MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE
Admiral Makarov National University of Shipbuilding
Educational and Scientific Institute of Shipbuilding

Department of Structural Mechanics and Ship Hull Design

"Approved for defense"
Head of the Department
Korostylyov L.I.
«_26_»___12___2024 y.

QUALIFICATION WORK

for obtaining the degree of higher education "Master"

Thesis topic: Investigation of the Stress-Strain State in Conductor Elements during the Welding of Floating Pontoon Elements

Performed by: Student of group 6116M

____________ Grechyn V.Y.
(signature)

Supervisor:
Head of the Department of Ship Structural Mechanics and Ship Hull Design, Doctor of Technical Sciences, Professor.
(position, academic degree, academic title)

____________ Korostylyov L.I.
(signature)

Reviewer:
Project Manager
Marine Design Engineering Galati

____________ Popov O.I.
(signature)

Mykolaiv – 2024 y

MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE
Admiral Makarov National University of Shipbuilding
Educational and Scientific Institute of Shipbuilding

Department: Structural Mechanics and Ship Hull Design
Educational-Qualification Level: Master
Specialty: 135 “Shipbuilding”
Educational Program: “Design and Strength of Ship Hulls and Floating Structures”

“APPROVED”
Educational Program Guarantor
____
(signature)
«26» 12 2024 y.

ASSIGNMENT
FOR THE QUALIFICATION WORK
for obtaining the degree of higher education “Master”
Grechyn Viacheslav Yaroslavovych

1. Thesis topic: Investigation of the Stress-Strain State in Conductor Elements during the Welding of Floating Pontoon Elements
Supervisor: Head of the Department of Ship Structural Mechanics and Ship Hull Design, Doctor of Technical Sciences, Professor. Korostylyov L.I.
Approved by Rector's Order No. dated « » 2024 year
2. Deadline for submission of the work:
3. Initial data for the work:
4. List of questions related to development (section titles): Methodology for conducting thermal and stress-strain analysis; Application of the finite element method considering thermal and mechanical boundary conditions; Results of thermal and stress-strain analysis of conductor elements during welding.
5.List of presentation materials: The presentation was created in PowerPoint.

Cha.	Page	№ docum.	Signature	Data	19.ДРМ.135.6116м.ПЗ.03	Page 1

6. Consultants for sections of the work

Section	Surname, initials, and position of the consultant	Signature, date	
		Assignment provided by	Assignment received by
1	Korostylyov L.I. Head of the Department		
2	Korostylyov L.I. Head of the Department		
3	Korostylyov L.I. Head of the Department		

7. Date of Assignment Issuance______________________________

CALENDAR PLAN

Number	Name of work stages	Deadlines for work stages	Note
1.	Study of literature sources on the research subject	September 2024	completed
2.	Development of a detailed plan for the master's thesis and review of the qualification work plan by the supervisor	September 2024	completed
3.	Writing Chapter 1	October 2024	completed
4.	Writing Chapter 2	October 2024	completed
5.	Writing Chapter 3	November 2024	completed
6.	Formatting the master's thesis	December 2024	completed
7.	Submission of the master's thesis to the reviewer for evaluation	December 2024	completed
8.	Submission of the master's thesis to the academic supervisor for feedback	December 2024	completed
9.	Defense of the master's thesis		
10.			

Student ____________________ Grechyn V.Y.
(signature) (Full Name)

Supervisor ____________________ Korostylyov L.I.
(signature) (Full Name)

TABLE OF CONTENTS

ABSTRACT

Master's Thesis Title:

"Investigation of the Stress-Strain State in Conductor Elements during the Welding of Floating Pontoon Elements"

This master's thesis is dedicated to investigating the stress-strain state in metallic conductor elements during the welding of modular floating pontoon components. The study is relevant in the context of ensuring assembly accuracy and improving the quality of welded structures for modern shipbuilding needs.

The research analyzes the thermal and mechanical processes occurring during welding using the Finite Element Method (FEM) in the ANSYS software environment. A three-dimensional model of the conductor and pontoon was developed, thermal and mechanical analyses were performed, and deformations caused by thermal loads were evaluated.

The primary objective of the research was to determine the effectiveness of the conductor design in maintaining the geometric parameters of pontoon elements within the allowable deviation of 0.5 mm after the welding process. The conducted simulations confirmed the conductor's capability to reduce deformations and ensure precise positioning of elements.

The results of the study are of significant practical importance for improving the design of metallic conductors and optimizing welding processes. The proposed methods and approaches can be applied to other types of floating structures and shipbuilding components.

Keywords: welding, modular pontoon, stress-strain state, thermal analysis, mechanical analysis, finite element method, ANSYS, conductor.

АНОТАЦІЯ

Магістерська робота на тему:

"Дослідження напружено-деформованого стану в елементах кондуктора під час зварювання елементів плавучого понтона"

Магістерська робота присвячена дослідженню напружено-деформованого стану в елементах металевого кондуктора під час зварювання елементів модульних плавучих понтонів. Робота актуальна в контексті забезпечення точності складання та підвищення якості зварювальних конструкцій для потреб сучасного суднобудування.

У дослідженні проаналізовано термічні та механічні процеси, які відбуваються під час зварювання, з використанням методу скінченних елементів (FEM) у програмному середовищі ANSYS. Розроблено тривимірну модель кондуктора та понтона, виконано тепловий і механічний аналіз, а також оцінено деформації, спричинені термічними навантаженнями.

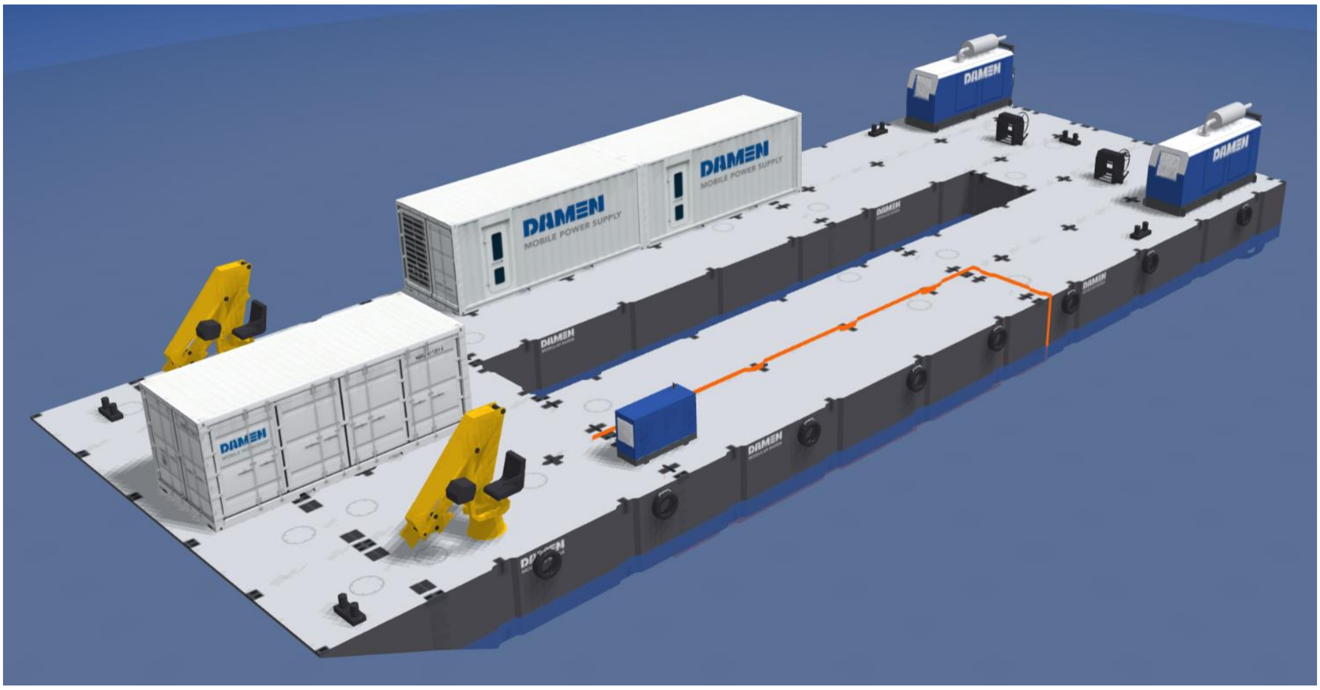
Основною метою дослідження було визначення ефективності конструкції кондуктора в забезпеченні збереження геометричних параметрів елементів понтона у межах допустимого відхилення 0,5 мм після процесу зварювання. Проведені моделювання підтвердили здатність кондуктора зменшувати деформації та підтримувати точність позиціонування елементів.

Результати роботи мають важливе практичне значення для вдосконалення конструкцій металевих кондукторів та оптимізації зварювальних процесів. Запропоновані методи й підходи можуть бути застосовані для інших типів плавучих конструкцій і компонентів суднобудування.

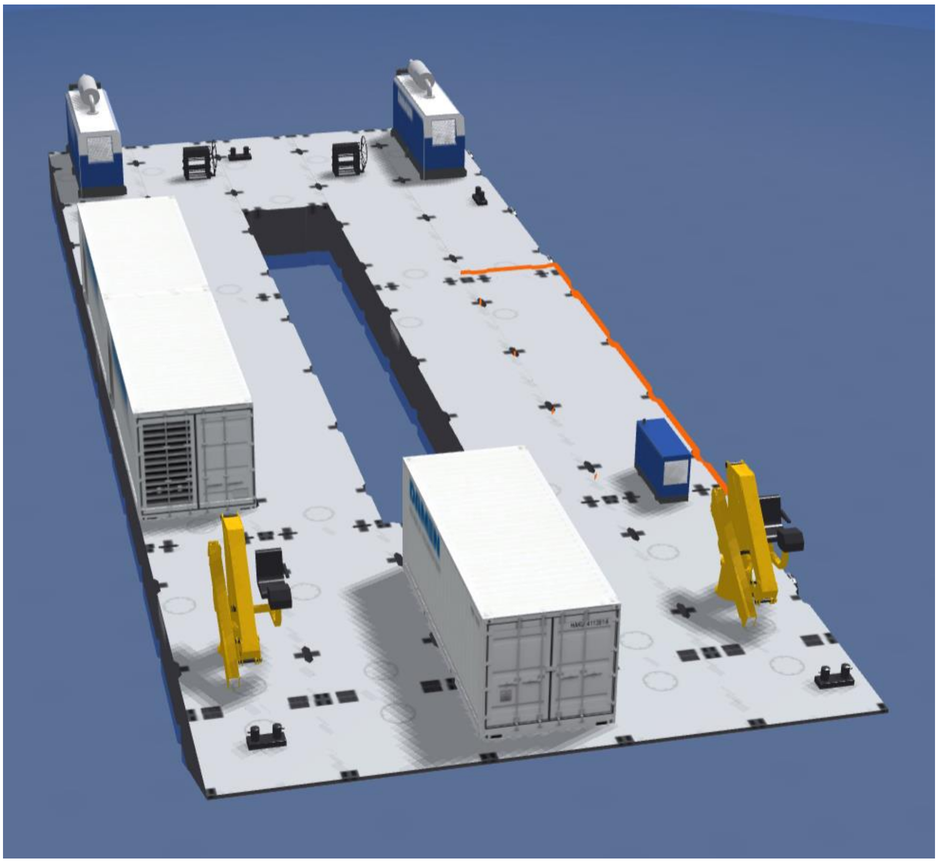
<u>Ключові слова:</u> зварювання, модульний понтон, напружено-деформований стан, термічний аналіз, механічний аналіз, метод скінченних елементів, ANSYS, кондуктор.

INTRODUCTION

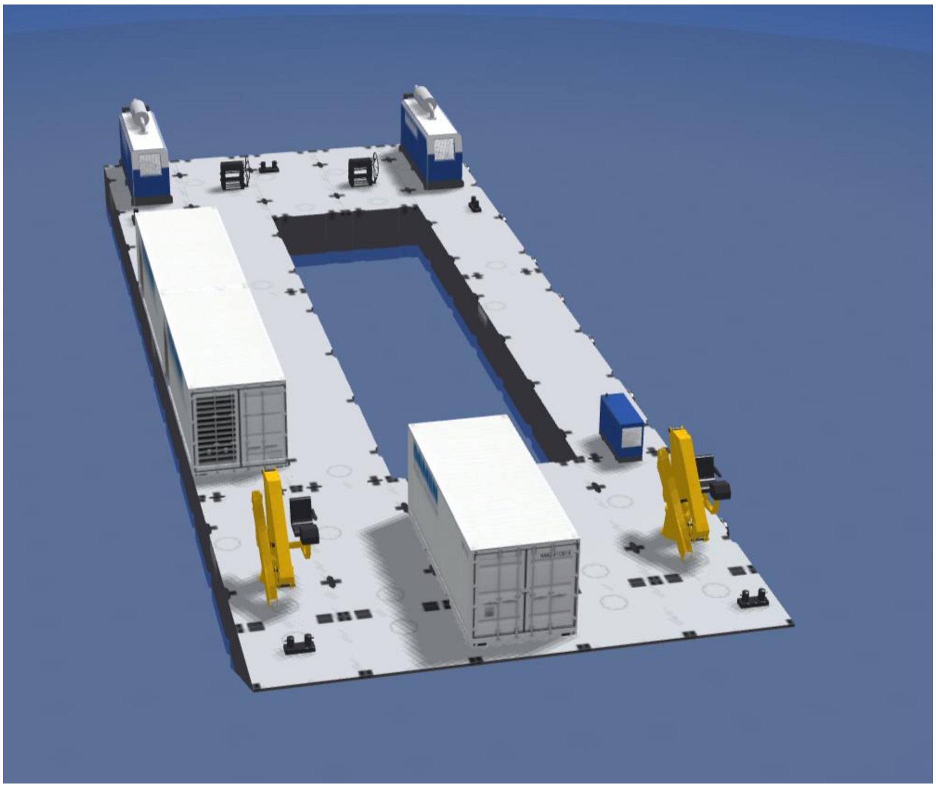
Relevance of Study. The relevance of studying the stress-strain state in conductor elements during the welding process of floating pontoon elements lies in the growing demand for modular steel pontoons in modern engineering applications. Floating modular pontoons are widely recognized for their versatility, ease of assembly, and adaptability to various marine environments. Their modularity allows for the construction of complex floating platforms by combining individual units, making them suitable for a wide range of tasks, including construction, transportation, and temporary support structures on water. Furthermore, the use of modular pontoons simplifies logistics, reduces construction times, and provides adaptability in scenarios requiring temporary or mobile infrastructure.



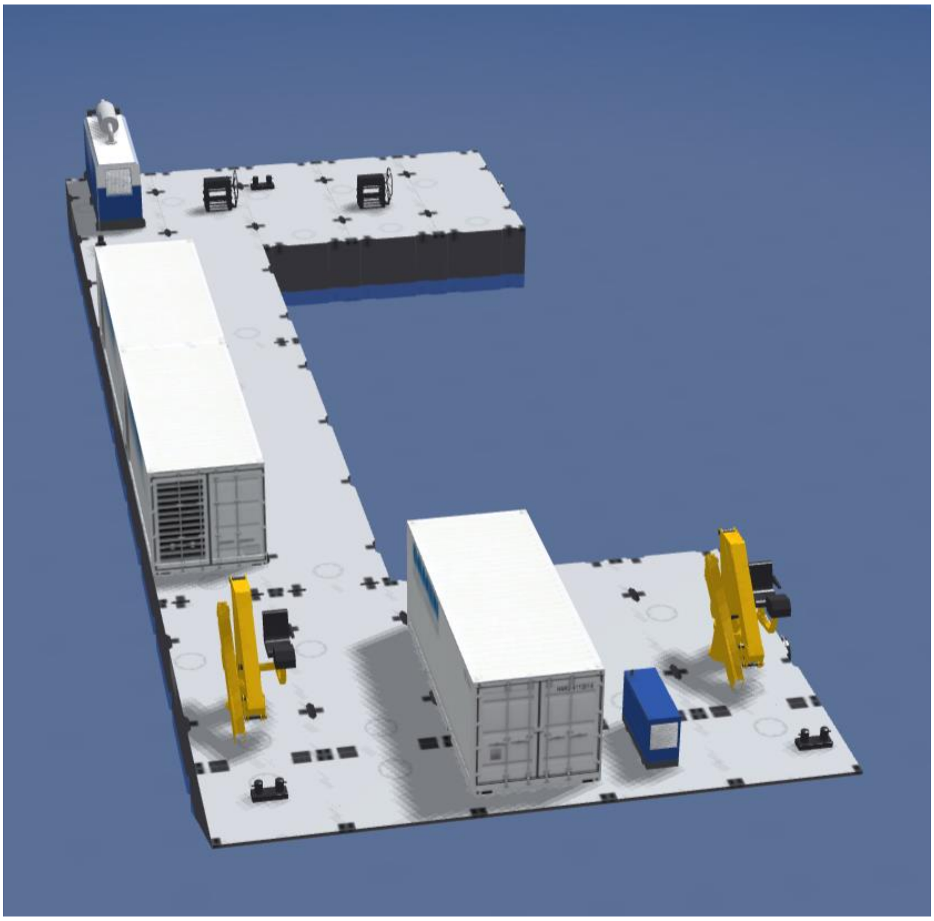
Picture 1: An example of a construction assembled based on pontoons. [1]



Picture 2: An example of a construction assembled based on pontoons. [1]



Picture 3: An example of a construction assembled based on pontoons.[1]



Picture: 4: An example of a construction assembled based on pontoons. [1]

Role of the Conductor. The conductor plays a vital role in the welding and assembly of pontoon elements by providing precise alignment and support during the welding process. Its primary function is to maintain the correct spatial positioning of individual components, ensuring that the geometrical characteristics meet the required specifications. This is crucial for achieving the precise alignment needed for modular pontoon structures, where each component must fit together seamlessly during final assembly.

The design of the conductor directly influences the accuracy and quality of the welding process. Factors such as the cross-sectional dimensions and thickness of the conductor elements contribute to minimizing deformations during welding, thus ensuring that the overall tolerances are met. A well-designed conductor ensures that each welded element remains within the specified tolerance limits, minimizing the risk of structural misalignment. By controlling the positioning of pontoon elements, the conductor helps maintain consistent weld quality, ultimately contributing to the overall integrity and strength of the floating structure.

Research Objective. The primary objective of this research is to determine whether the conductor can effectively secure the pontoon elements in their specified positions, with an allowable tolerance of 0.5 mm, after welding them to the main structure of the pontoon. This involves analyzing the impact of thermal loads during welding and assessing whether the deformation of the components remains within acceptable limits.

Research Tasks. To achieve the research objective, the following tasks have been identified:

- Determine the magnitude of thermal loads generated during the welding process.
- Investigate the stress-strain state of the pontoon components under the influence of thermal loads during welding.
- Measure the deformation of the components and verify whether it exceeds the specified tolerance of 0.5 mm.
- Make conclusions based on the findings.

Research Object. The object of this study is the conductor – a metallic structure designed to hold and stabilize pontoon elements during the welding process. The conductor's primary role is to ensure that the pontoon components are positioned accurately relative to one another while being welded to the main structure of the pontoon.

Cha.	Page	№ docum.	Signature	Data
Student		Grechyn V.Y.		
Consultant		Korostylyov L.I.		
Supervisor		Korostylyov L.I.		
Head of Dep.		Korostylyov L.I.		

19.ДРМ.135.6116м.ПЗ.03

Methodology for thermal and stress-strain analysis

Liter.	Page	Pages

NUS

1. METHODOLOGY FOR THERMAL AND STRESS-STRAIN ANALYSIS

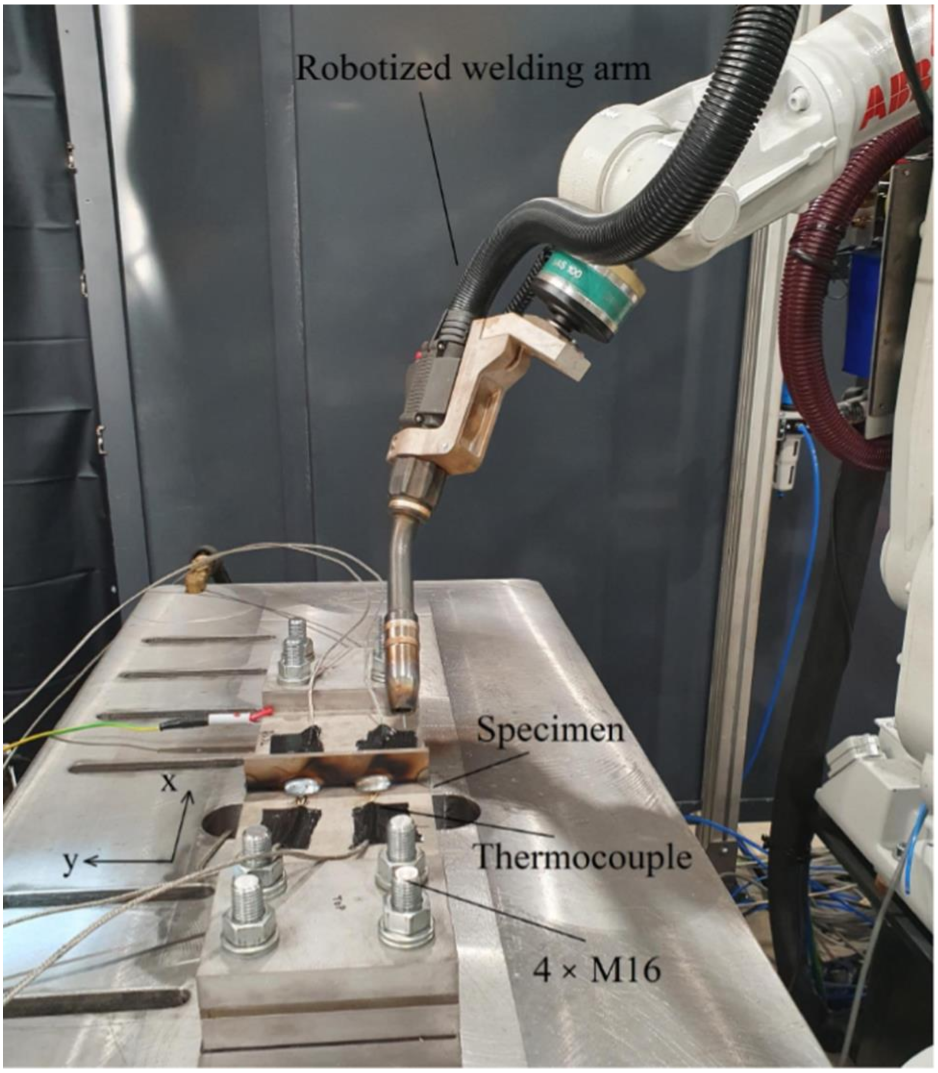
1.1 Overview of Thermal and Stress-Strain Analysis Techniques

Thermal and stress-strain analysis techniques are important for understanding the behavior of materials during welding, especially when working with complex structures like modular pontoons. These analyses help predict how materials will respond to thermal loads generated during welding and how these responses affect the overall structural integrity of the welded assembly. This section will provide an overview of commonly used methods for analyzing both thermal and stress-strain behaviors, including analytical approaches, numerical simulations using the Finite Element Method (FEM), and experimental validation techniques.

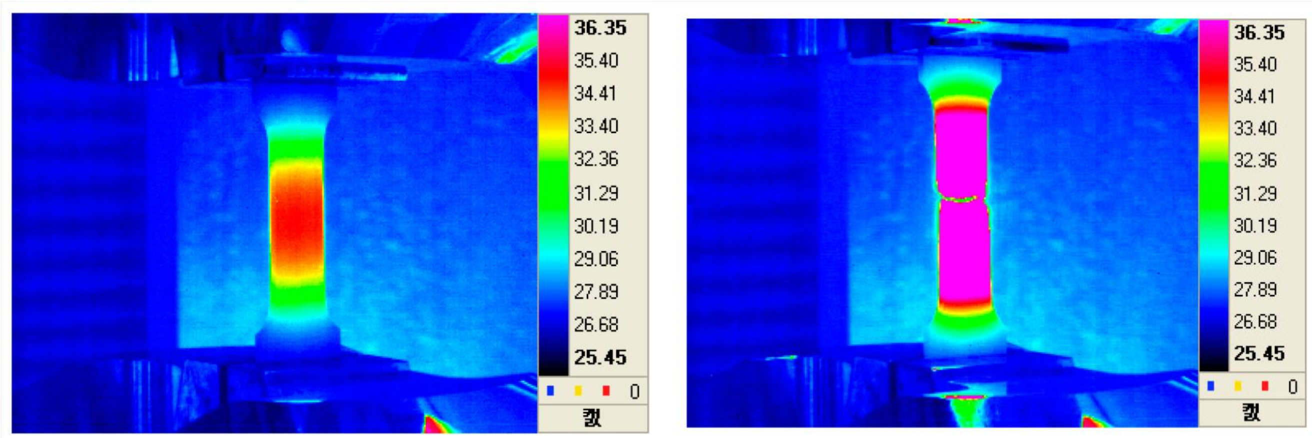
Thermal analysis techniques are used to determine the distribution of heat throughout a structure during welding. The heat generated during welding can significantly affect material properties and, the overall performance of the structure. One of the most fundamental methods of thermal analysis is the analytical solution of the heat conduction equation, such as Rosenthal's equation. This equation provides a mathematical description of how heat propagates through a material from a point or line heat source. It helps estimate temperature fields around the weld pool. Analytical solutions, such as those described by Rosenthal, are computationally efficient, but they rely on several assumptions, like uniform material properties and simple geometries, which limit their applicability for more complex scenarios [2].

The Finite Element Method (FEM) analysis breaks down the complex geometry of a structure into smaller finite elements, making it possible to model heat flow through different regions of the structure with varying material properties and boundary conditions. This approach can predict temperature distributions and heat-affected zones (HAZ) but requires significant computational resources and accurate input data. FEM is particularly useful for evaluating how different welding parameters such as welding speed, heat input, and cooling rates impact the thermal behavior of the material [3].

Experimental methods such as thermocouples and infrared thermography are also used to monitor temperature changes during welding. Such methods are valuable for validating numerical models, ensuring that the simulations accurately represent real-world welding conditions. Thermocouples are used to directly measure temperature at specific locations, while infrared thermography can provide a temperature map of the entire weld area. Experimental validation is important in complex welding applications, where accurate temperature control is needed to avoid excessive deformation and ensure the final product meets design specifications.



Picture 5: Example of Thermocouples. [4]



Picture 6: Example of Thermography. [5]

Stress-Strain Analysis Techniques. Stress-strain analysis is critical to evaluate how materials respond to thermal loading during welding, particularly in terms of mechanical deformation and residual stress. Welding generates significant thermal expansion and contraction, which can induce stresses and cause permanent deformation of the welded structure. The Finite Element Method (FEM) is again the primary tool used for stress-strain analysis. By coupling thermal and mechanical analyses, FEM can predict the development of residual stresses and plastic deformation. This coupled analysis provides a comprehensive understanding of how temperature changes during welding affect the mechanical properties of the material and the overall stability of the structure [3].

Coupled thermal-mechanical analysis is effective for welding applications, where the temperature field obtained from the thermal analysis is used as an input for the mechanical analysis. This type of analysis allows for accurate prediction of residual stress and strain distributions within the material. However, this method requires significant computational resources and detailed knowledge of the material properties at different temperatures, which can be challenging to obtain.

Summary of Techniques. The choice of analysis technique depends on the complexity of the structure and the specific requirements of the study. Analytical methods provide a quick and cost-effective means to estimate thermal and mechanical behaviors but are limited by their assumptions and simplifications. Numerical methods like FEM are far more detailed and can accurately predict temperature distributions, stress fields, and deformation patterns, but they require substantial computational resources and accurate input data. Finally, experimental validation is essential to ensure that numerical models accurately represent the real-world conditions and that the predictions made by the models are reliable.

In this study, a combination of FEM for both thermal and stress-strain analysis, will be used.

1.2 Finite Element Method (FEM) in Welding Analysis

The Finite Element Method (FEM) is a powerful numerical technique that plays a significant role for understanding the behavior of complex structures under the influence of thermal and mechanical loads. FEM analysis enables detailed modeling of the material response, including temperature distribution, stress development, and resulting deformation.

Overview of FEM in Welding. FEM is based on the subdivision of a complex structure into smaller, simpler elements, known as finite elements. By discretizing the structure, FEM allows for the calculation of field variables (such as temperature, stress, and strain) across the entire domain of the model. This method is advantageous in welding analysis as it can simulate the complex interactions between thermal and mechanical phenomena that occur during welding. These interactions include heat transfer, thermal expansion, plastic deformation, and the development of residual stresses.

One of the primarily uses of FEM in welding is thermal analysis. During welding, the localized heat from the welding arc generates a significant thermal gradient, leading to temperature variations across the structure. FEM is used to solve the heat conduction equation numerically (equation 1), providing detailed temperature distribution across different parts of the welded assembly.

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)+q_V=\rho c_p\frac{\partial T}{\partial t}$$

Equation 1: Heat conduction equation

Where: k – is the materials conductivity $\left[\frac{W}{mK}\right]$

q_V *– is the rate at which energy is generated per volume of the medium* $\left[\frac{W}{m^3}\right]$

ρ *– is the density* $[kgm^3]$

c_p *– is the specific heat capacity* $\left[\frac{J}{kgK}\right]$

This temperature distribution is crucial for predicting the size of the heat-affected zone (HAZ) and the changes in material properties resulting from temperature exposure [3].

After determining the temperature profile, the next step involves mechanical analysis. In this phase, the temperature data obtained from thermal analysis is used to evaluate the resulting mechanical effects, such as stress and strain. The temperature gradients induce thermal expansion and contraction, causing residual stresses and potential deformations in the structure. FEM can predict these stresses and deformations, helping engineers understand how welding parameters influence the mechanical integrity of the welded joint [6].

Thermal-Mechanical Coupling in FEM. In these coupled simulations, thermal analysis is directly linked to mechanical analysis, allowing for a comprehensive understanding of how temperature changes affect the stress-strain state of the material. By accounting for these coupled effects, FEM can provide a more accurate prediction of residual stresses and deformations compared to standalone thermal or mechanical analysis. Coupled thermal-mechanical FEM analysis is used to identify critical stress regions within the welded structure. However, coupled analysis is computationally intensive, requiring significant resources and accurate input data, including material properties at elevated temperatures. [7].

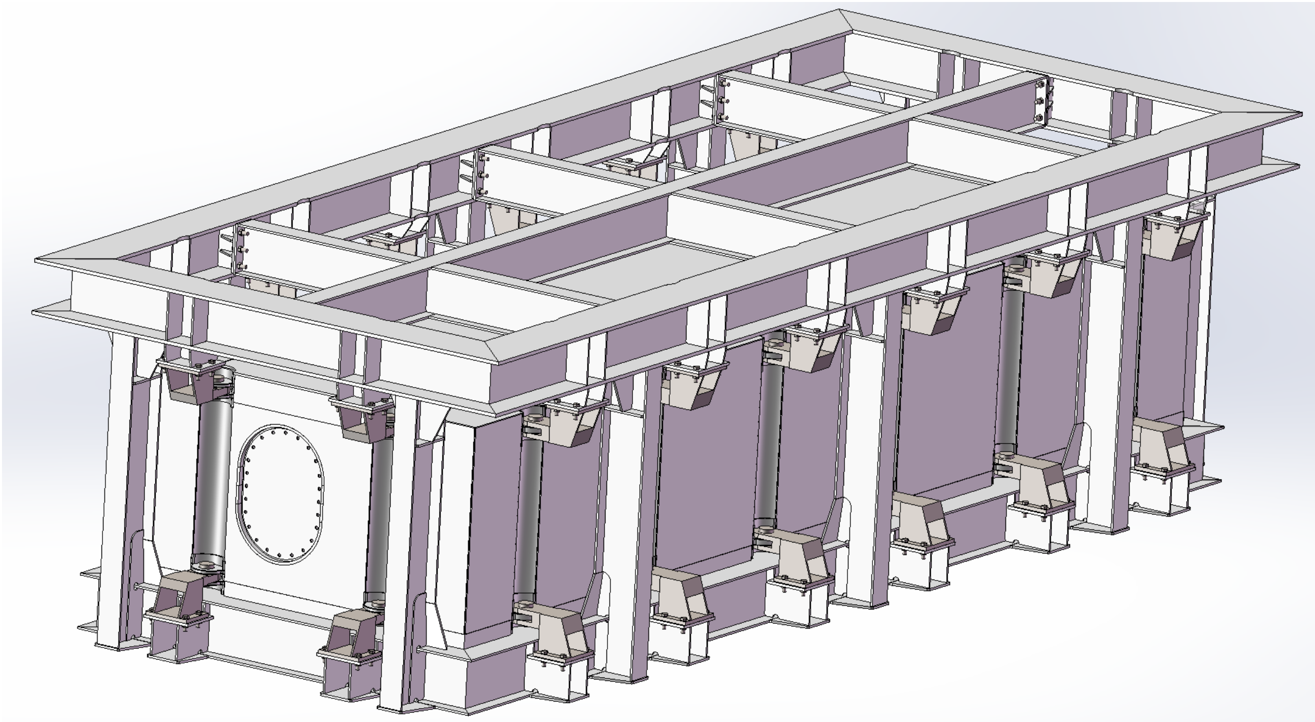
Application of FEM in the Current Study. In the context of this study, FEM is used to simulate both the thermal and mechanical aspects of welding to evaluate the effectiveness of the conductor in maintaining the proper alignment of pontoon elements.

Due to limited computational resources, the welding process was represented by applying a specific heat flow along the entire length of all weld seams simultaneously. This approach simplifies the modeling process while still providing insight into the overall thermal behavior of the structure.

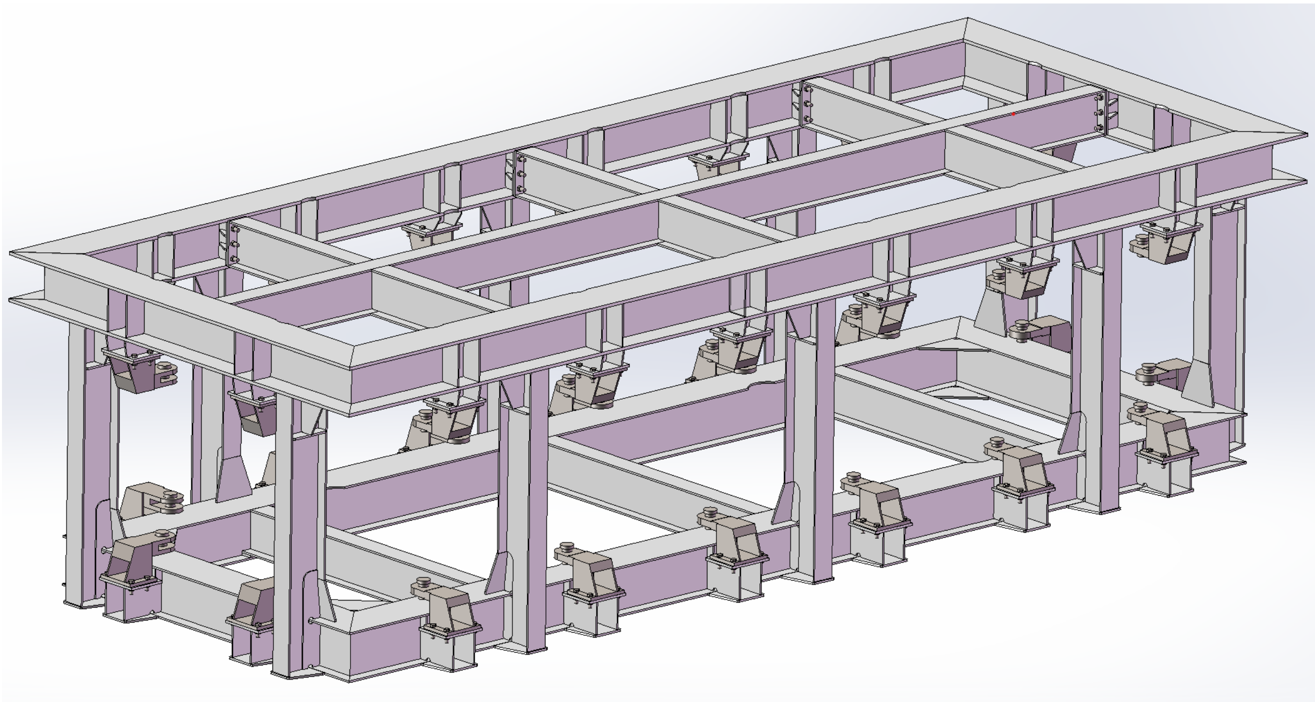
Benefits and Limitations of FEM in Welding Analysis. The use of FEM in welding analysis provides numerous benefits, including the ability to model complex geometries and simulate the interactions between thermal and mechanical phenomena. FEM allows for detailed predictions of temperature fields, residual stresses, and deformation, which are essential for assessing the quality of welded structures. FEM simulations can be computationally expensive, particularly for large and complex models that require fine mesh resolution for accurate results.

1.3 Model Development

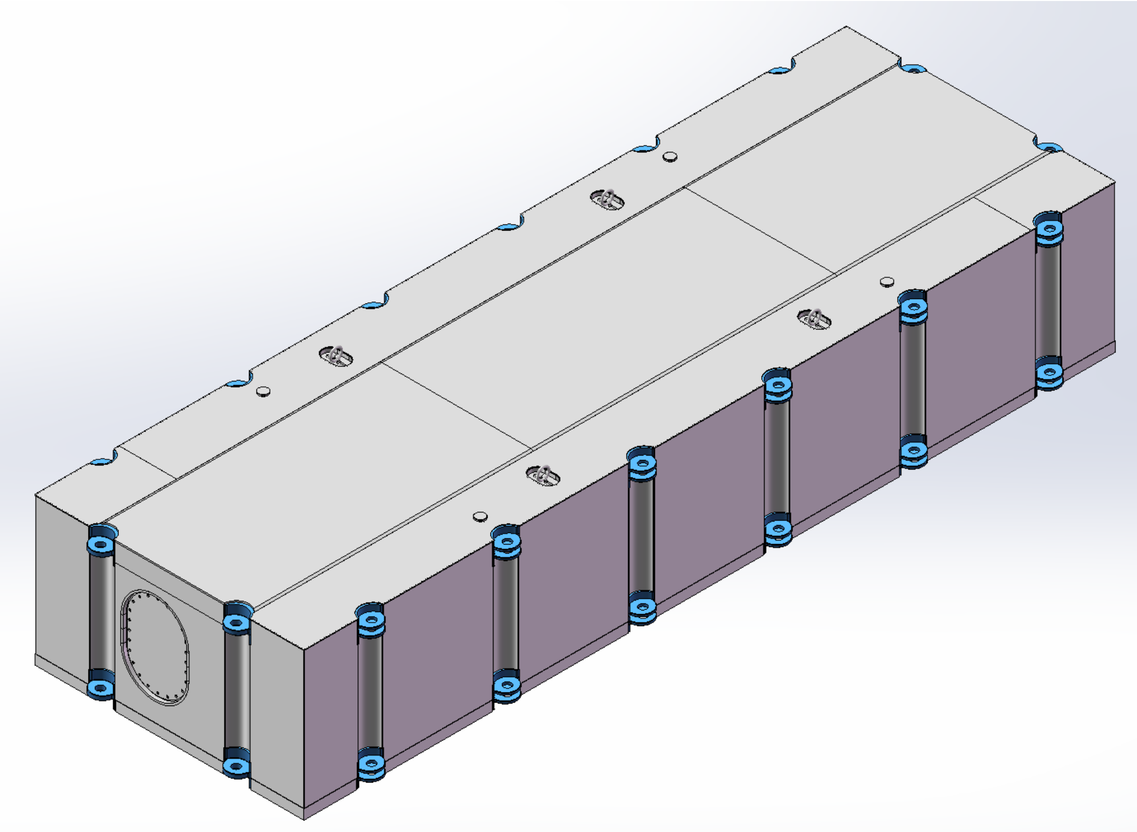
The development of the conductor and pontoon models was based on a real project designed by the author.



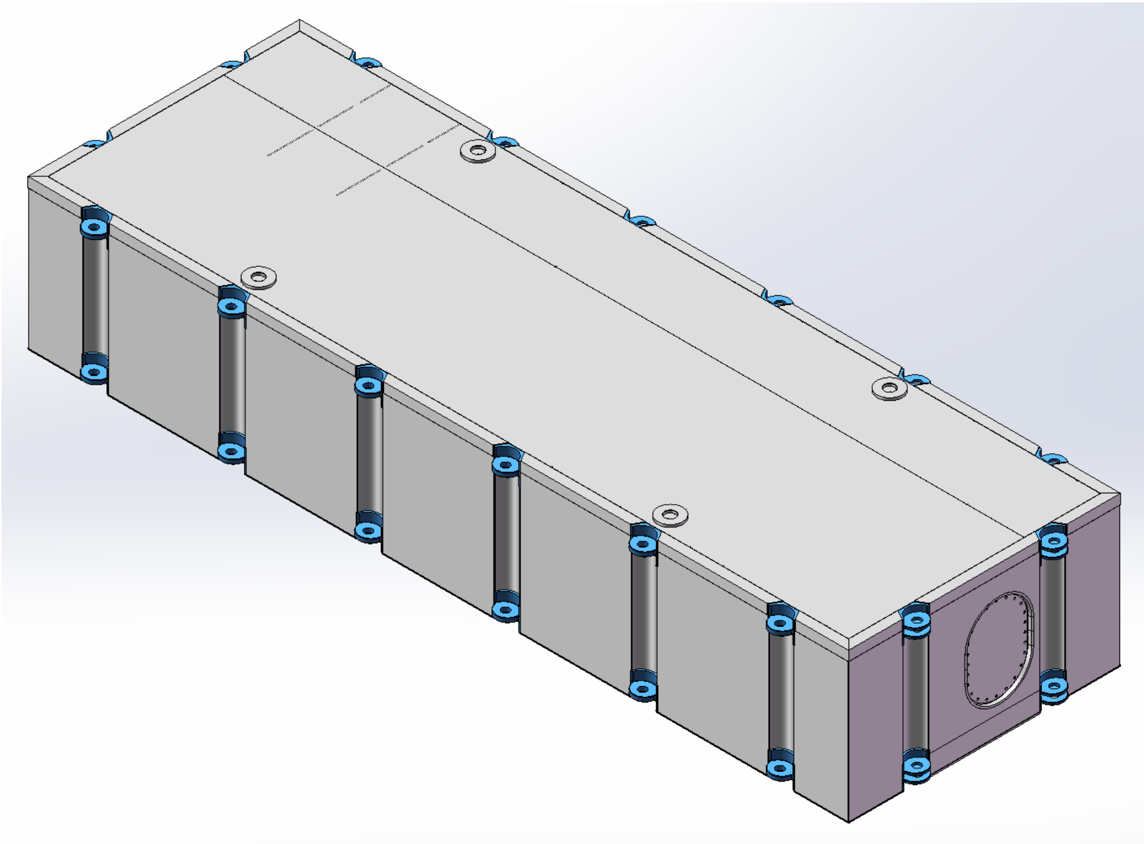
Picture 7: Real project Conductor with Pontoon.



