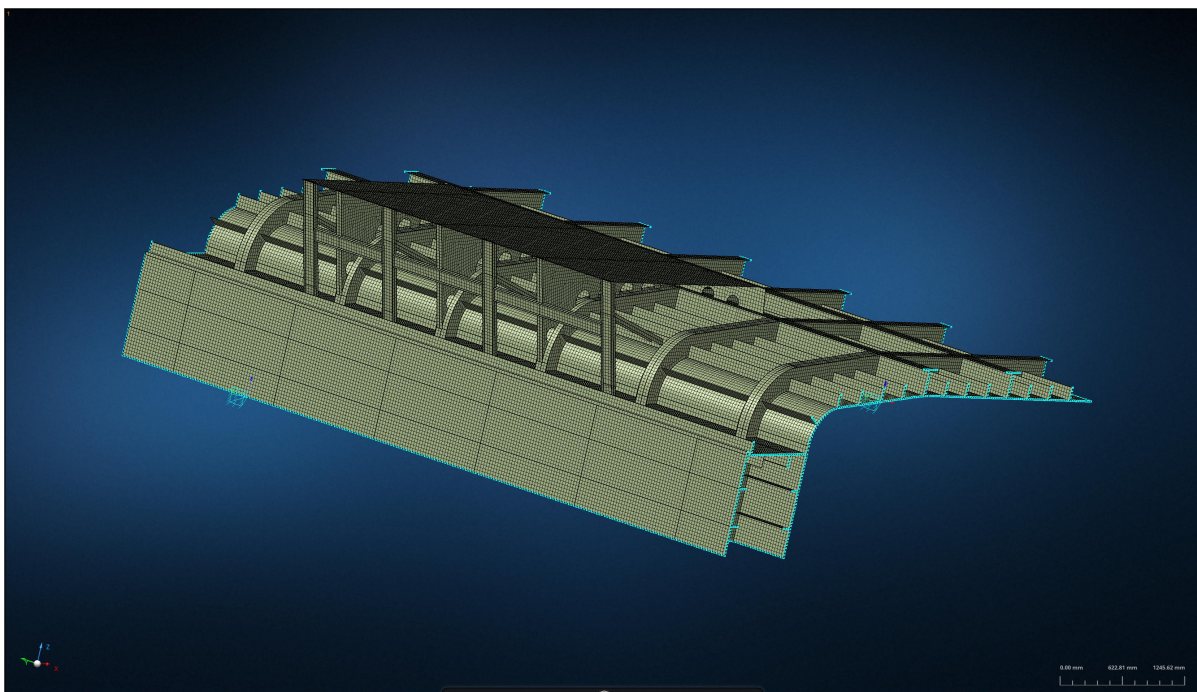


Figure 20: Constrains

3.5 Results & Discussions

3.5.1 Von Mises Stress Distribution

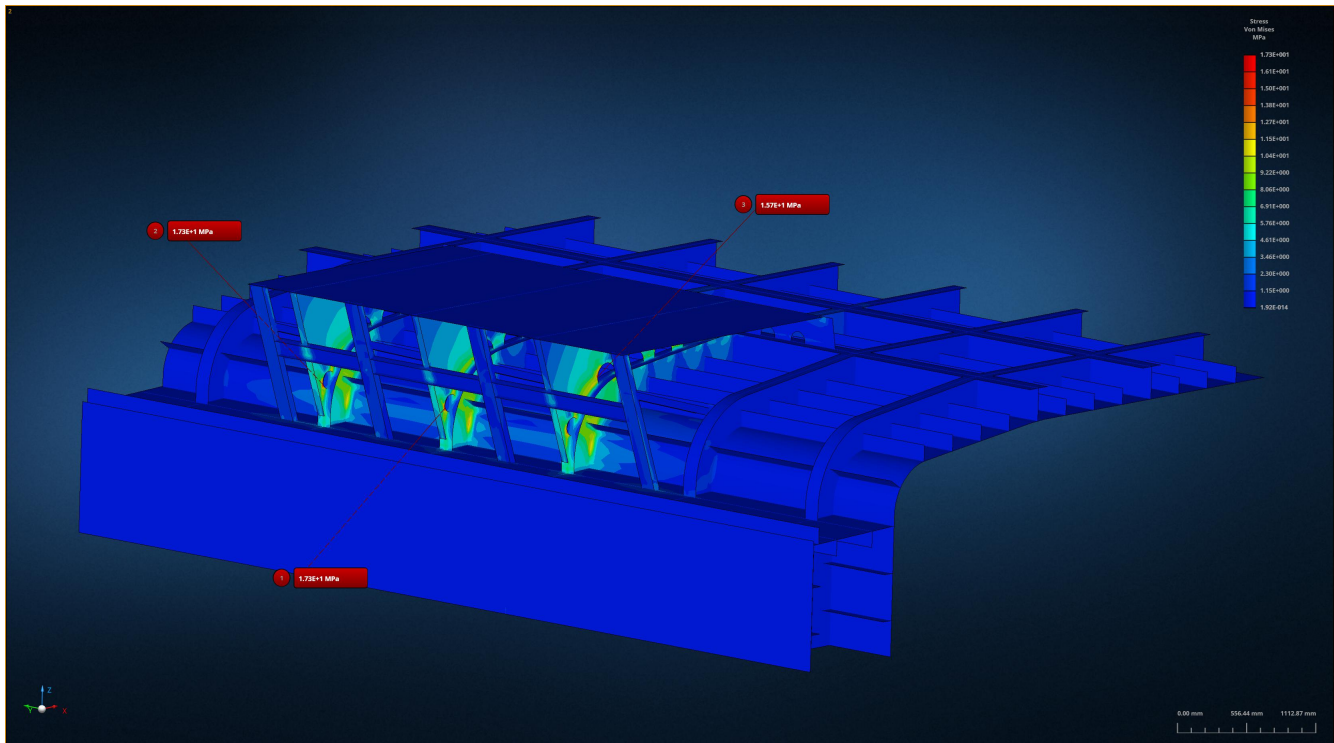


Figure 21: Stress Distribution

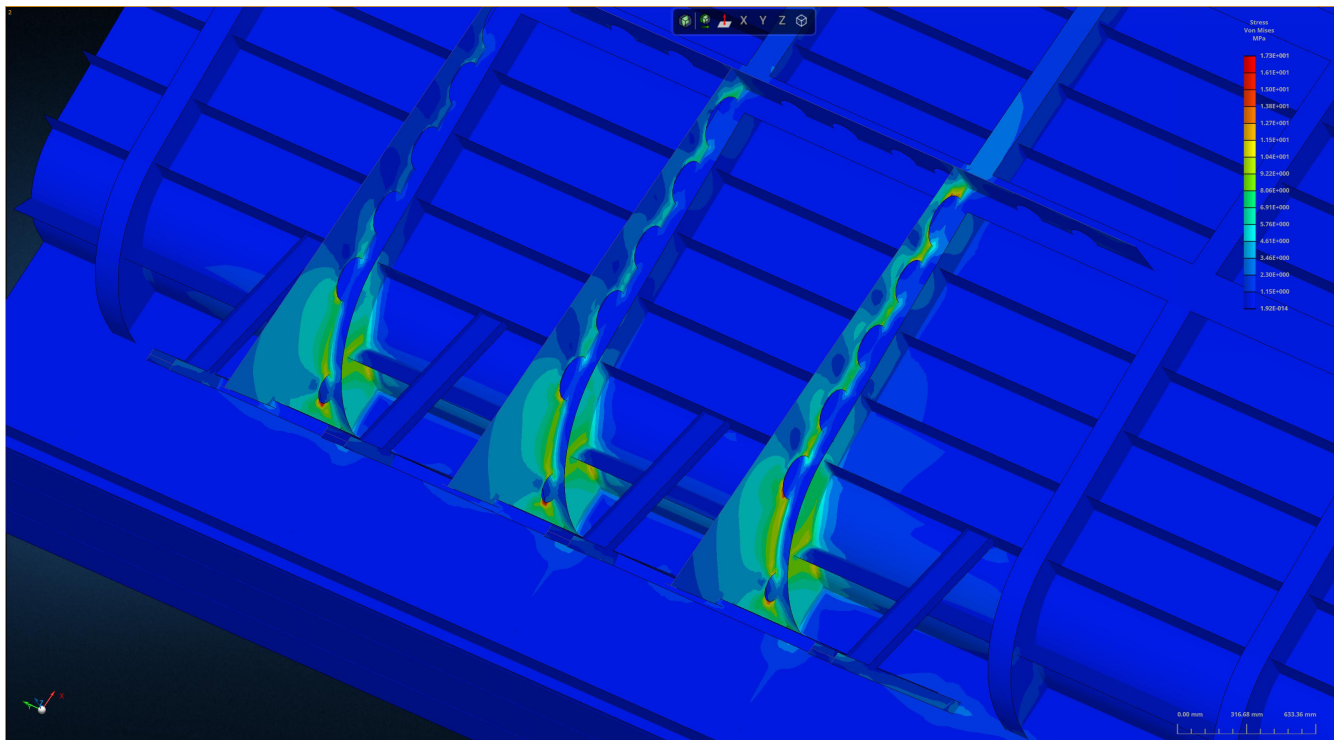