Picture 8: Real project Conductor.



Picture 9: Real project Pontoon with seledted elements that should be welded.



Picture 10: Real project Pontoon with seledted elements that should be welded.

Construction of conductor:

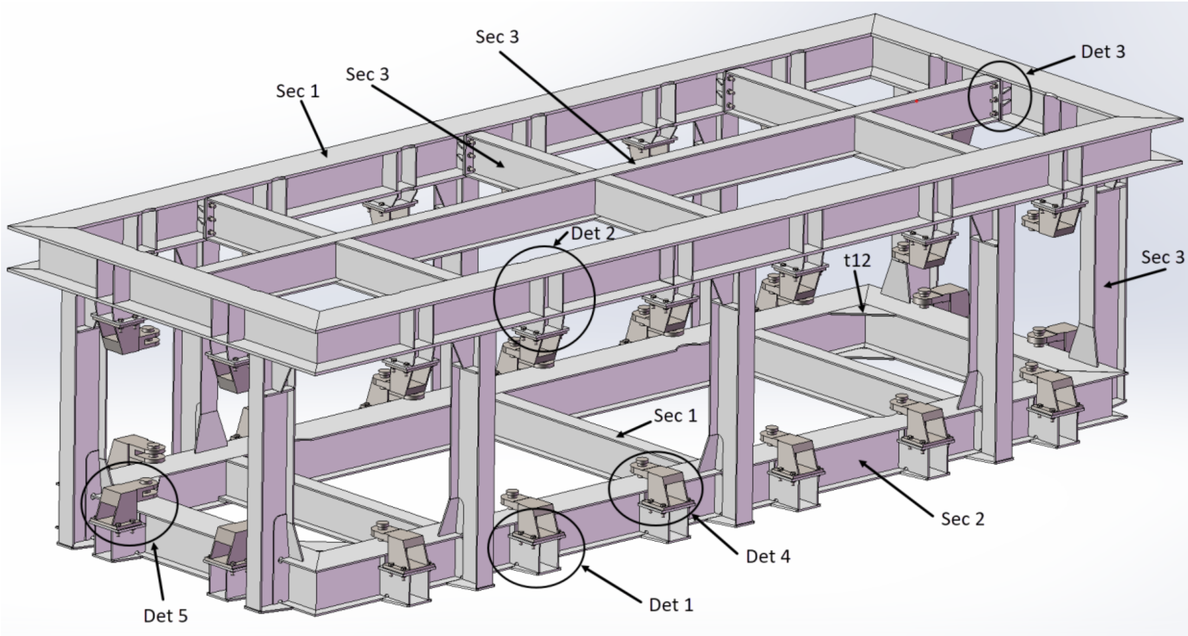
All elements are made from structural steel (See Picture 11).

Structural Steel

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

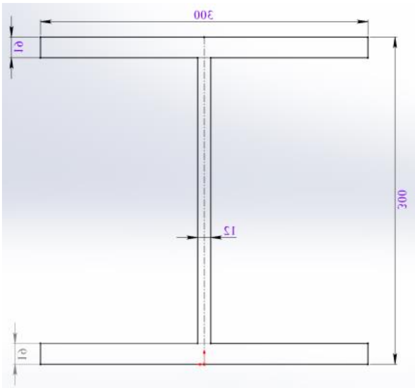
Density	7,85e-06 kg/mm³
Structural	
Isotropic Elasticity	
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	2e+05 MPa
Poisson's Ratio	0,30000
Bulk Modulus	1,6667e+05 MPa
Shear Modulus	76923 MPa
Isotropic Secant Coefficient of Thermal Expansion	1,2e-05 1/°C
Compressive Ultimate Strength	0 MPa
Compressive Yield Strength	250,00 MPa
Strain-Life Parameters	-6.6e-1, -5.4e+0, 0.0e+0, 1.0e+1
S-N Curve	3.6e+0, MPa log(10), 1.9e+0, 1.0e+0, log(10), 6.0e+0
Tensile Ultimate Strength	460,00 MPa
Tensile Yield Strength	250,00 MPa
Thermal	
Isotropic Thermal Conductivity	0,060500 W/mm·°C
Specific Heat Constant Pressure	4,34e+05 mJ/kg·°C

Picture 11: Properties of structural steel

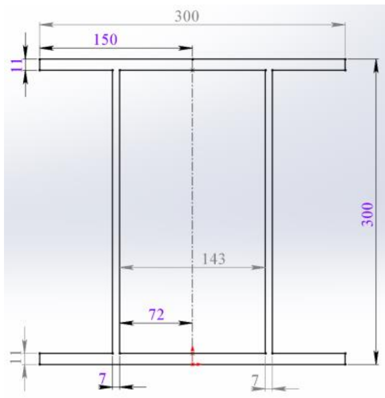


Picture 12: Conductor.

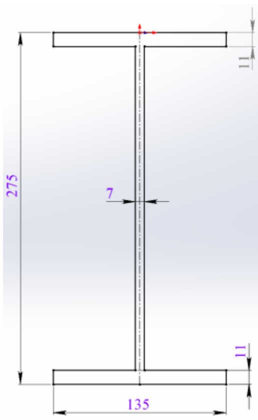
Setion dimentions:



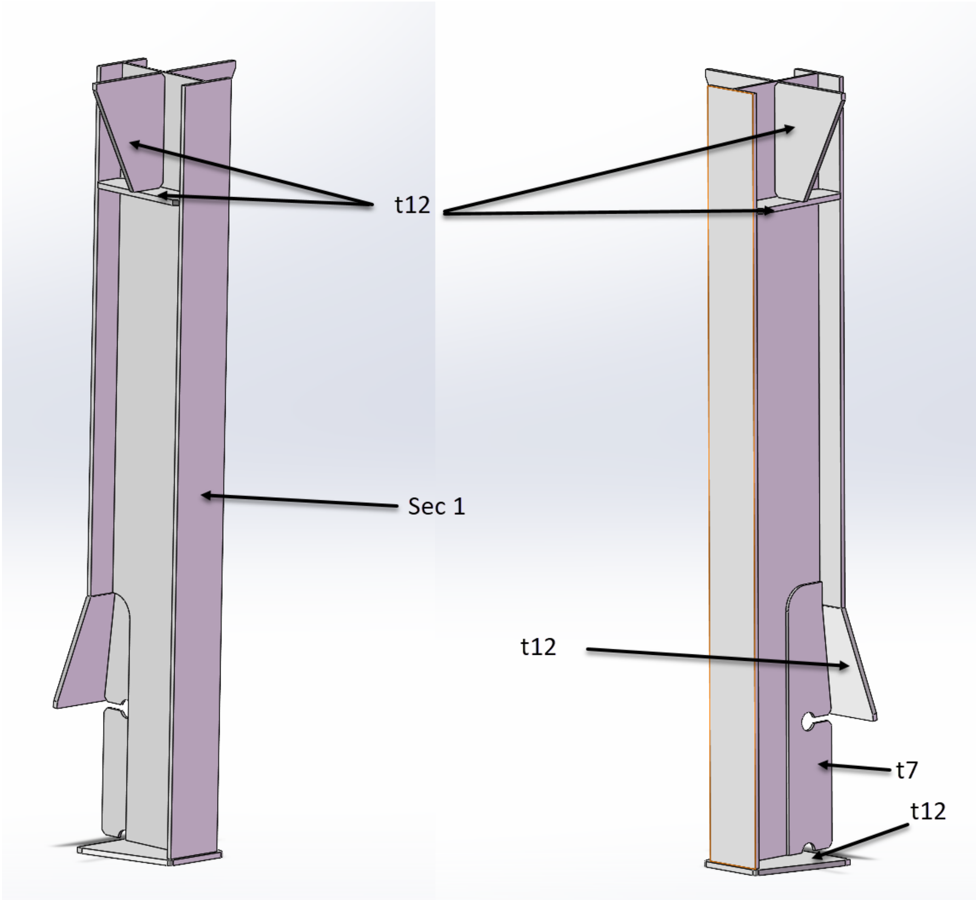
Picture 13: Section 1 dimensions.



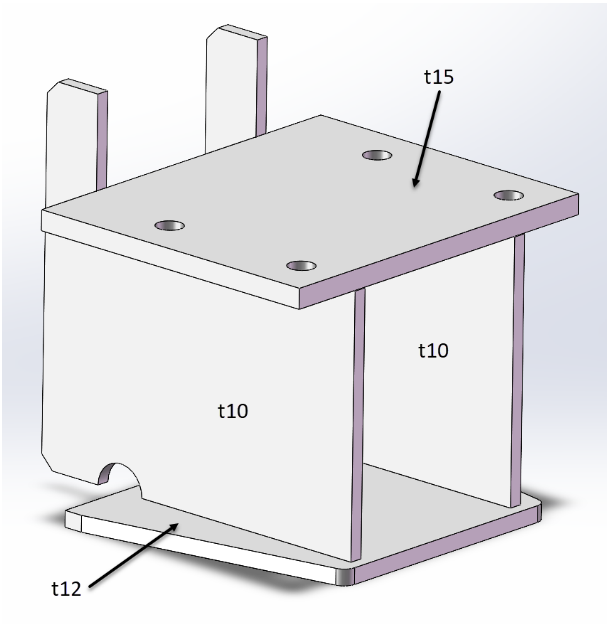
Picture 14: Section 2 dimensions.



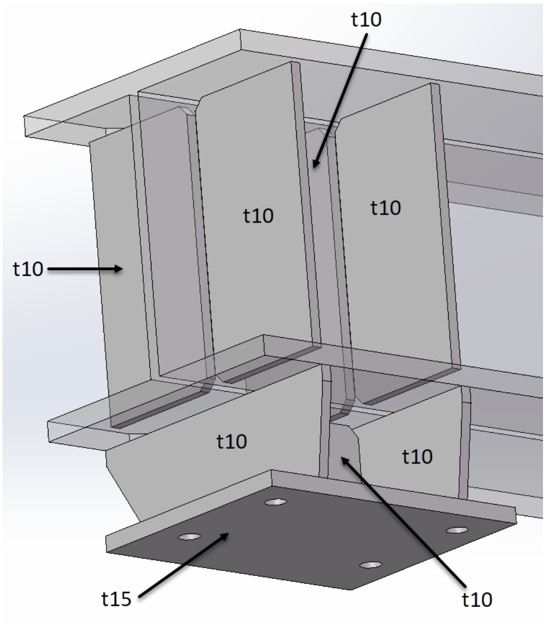
Picture 15: Section 3 dimensions.



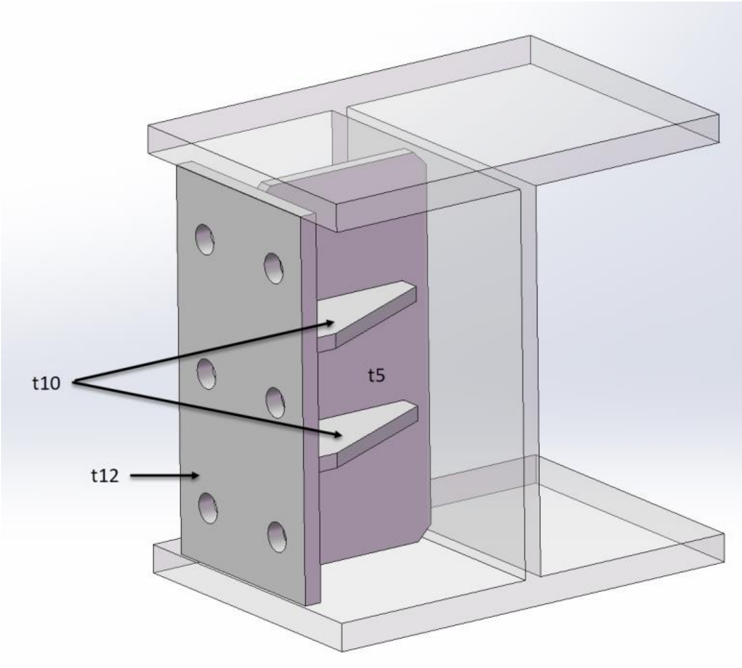
Picture 16: Constructions dimensions.



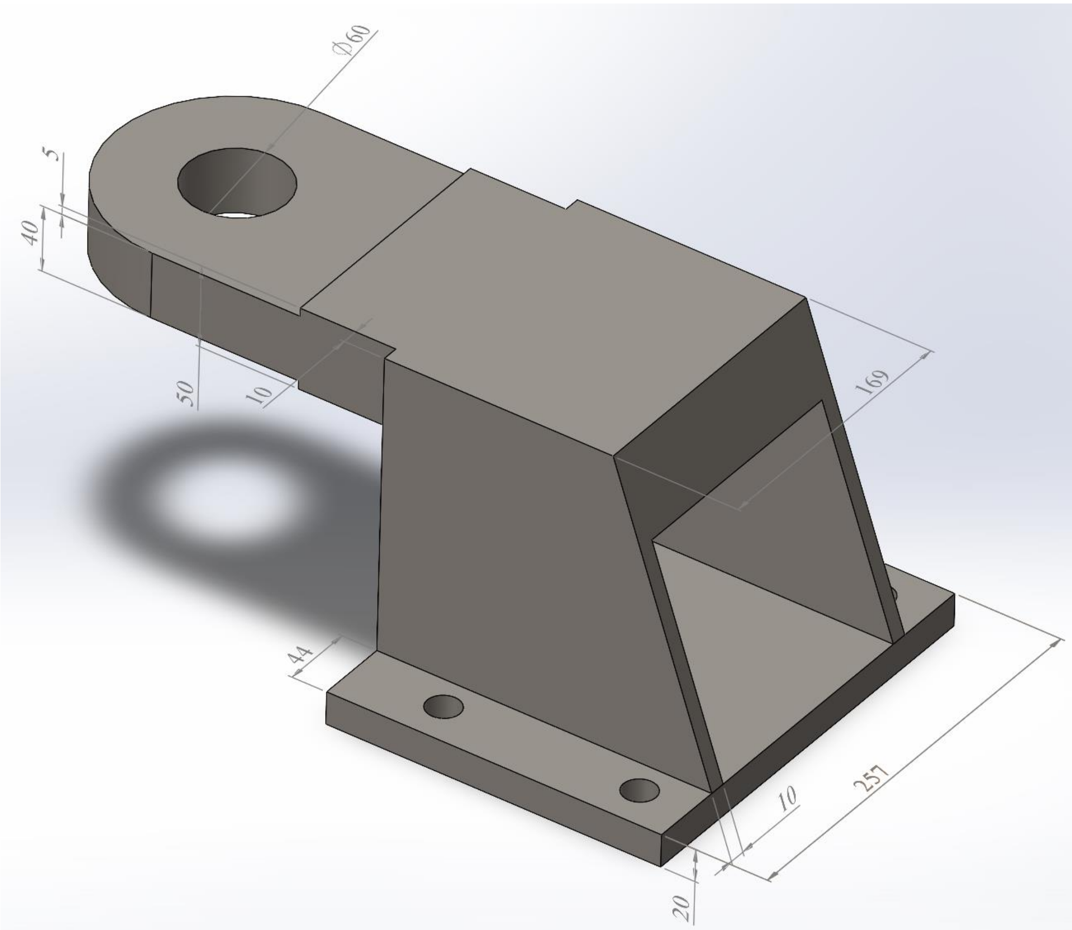
Picture 17: Det. 1



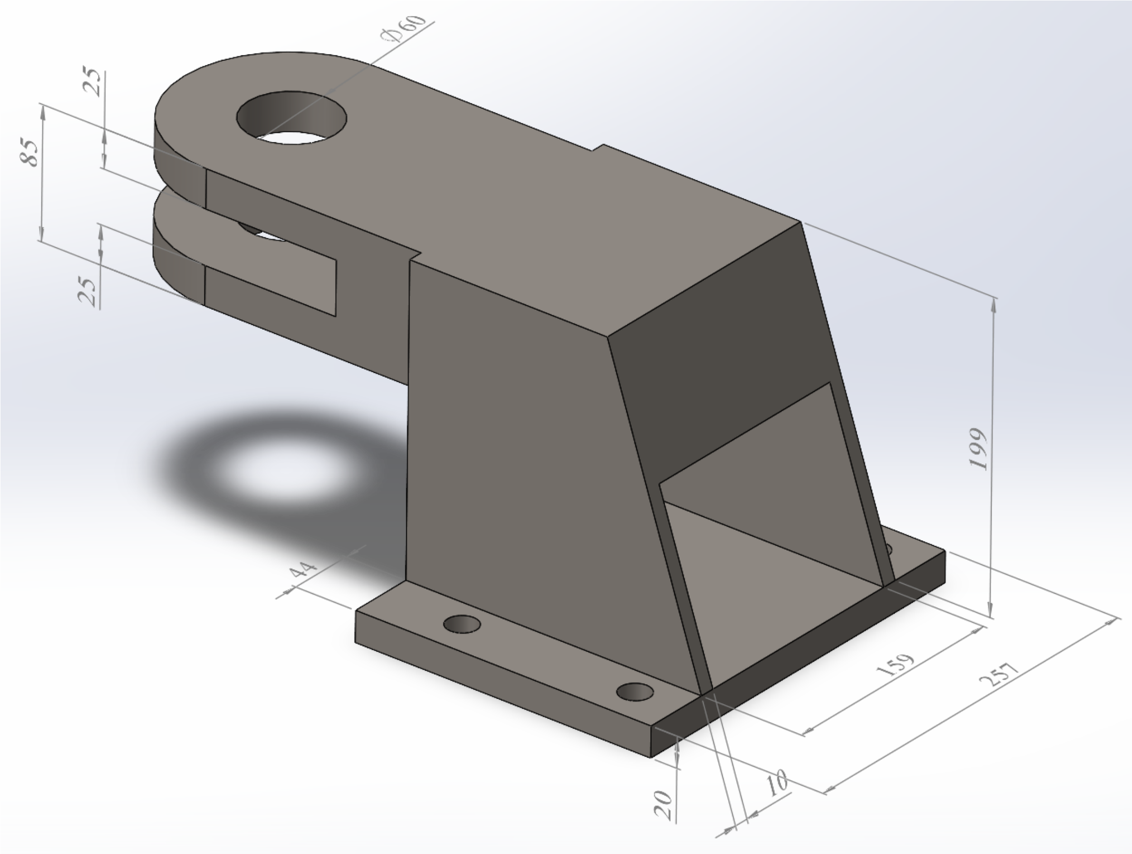
Picture 18: Det. 2



Picture 19: Det. 3



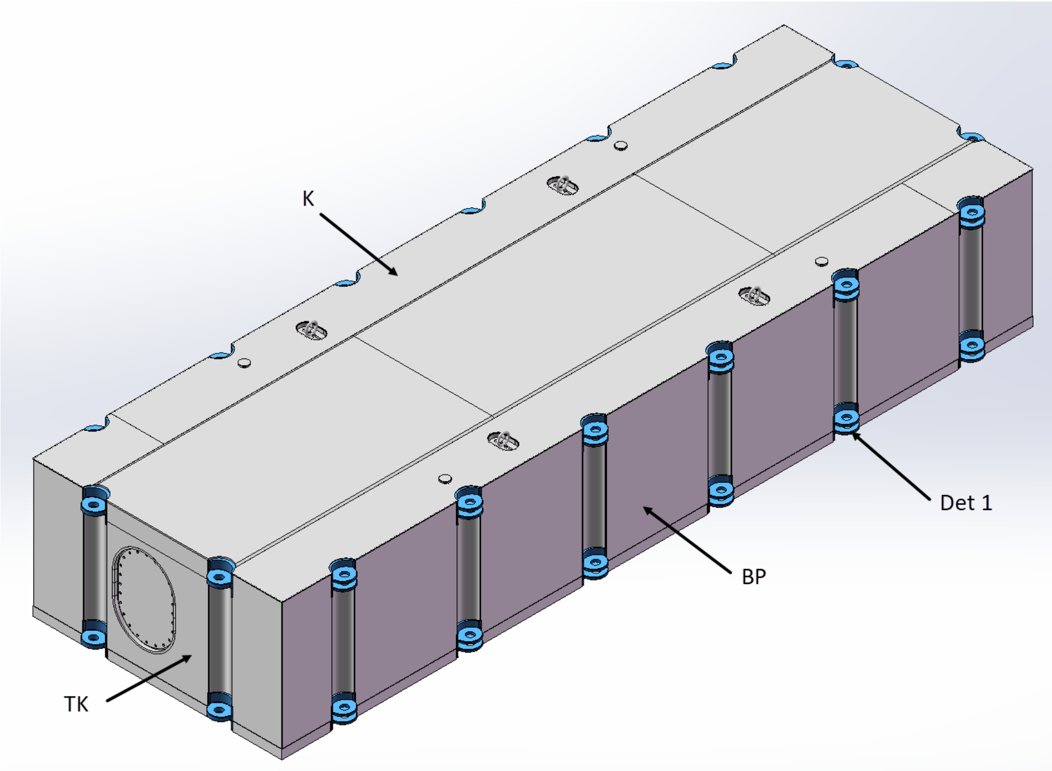
Picture 20: Det. 4



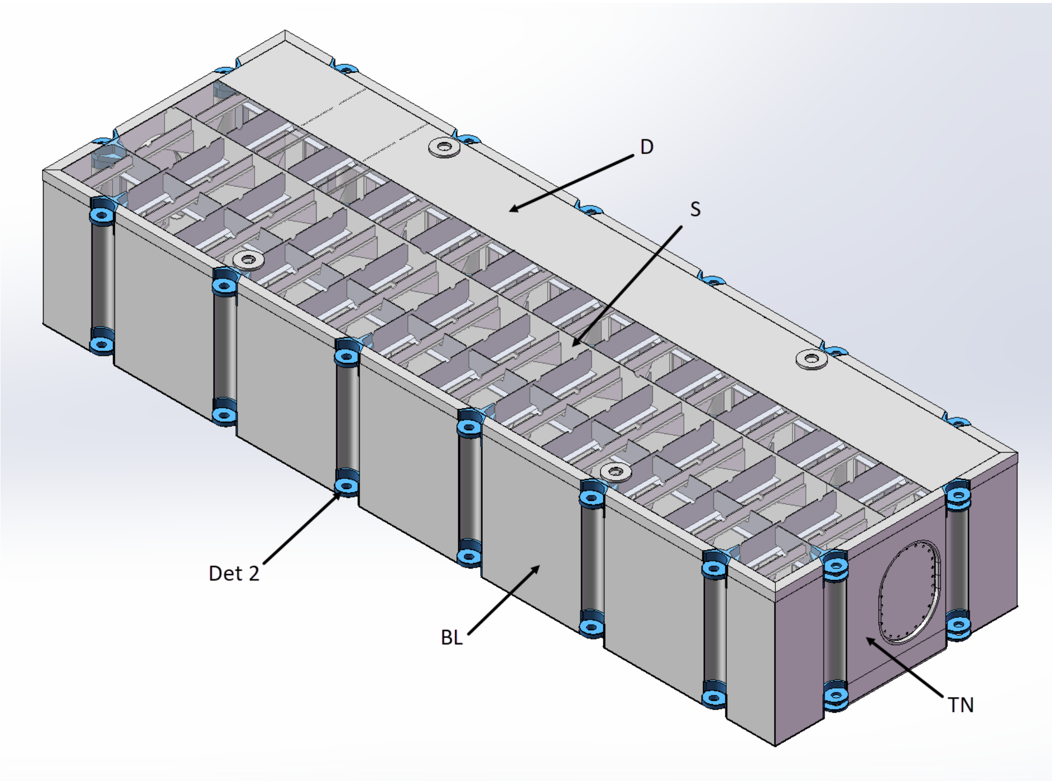
Picture 21: Det. 5

Construction of conductor:

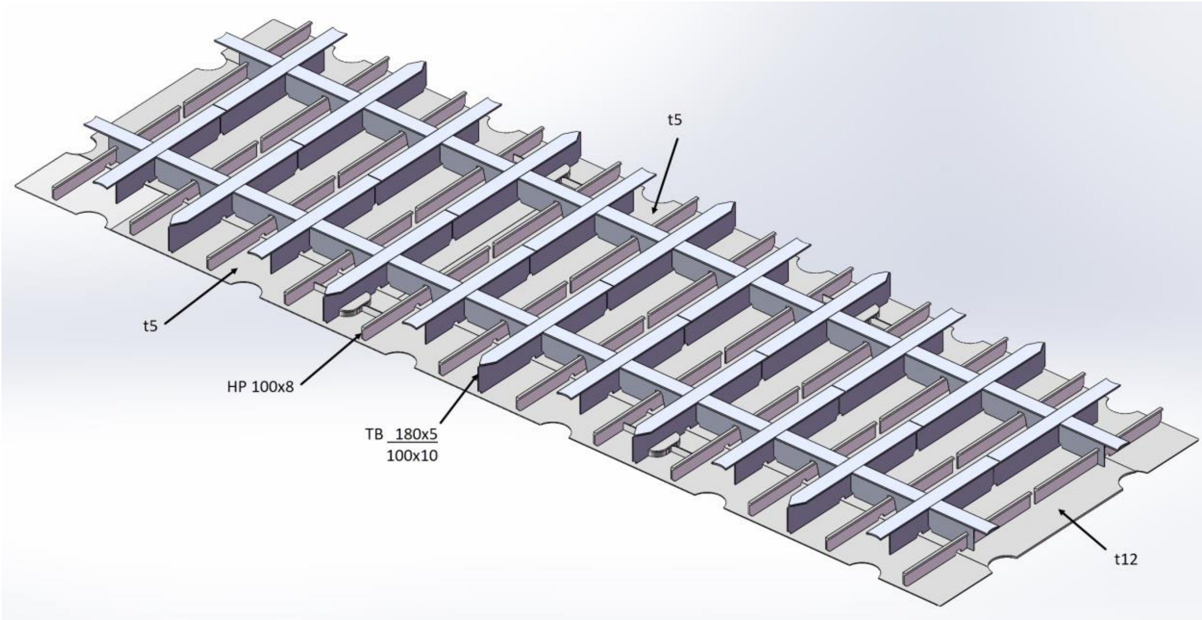
All elements are made from structural steel (sea Picture 11).



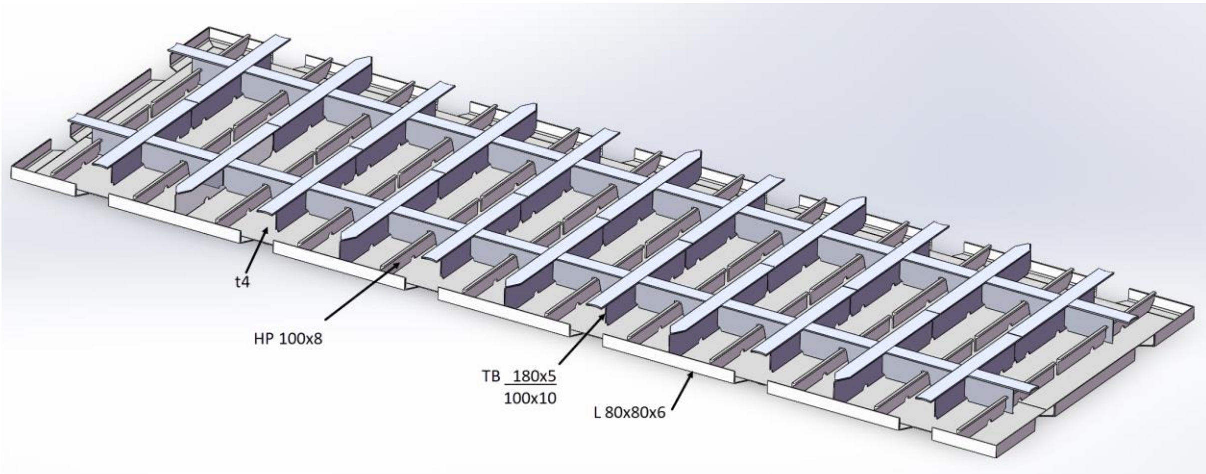
Picture 22: Pontone



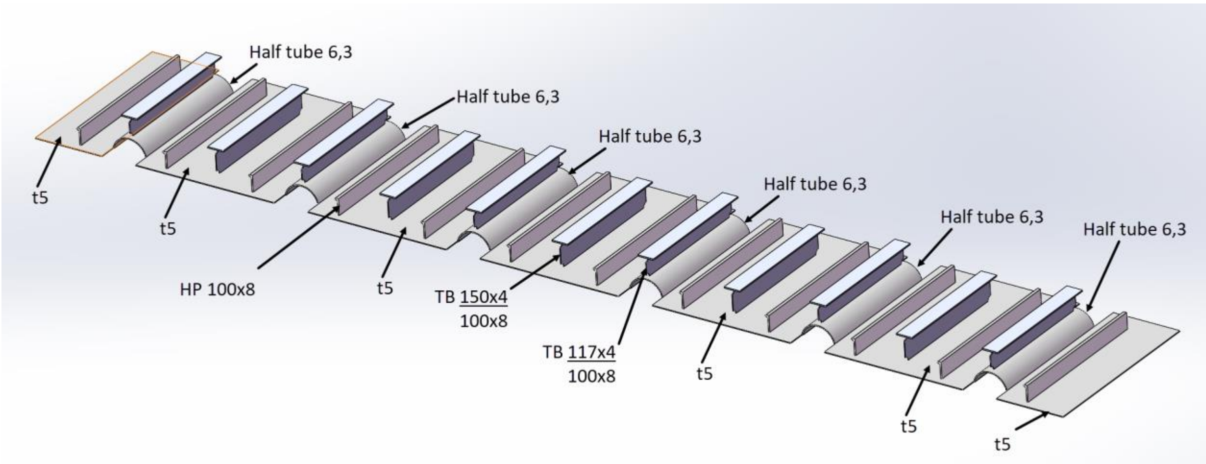
Picture 23:



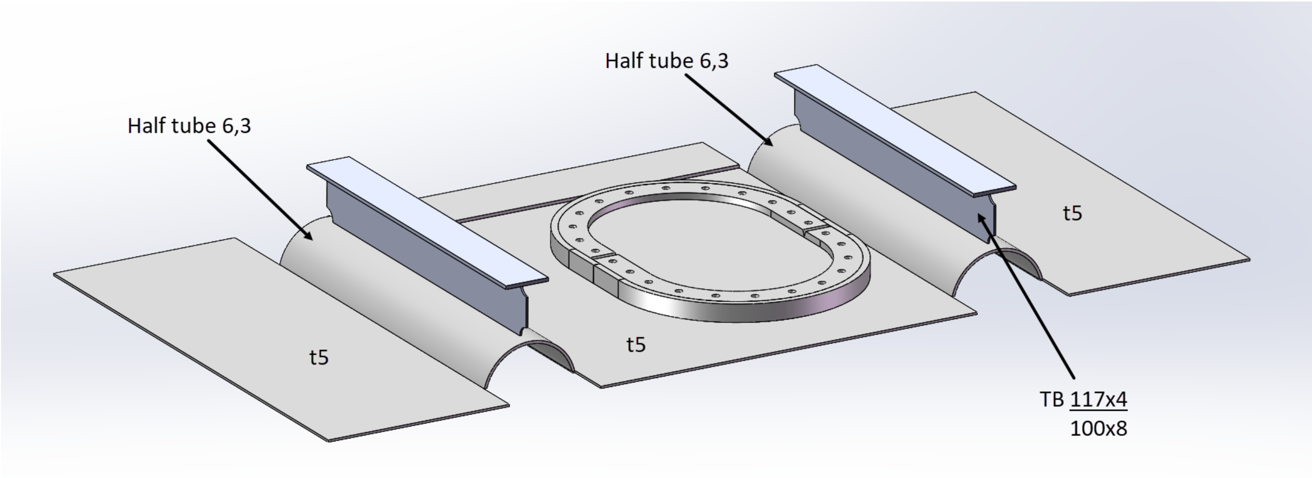
Picture 24: K



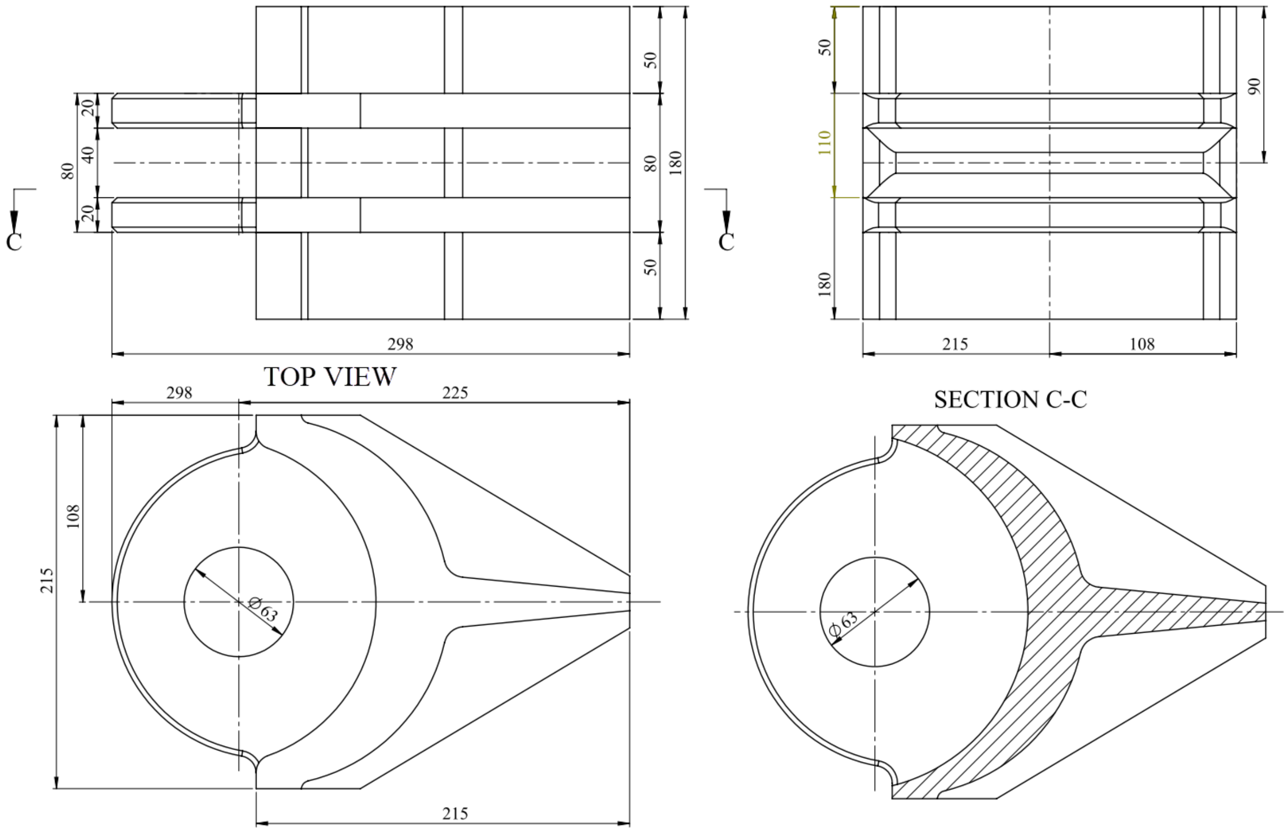
Picture 25: D



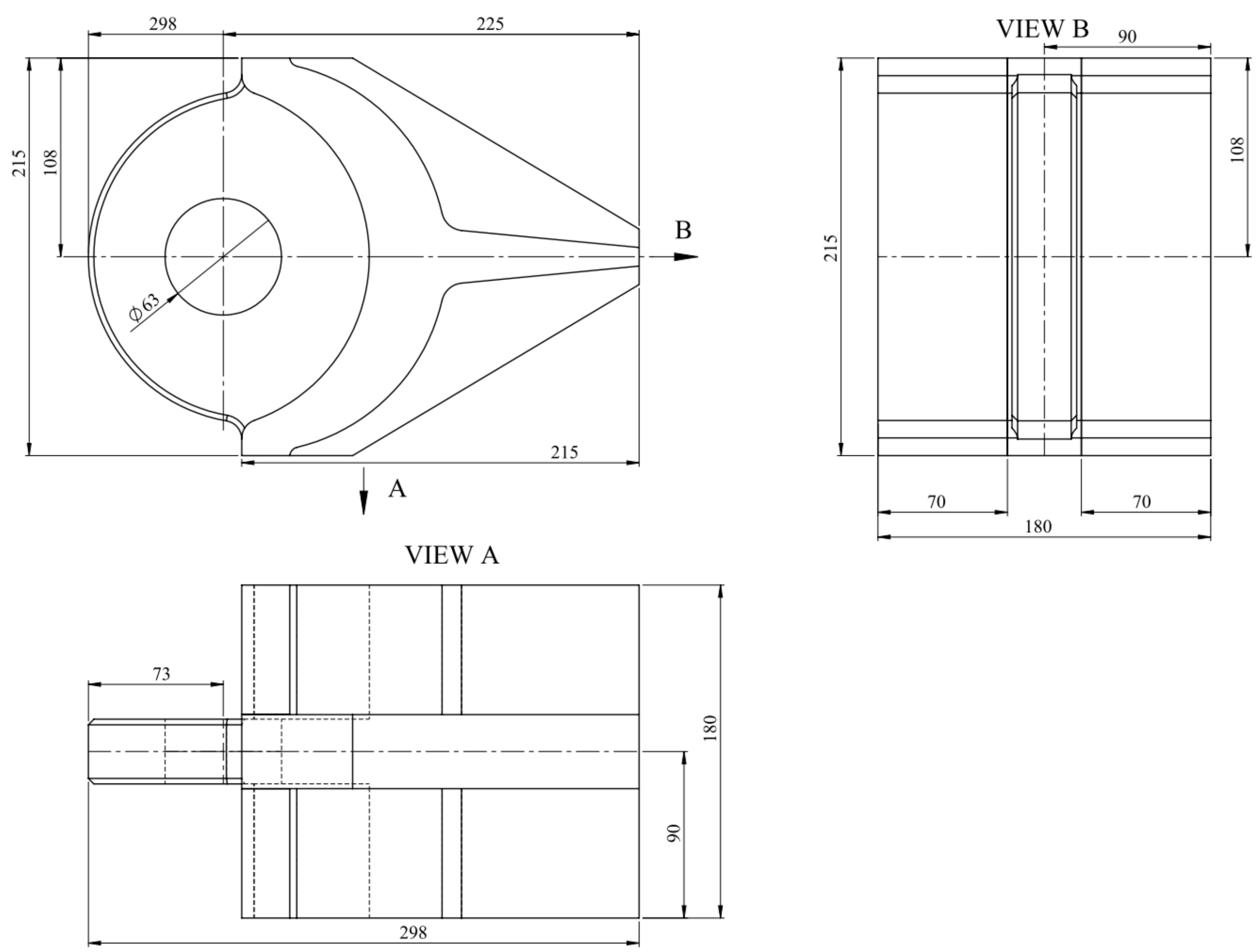
Picture 26: BP, Apply to BL



Picture 27: TK, Apply to TN



Picture 28: Det 1



Picture 29: Det 2

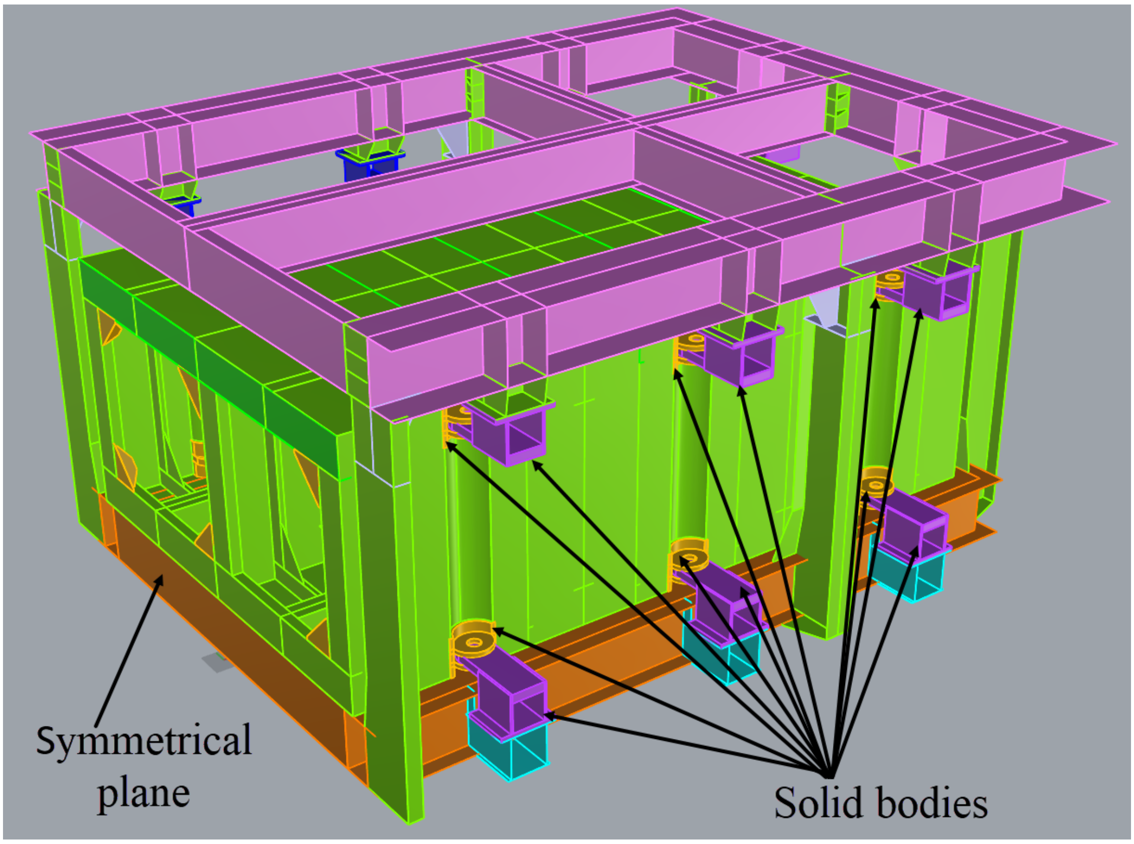
The initial modeling was carried out in **SolidWorks**. Due to limited computational resources, the model was simplified by removing unnecessary features and components that were not significant for the analysis. This included eliminating small cuts, fillets, and other details that had minimal impact on the stress-strain behavior during welding. The simplification was done to ensure that the model remained computationally manageable while still accurately representing the structure's response to thermal and mechanical loads.

The simplification process continued with further modifications in **Rhino**, where the model was adjusted to reduce computational demands even more. Since the structure was nearly symmetrical relative to the XY plane, only half of the model was retained. This symmetry assumption allowed for a significant reduction in the number of elements needed for finite element analysis, effectively reducing the computational cost. Furthermore, several components originally designed as angles and Holland profiles were replaced with sheet metal representations, making it easier to model and analyze the structure.

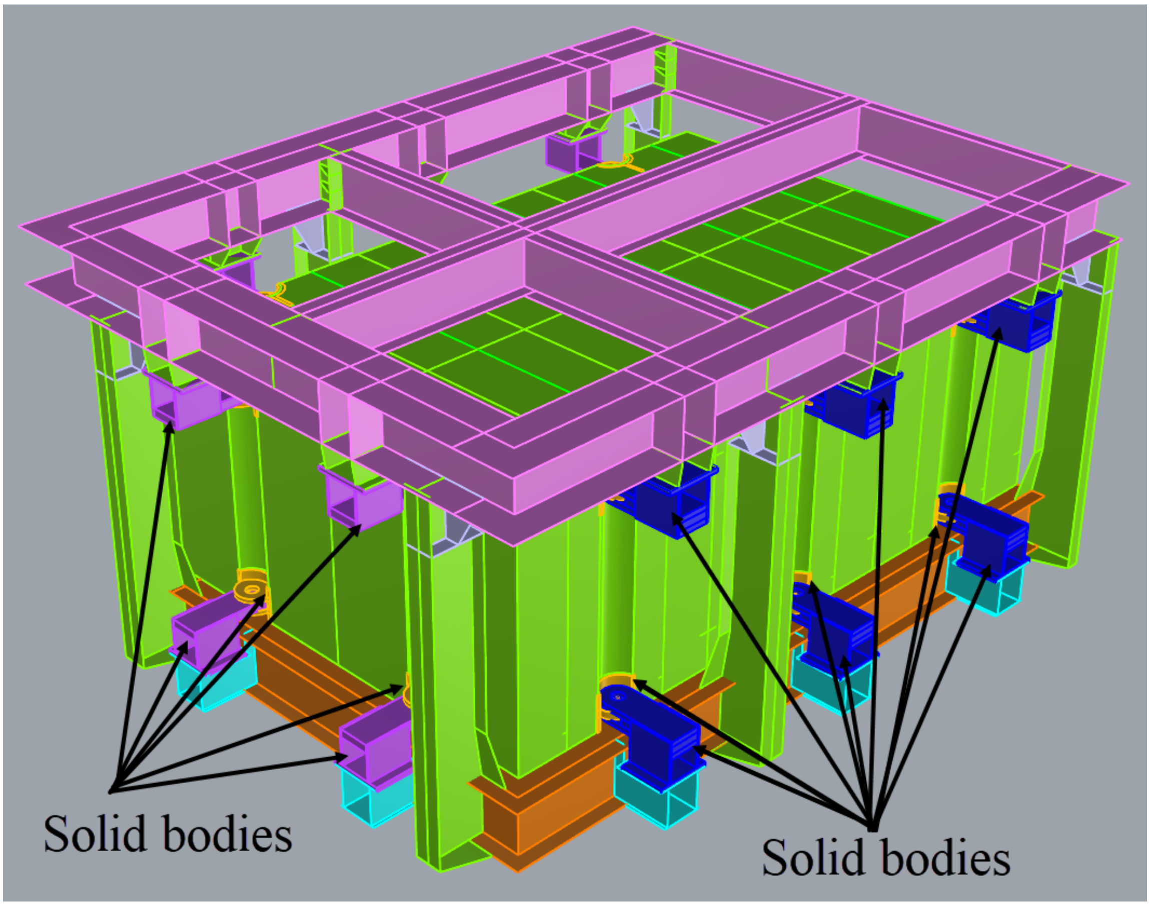
The sheet metal components were further simplified by replacing them with shell components. This step was crucial for reducing the mesh complexity and ensuring that the analysis could be completed within the available computational resources. Only components with complex geometries were left as solid bodies, as their intricate shapes had a considerable effect on the overall stress-strain distribution. These remaining components were also simplified and divided into more manageable sub-elements to facilitate meshing and analysis.

The approach of reducing the model complexity by simplifying components and retaining only essential geometries is a common practice in finite element analysis, especially when working with limited computational resources. It allows for a balance between maintaining accuracy and ensuring that simulations can be completed in a reasonable timeframe. By focusing on the critical load-bearing components and removing non-essential details, the model retains its ability to provide accurate insights into the thermal and mechanical behavior of the conductor and pontoon during welding.

The modifications in Rhino ensured that the model was optimized for thermal and stress-strain analysis while keeping the computational cost low. The use of half-model symmetry and simplification of sheet metal components into shell components significantly reduced the number of elements in the finite element mesh, leading to faster computation times. This approach was particularly important given the limited computational resources available for this study.



Picture 30: Geometry at the Rhino.



Picture 31: Geometry at the Rhino.

Application in the Current Study. In the current study, the simplified model is used to analyze the thermal and mechanical behavior of the conductor and pontoon elements during welding. The thermal analysis involves applying a uniform heat flux across the entire length of the weld seams, which represents the heat input during welding. This approach simplifies the welding process representation while providing valuable insights into the temperature distribution throughout the structure. The mechanical analysis follows the thermal analysis, using the temperature data to predict the stress-strain response and assess the resulting deformation.

The simplifications made to the model are intended to ensure that the key aspects of the conductor's performance such as maintaining alignment accuracy and minimizing deformation are accurately captured. The use of SolidWorks for initial modeling and Rhino for further simplification demonstrates the importance of using different CAD tools to achieve an optimal balance between model fidelity and computational efficiency.

1.4 Simulation Setup in ANSYS

The simulation setup in ANSYS for this study involves two primary stages: thermal analysis and structural analysis. These stages are crucial for evaluating the effects of welding on the conductor and pontoon components, considering both thermal and mechanical impacts. This section provides a detailed explanation of the theoretical foundations and practical setup for these simulations.

1.4.1 Thermal Analysis Setup

The first stage of the simulation is the thermal analysis, which calculates the temperature distribution across the conductor and pontoon structures during welding. In the case of arc welding, the energy input is usually defined as the arc energy input, which represents the amount of energy transferred per unit length of the weld. This can be expressed as the ratio of total input power (P) to the travel velocity (R) of the heat source:

$$H = \frac{P}{R}$$

Equation 2

Where: H: Heat input per unit length [J/m]

P: Total input power [W]

R: Travel velocity of the heat source [m/s]

If the heat source is an arc, then the heat input energy can also be described as:

$$H = \frac{EI}{R}$$

Equation 3

Where: H: Heat Input for Arc Welding [J/m]

E: is the voltage [V]

I: is the current [A].

R: Travel velocity of the heat source [m/s]

In this study, the welding heat is represented by applying a uniform heat flow along the entire length of the weld seams. This simplification was adopted to minimize computational requirements while still capturing the key effects of heat input on the structural components. The heat flow, represented by the total input power P [W], simulates the energy delivered by the welding arc and is uniformly distributed along the weld lines.

The heat conduction equation used in this stage is based on Fourier's Law of heat conduction:

$$q = -k\nabla T$$

Equation 4

Where: q: represents the heat flux,

k: is the thermal conductivity of the material $\left[\frac{W}{mK}\right]$,

∇T: is the temperature gradient.

This equation is crucial for understanding how heat propagates through the conductor and pontoon materials.

Additionally, convection is modeled to account for heat loss to the surrounding air, with an ambient temperature of 22°C. The convective heat transfer coefficient (Film coefficient) $= 1{,}1e^{-5}\left[\frac{W}{mm^2C}\right]$ is applied to all surfaces exposed to air to simulate the cooling effect.

The material properties used in the thermal analysis, such as thermal conductivity, specific heat, and density, are defined according to the actual materials used in the conductor and pontoon structures. This ensures that the temperature distribution calculated by ANSYS reflects realistic thermal behavior.

The output of the thermal analysis is a temperature distribution over time, which helps identify areas with high thermal gradients and potential heat-affected zones (HAZ). These zones are critical for predicting changes in material properties and assessing the impact of welding on the structural integrity of the conductor and pontoon elements. The heat flux simulates the energy delivered by the welding arc and is distributed evenly along the weld lines.

1.4.2 Structural Analysis Setup

Following the thermal analysis, the structural analysis is performed to evaluate the stress-strain state of the conductor and pontoon components under the influence of the thermal loads. The temperature data from the thermal analysis is used as input for the structural analysis, allowing for a coupled thermal-mechanical assessment of the welded structure.

The mechanical response of the structure is governed by Hooke's Law for linear elastic materials, modified to include thermal expansion effects [3]:

$$\sigma = E(\varepsilon - \alpha\Delta T)$$

Equation 5

Where: σ:represents stress,

E: is the Young's modulus,

ε: is the strain,

α: is the coefficient of linear thermal expansion,

ΔT: is the change in temperature.

This relationship allows the simulation to predict the residual stresses and thermal strains induced in the structure as a result of welding.

Additionally, gravitational forces are considered in the structural analysis to account for the weight of the conductor and pontoon components. This ensures that the analysis reflects realistic operating conditions, where both thermal expansion and gravitational forces influence the overall stress-strain behavior of the welded assembly.

Boundary conditions are carefully defined to replicate the real-world constraints of the conductor during welding. Fixed supports are applied in necessary locations to ensure stability, ensuring that the model accurately represents the mechanical constraints and captures the real behavior of the structure.

					19.ДРМ.135.6116м.ПЗ.03
Cha.	*Page*	*№ docum.*	*Signature*	*Data*	

				Application of finite element method considering thermal and mechanical boundary conditions.	Liter.	*Page*	*Pages*
Student	Grechyn V.Y.						
Consultant	Korostylyov L.I.				NUS		
Supervisor	Korostylyov L.I.						
Head of Dep.	Korostylyov L.I.						

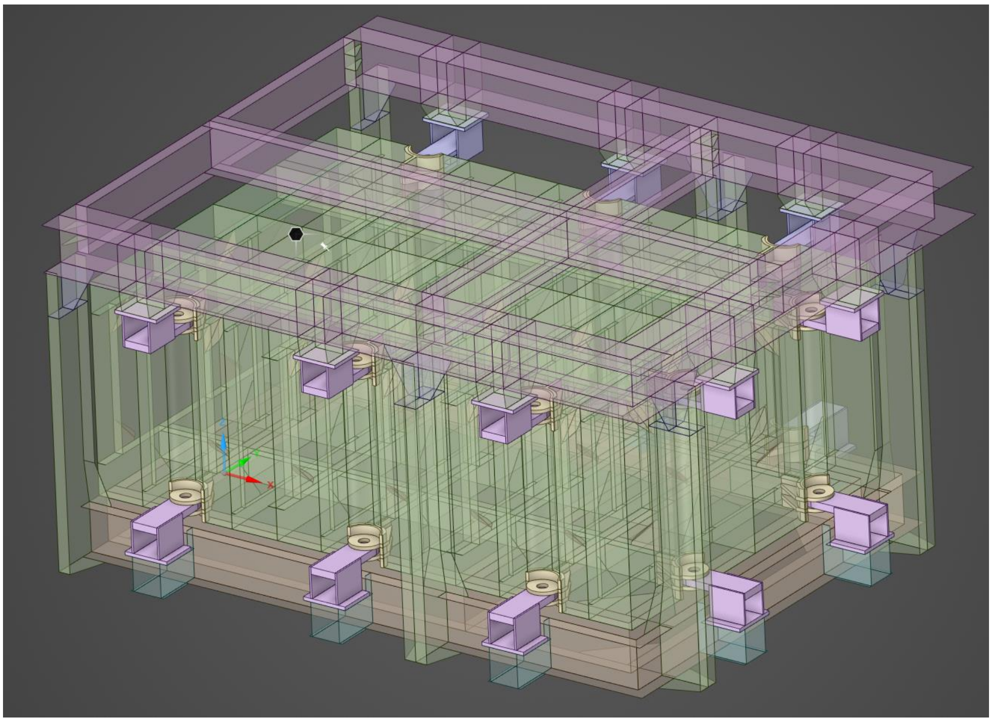
2. APPLICATION OF FINITE ELEMENT METHOD CONSIDERING THERMAL AND MECHANICAL BOUNDARY CONDITIONS.

2.1 Geometry Description

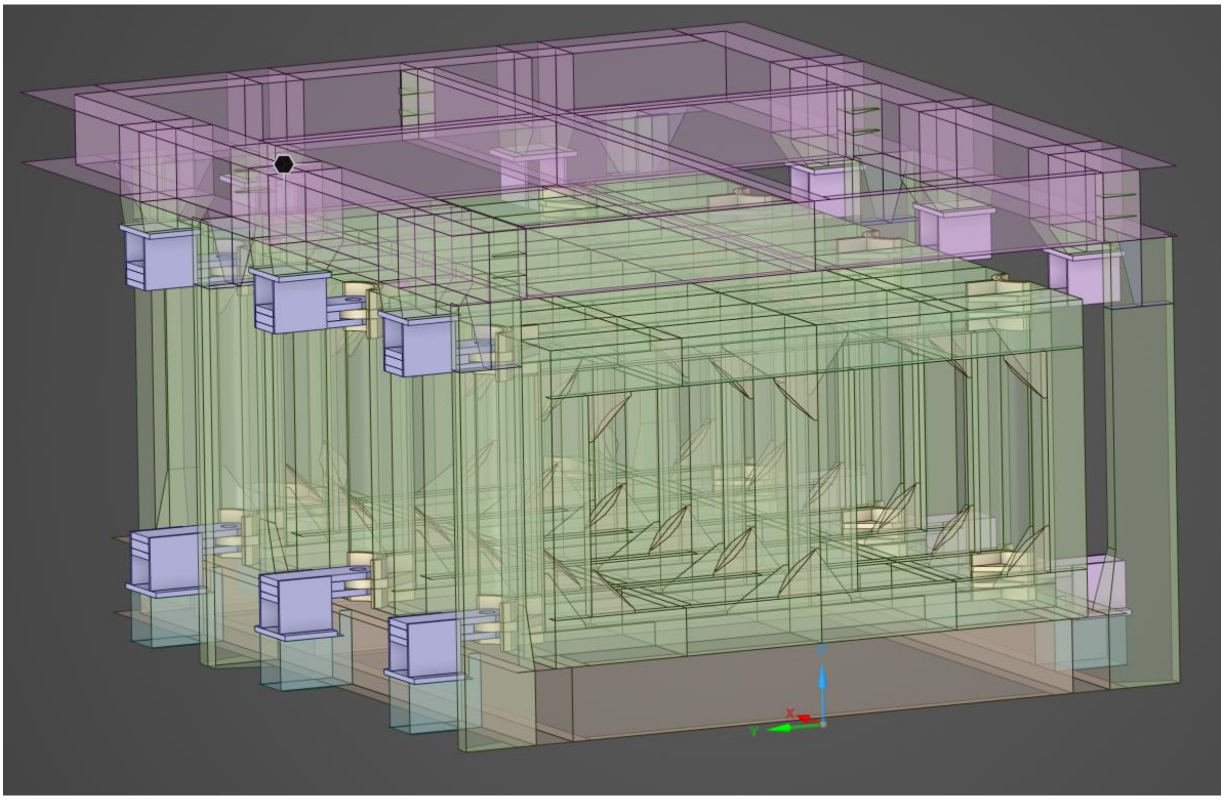
2.1.1 Geometry for Thermal Analysis

Preparation of the Model: The final model, created in *Rhino*, is imported into ANSYS using the Discovery and the SpaceClaim module. Within *SpaceClaim*, the *Share Topology* command is employed to unite most of the structure, excluding the bottom of the pontoon and the conductor elements that are in contact with the pontoon base. The Share Topology command does not merge the entire structure into a single entity but eliminates duplicate elements at contact points. This is essential for ensuring that the interacting surfaces are automatically taken into account during mesh generation, which enhances heat distribution accuracy during thermal analysis.

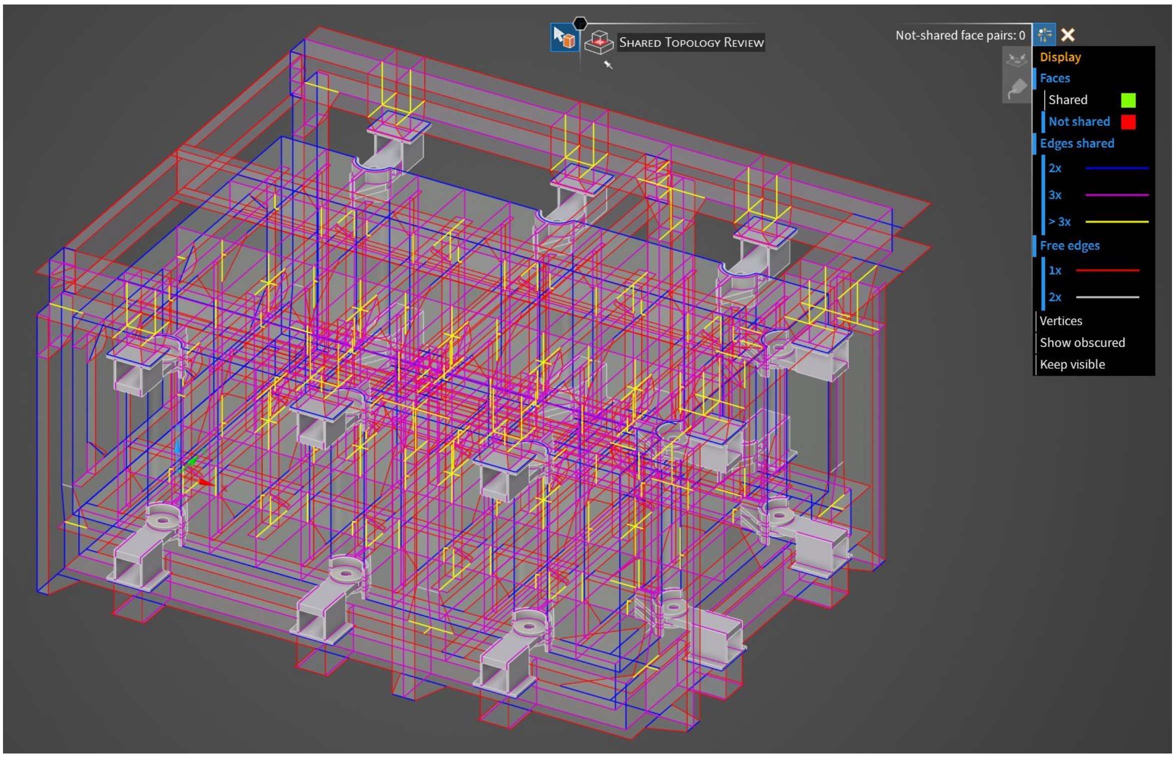
Loading the Model in ANSYS Mechanical: Once the preparation is complete, the model is loaded into ANSYS Mechanical. Material properties are assigned to each element of the model to ensure that the simulation reflects realistic thermal behavior. For shell components, additional parameters such as material thickness and offset direction are also defined. These settings are critical for accurately modeling the heat flow through different regions of the structure.



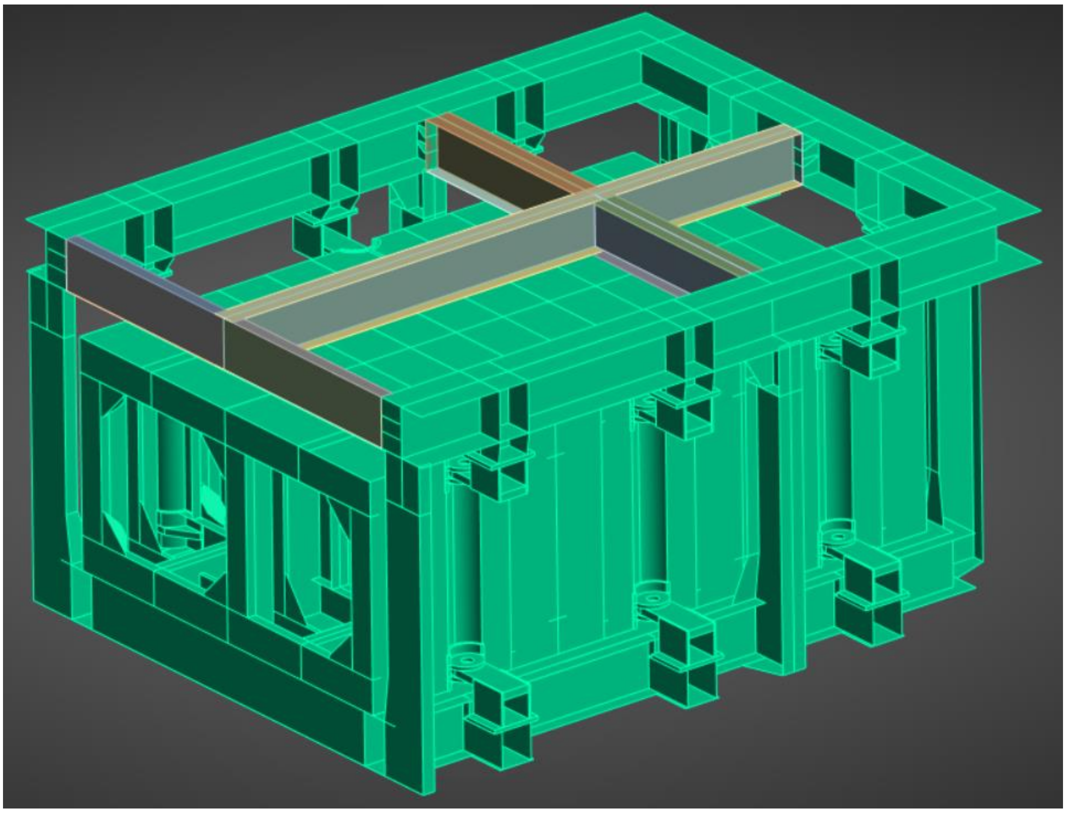
Picture 32: Geometry at the Discovery Module.



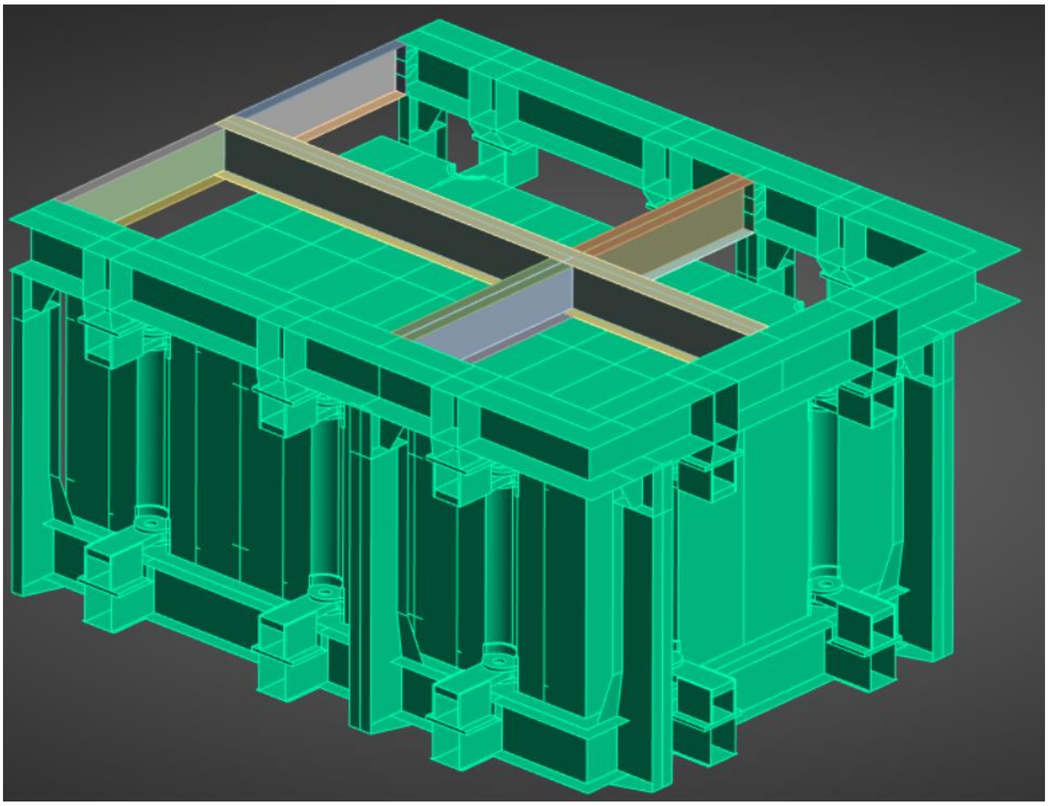
Picture 33: Geometry at the Discovery Module.



Picture 34: Geometry at the Discovery Module. Shared topology Preview.



Picture 35: Geometry at the Mechanical Module. The Construction in green color shares its topology.



Picture 36: Geometry at the Mechanical Module. The Construction in green color shares its topology.

2.1.2 Geometry for structural analysis

Features of Element Interaction: In the structural analysis model, both solid and shell elements are utilized. This approach is crucial because the degrees of freedom (DOF) for each type of element must be managed carefully. Solid elements have three degrees of freedom, involving translations in the x, y, and z directions, while shell elements have six degrees of freedom, which include translations and rotations. This discrepancy necessitates special handling of the interactions between these elements.

At locations where shell elements contact solid elements, using Share Topology is not advisable as it can lead to conflicts in degrees of freedom. Instead, contact interactions must be defined to properly constrain the degrees of freedom at the points of interaction. This ensures accurate modeling of the connection between shell and solid elements and helps avoid errors related to incompatible constraints during the finite element analysis.

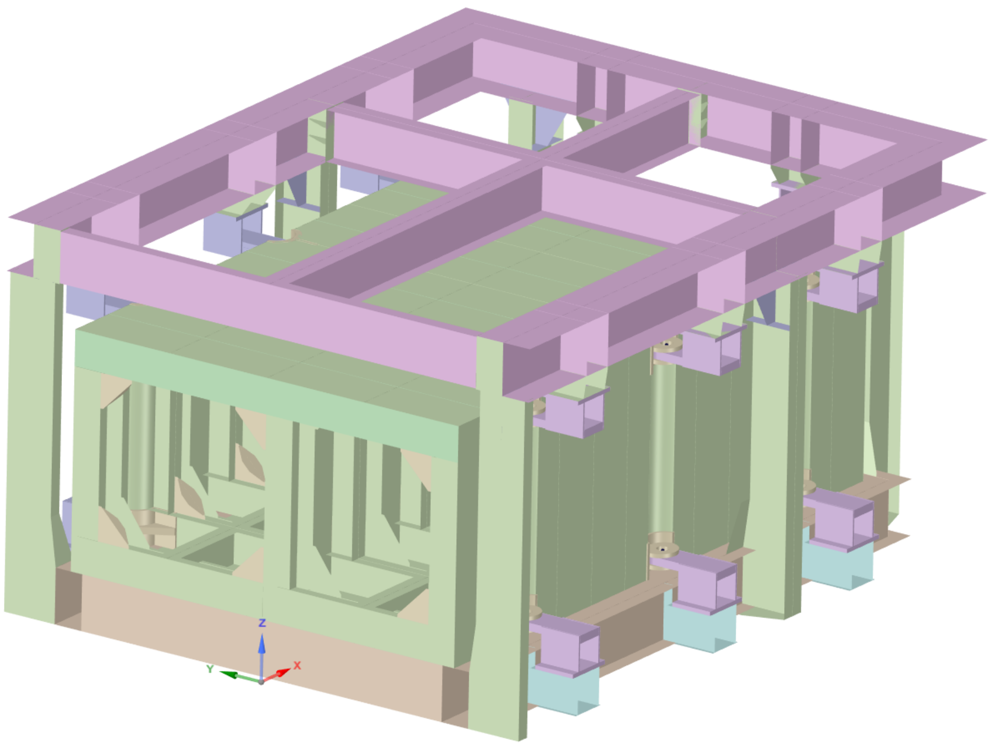
Preparation of the Model: The final model, developed in Rhino, is imported into ANSYS using the SpaceClaim module. In SpaceClaim, the Share Topology command is used to unify almost all structural elements, with the following exceptions:

- The bottom of the pontoon and the conductor elements that are in contact with the pontoon bottom are not merged using Share Topology. This ensures that proper contacts can be defined to handle the complex interaction between the conductor and the pontoon.
- Solid and shell elements within the pontoon and conductor structures are also not unified with Share Topology. This is to prevent conflicts due to their differing degrees of freedom, as discussed earlier. Instead, contacts are defined to accurately model the interaction between solid and shell parts.

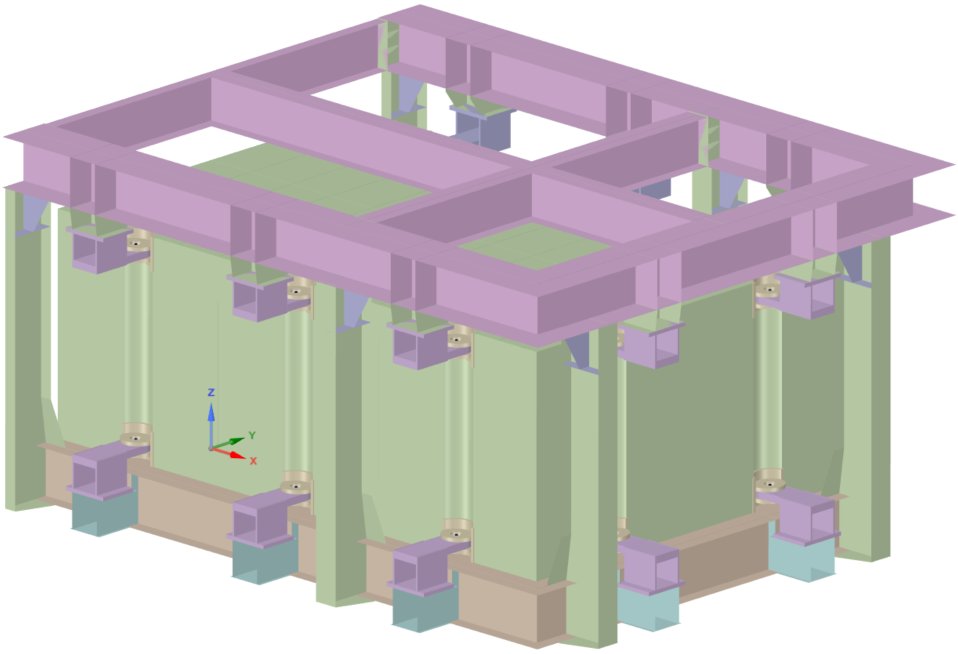
Loading the Model in ANSYS Mechanical: After preparing the model, it is loaded into ANSYS Mechanical for analysis. During this process:

Materials are assigned to each component to ensure that the model accurately represents the real physical properties of the structure.

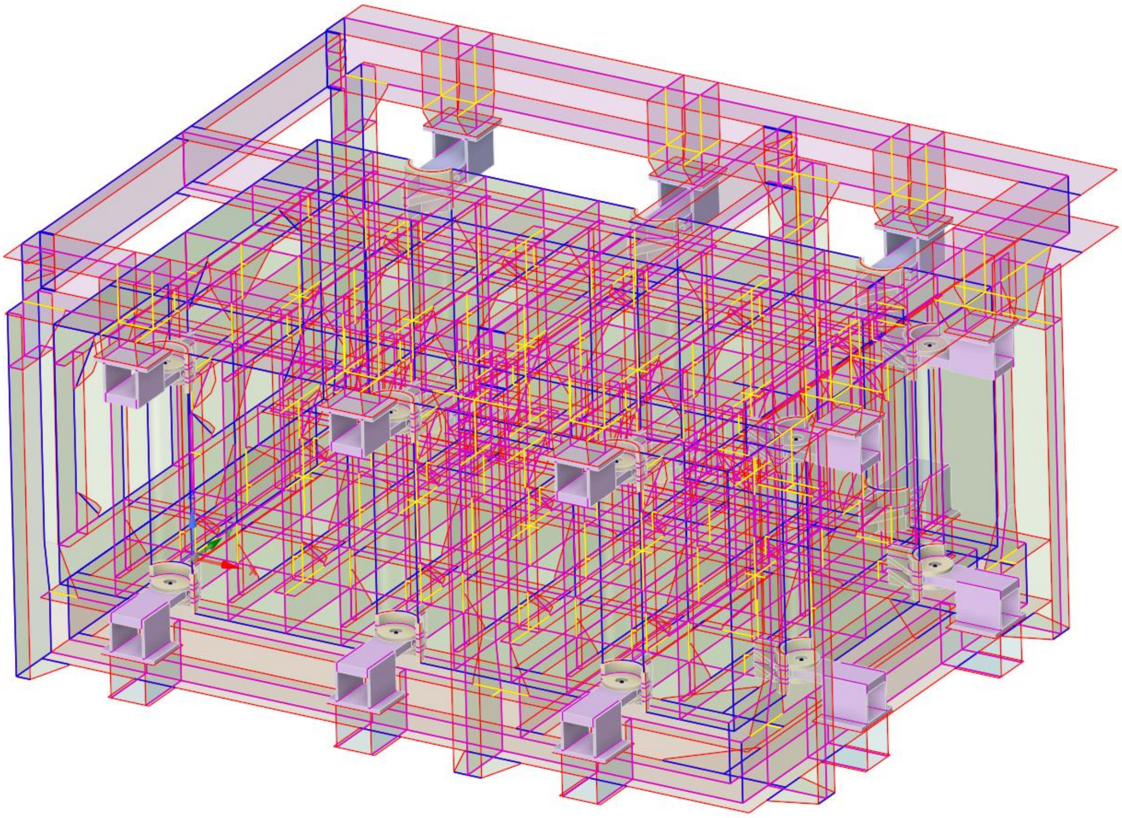
For shell components, additional parameters are specified, including the thickness of the material and the offset direction. These parameters are critical for capturing the correct behavior of the shell elements under loading conditions.



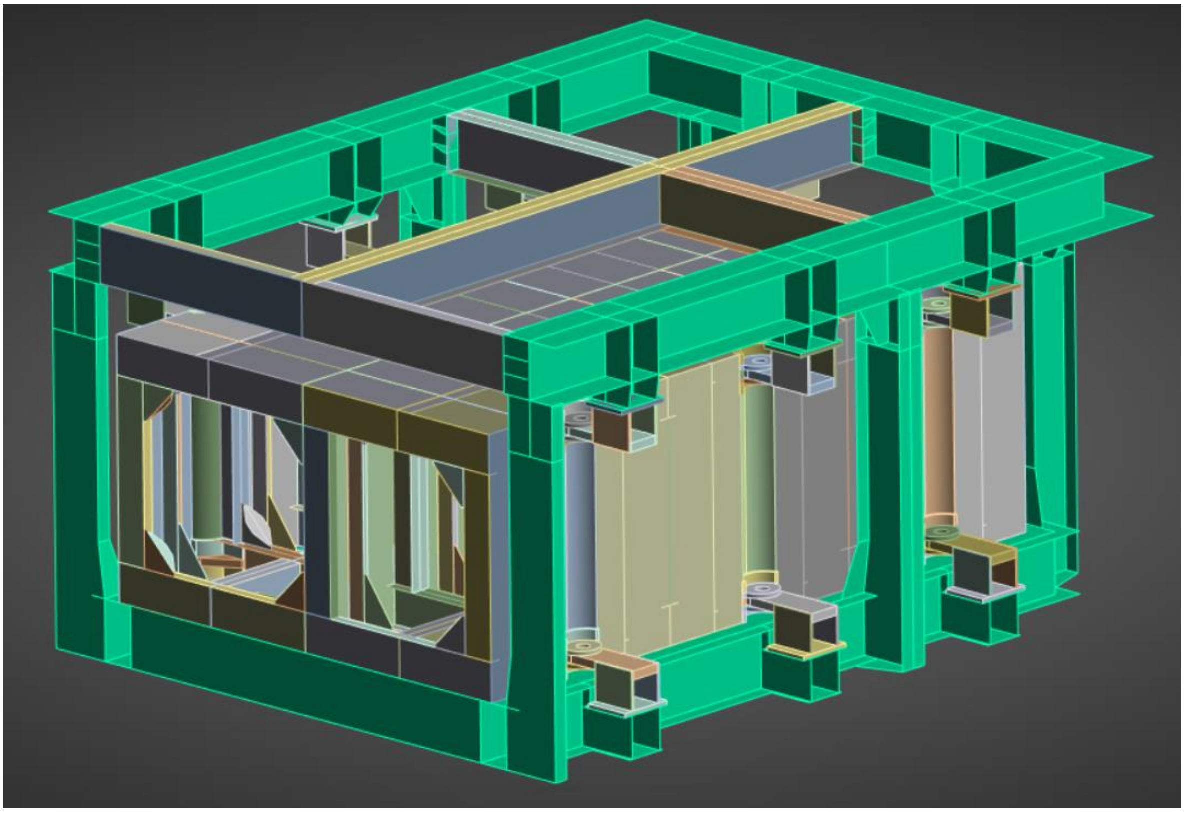
Picture 37: Geometry at the SpaceClaim Module.



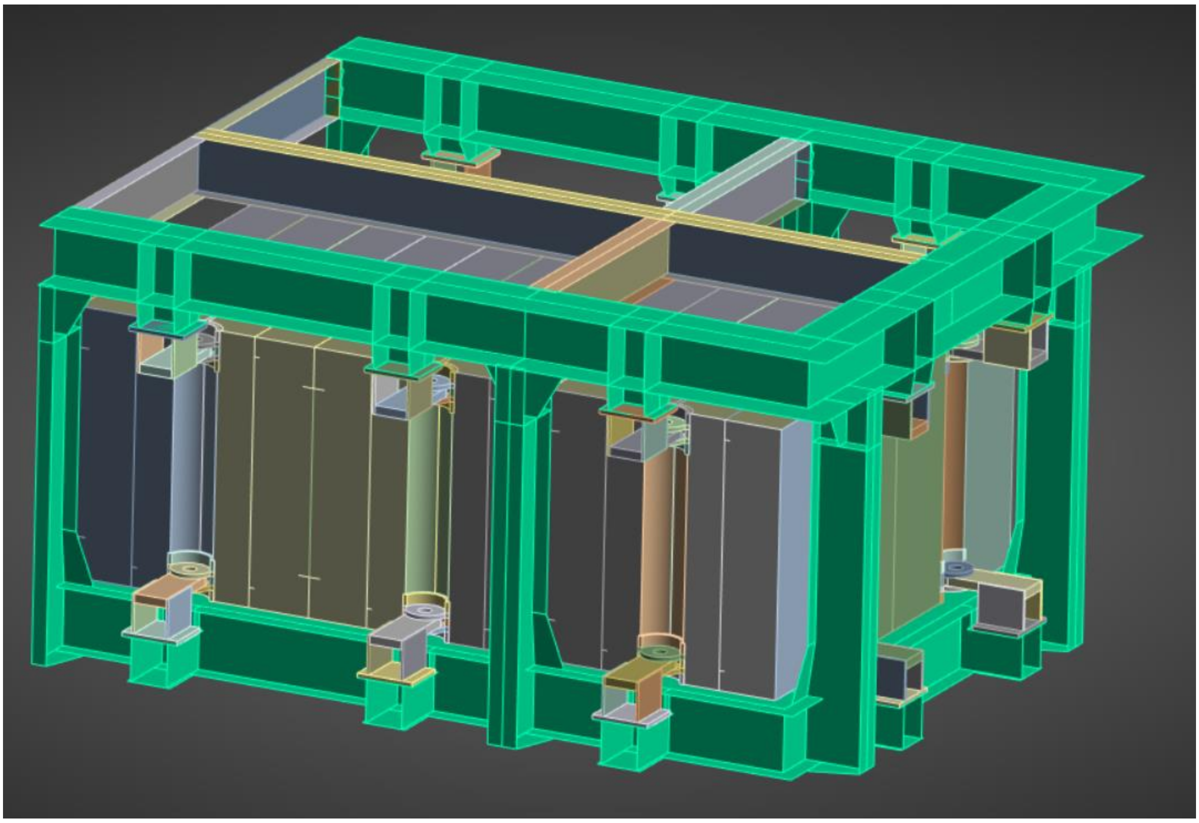
Picture 38: Geometry at the SpaceClaim Module.



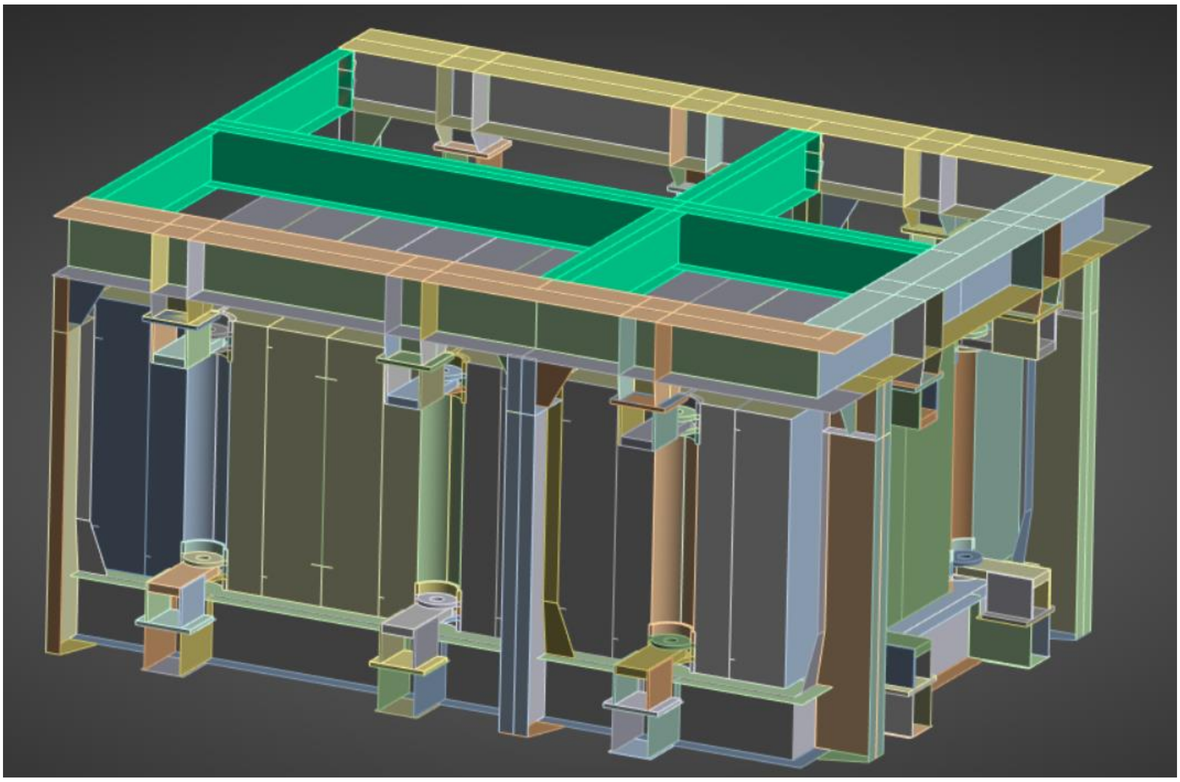
Picture 39: Geometry at the SpaceClaim Module. Shared topology Preview.