Figure 22: Stress Distribution Near Max. Stress Region.

3.5.2 Conclusion

Maximum stress is about 17.3 MPa on the web support edges. The structure was examined under the conditions specified in the report. Since the maximum stress values were far below the permissible values, it is seen that existing structures and additional deckhouse supports are sufficient. Max. Stress region can be seen in Figure. 23 and Figure. 24

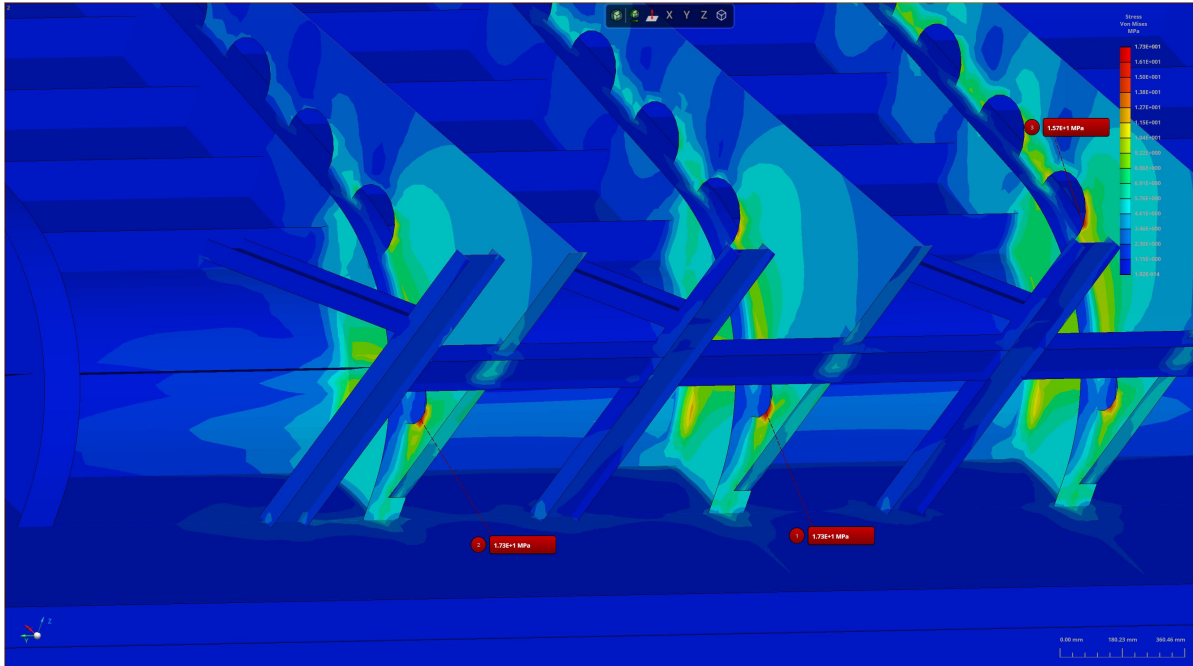


Figure 23: Stress Distribution Near Max. Stress Region.



Figure 24: Stress Distribution