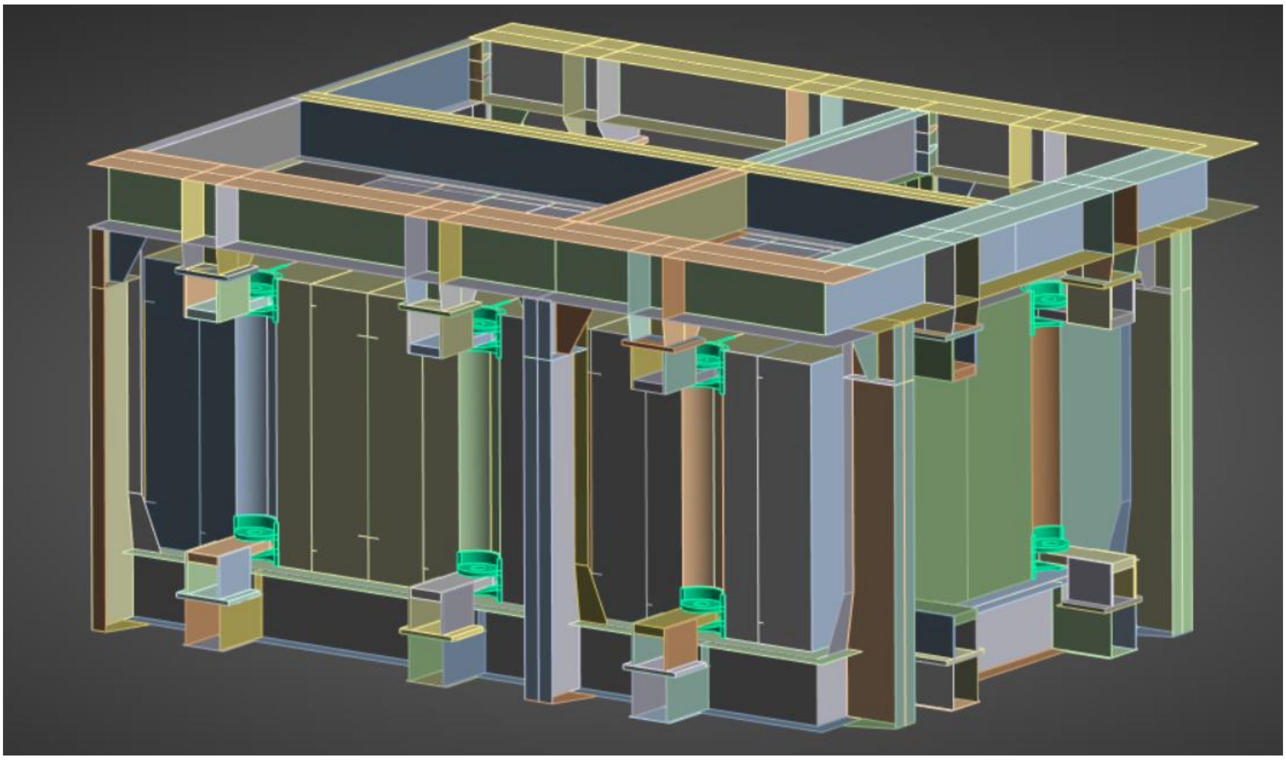
Picture 40: Geometry at the Mechanical Module. The Construction in green color shares its topology.



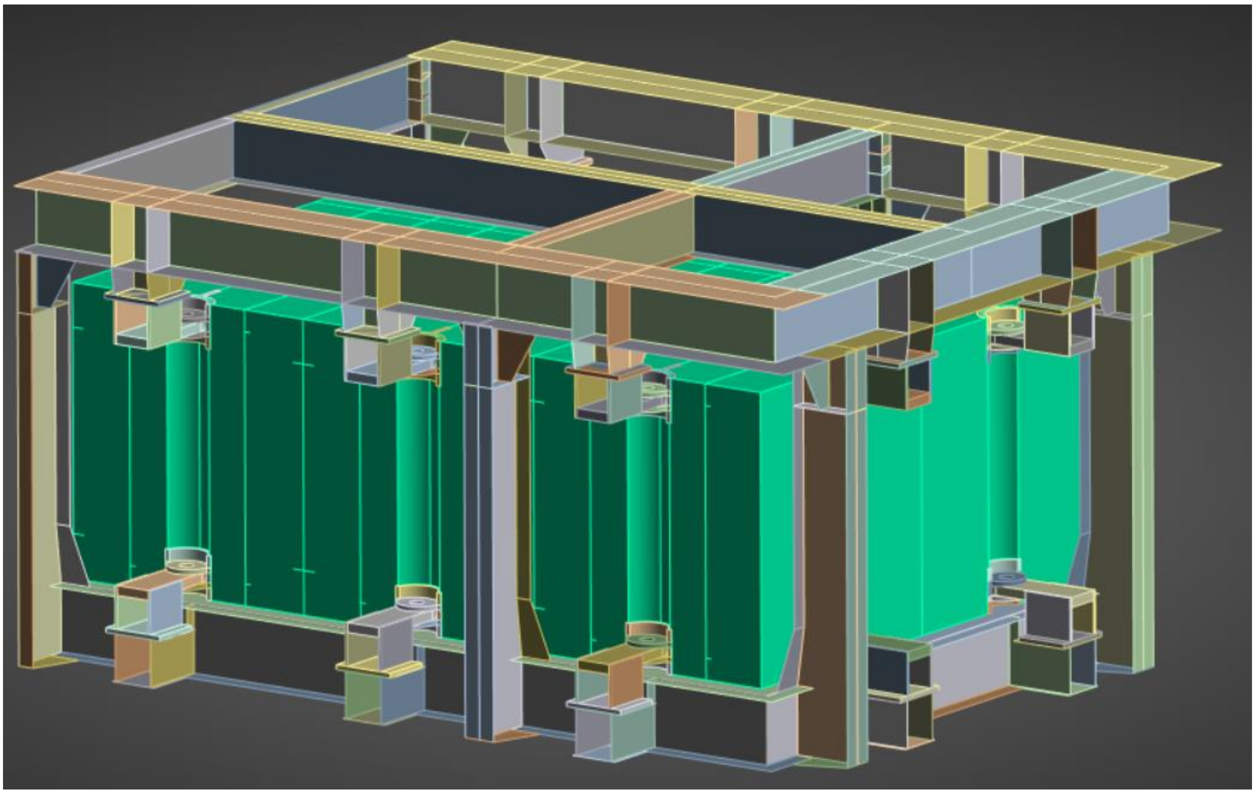
Picture 41: Geometry at the Mechanical Module. The Construction in green color shares its topology.



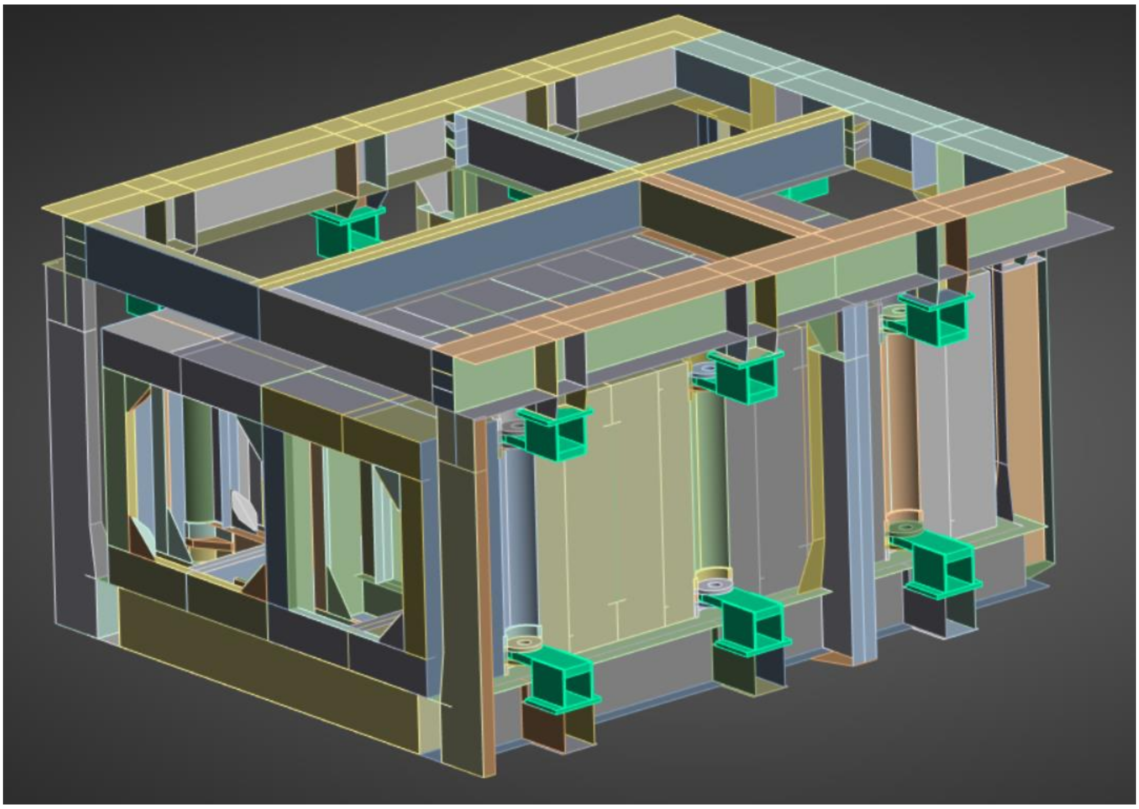
Picture 42: Geometry at the Mechanical Module. The Construction in green color shares its topology.



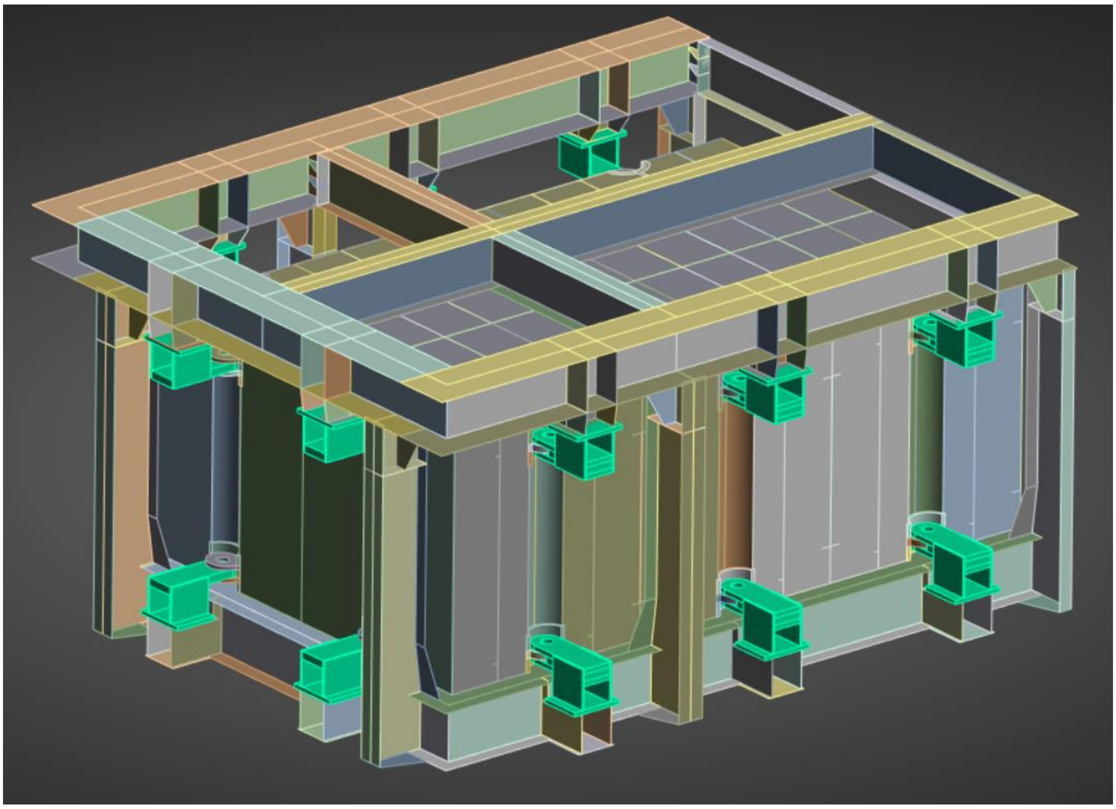
Picture 43: Geometry at the Mechanical Module. The Construction in green color shares its topology.



Picture 44: Geometry at the Mechanical Module. The Construction in green color shares its topology.



Picture 45: Geometry at the Mechanical Module. The Construction in green color shares its topology.



Picture 46: Geometry at the Mechanical Module. The Construction in green color shares its topology

2.2 Finite Element Models

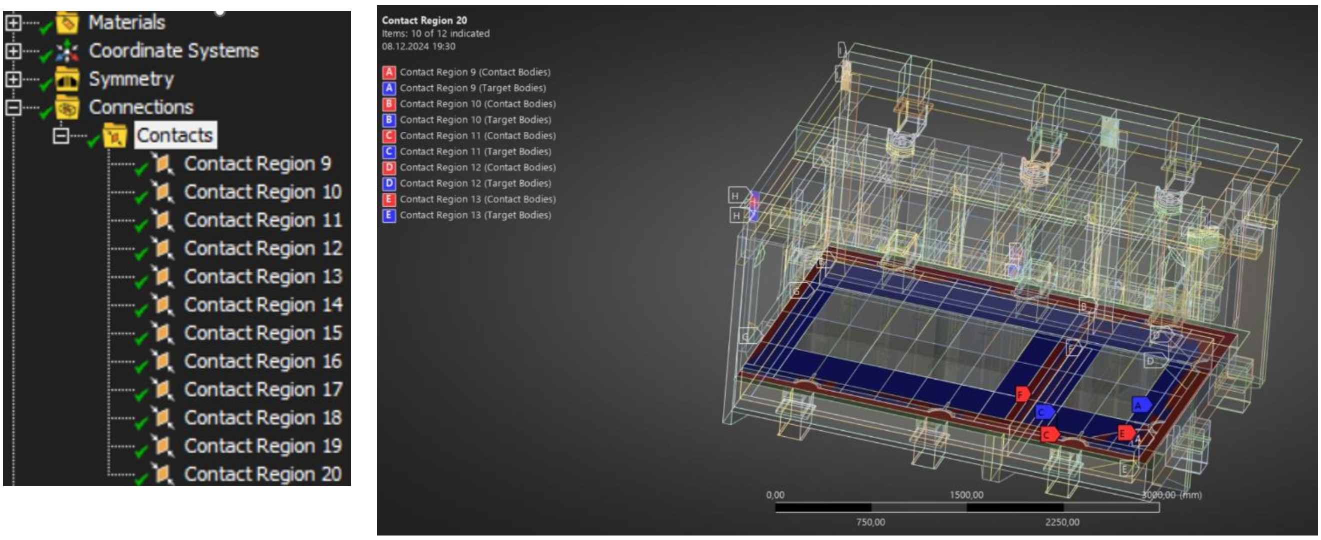
2.2.1 Model for thermal analysis

2.2.1.1 Conductor-Pontoon Interaction

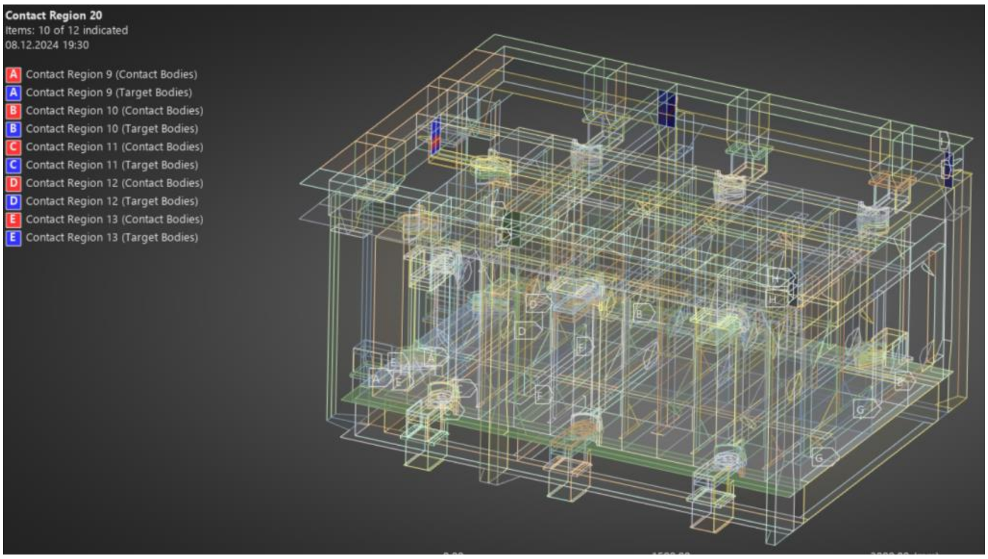
Preparation of the Model:

For the previously imported model, which already has materials and thicknesses assigned, the software automatically searches for contact surfaces between different components. Based on these detected surfaces, contacts are generated with a default parameter - bonded contact. Bonded contacts ensure that the connected components do not experience relative movement during the analysis, which is critical for accurately predicting thermal behavior.

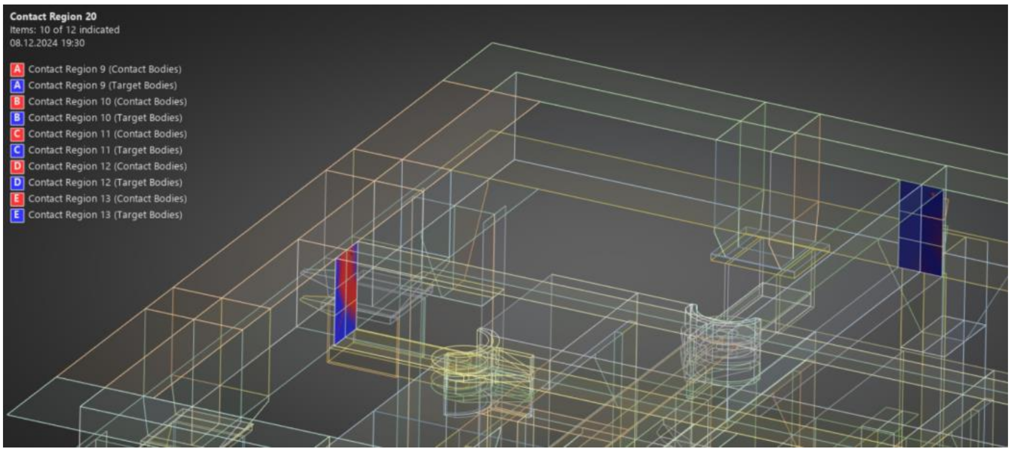
After the automatic *contact generation*, a thorough verification of the contacts is performed, with particular attention to the bottom of the pontoon and the conductor structures that are in contact with it. This verification step is essential to ensure that all critical areas are correctly modeled, as inaccuracies in contact definitions can lead to erroneous thermal and mechanical responses in the simulation.



Picture 47: List of contact groups and Contact regions.



Picture 48: Contact regions.



Picture 49: Contact regions.

2.2.1.2 Mesh Generation

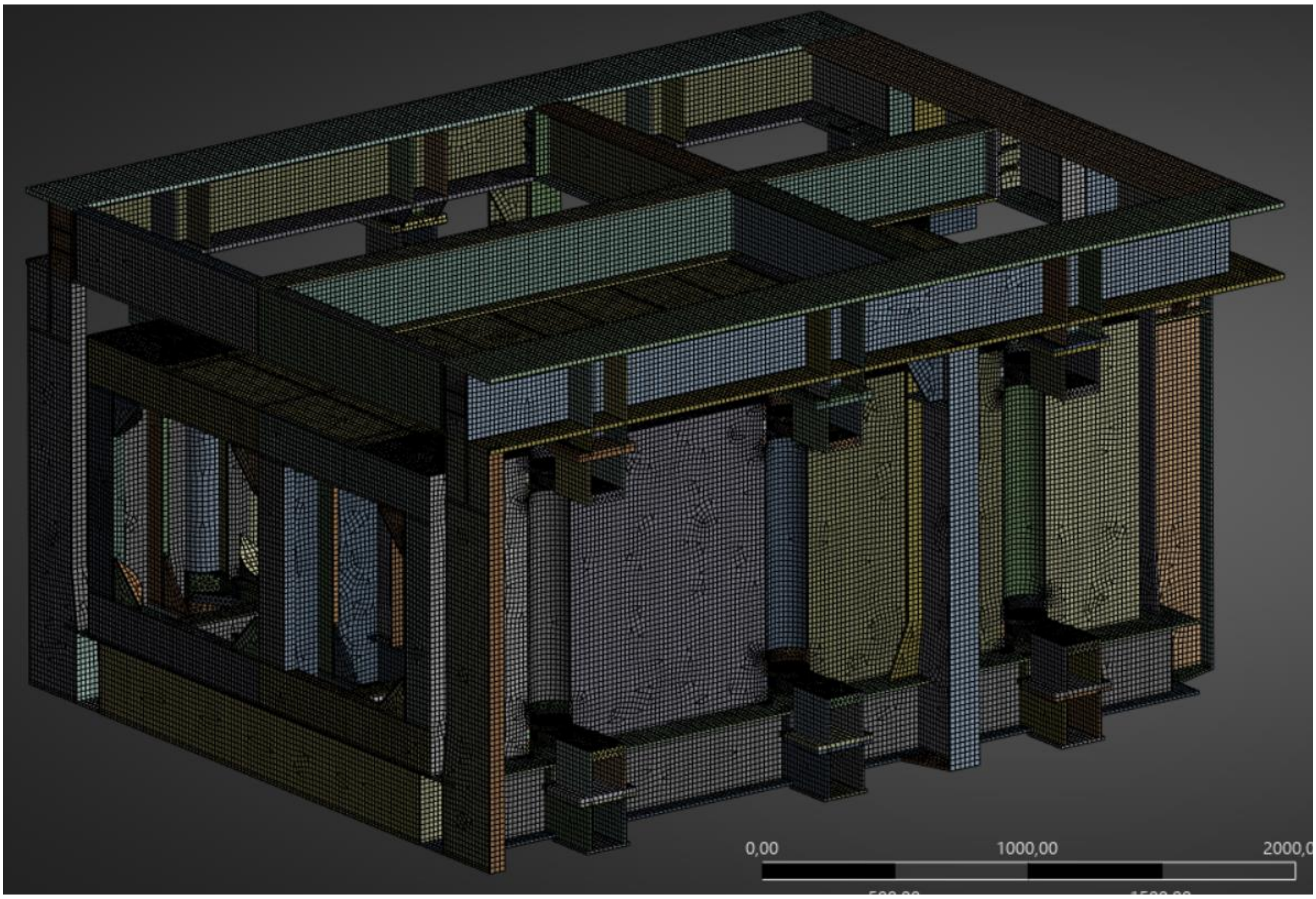
Mesh Generation Process: The mesh generation for the thermal analysis of the conductor and pontoon model is a critical step to ensure accuracy in the finite element simulation. The mesh generation process involves several key actions to define the discretization of the geometry:

Mesh Preparation: To create an effective mesh, different groups of elements within the pontoon and conductor structures are selected. For these groups, named selections are created, allowing specific mesh parameters to be set for each group individually. This helps in maintaining control over the mesh density, ensuring that critical areas receive higher resolution.

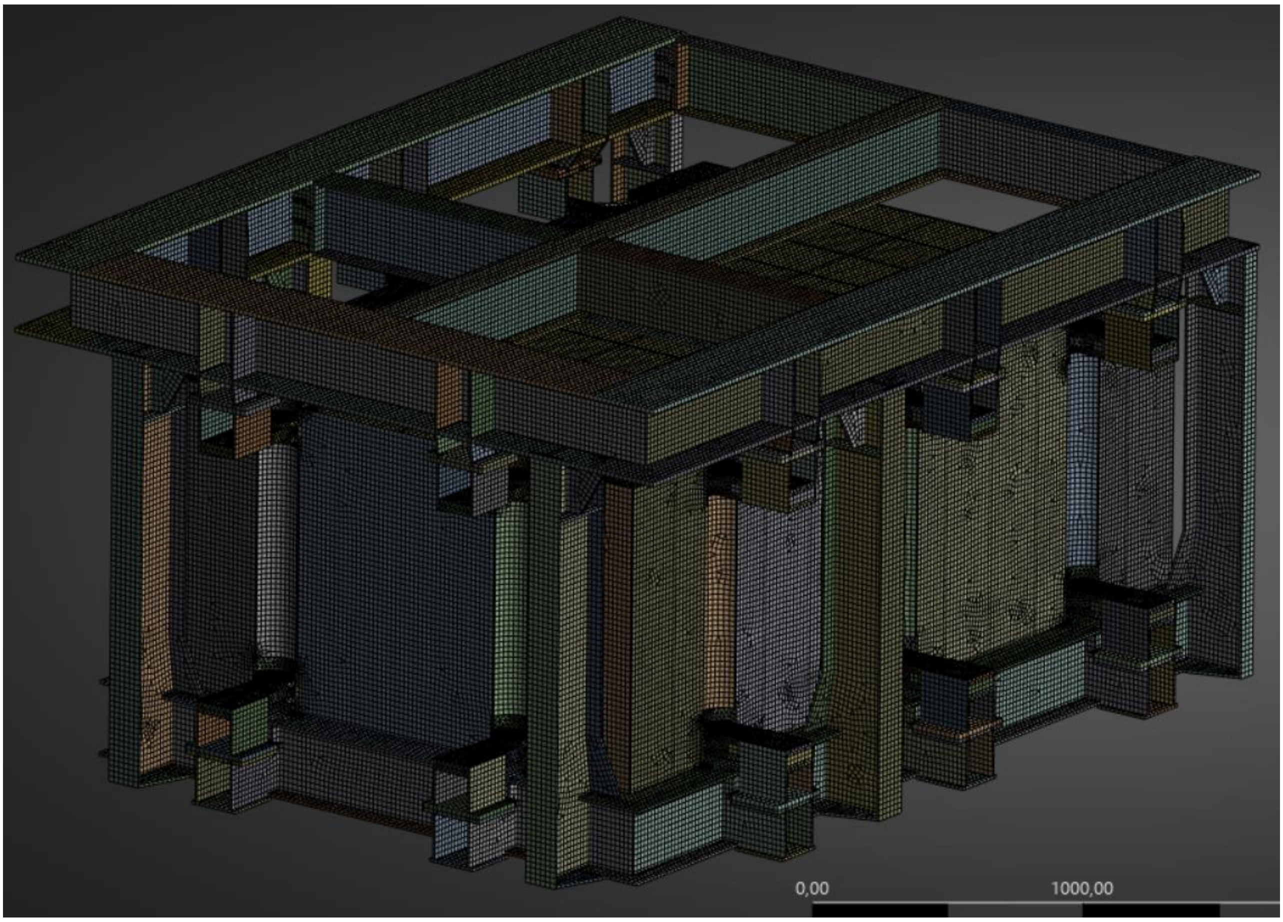
For the welded elements of the pontoon and the conductor fixing elements, particularly in regions with holes for fixing elements, a parameter called face sizing is applied. This ensures that the mesh is more refined in these areas, providing higher accuracy for thermal gradient calculations. This is particularly important in regions of stress concentration and thermal flux, where an accurate representation of the mesh is necessary to capture localized effects.

The element order for the mesh is chosen as quadratic for all parts of the model. Quadratic elements, which have mid-side nodes, are known to provide better precision for problems involving high gradients in the solution, such as thermal and stress-strain analyses. These elements improve the convergence rate and accuracy of the calculations, making them ideal for the current welding analysis.

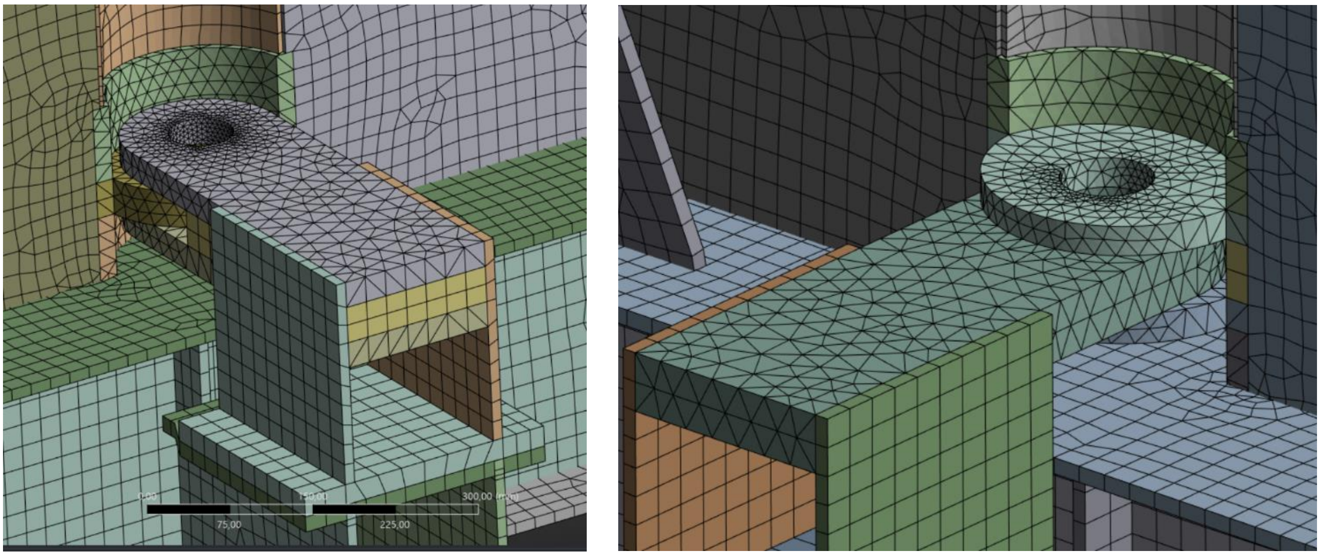
The total number of elements in the model is 577563.



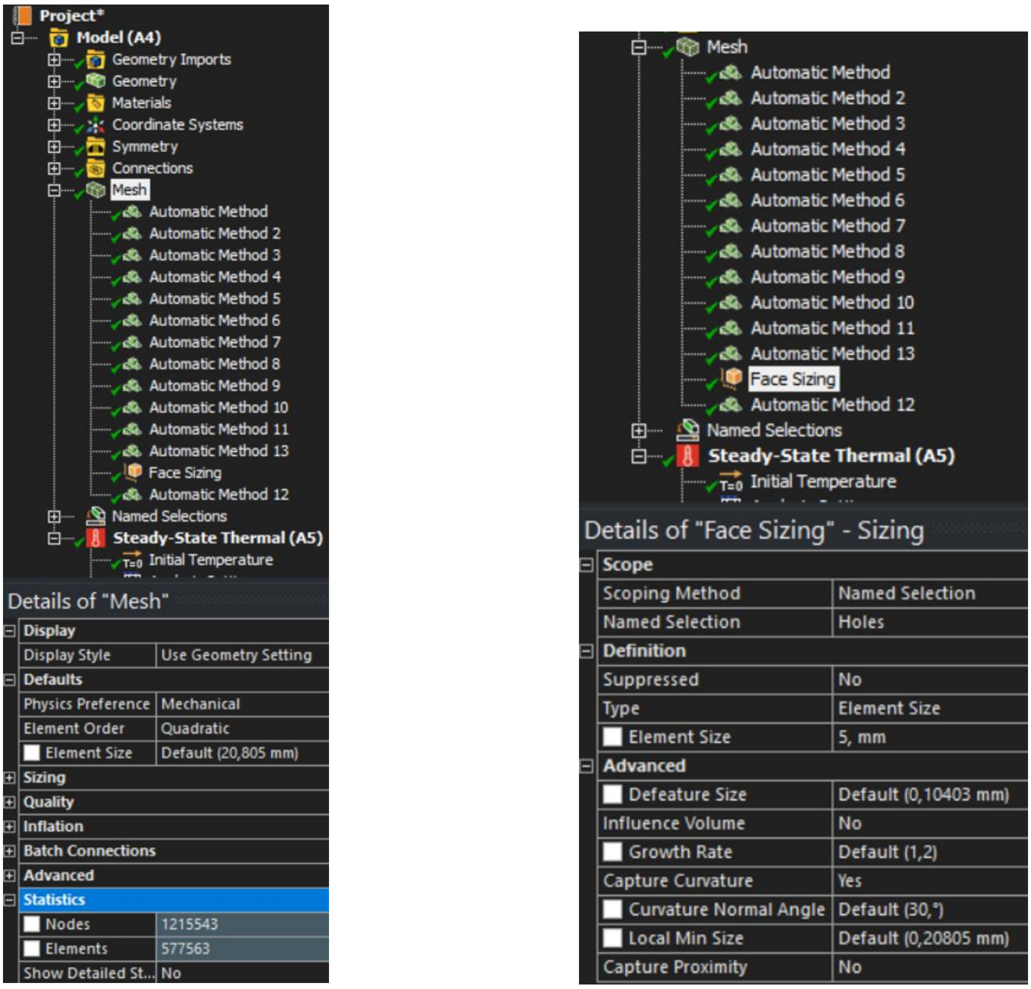
Picture 50: Meshed elements of the structure.



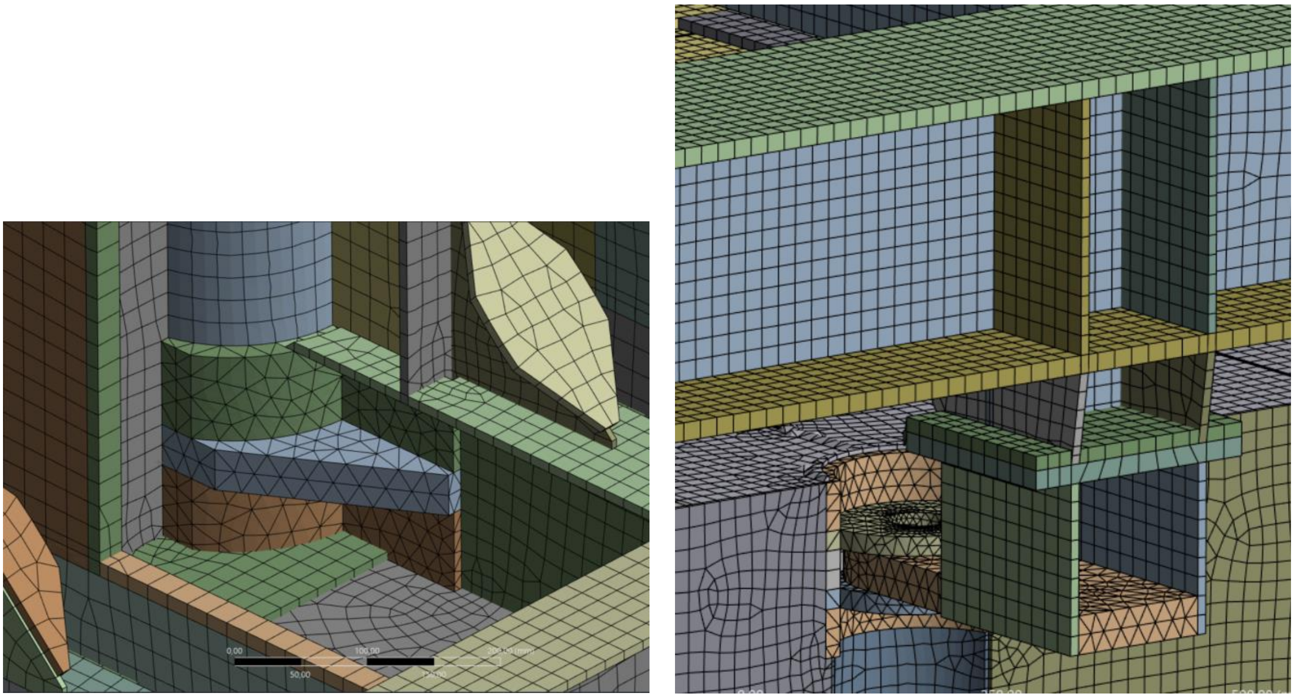
Picture 51: Meshed elements of the structure.



Picture 52: Refined meshed elements of the structure.



Picture 53: Characteristics of mesh components.



Picture 54: Refined meshed elements of the structure.

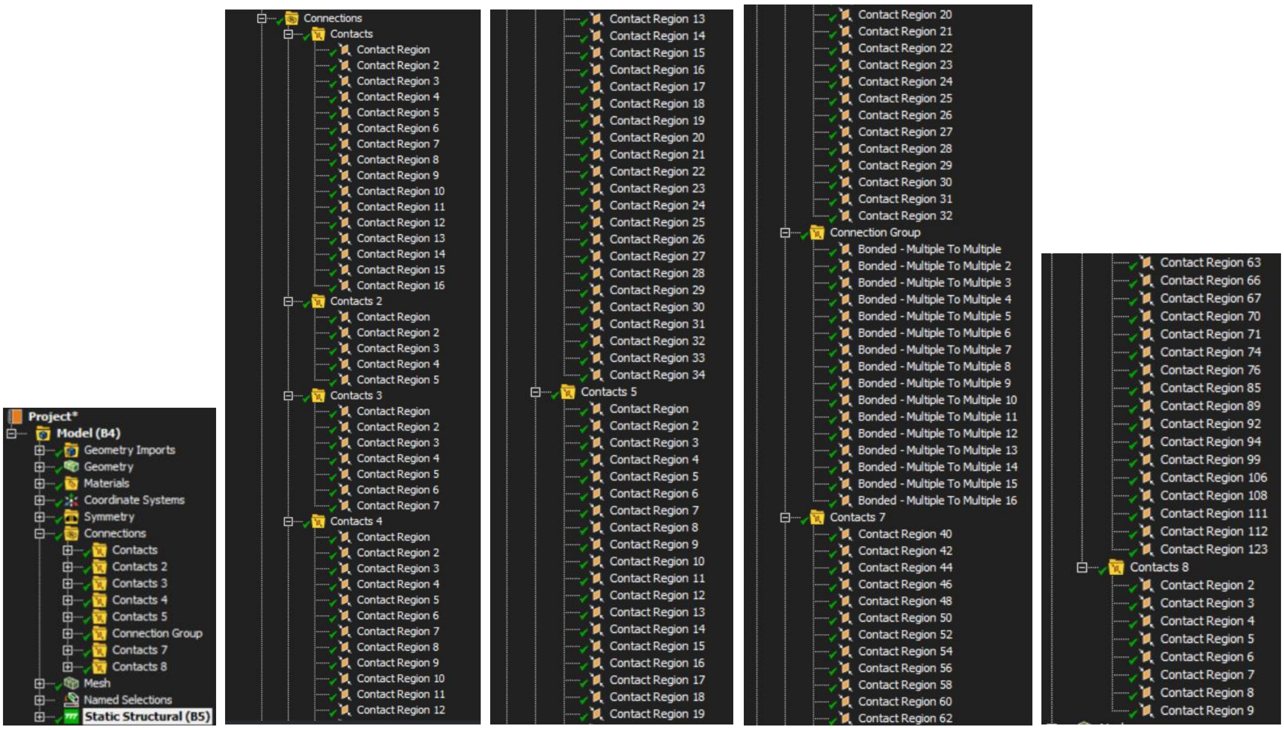
2.2.2 Model for structural analysis

2.2.2.1 Conductor-Pontoon Interaction

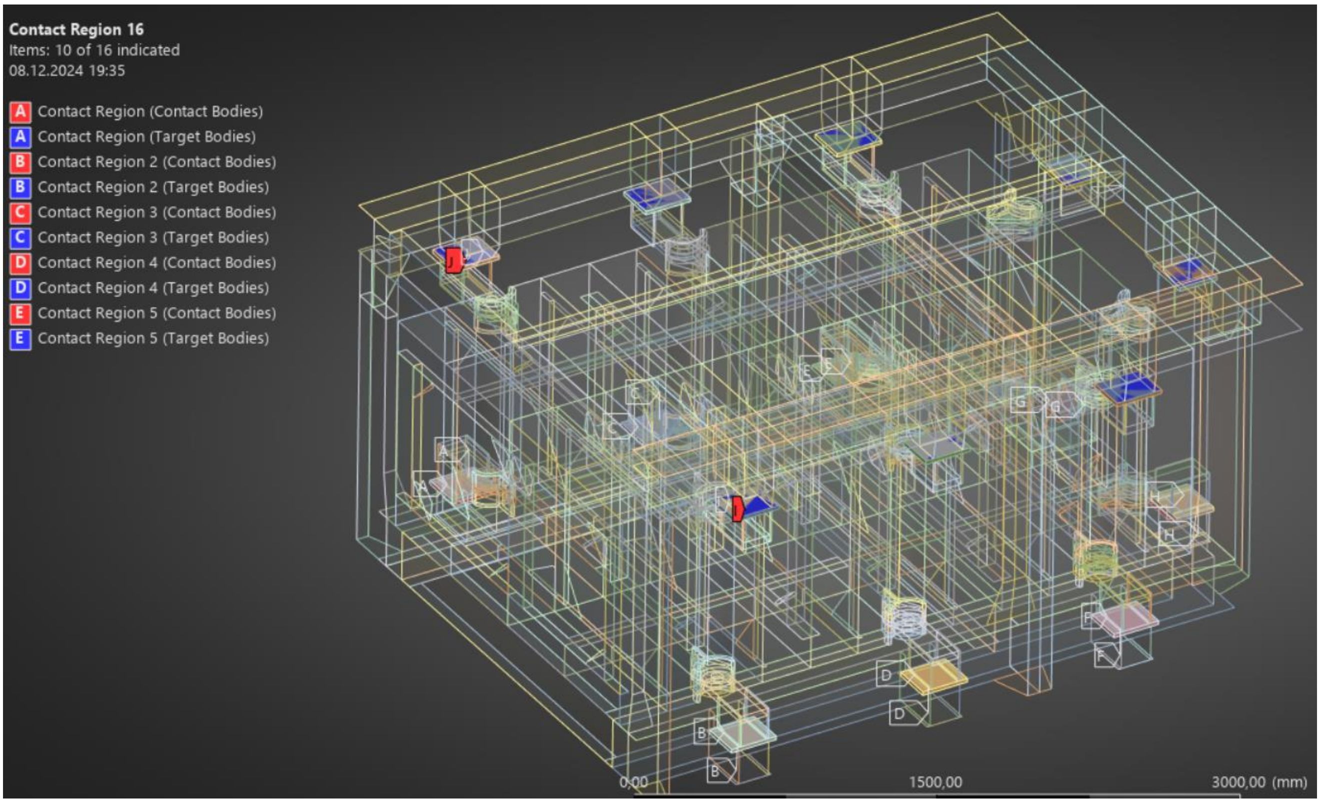
Preparation of the Model: For the previously imported model, with materials and thicknesses assigned, the software automatically searches for contact surfaces between the conductor and pontoon elements. Given that the structural analysis involves both solid and shell elements, specific distinctions must be taken into account, especially in areas where these elements interact. The model is prepared so that the program can automatically determine the appropriate contact areas.

The bonded contacts generated for solid and shell elements ensure proper fixation of additional degrees of freedom specific to shell elements, providing the necessary constraints to manage their interaction effectively. This type of contact is crucial for accurately simulating welded connections by preventing relative displacement between elements and ensuring that all constraints are handled appropriately.

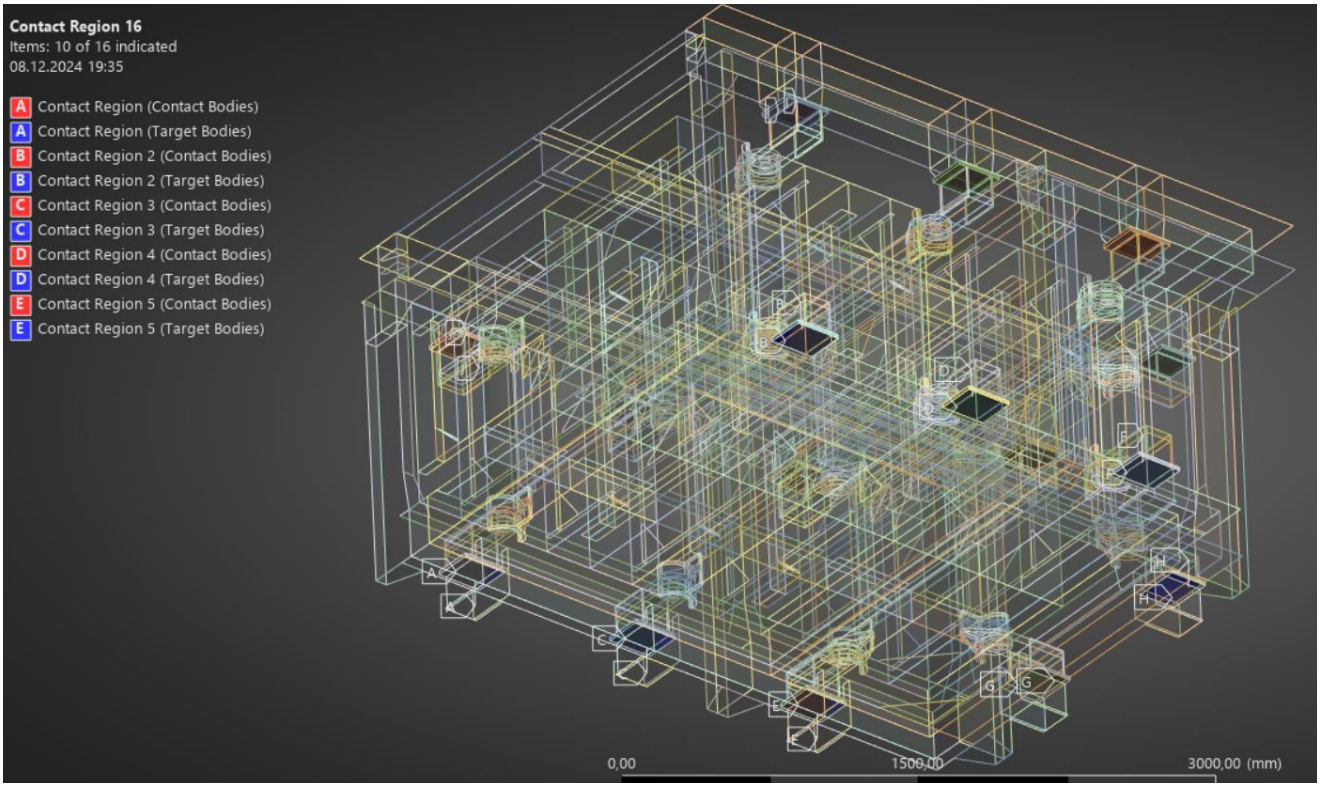
After the automatic generation of bonded contacts, a thorough verification process is conducted.



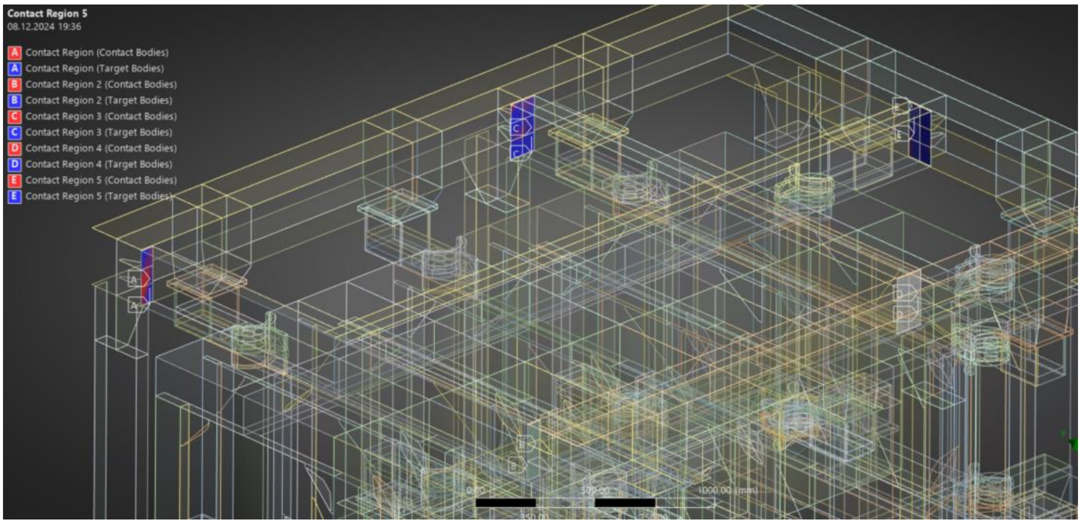
Picture 55: List of contact groups and contacts.



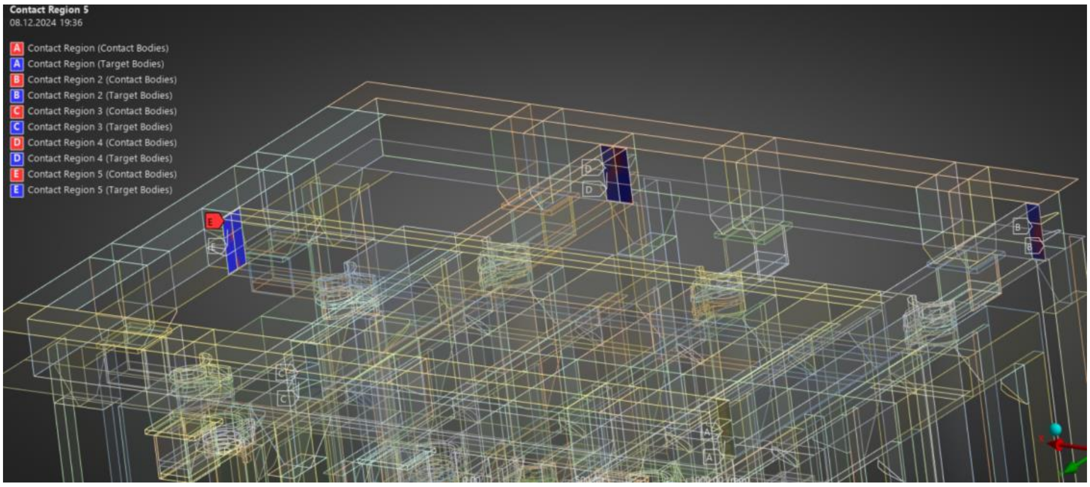
Picture 56: Contact regions.



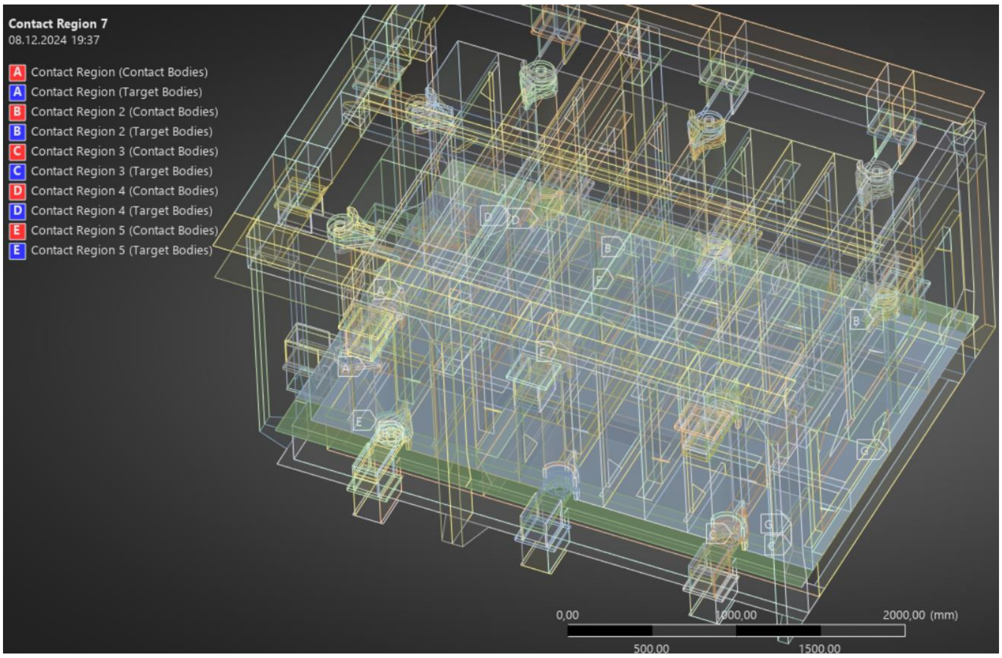
Picture 57: Contact regions.



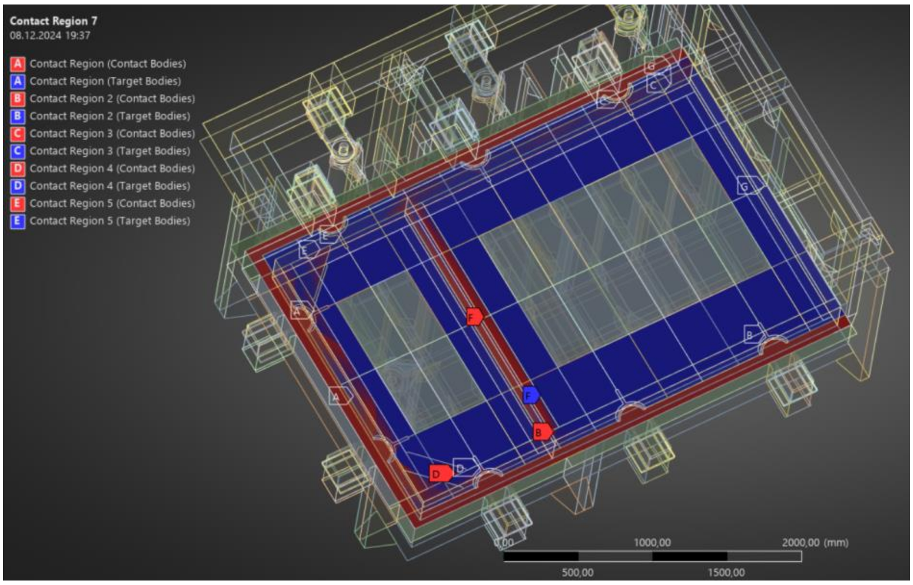
Picture 58: Contact regions.



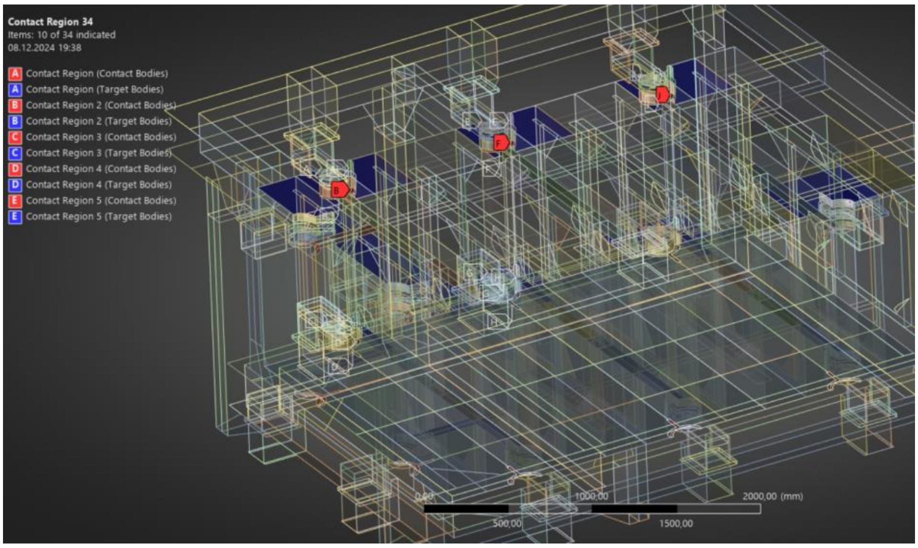
Picture 59: Contact regions.



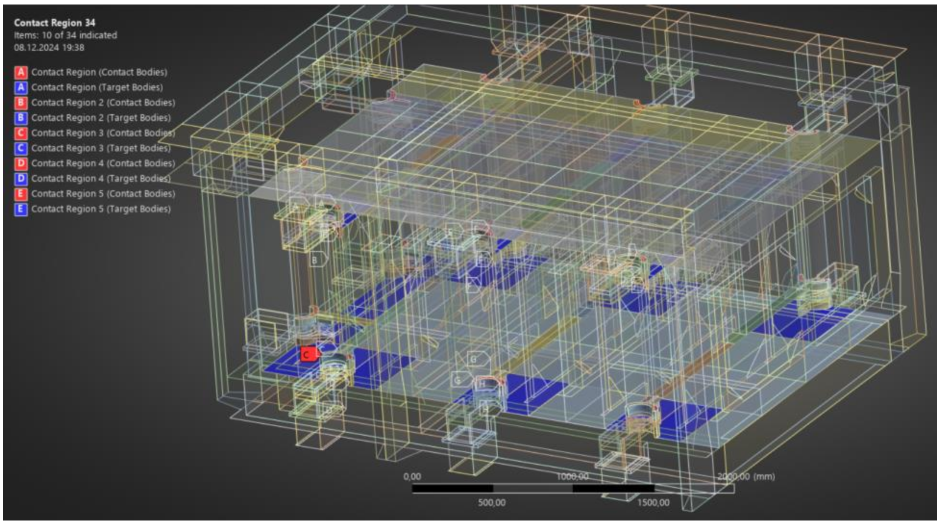
Picture 60: Contact regions.



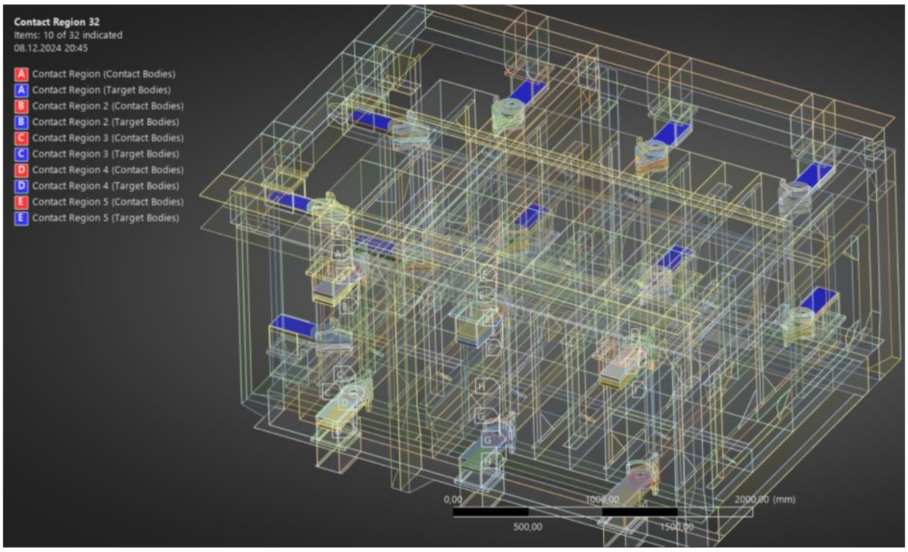
Picture 61: Contact regions.



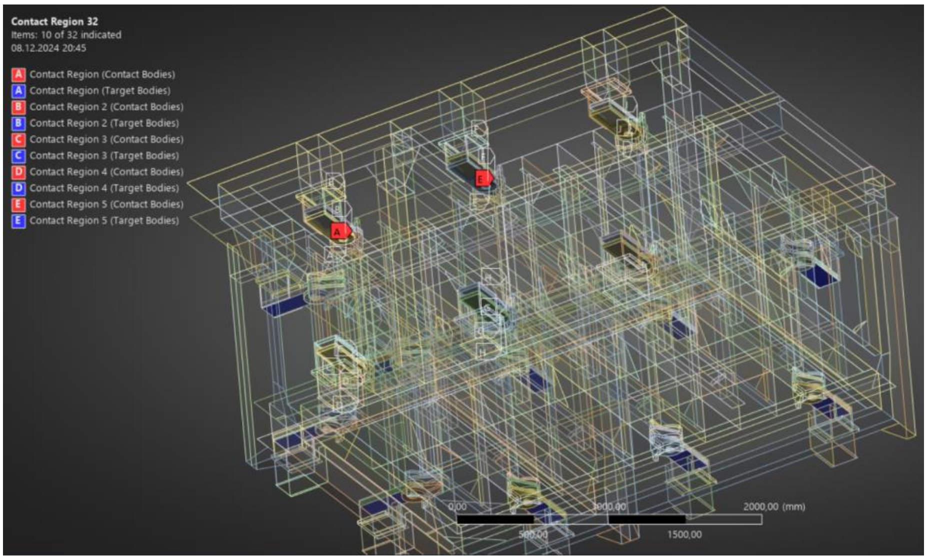
Picture 62: Contact regions.



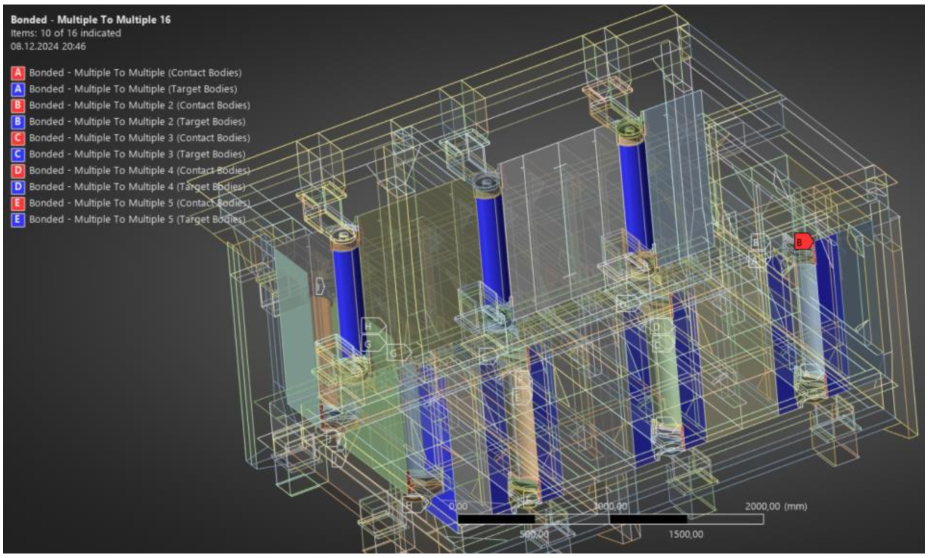
Picture 63: Contact regions.



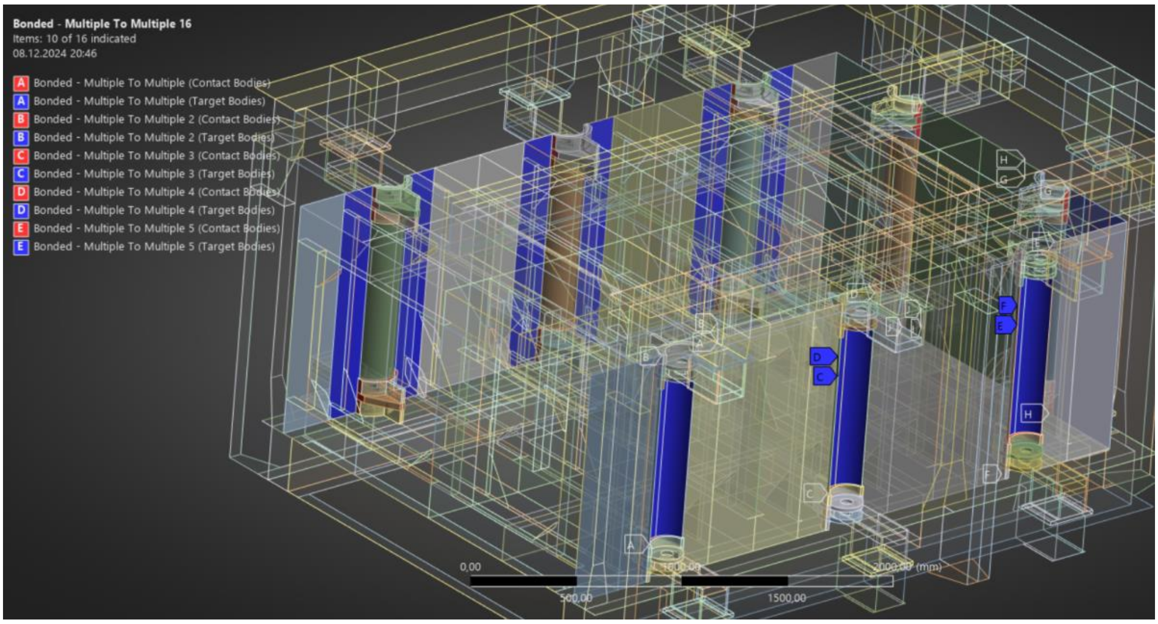
Picture 64: Contact regions.



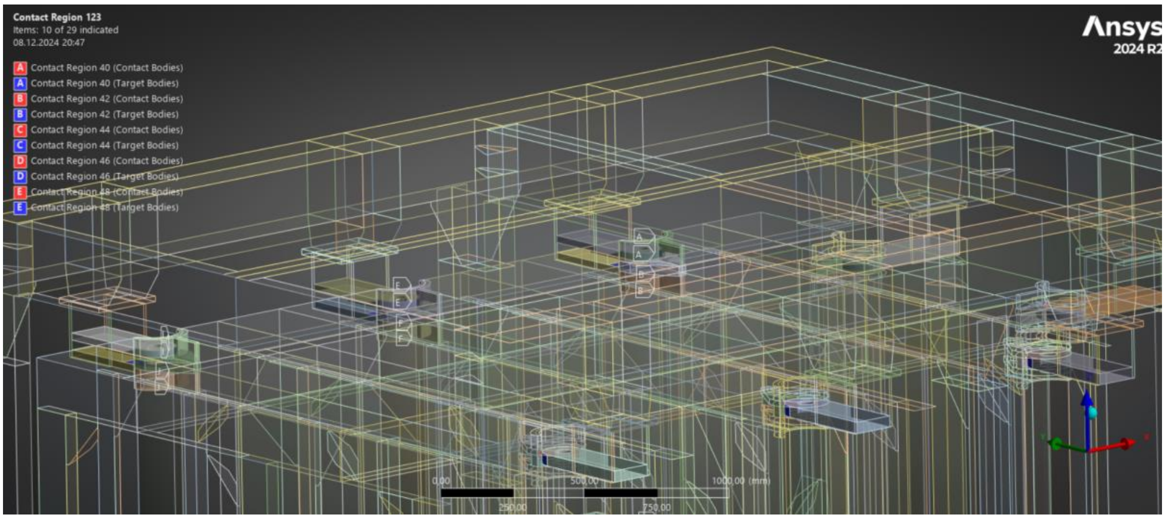
Picture 65: Contact regions.



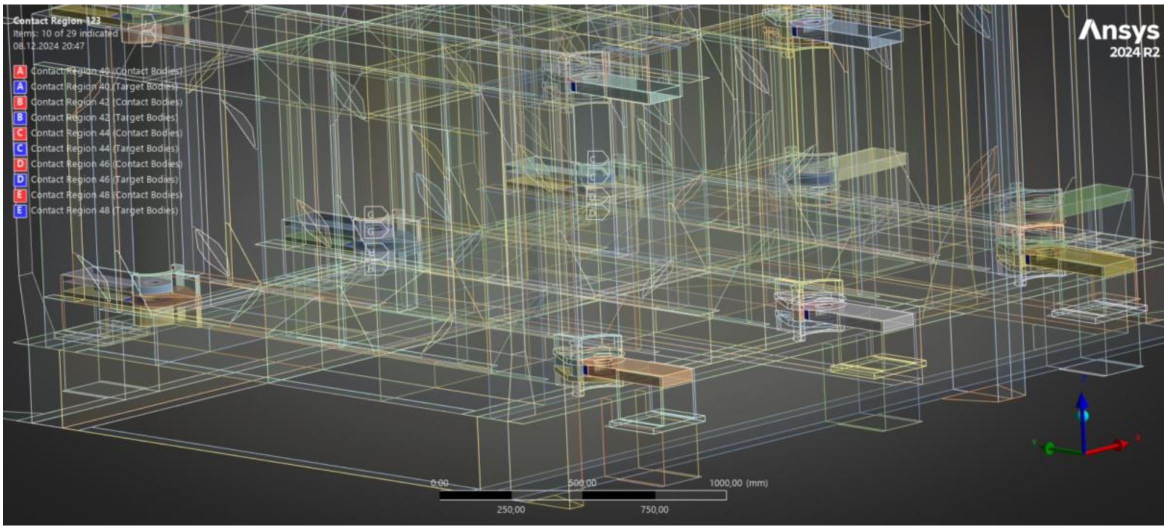
Picture 66: Contact regions.



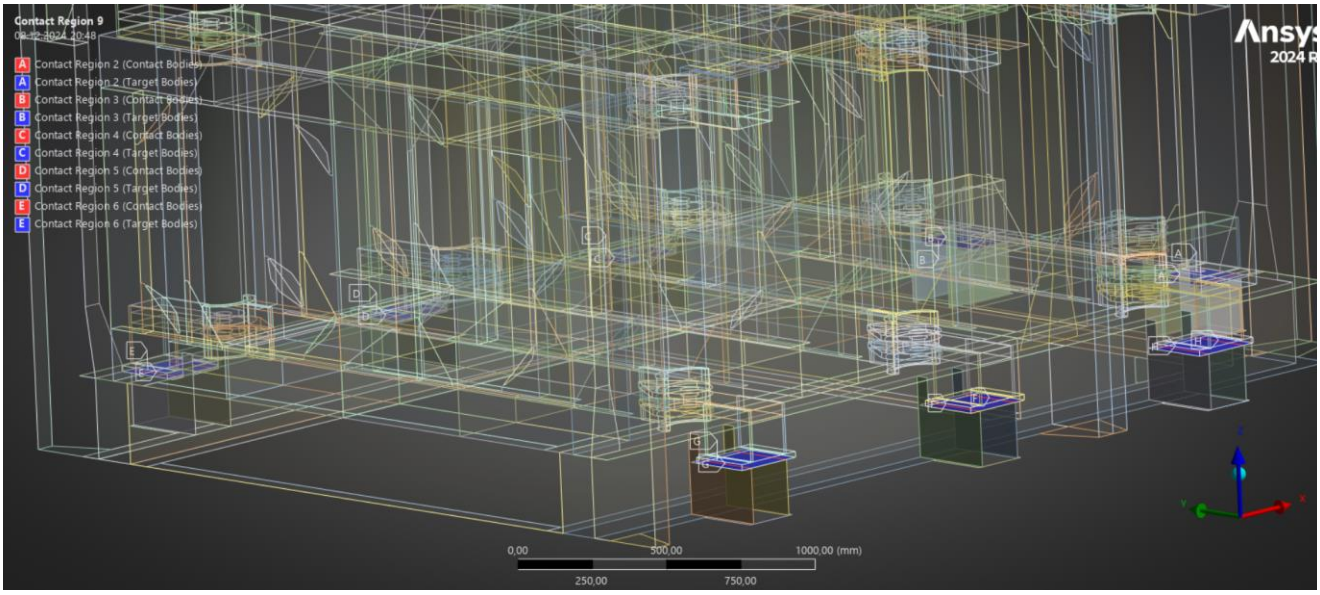
Picture 67: Contact regions.



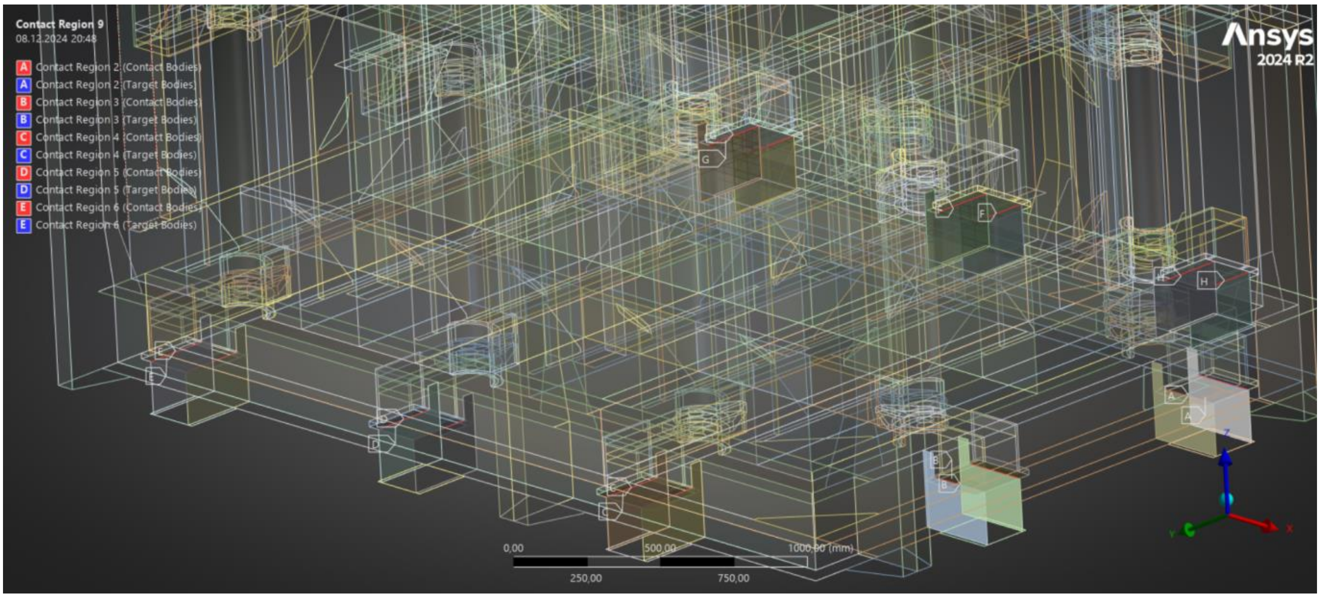
Picture 68: Contact regions.



Picture 69: Contact regions.



Picture 70: Contact regions.



Picture 71: Contact regions.

2.2.1.2 Mesh Generation

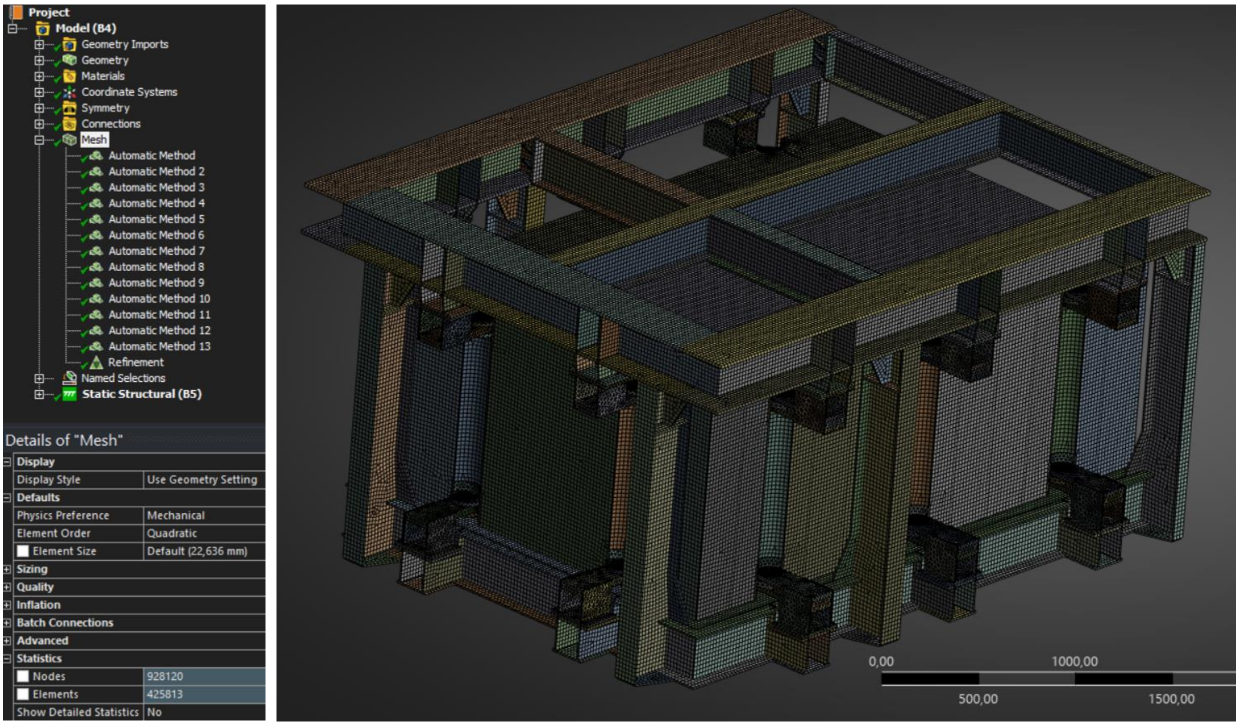
The mesh generation process for structural analysis is carried out similarly to the approach used for thermal analysis, as described in Section 2.2.1.2. However, specific attention is given to the mesh parameters to ensure the accuracy needed for structural evaluation under combined thermal and mechanical loads.

Mesh Preparation: The preparation of the mesh for structural analysis follows a similar approach to that used in the thermal model, where different groups of elements within the pontoon and conductor are selected, and named selections are created for effective mesh management. This segmentation ensures that high-stress areas or regions of complex geometry receive a more refined mesh, thus improving the accuracy of the analysis.

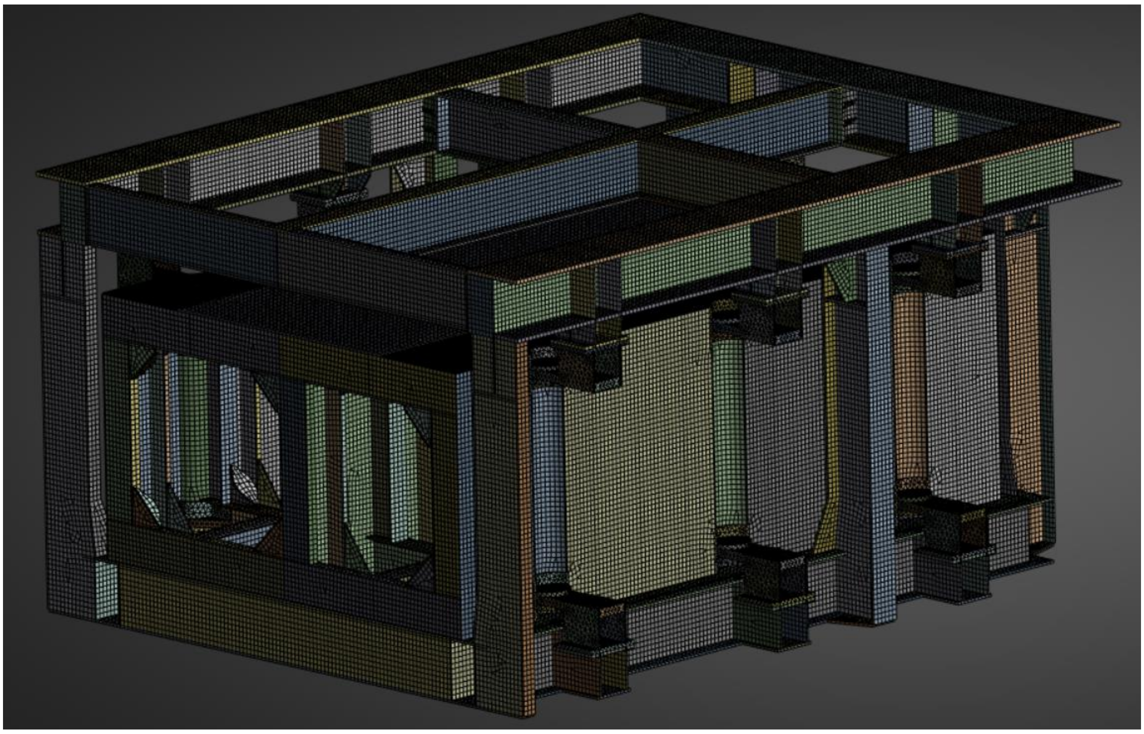
For critical regions, such as welded elements of the pontoon and conductor fixing elements, particularly around fixing holes, the face sizing parameter is applied to generate a finer mesh. This enhanced resolution is crucial for accurately capturing stress concentrations and deformations in these areas. Proper mesh refinement in regions of interest ensures that the structural behavior under mechanical loads is accurately modeled.

The element order used is quadratic for the structural analysis as well, allowing for better precision in capturing stress and deformation gradients. Quadratic elements, which include mid-side nodes, offer improved accuracy for simulations involving mechanical loading, providing a reliable representation of the structure's behavior during the welding process.

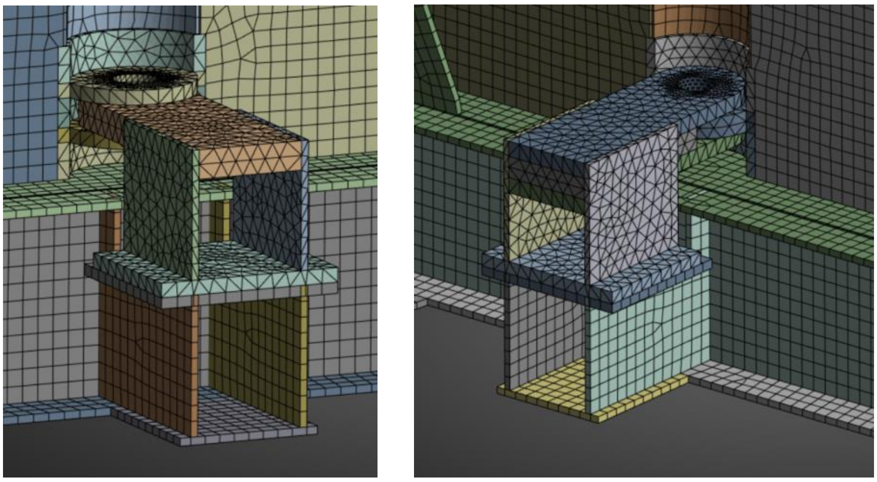
The total number of elements in the model is 425813.



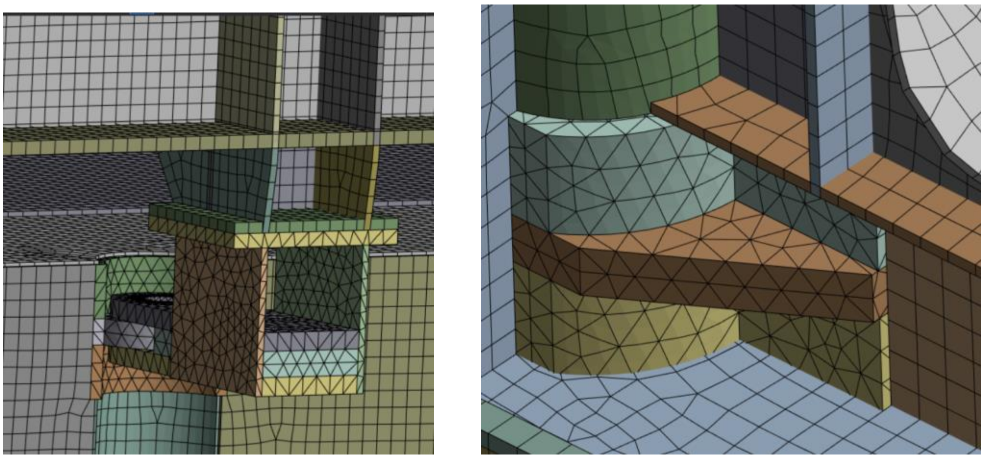
Picture 72: Characteristics of mesh components. Meshed elements of the structure.



Picture 73: Meshed elements of the structure.



Picture 74: Refined meshed elements of the structure.



Picture 75: Refined meshed elements of the structure.

2.3 Simulation Conditions

2.3.1 Thermal Boundary Conditions

This section describes the Thermal Boundary Conditions used in the analysis of the welding process for the pontoon structure. The welding sequence is a critical factor in determining the thermal and mechanical behavior during the welding process. For this study, certain assumptions have been made to simplify the welding sequence while ensuring that the analysis remains safe and conservative. Due to limited computational resources, the welding process has been modeled in a simplified manner, which still captures the essential aspects of thermal behavior and potential deformations.

Assumptions. In real-world conditions, welding is performed at a single point that gradually moves along the weld seam. As the welding point moves, previously welded regions begin to cool, resulting in a temperature gradient across the welded structure. Typically, no more than two elements are welded to the pontoon simultaneously, and often only one is welded at a time. However, for computational efficiency, the welding sequence in this study has been simplified.

The simplification involves selecting all the weld seams at once and applying a uniform heat flow, represented in watts, across all welding surfaces simultaneously. This approximation allows us to reduce the computational complexity while still ensuring that the maximum potential thermal effects are captured, providing a conservative assessment of the thermal loads and structural deformations that could arise during welding.

Manual arc welding is used as the welding technique, with specific welding parameters defined as follows:

- Electrode Diameter: 3.2 mm
- Current: 145 Amperes
- Voltage: 24 Volts
- Welding Speed: 7 cm/min
- Heat Transfer Efficiency Coefficient (η_t): 0.8

Using these parameters, the heat input can be calculated to determine the thermal flow applied during the simulation. The heat input (P) [W] is calculated as[8]:

$$P = \eta_t \cdot E \cdot I$$

Equation 6

Inserting the given parameters:

$$P = 0.8 \cdot 145 \cdot 24 = 2784\ [W]$$

Equation 7

Since the welding will be performed on both sides of the attached plate (T-welding), the resulting heat flow is doubled.

$$P_{Final} = 2784\ \cdot 2 = 5568[W]$$

Equation 8

This heat input uniformly distributed across the weld seams to simulate the heat generated during the welding process.

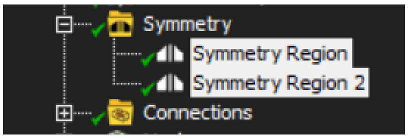
The assumption that all weld seams are heated simultaneously represents a worst-case scenario, where the maximum possible thermal load is considered. This ensures that any potential deformations or stresses are adequately captured, providing a safe margin for evaluating the structural integrity of the welded components.

In addition to the heat flow, *convection* is applied to all surfaces of the structure to simulate the cooling effect of ambient air. The convective boundary condition represents the heat loss from the structure to the surrounding environment, with an ambient temperature of 22°C. The convective heat transfer coefficient (Film coefficient) = $1{,}1e^{-5}\left[\frac{W}{mm^2 C}\right]$ is applied uniformly across all exposed surfaces, allowing the model to account for natural cooling as the welding progresses.

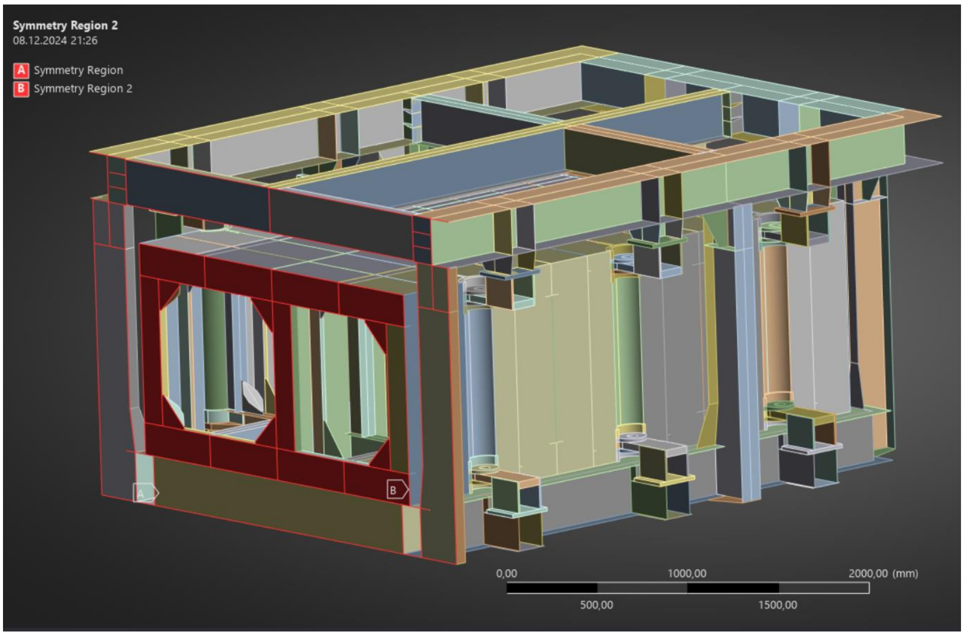
The heat input and cooling conditions are crucial for accurately simulating the thermal profile of the conductor and pontoon elements during welding. The thermal analysis results, including temperature distribution and heat-affected zones (HAZ), form the basis for the subsequent structural analysis, where the mechanical response to thermal loads is evaluated.

Boundary Conditions:

1. Symmetry:

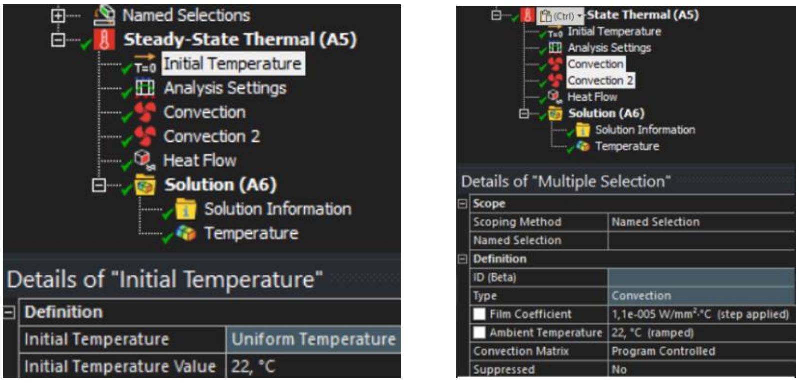


Picture 76: Symmetry

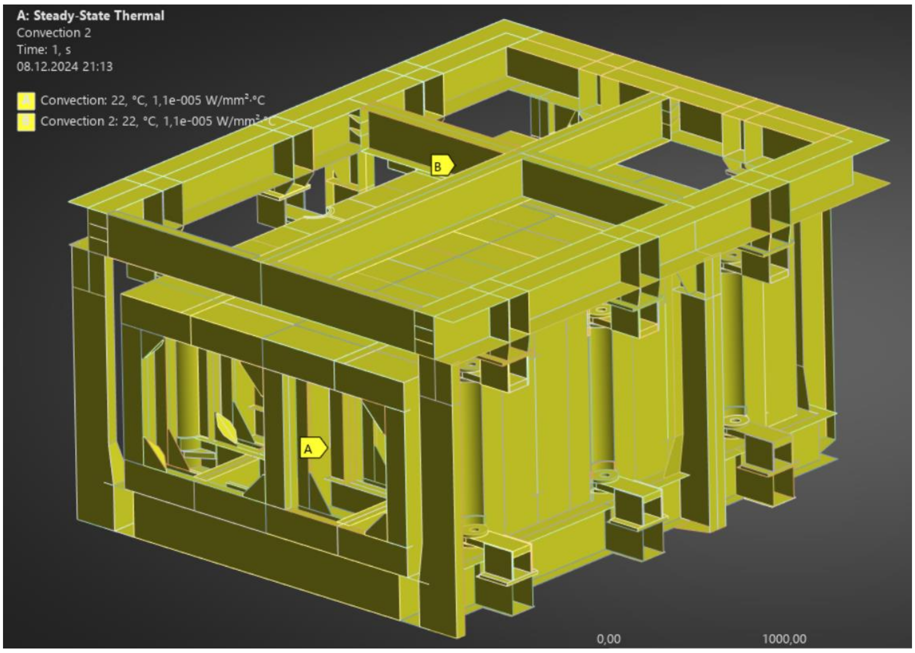


Picture 77: Planes and symmetry faces are shown in red.

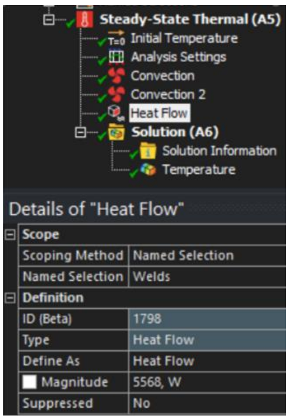
2. Thermal Boundary Conditions

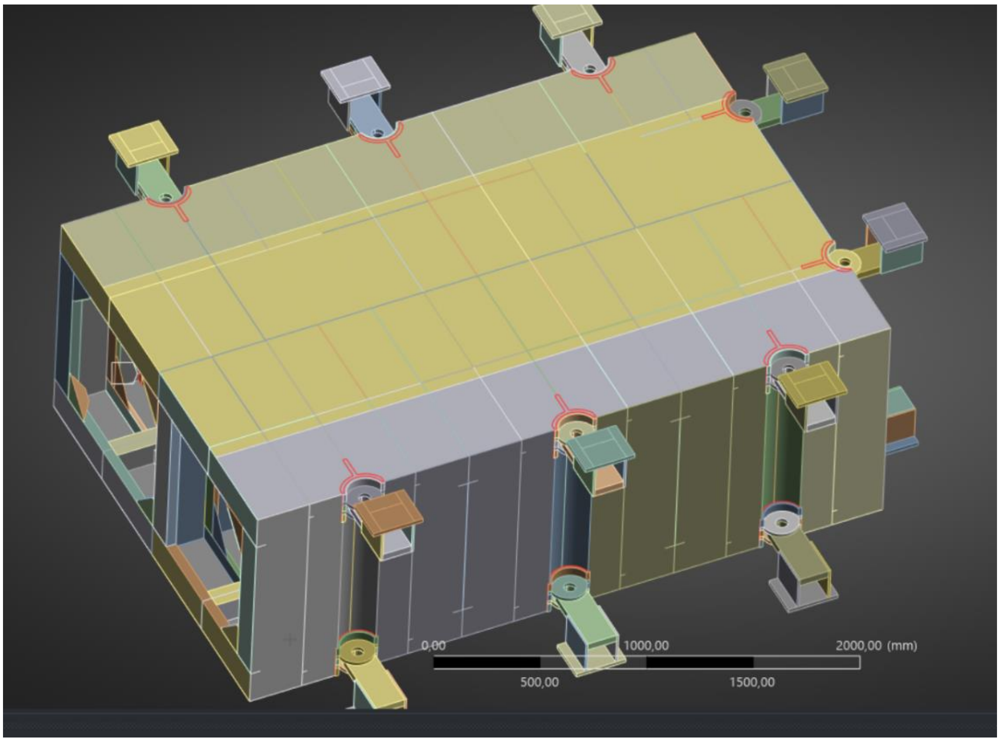


Picture 78: Initial temperature and environmental parameters (on the left), convection parameters (on the right).

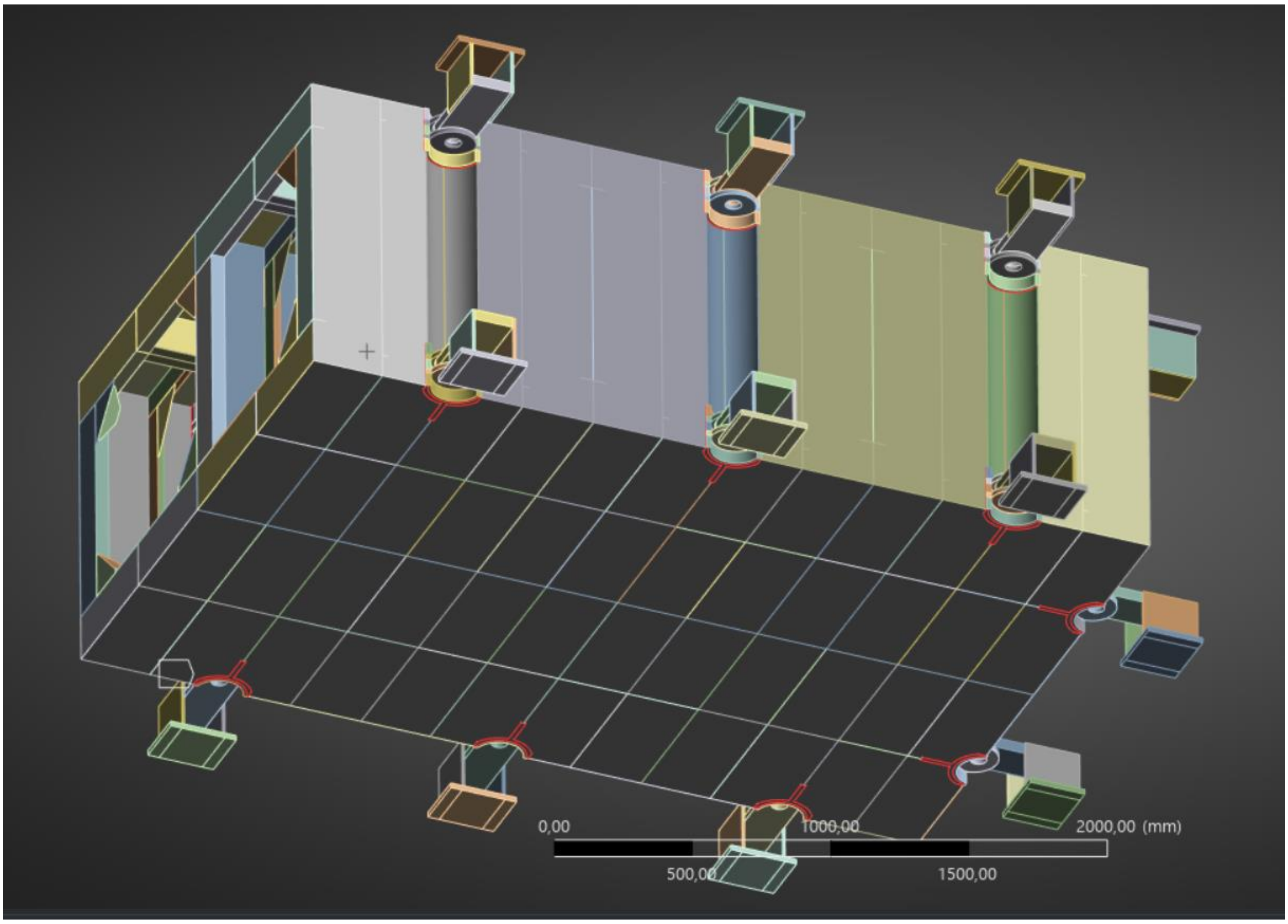


Picture 79: Surfaces with applied convection.

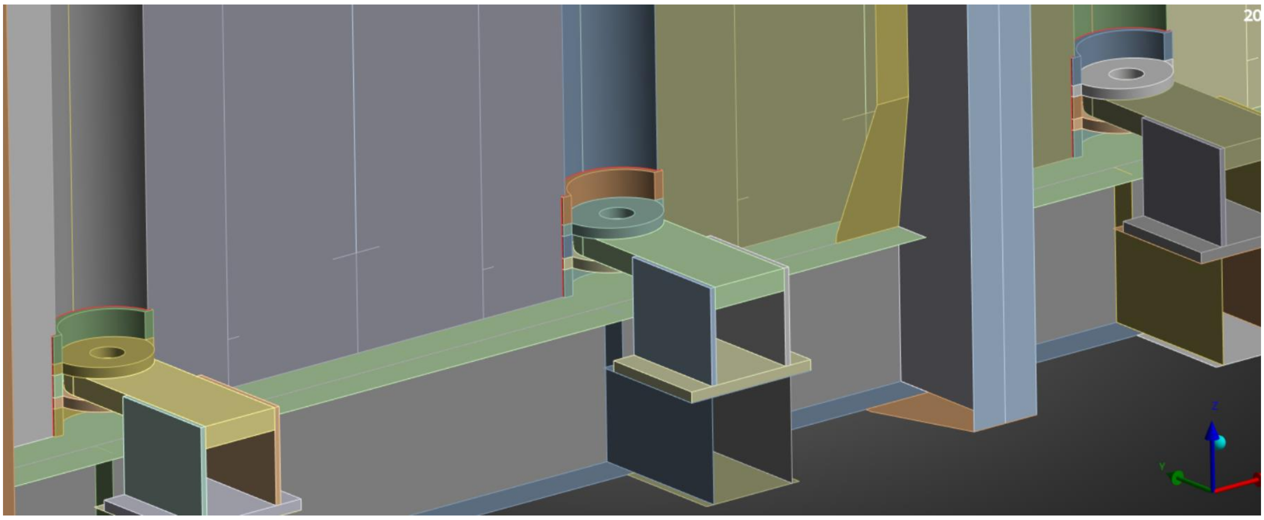




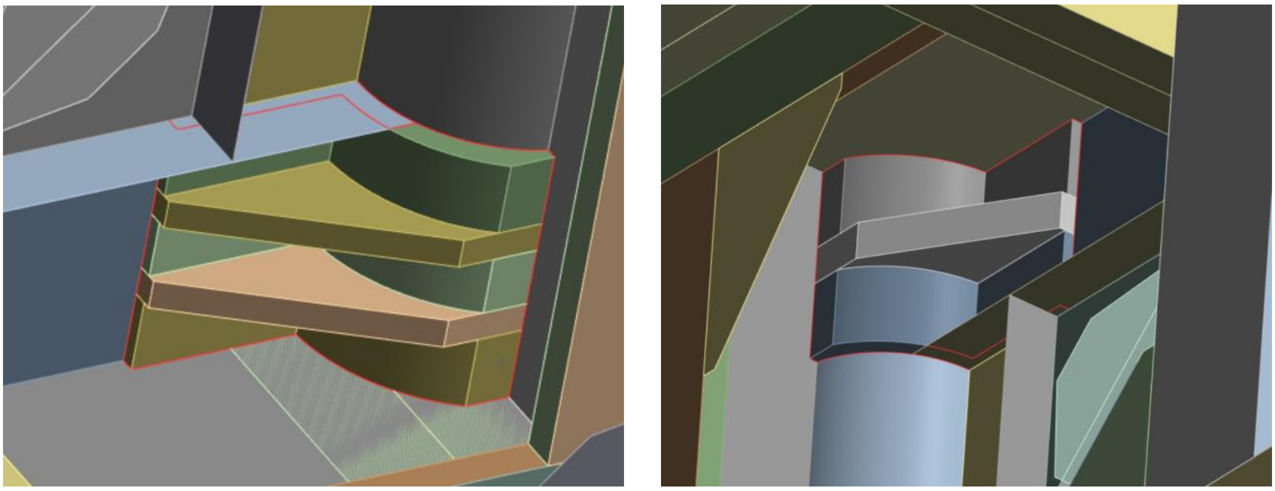
Picture 80: Heat flow parameters. Picture 81: The welded edges are shown in red.



Picture 82: The welded edges are shown in red.



Picture 83: The welded edges are shown in red.



Picture 84: The welded edges are shown in red.

2.3.2 Mechanical Boundary Conditions

The boundary conditions for the structural model include several key components:

Symmetry Planes and faces: Certain planes and surfaces are defined as symmetry boundaries to reduce computational effort while maintaining accuracy in the model. Symmetry conditions are applied to appropriately reflect the behavior of the structure, particularly for elements that are mirrored across certain axes.

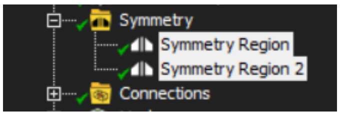
Fixed Supports: Rigid support conditions are applied to the lower surfaces and edges of the conductor. These fixed supports eliminate any potential movement in the specified areas, representing the real-world scenario where the structure is restrained and cannot shift during the welding process. By restricting all degrees of freedom at these locations, the model accurately simulates the effects of supports and fixtures that hold the structure in place during fabrication.

Gravitational Load: All components of the structure are assigned gravitational loading. This load accounts for the weight of the conductor and pontoon elements, which contributes to the overall stress state during welding. The gravitational effect is particularly important in ensuring that the structural analysis accounts for any deformation or stress that could result from the weight of the components.

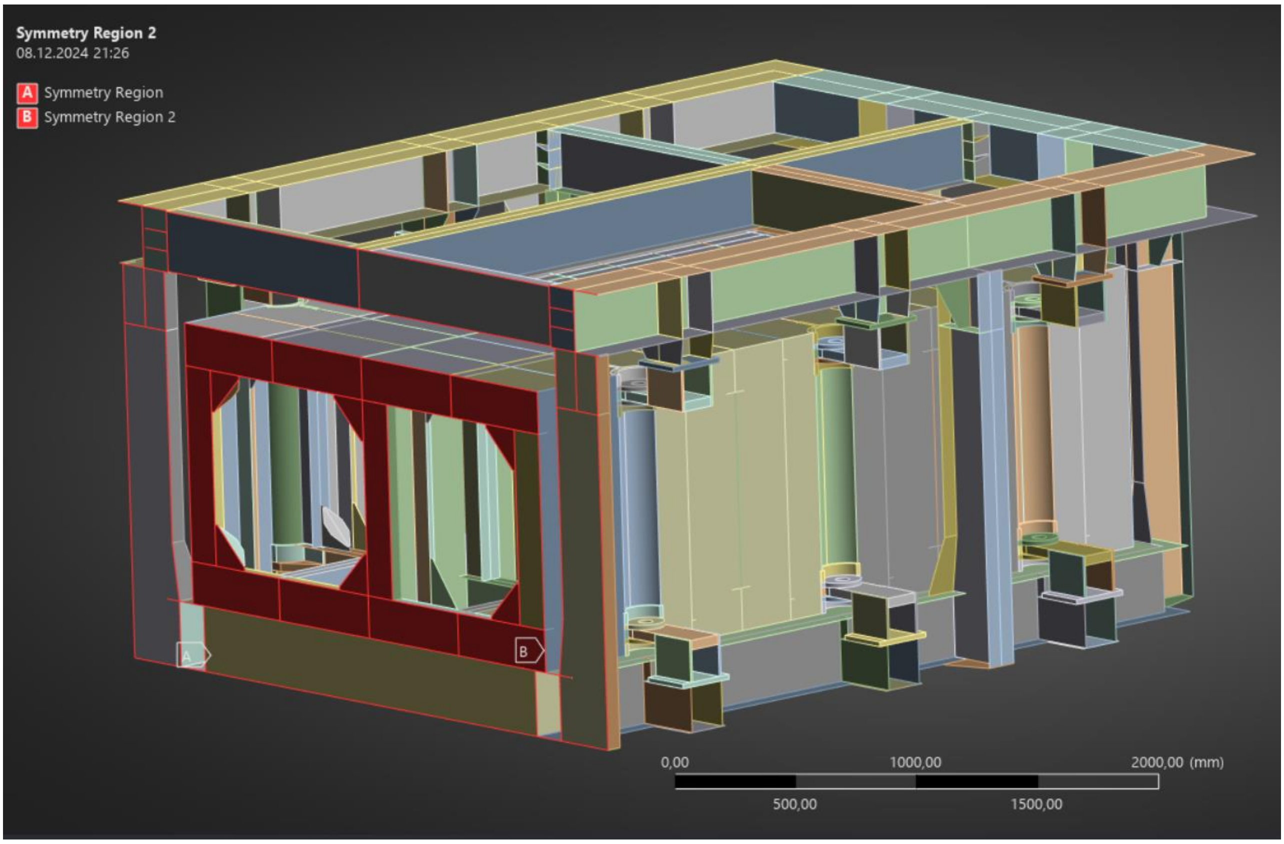
Thermal Load: The results from the thermal analysis, which include the calculated temperature distribution, are imported and used as a thermal load in the structural analysis. This thermal load is essential for capturing the stress and deformation effects that arise due to the heating and cooling cycle experienced by the structure during welding. The interaction between thermal and mechanical loads is a critical factor in determining the overall behavior of the welded components.[9]

Boundary Conditions:

1. Symmetry :

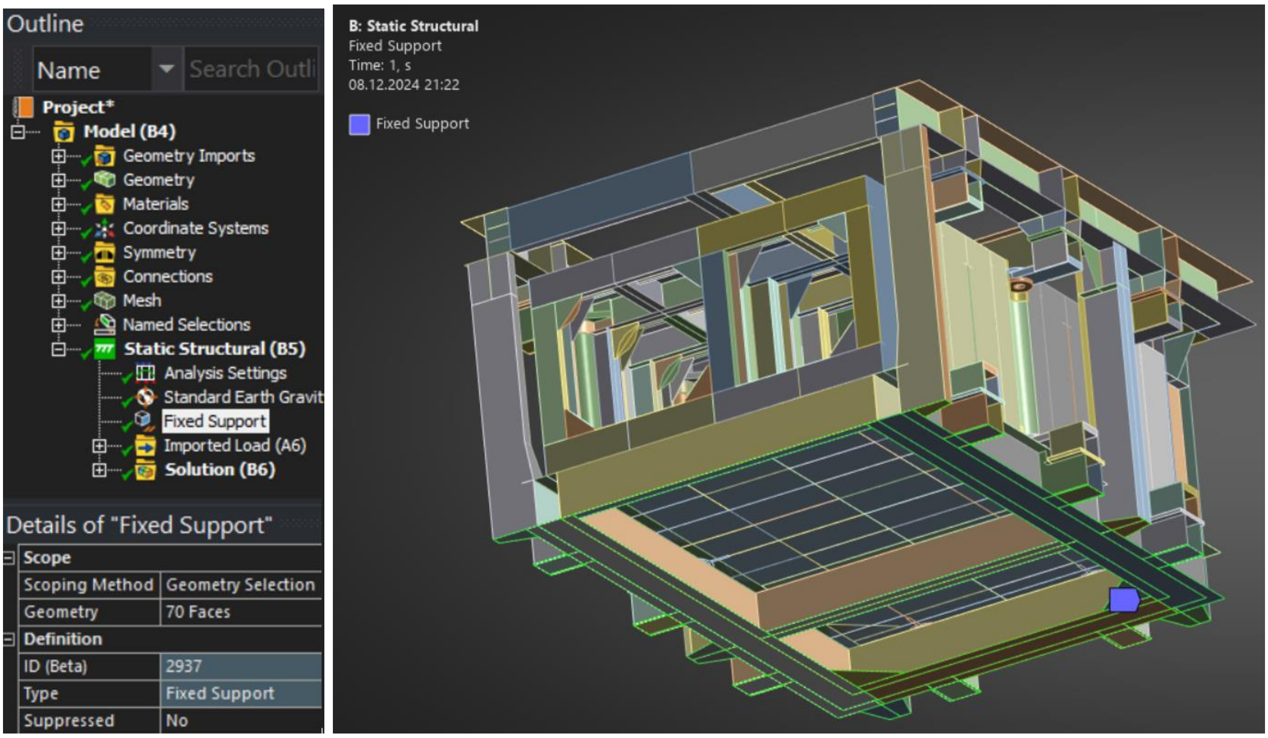


Picture 85: Symmetry



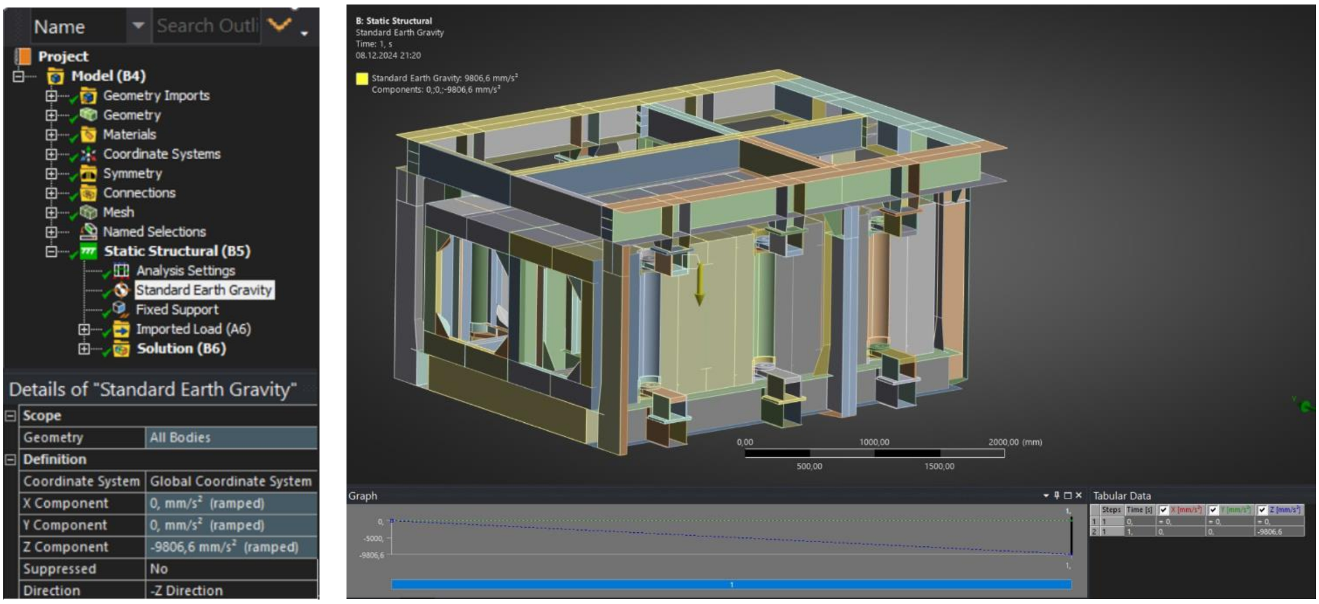
Picture 86: Planes and symmetry faces are shown in red.

2. Fixed Supports:



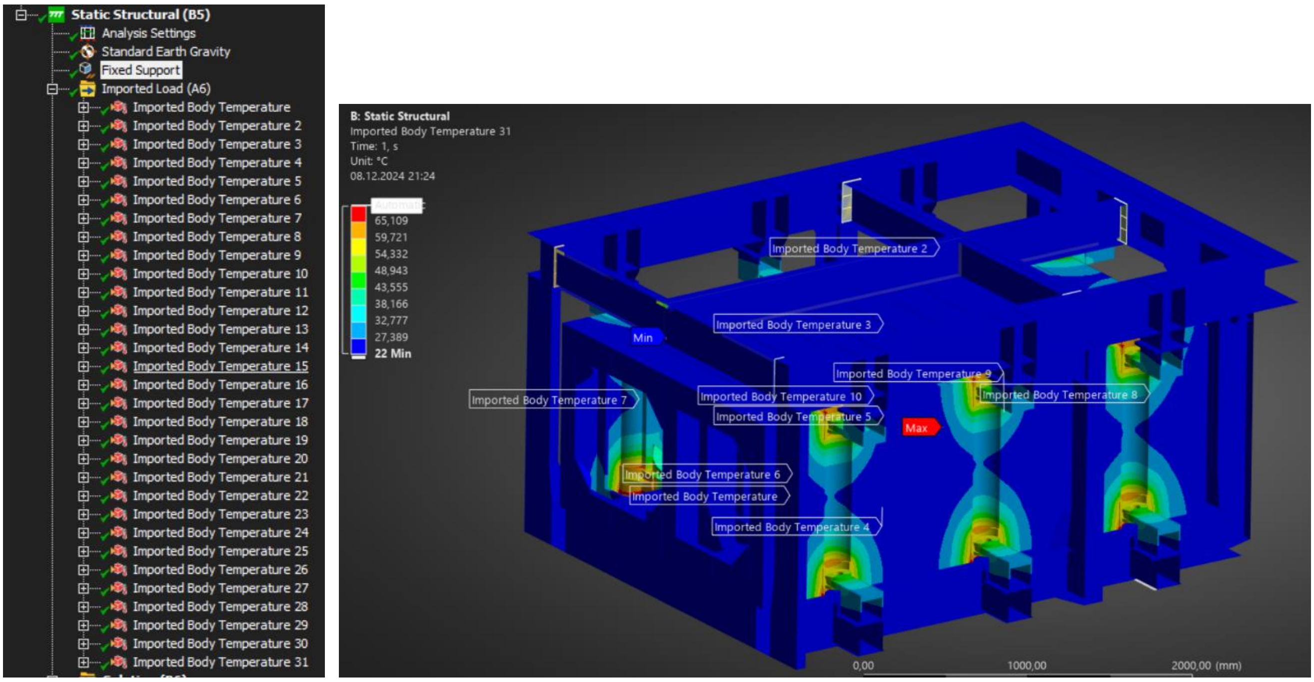
Picture 87: The area of application of the Fixed Supports.

3. Gravitational Load

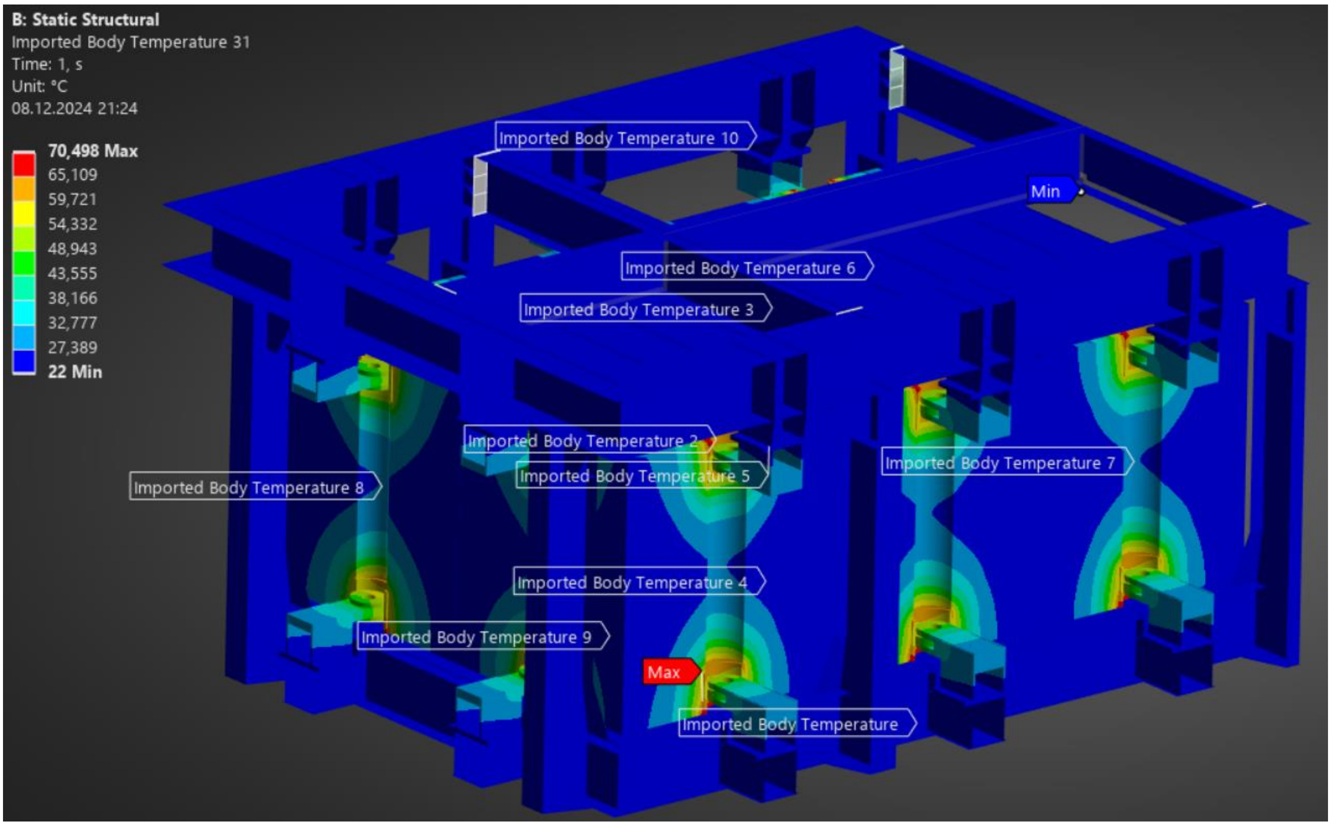


Picture 88: Application of the Gravity force.

4. Thermal Load:



Picture 89: Thermal loads imported from the thermal analysis.



Picture 90: Thermal loads imported from the thermal analysis.

					19.ДРМ.135.6116м.ПЗ.03
Cha.	*Page*	*№ docum.*	*Signature*	*Data*	

				Results of thermal and stress-strain analysis	Liter.	*Page*	*Pages*
Student	Grechyn V.Y.						
Consultant	Korostylyov L.I.				NUS		
Supervisor	Korostylyov L.I.						
Head of Dep.	Korostylyov L.I.						

3 RESULTS OF THERMAL AND STRESS-STRAIN ANALYSIS

3.1 Thermal Analysis Results

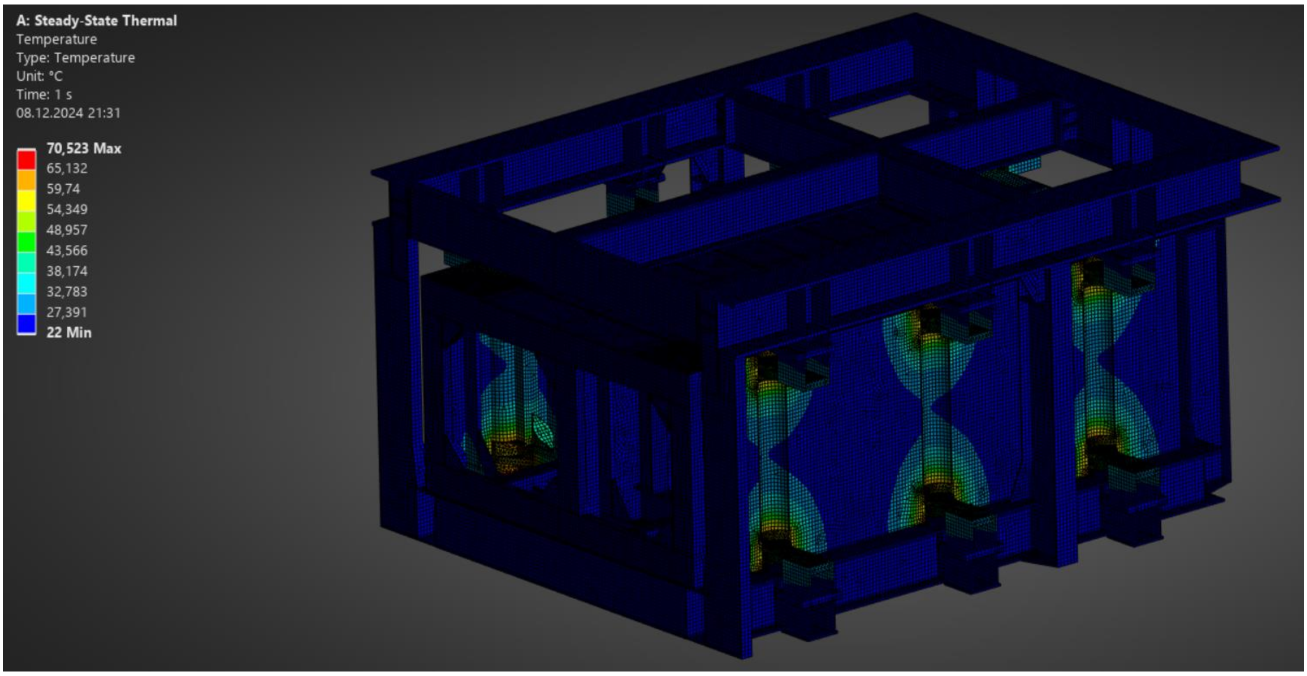
3.1.1 Temperature Distribution

The results of the thermal analysis provide a comprehensive understanding of the temperature distribution across the conductor and pontoon structures. The temperature distribution is a crucial factor in assessing the thermal loads experienced by different parts of the structure during welding. The data obtained from this analysis reveals the regions that are subjected to the highest and lowest temperatures, which is critical for identifying heat-affected zones (HAZ) and evaluating the impact of thermal stresses on the overall structural integrity.

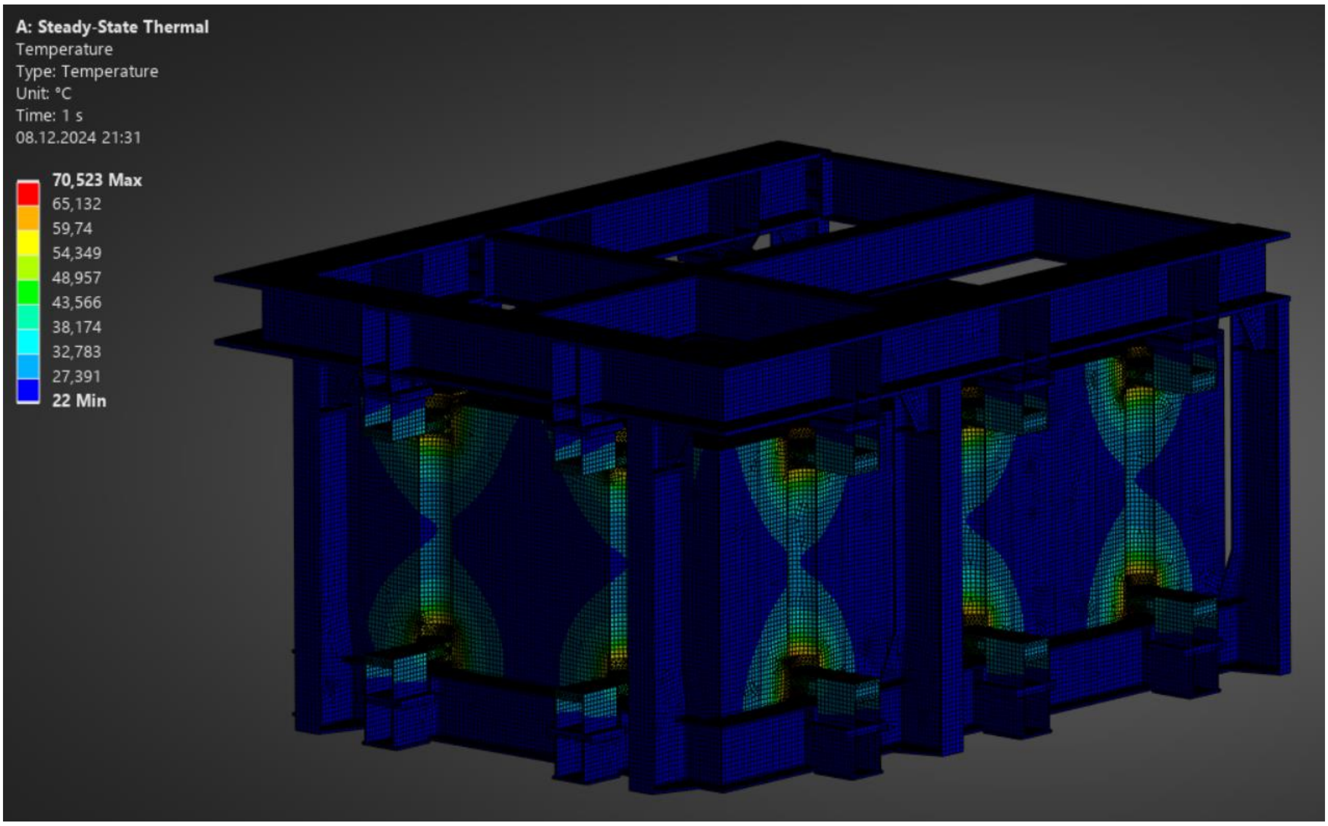
The thermal analysis indicates that the areas directly adjacent to the weld seams experience the highest temperatures due to direct exposure to the welding heat flux. As the distance from the weld seam increases, the temperature gradually decreases, forming a temperature gradient across the structure. This gradient is essential for understanding how heat propagates through the conductor and pontoon components and for evaluating the subsequent cooling behavior of these elements.

Regions of maximum temperature are typically located along the weld seams, where the heat input is concentrated. These areas are most susceptible to thermal expansion and potential changes in material properties due to prolonged exposure to elevated temperatures. In contrast, regions of minimum temperature are found further away from the weld seams, where the cooling effect of ambient air is more pronounced. This temperature difference creates non-uniform thermal expansion, which can lead to residual stresses and deformations in the structure.

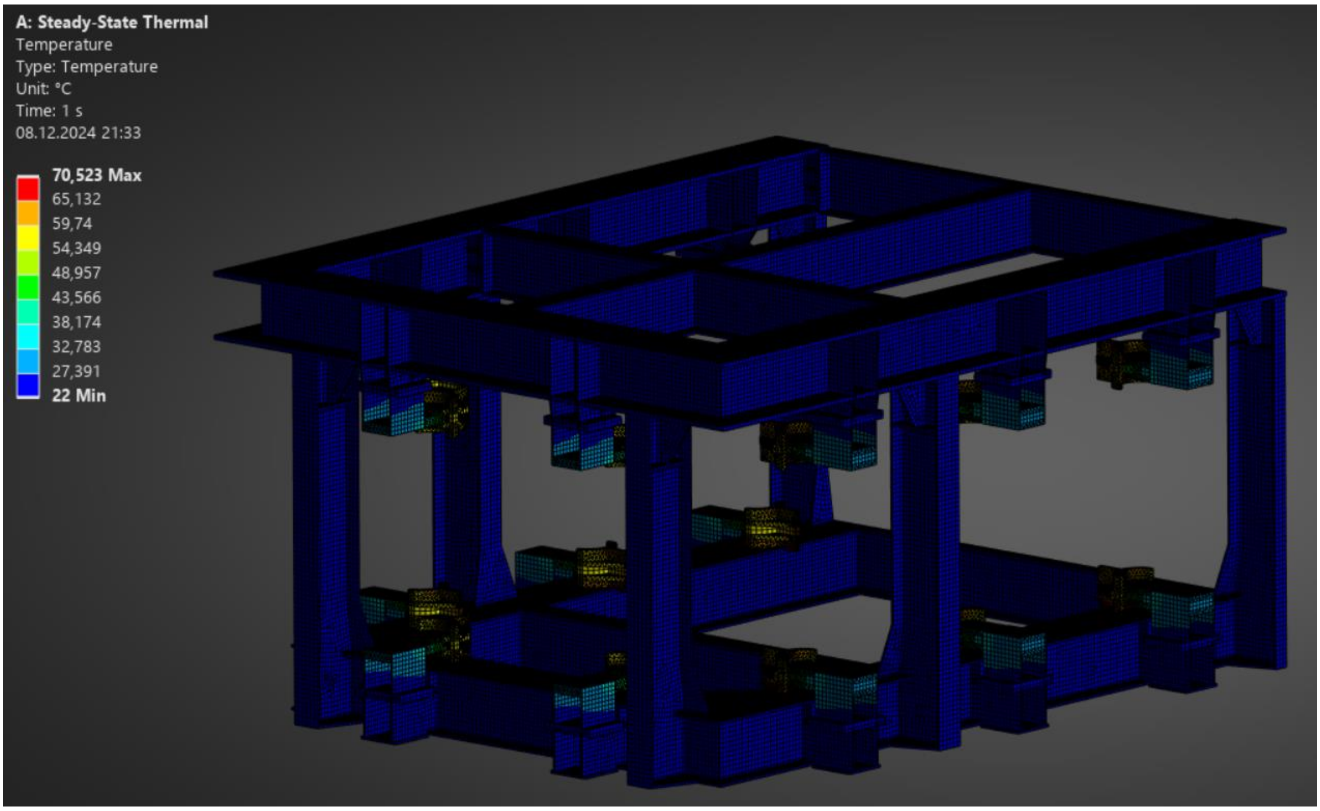
The analysis also highlights the heat-affected zones (HAZ), which are critical areas where the material properties may change due to the influence of welding heat. The HAZ is characterized by significant thermal gradients, which can lead to variations in material hardness and strength, potentially affecting the overall performance of the welded structure. Understanding the extent and characteristics of the HAZ is essential for predicting the mechanical behavior of the conductor and pontoon under operational loads.



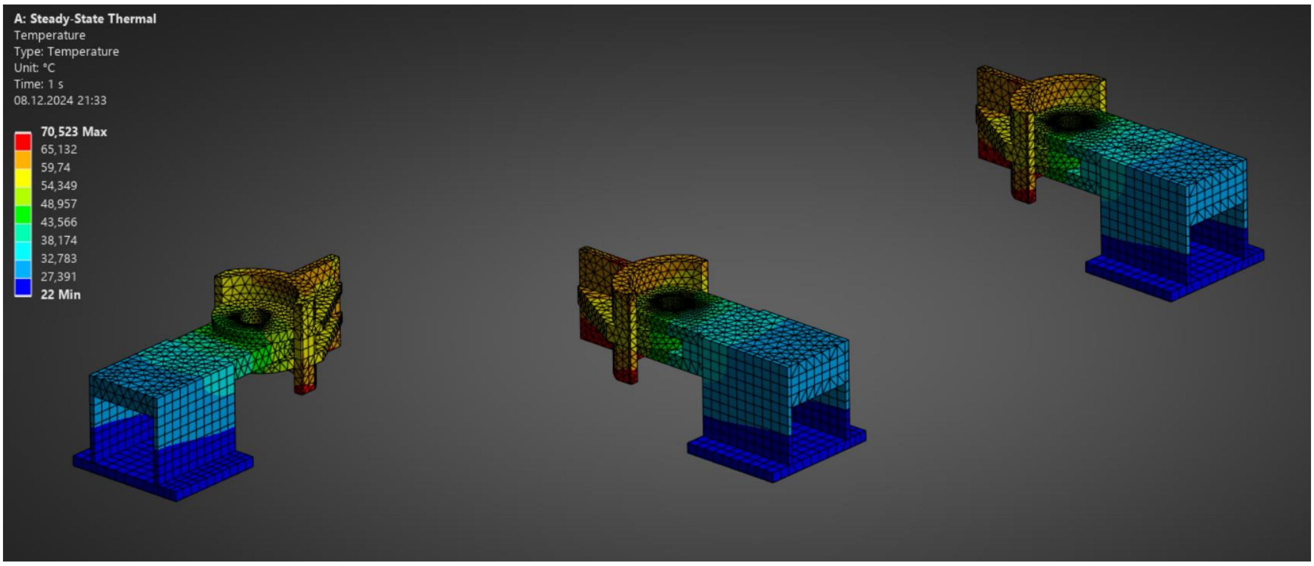
Picture 91: Temperature distribution across the pontoon and conductor structures.



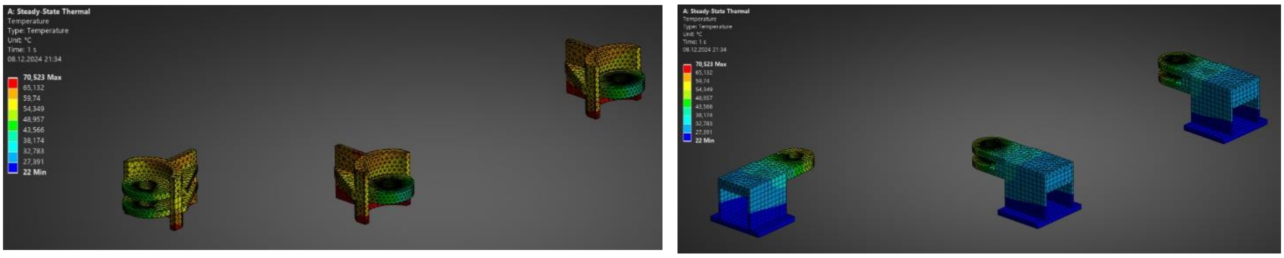
Picture 92: Temperature distribution across the pontoon and conductor structures.



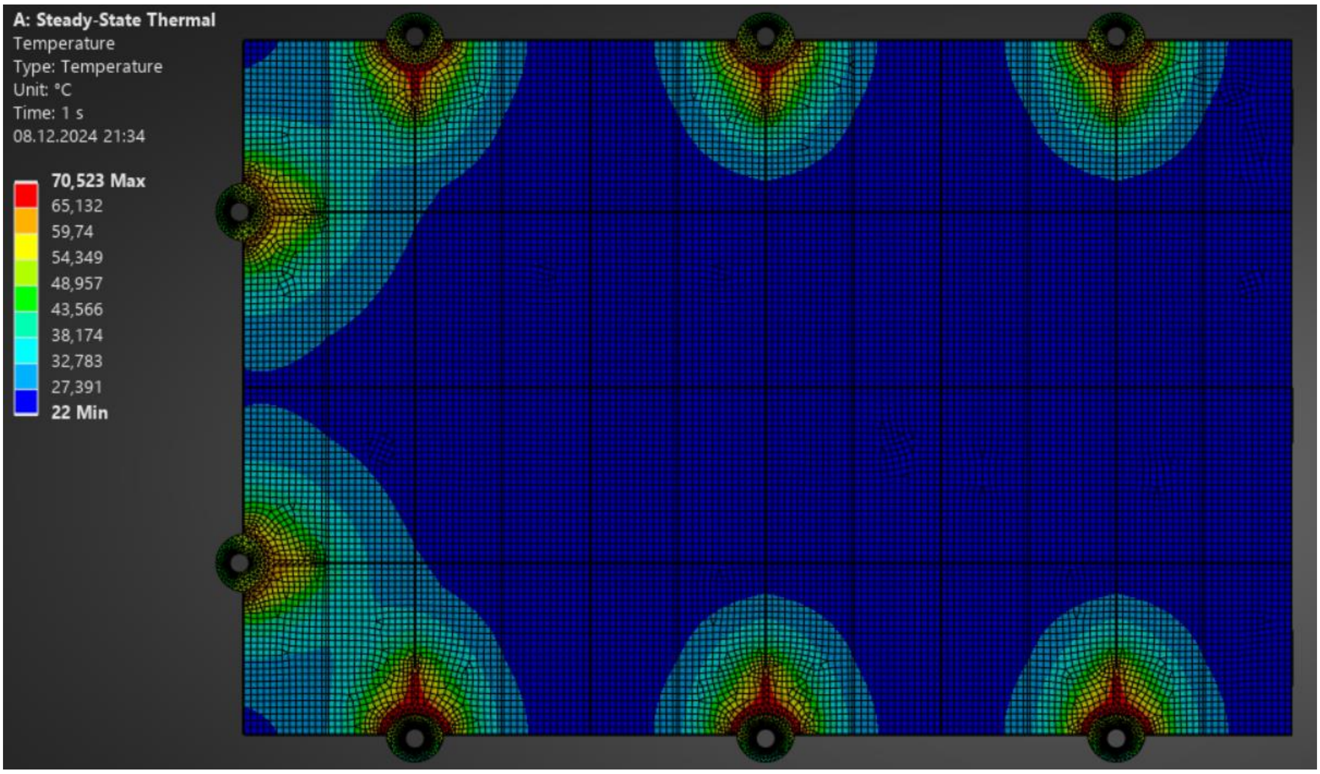
Picture 93: Temperature distribution across the conductor structures.



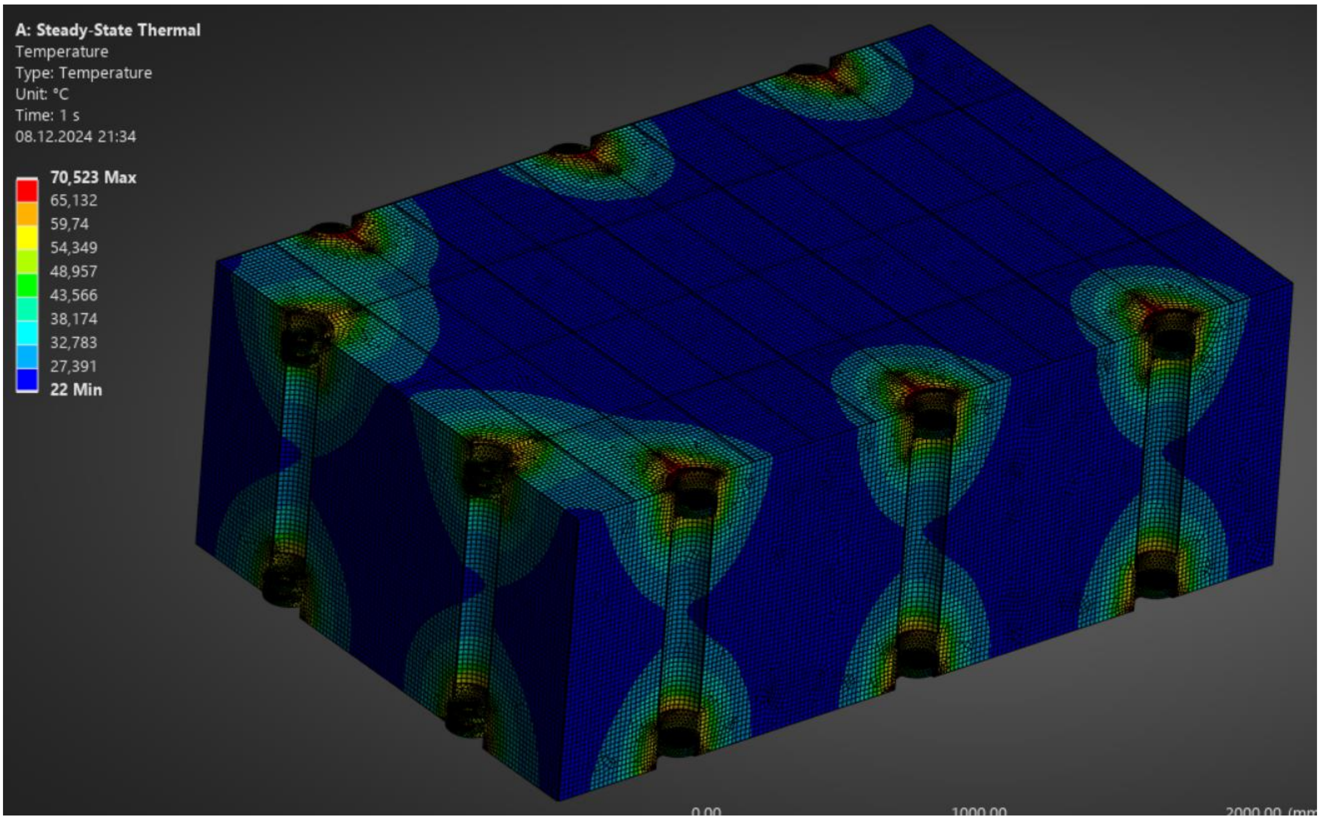
Picture 94: Temperature distribution across some element’s structures.



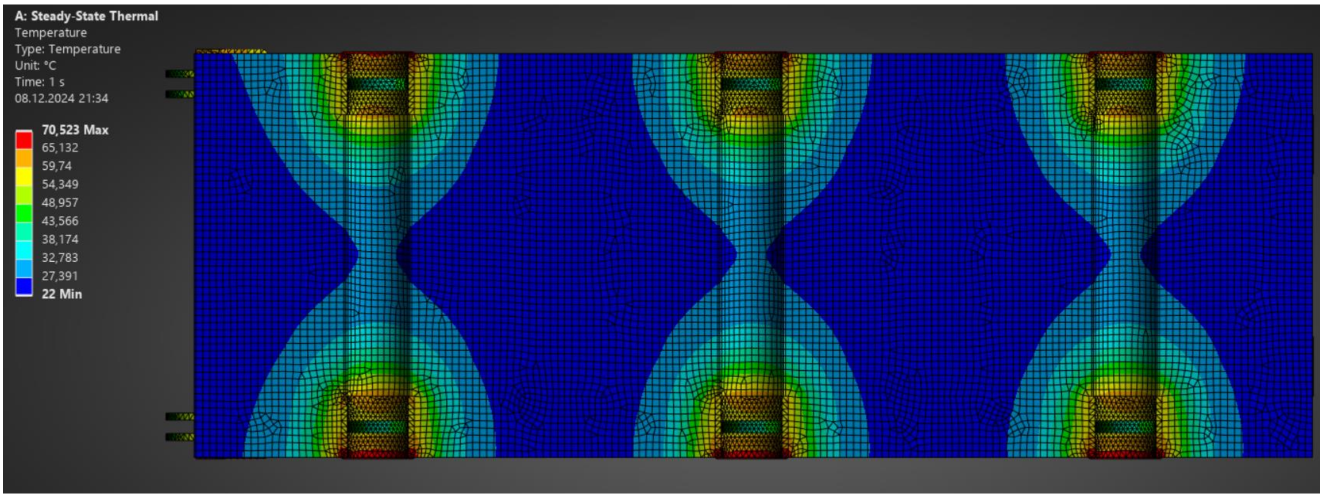
Picture 95: Temperature distribution across some element’s structures.



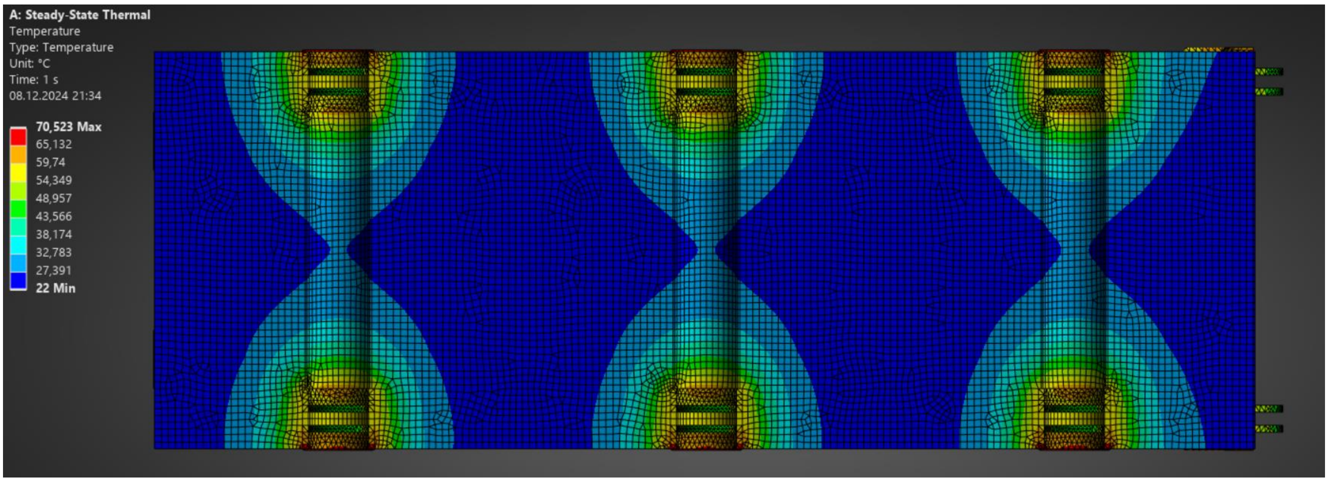
Picture 96: Temperature distribution across the pontoon structures.



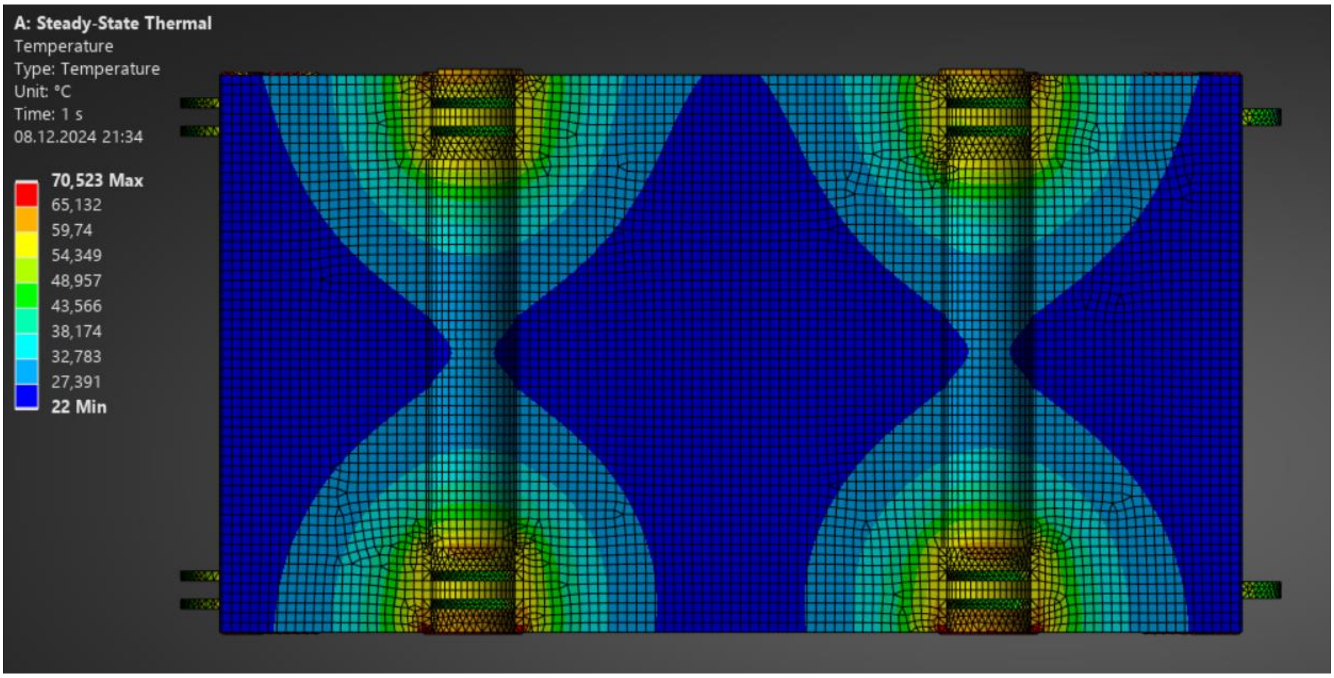
Picture 97: Temperature distribution across the pontoon structures.



Picture 98: Temperature distribution across the pontoon structures.



Picture 99: Temperature distribution across the pontoon structures.



Picture 100: Temperature distribution across the pontoon structures.

3.2 Mechanical Analysis Results

3.2.1 Stress-Strain Distribution

The results of the mechanical analysis provide insight into the distribution of stress and strain across the conductor and pontoon structures under the combined effects of thermal loads and gravitational forces. The mechanical analysis is crucial for understanding how these structures respond to the thermal effects generated during welding, as well as the self-weight of the components.

The data from the mechanical analysis reveal that the maximum deformations occur at critical areas of the structure, reaching 0,237 mm at specific locations of interest. These deformations are largely a result of the temperature gradients established during the welding process, which lead to thermal expansion and contraction of the materials. The areas experiencing the highest temperatures, as identified in the thermal analysis, are also those most prone to deformation due to the thermal stress generated.

In addition to deformation, the analysis also identifies regions of maximum stress, with values reaching up to 416,7 MPa. These stress concentrations are primarily found near the weld seams, where the thermal load and mechanical constraints interact to create significant internal stresses.

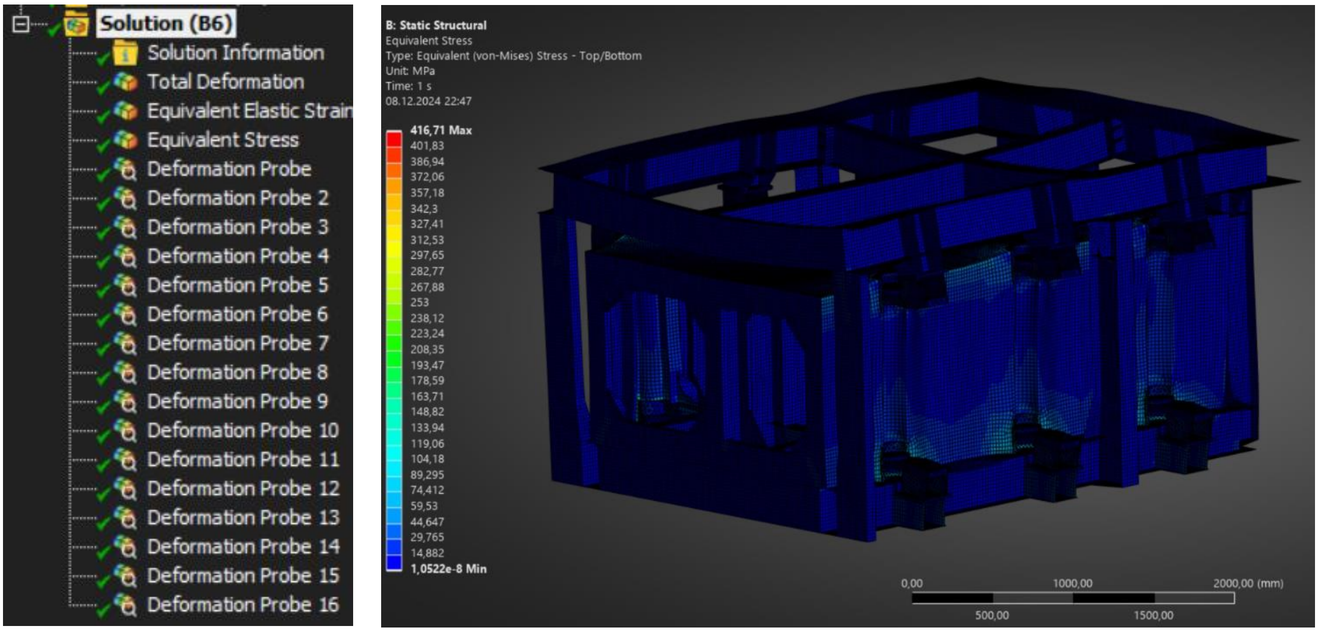
Minimum [MPa]	Maximum [MPa]	Average [MPa]
1,05E-08	416,71	9,002

Table 1: Maximum - Minimum stress.

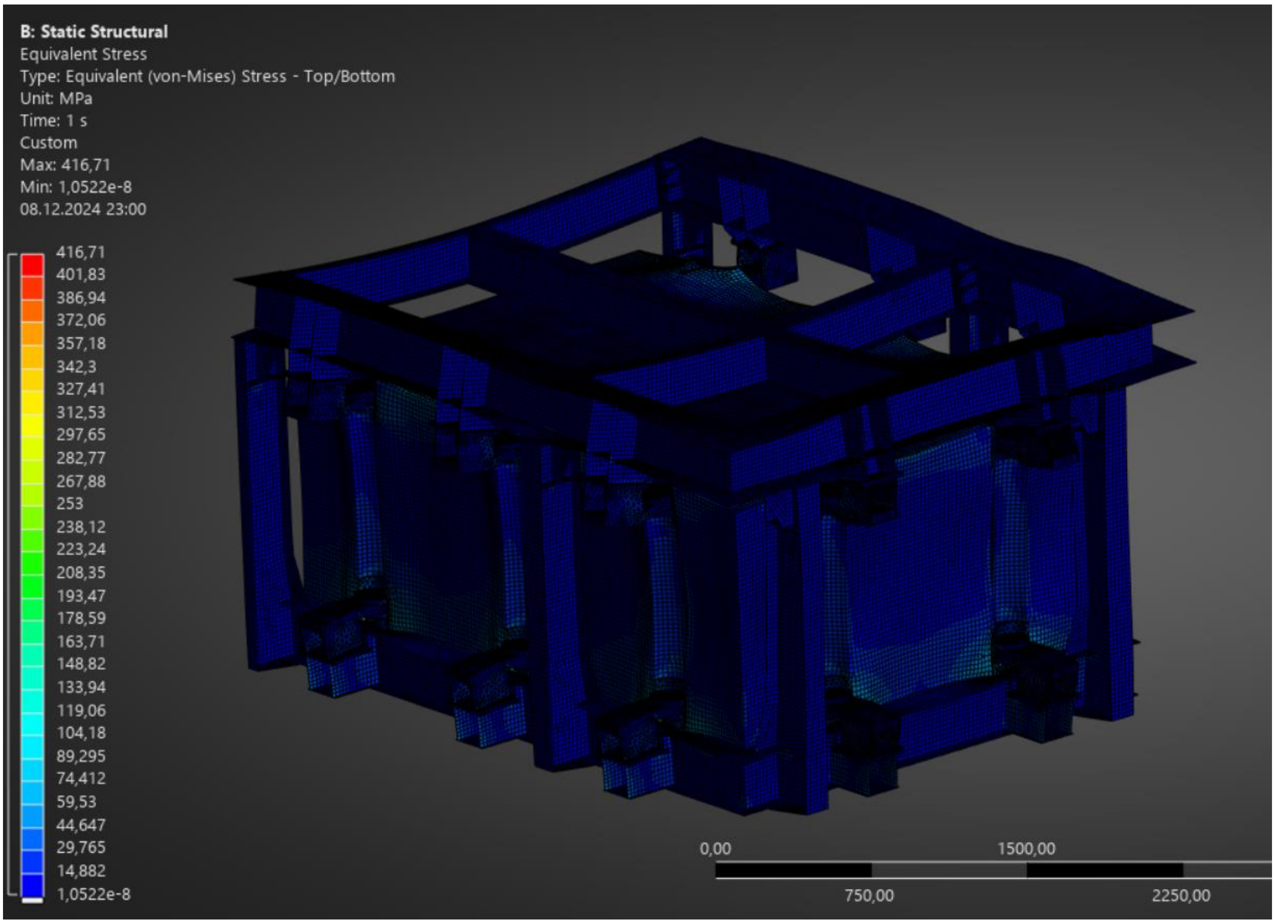
The stress value of 416.7 MPa significantly exceeds the yield strength of the structural material, which is 250 MPa. As shown in Figure 105, the areas with such high stress values are extremely small and localized near certain mesh nodes. The image also shows that the stress value at nodes adjacent to the one with the peak stress value drops sharply to 57 MPa and below. This abrupt change in these points is associated with mesh imperfections in these areas. However, even if such localized stresses were realistic, this small volume of material would deform, and the stress field in this region would normalize and align with the surrounding points.

It should also be noted that these zones belong to elements of the part of the pontoon structure that is not the focus of our study, and they can therefore be disregarded.

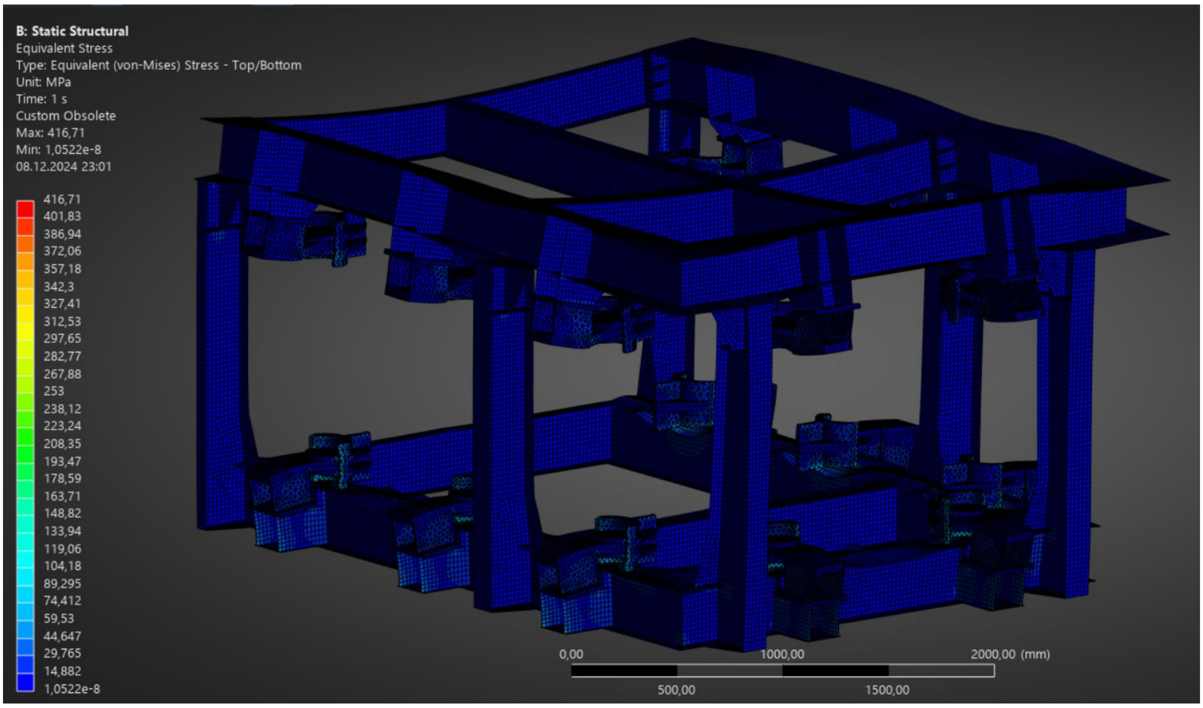
The analysis of stress-strain distribution is vital for identifying potential failure points and for making informed decisions regarding the design and reinforcement of the welded structure. Understanding the interplay between thermal and mechanical loads allows for a more accurate prediction of how the structure will behave under operational conditions, thus aiding in the development of effective mitigation strategies for stress-related failures.



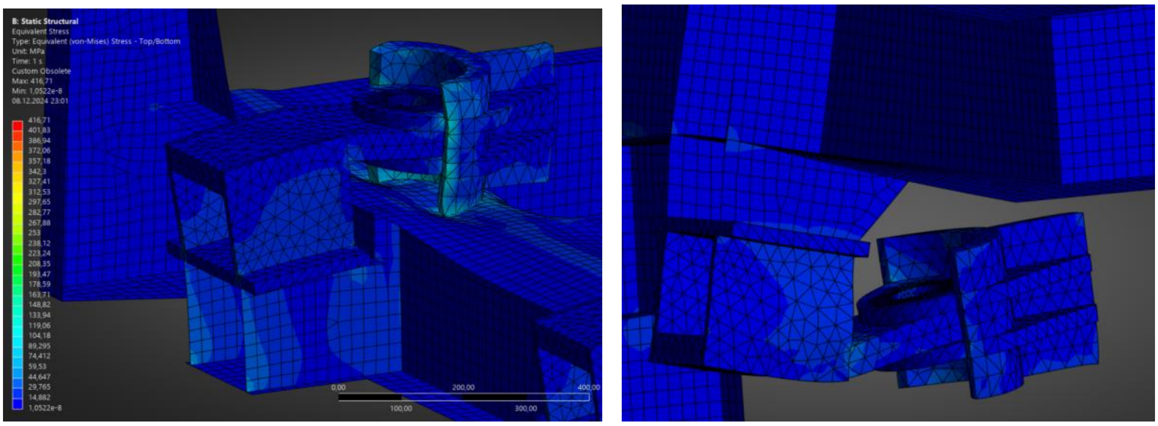
Picture 101: Stress distribution across the pontoon and conductor structures.



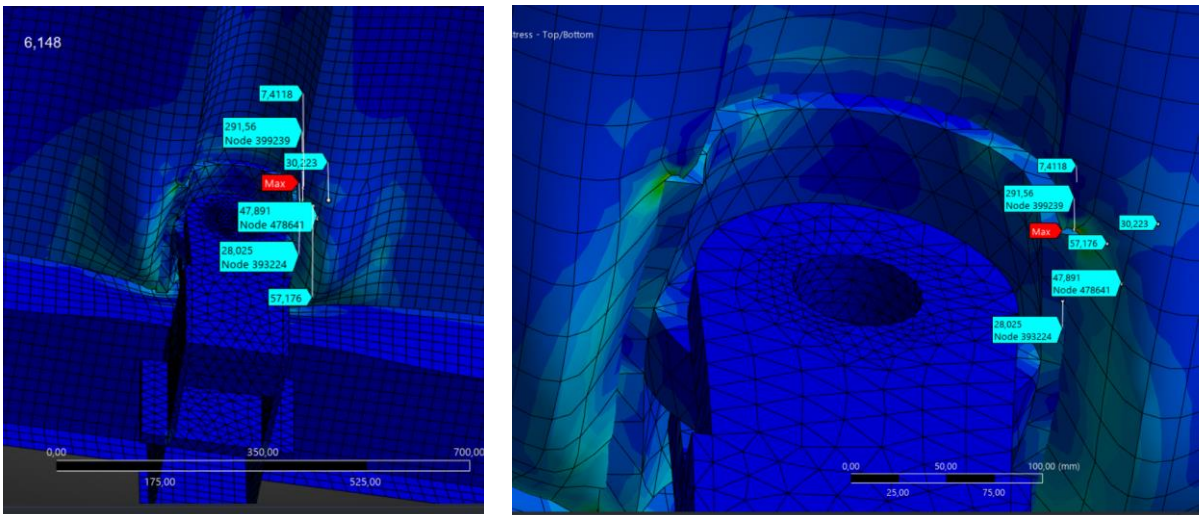
Picture 102: Stress distribution across the pontoon and conductor structures.



Picture 103: Stress distribution across the conductor structures.



Picture 104: Stress distribution across the conductor structures.



Picture 105: Stress distribution across the conductor structures.

3.2.2 Deformation Assessment: Exceeding Tolerance Limits

The deformation assessment of the conductor and pontoon structures is crucial to ensure that the welded components meet the specified tolerance limits. The results of the mechanical analysis show that the maximum deformation is 0.307 mm, which is below the specified limit. However, this value represents the maximum for the entire structure, while our study focuses on the deformation of the welded components.

To determine the deformation values in the elements of interest, “deformation probes” were added in these areas, which provided deformation values obtained from the calculations. These values are presented in Table 3. As seen in the table, the maximum deformation in the areas of interest is 0.273 mm, which is almost half the allowable value of 0.5 mm. This indicates that the structure complies with the established requirements and does not exceed the permissible deformation limits, thus ensuring its reliability and stability under operational conditions.

The deformation assessment is conducted by analyzing the displacement data obtained from the structural analysis, which takes into account both the thermal load (calculated during the thermal analysis) and the gravitational load acting on the structure. The results demonstrate that the deformation experienced by the conductor and pontoon elements remains well within acceptable limits, even in regions subjected to high temperatures and stress concentrations. This is an important outcome as it validates the effectiveness of the design in maintaining deformation within permissible limits during and after the welding process.

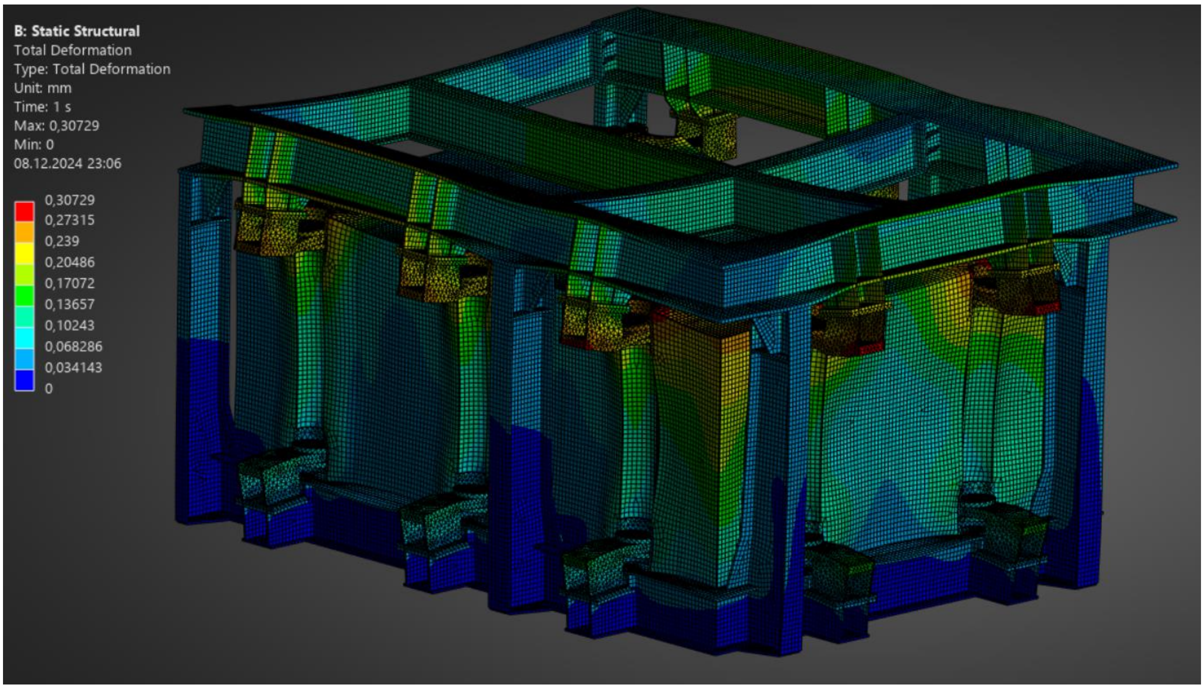
Minimum, [mm]	Maximum, [mm]	Average, [mm]
0	0,307	0,105

Table 2: Maximum - Minimum Deformations.

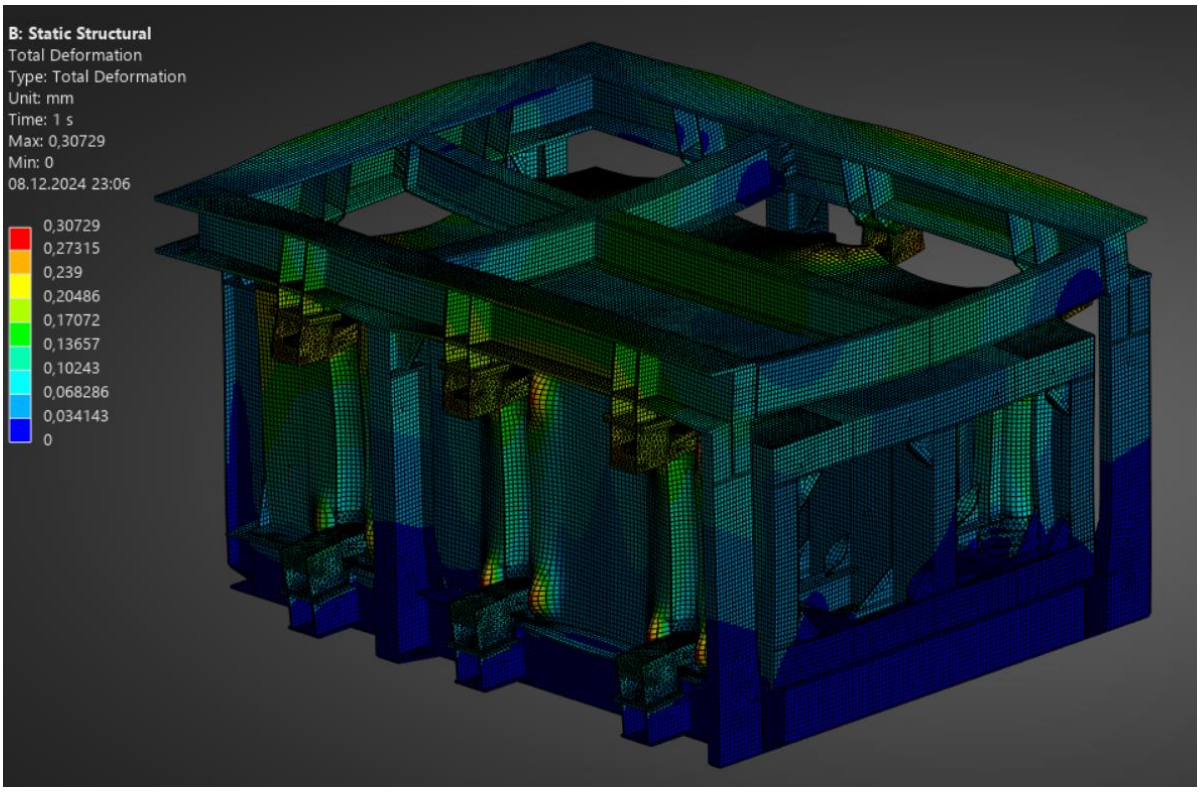
Probe N	1	2	3	4	5	6	7	8	9
Deformations, mm	0,044	0,045	0,056	0,068	0,068	0,064	0,053	0,052	0,202
Probe N	10	11	12	13	14	15	16		
Deformations, mm	0,203	0,218	0,237	0,233	0,207	0,187	0,185		

Table 3: Deformation probes.

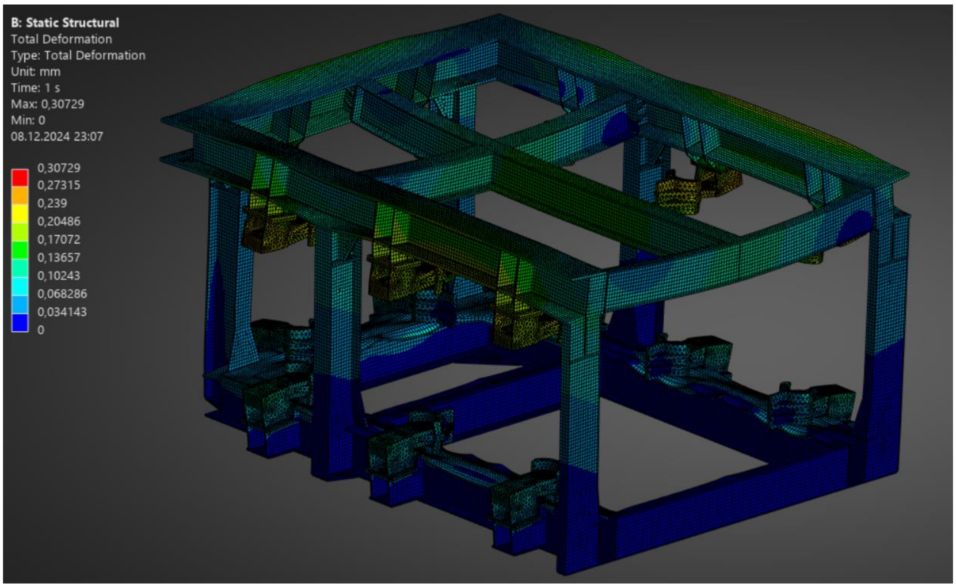
The deformation assessment also highlights the importance of accurately defining the thermal and mechanical boundary conditions to ensure that the model reflects real-world behavior. The interaction between thermal expansion, gravitational loading, and mechanical constraints is complex, and careful consideration must be given to each factor to accurately predict the deformation behavior of the structure. The results of this analysis provide confidence that the welded pontoon elements will remain within the required geometric tolerances, thereby ensuring proper alignment and fitment during assembly and operation.



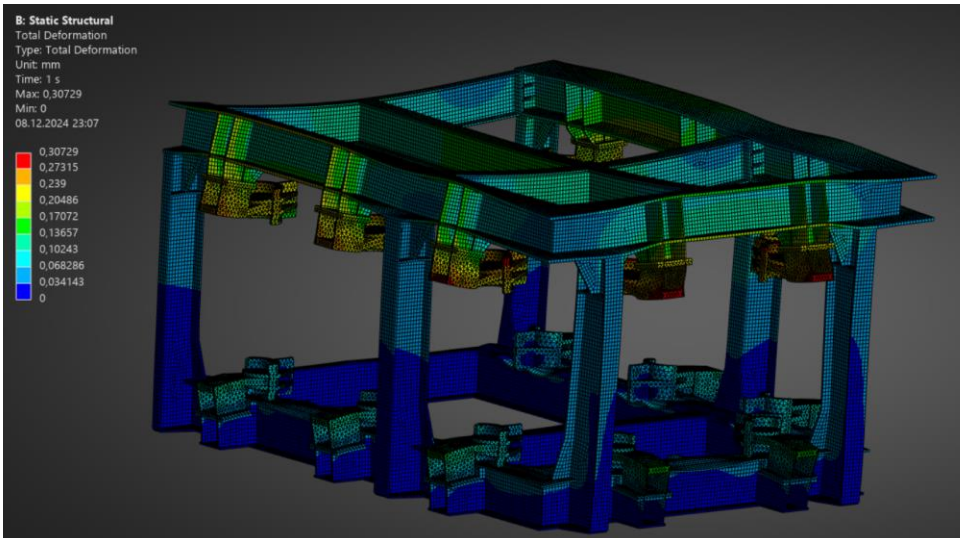
Picture 106: Deformation distribution across the pontoon and conductor structures.



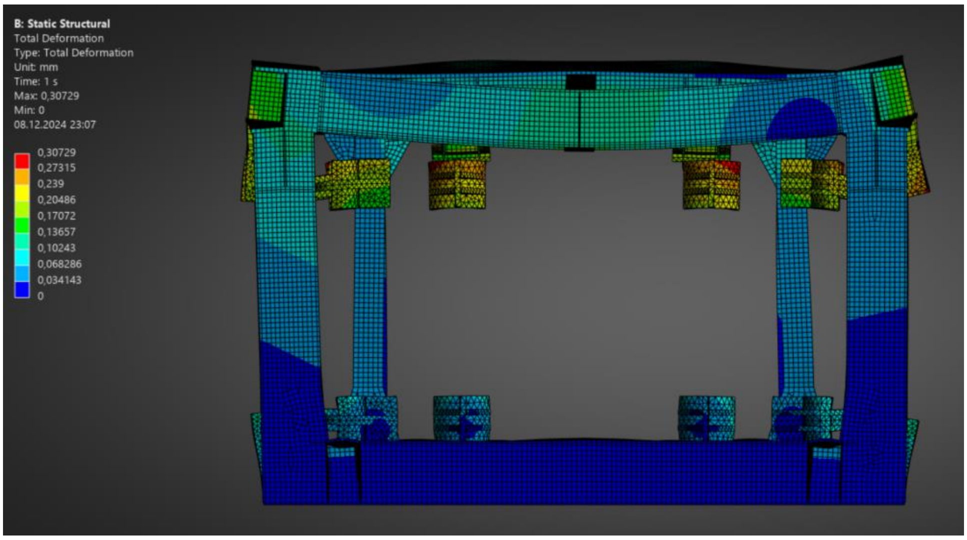
Picture 107: Deformation distribution across the pontoon and conductor structures.



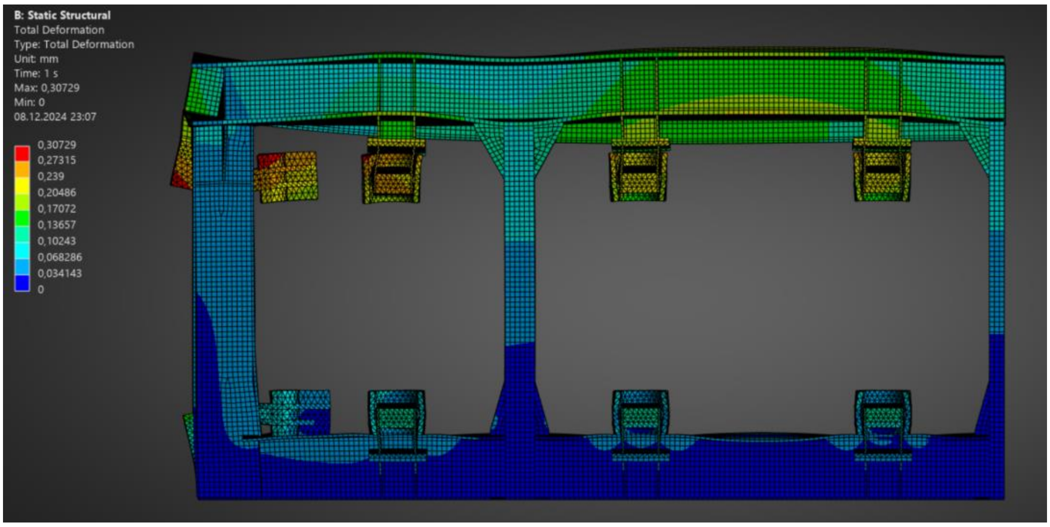
Picture 108: Deformation distribution across the conductor structures.



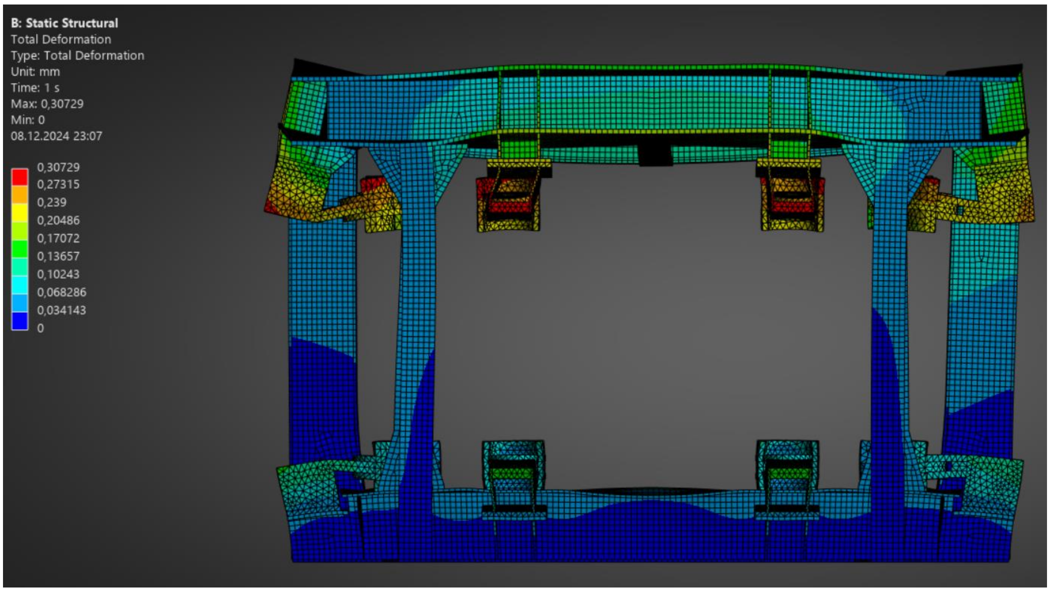
Picture 109: Deformation distribution across the conductor structures.



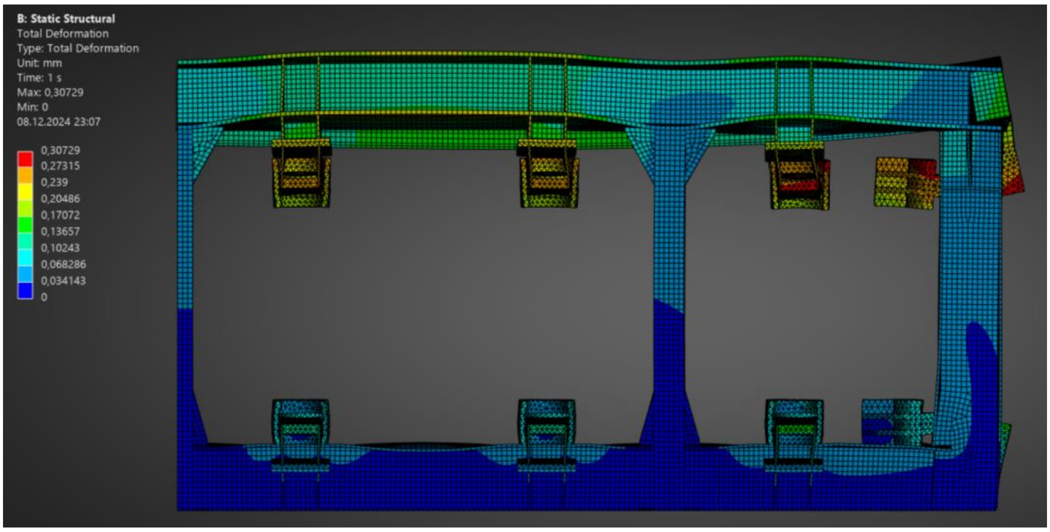
Picture 110: Deformation distribution across the conductor structures.



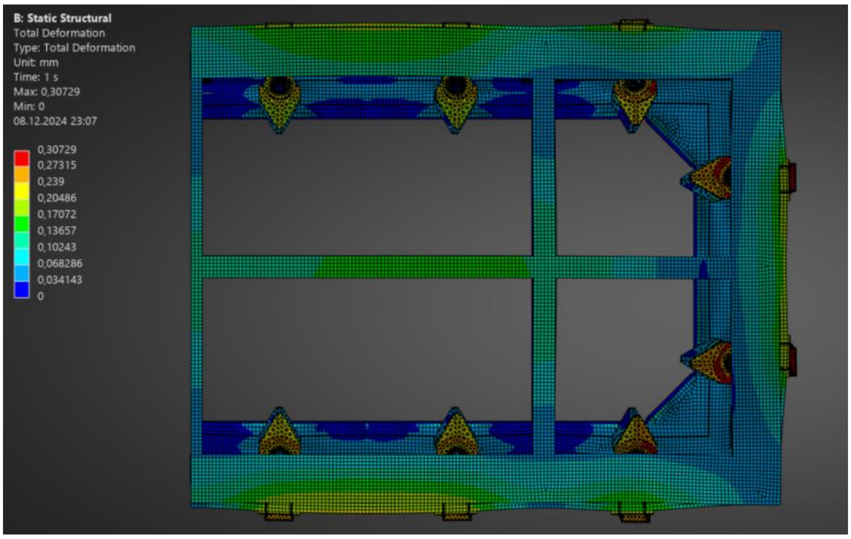
Picture 111: Deformation distribution across the conductor structures.



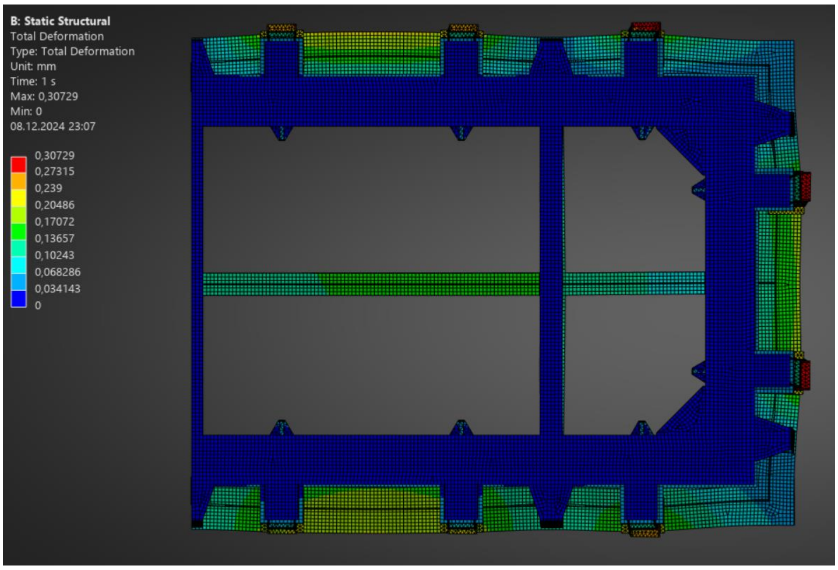
Picture 112: Deformation distribution across the conductor structures.



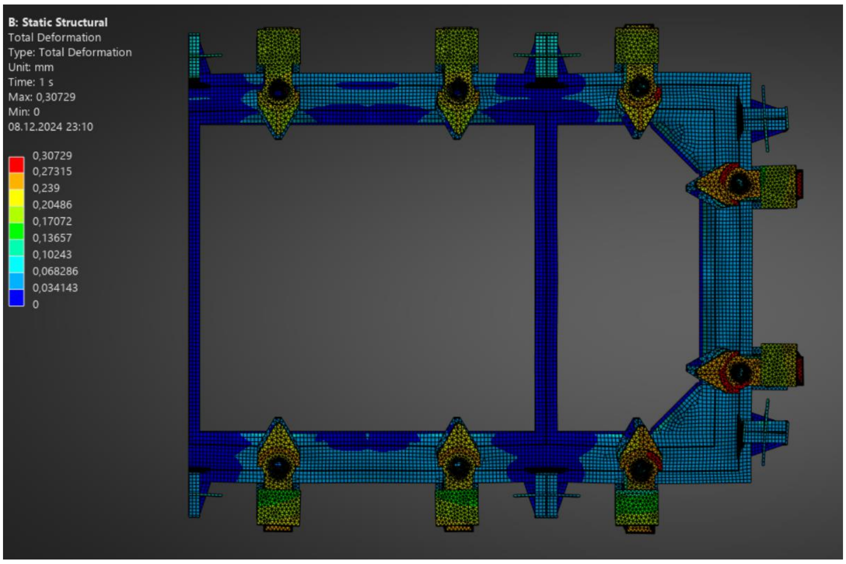
Picture 113: Deformation distribution across the conductor structures.



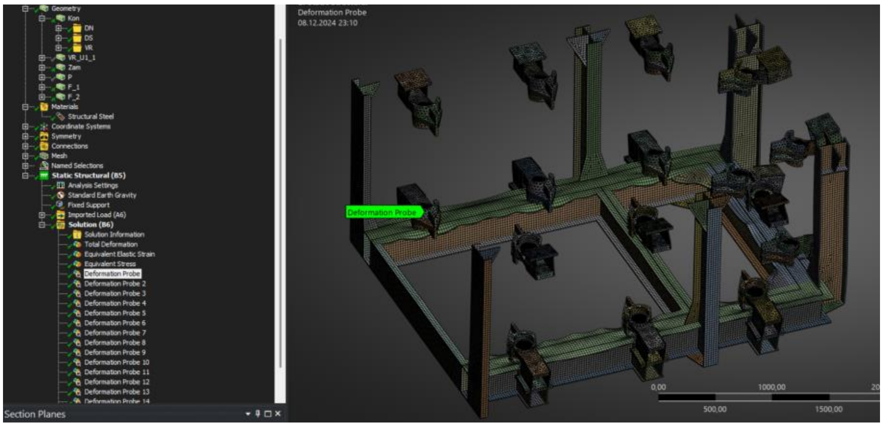
Picture 114: Deformation distribution across the conductor structures.



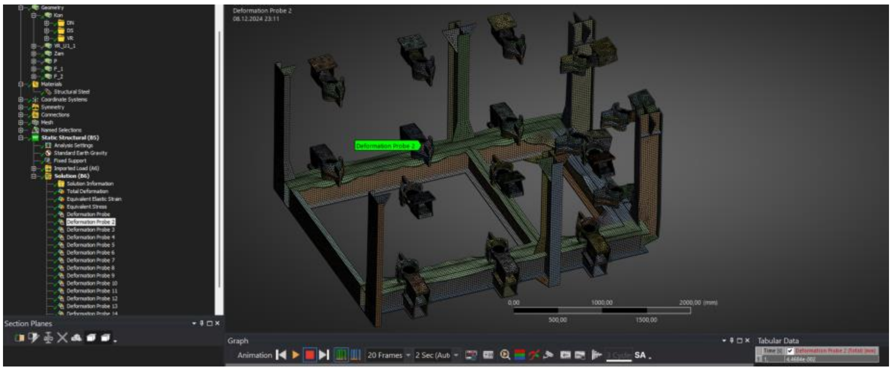
Picture 115: Deformation distribution across the conductor structures.



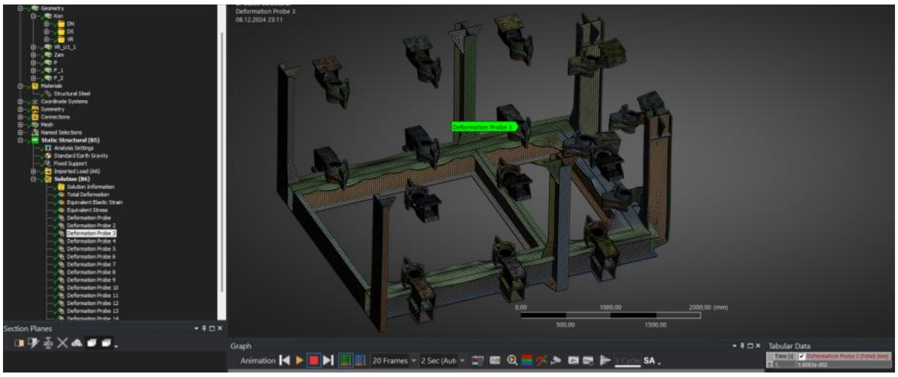
Picture 116: Deformation distribution across the conductor structures.



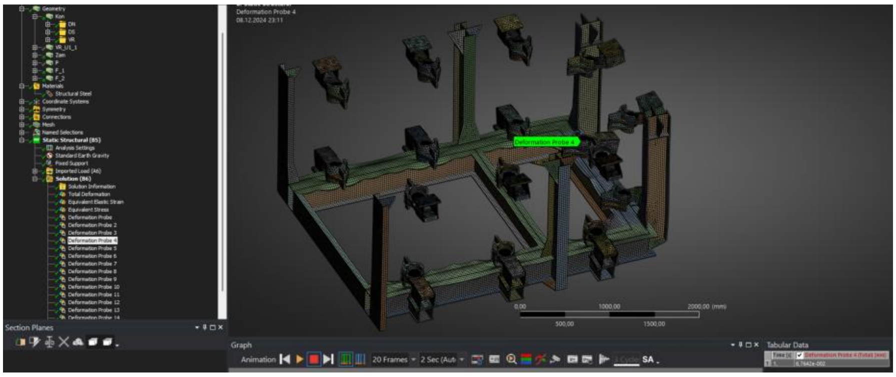
Picture 117: Deformation Probe 1 of the element of interest.



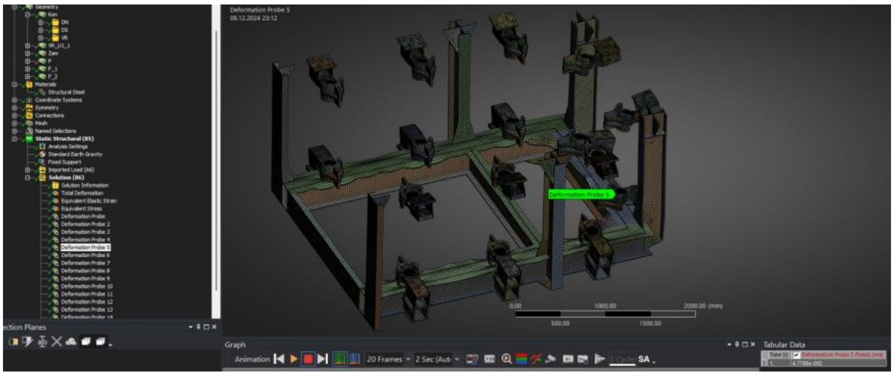
Picture 118: Deformation Probe 2 of the element of interest.



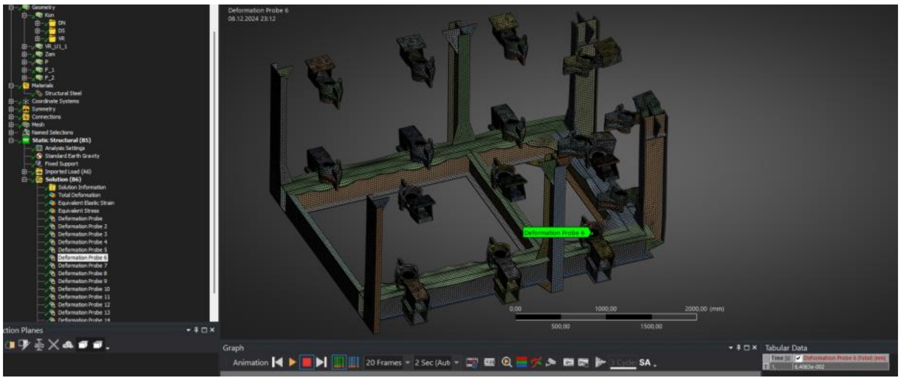
Picture 119: Deformation Probe 3 of the element of interest.



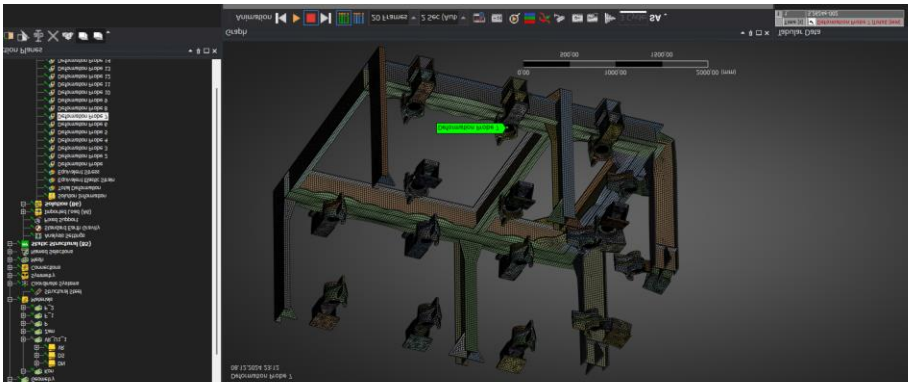
Picture 120: Deformation Probe 4 of the element of interest.



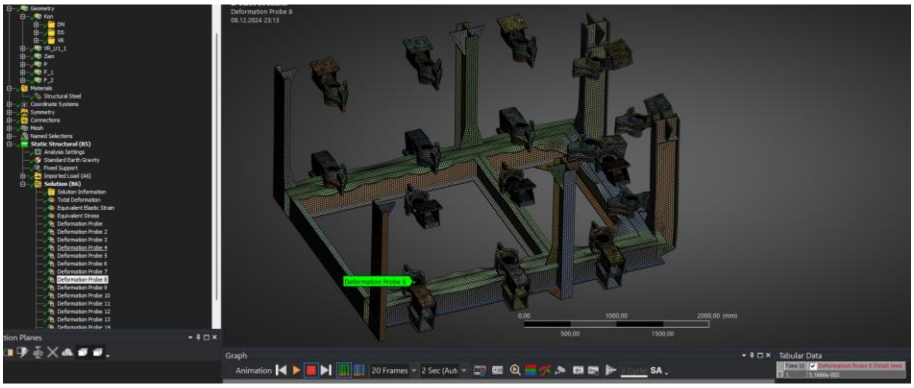
Picture 121: Deformation Probe 5 of the element of interest.



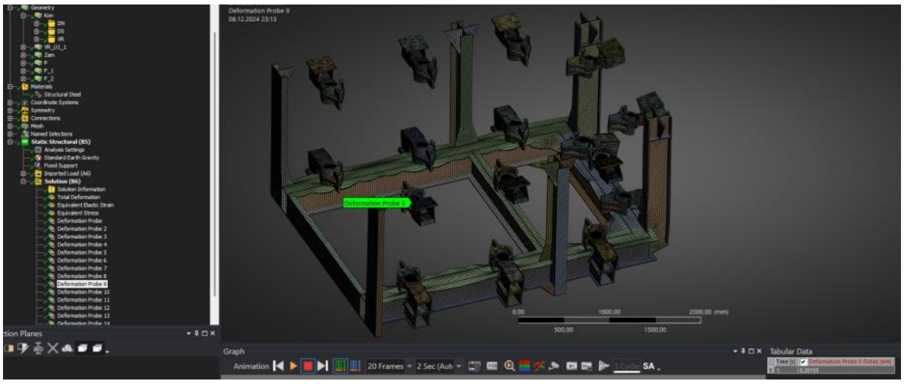
Picture 122: Deformation Probe 6 of the element of interest.



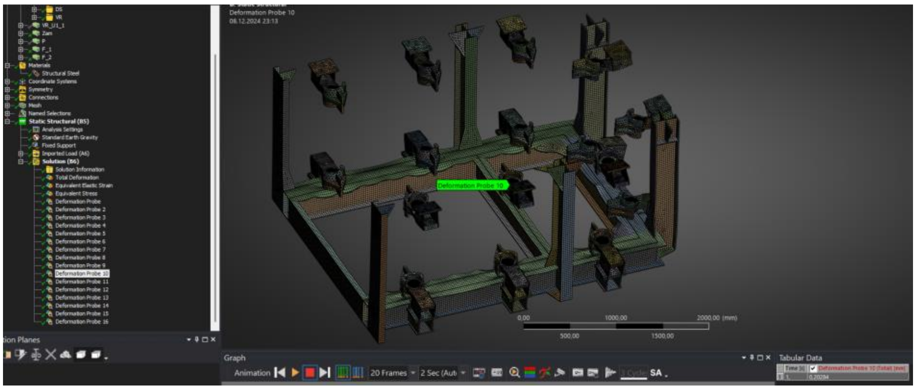
Picture 123: Deformation Probe 7 of the element of interest.



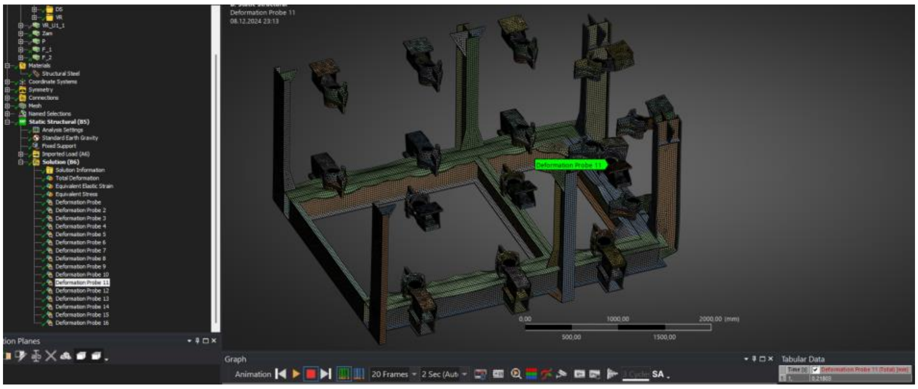
Picture 124: Deformation Probe 8 of the element of interest.



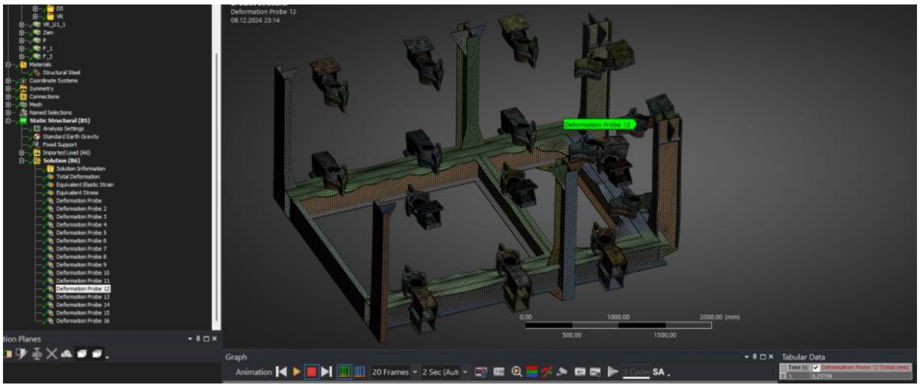
Picture 125: Deformation Probe 9 of the element of interest.



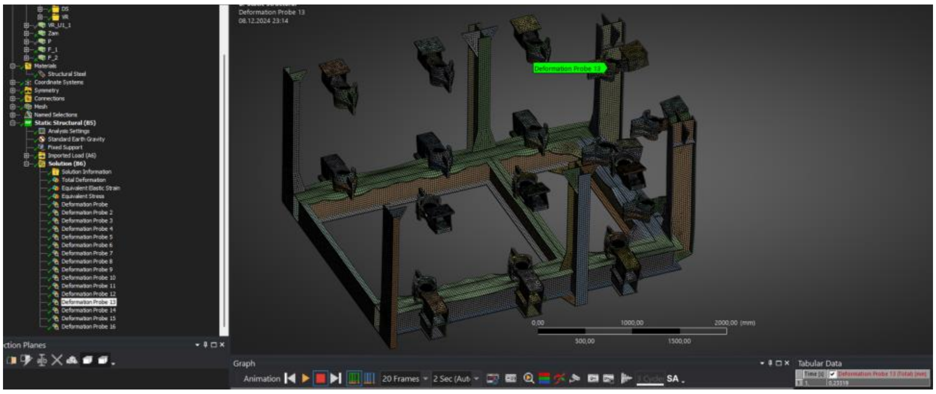
Picture 126: Deformation Probe 10 of the element of interest.



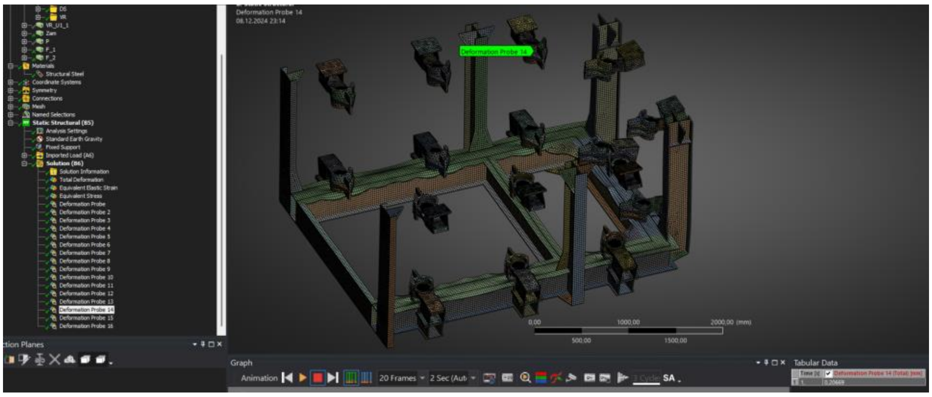
Picture 127: Deformation Probe 11 of the element of interest.



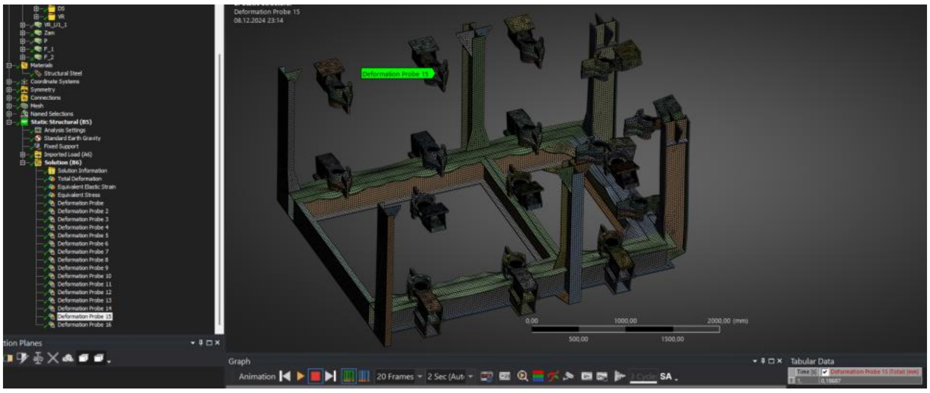
Picture 128: Deformation Probe 12 of the element of interest.



Picture 129: Deformation Probe 13 of the element of interest.



Picture 130: Deformation Probe 14 of the element of interest.



Picture 131: Deformation Probe 15 of the element of interest.

Picture 132: Deformation Probe 16 of the element of interest.

CONCLUSIONS

The conclusion of this study presents the findings related to the influence of the conductor design on alignment accuracy and overall structural integrity during the welding process. The results of the thermal and mechanical analysis highlight how the conductor's design affects the ability to maintain the specified alignment tolerance and minimize deformations.

Influence of Conductor Design on Alignment Accuracy. The conductor's design plays a crucial role in ensuring that the welded elements are accurately positioned and that the deformation remains within acceptable limits. The conductor is designed to securely hold the pontoon components in their designated positions during the welding process, thereby minimizing the risk of misalignment. The ability to maintain the correct positioning of components is essential, as even minor deviations can lead to significant challenges during the assembly of the pontoon structure.

One of the key aspects of the conductor design that contributes to alignment accuracy is the size of the cross-sections and the thickness of the elements used. By selecting appropriate dimensions for the conductor's structural members, the overall stiffness of the conductor is increased, which in turn minimizes deformation during welding.

The analysis results indicate that the deformation experienced by the conductor and pontoon elements is well within the allowable tolerance of 0,5 mm. The maximum displacement observed in the elements of interest was 0,273 mm, which demonstrates that the conductor design is effective in managing the thermal and mechanical loads applied during welding. This outcome is directly related to the choice of materials, cross-sectional dimensions, and the overall design strategy employed to ensure stability and rigidity.

Another important design feature is the use of reinforcement elements in areas where higher loads or significant thermal gradients are expected. These reinforcements help to distribute the stresses more evenly across the structure, thereby reducing the risk of localized deformation. By strategically placing reinforcements, the conductor can better withstand the thermal expansion of the welded elements, ensuring that the alignment is maintained throughout the welding process.

The findings from this study demonstrate that the conductor's design effectively mitigates the effects of thermal and mechanical loads, ensuring that the welded components remain within the specified geometric tolerances. The choice of appropriate cross-sectional dimensions, material properties, and reinforcement plays a main role in achieving this outcome.

Limitations of the Study. The analysis presented in this study involved several key assumptions that helped to simplify the modeling process and reduce computational requirements. However, these assumptions may introduce certain limitations that should be considered when interpreting the results.

One of the primary assumptions made during the modeling process was the application of heat flow across the entire length of the weld seams, rather than simulating the actual movement of the welding point. In real-world conditions, the welding heat is applied at a specific point that gradually moves along the seam, creating a temperature gradient as the material cools after the welding torch passes. By applying the heat flow uniformly across all weld seams simultaneously, the analysis represents a worst-case scenario, which is beneficial for safety considerations but may not accurately capture the dynamic thermal behavior observed during an actual welding process. This simplification could lead to overestimation of thermal loads and associated deformations.

Another assumption involved the use of symmetry to reduce computational resources. Symmetry boundary conditions were applied to model only a portion of the entire structure, assuming that the rest of the structure behaves similarly. While this approach helps to significantly reduce computational time and effort, it may not fully account for any asymmetries present in the actual structure or loading conditions. As such, the results may not capture localized effects that could arise due to deviations from symmetry in real-world scenarios.

The use of shell elements for modeling the main components of the structure was another significant assumption. Shell elements are effective for reducing computational requirements, particularly for large and thin-walled structures. However, shell elements may not fully capture the three-dimensional stress state in regions where thickness variations or complex geometries are present. This could lead to less accurate predictions of stresses and deformations in areas where solid elements would have been more appropriate for capturing the detailed behavior of the material.

Additionally, the replacement of bolted connections with bonded contacts was implemented to further simplify the model. Bolted connections introduce complexities due to the interaction between bolts and the surrounding material, including potential slippage and localized stress concentrations. By using bonded contacts instead, the model assumes that the connected elements are perfectly bonded without any relative movement. This assumption may result in an underestimation of local stresses and potential points of failure in regions where bolted connections are present.

Overall, while these assumptions allow for a more efficient analysis and provide a conservative estimate of the structural response, they may also introduce certain limitations in terms of accuracy. It is important to recognize that the results presented in this study should be interpreted with these assumptions in mind, and further detailed modeling, including more realistic boundary conditions and connection representations, may be necessary for more precise assessments.

Summary of Findings. The analysis conducted throughout this study confirms that the conductor and pontoon structures meet the required deformation limits, thereby ensuring stability during the welding process. The findings indicate that the maximum deformations observed during the thermal and mechanical analysis were well within the allowable tolerance.

The study highlights the importance of the conductor's geometric features in minimizing deformation. The combination of a well-designed conductor, adequate material selection, and optimal boundary conditions contributed to achieving the desired alignment accuracy.

Overall, the conclusions drawn from this study validate that the conductor design and the adopted welding procedures are robust enough to meet the stringent requirements of alignment accuracy and structural strength. The key findings, including the deformation limits and stress assessments, all point toward the successful application of finite element analysis in predicting the structural behavior under welding conditions.

These findings are significant for future work, as they confirm that the current design methodology and analysis approach can be extended to other similar welding processes. The validated model provides a basis for further optimization, particularly in enhancing the efficiency of welding simulations or exploring alternative conductor configurations that may yield even lower deformation rates. The current study thus lays the foundation for ongoing improvements in the floating modular structures.

Practical Implications. The findings of this research have practical implications for the development and optimization of welding technologies in marine engineering and other structural applications. The key practical outcomes from this study can be applied in several areas, as detailed below.

First, the results can be used to design new conductor models with optimized dimensions and thicknesses. By understanding the stress-strain response of existing conductor designs, engineers can determine which areas need reinforcement and make changes that reduce overall deformation. This optimization can help in reducing the material costs while ensuring that the structure meets the accuracy and strength requirements. Improved conductor design will also ensure that modular components, like pontoons, can be welded more reliably, with a minimal risk of post-welding misalignment.

Second, the study's findings offer insight into improving welding technologies to minimize deformations that occur during the welding process. One significant recommendation is to enhance the manual welding sequence by implementing an optimized welding pattern that distributes heat evenly. This would help in minimizing the concentration of thermal stresses, which is one of the major causes of excessive deformation. Furthermore, the findings indicate that by using a more controlled heat input, it may be possible to achieve even greater precision in maintaining the alignment of components. These enhancements in welding technologies can be crucial for industries that demand high precision, such as shipbuilding and offshore construction.

Future Research Prospects. The conclusions drawn from this study have highlighted several directions for future research to further enhance the understanding and optimization of conductor welding processes and structural performance.

One potential direction is to utilize more sophisticated welding simulation methods. This may involve implementing more detailed transient thermal analyses that account for the dynamic movement of the welding arc rather than simplifying the process into a uniform heat flow applied along the weld seams. Such a refined approach could provide a more accurate representation of the temperature distribution and subsequent stress-strain behavior across the welded structures.

Another valuable direction is to investigate the effect of applying welding loads individually to the elements being welded. In the current study, heat loads were applied simultaneously to all weld seams for computational efficiency. Future work could focus on applying these loads sequentially, simulating more realistic welding conditions, where only one or two elements are welded at any given time. This approach would help in better understanding the localized effects of welding on the structural stability and integrity of the pontoon and conductor.

An additional area for future research is the exploration of the possibility of reducing the thickness and cross-sectional dimensions of the structural components without compromising their strength or stiffness. The current design ensures that all deformations remain within acceptable tolerance limits. However, optimizing these parameters further could lead to a more cost-effective and lighter design, which would be beneficial in both manufacturing and operational phases.

LIST OF REFERENCES

1. Damen Modular Vessel Configurator, URL: https://configurator.damenmodularbarges.com/ (Last accessed: 12.03.2024).

2. Rosenthal D. Mathematical theory of heat distribution during welding and cutting. Welding journal. 1941;20(5):220s-34s.

3. Zienkiewicz OC, Taylor RL. The finite element method for solid and structural mechanics. Elsevier; 2005 Aug 9.

4. Ghafouri M, Ahola A, Ahn J, Björk T. Numerical and experimental investigations on the welding residual stresses and distortions of the short fillet welds in high strength steel plates. Engineering Structures. 2022 Jun 1;260:114269.

5. Kim J. Non-destructive characterization of railway materials and components with infrared thermography technique. Materials. 2019 Dec 6;12(24):4077.

6. Stonesifer RB, Rybicki EF. Development of models for warm prestressing. Computational Mechanics, Julian, PA (USA); PROSIG, Inc., Tulsa, OK (USA); Materials Engineering Associates, Inc., Lanham, MD (USA); Nuclear Regulatory Commission, Washington, DC (USA). Div. of Engineering Safety; 1987 Jan 1.

7. Goldak JA, Akhlaghi M. Welded Structures and Applications of Welding in Industrial Fields. Computational Welding Mechanics. 2005:247-75.

8. Ozcelik S, Moore K. Modeling, sensing and control of gas metal arc welding. Elsevier; 2003 Jun 11.

9. ANSYS. (n.d.). ANSYS Help Documentation. Avalable URL: https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/main_page.html?lang=en. (Last accessed: 12.04.2024).

10. Larson MG, Bengzon F. The finite element method: theory, implementation, and applications. Springer Science & Business Media; 2013 Jan 13.

11. British Standards Institution. BS EN ISO 9606-1:2013 - Qualification Testing of Welders - Fusion Welding. BSI Standards Publication. 2013.

12. Lloyd's Register. (2020). Rules and Regulations for the Classification of Ships - Welding and Structural Details.

13. American Welding Society. AWS D1.1 Structural Welding Code. 2000.

14. DIN Standards Committee. DIN EN 1011-3: Recommendations for Welding of Metallic Materials. 2019.

15. ISO. 5817: Welding — Fusion-welded joints in steel, nickel, titanium, and their alloys (quality levels for imperfections). 2016.

16. Özışık MN. Heat conduction. John Wiley & Sons; 1993 Mar 22.

17. Holman J. Heat Transfer tenth edition, ser. McGraw-Hill Series in Mechanical Engineering. McGraw Hill Higher Education. 2009.