Structural topology optimization of an active motion compensated gangway

An optimization study

M.C.J. Vergeer

Master thesis

Delft University of Technology

12 October, 2018

The work in this thesis was supported by Royal IHC. Their cooperation is gratefully acknowledged.

Delft University of Technology

Faculty of Mechanical Engineering

Department of Offshore and Dredging Engineering

Structural optimization of an active motion compensated gangway

By

Michael Christianus Jozef Vergeer

Master thesis
In order to obtain the master degree

Master of Science
in
Offshore and Dredging Engineering

Delft, the Netherlands
October 12, 2018

Information

About the thesis

This report presents the master thesis of Michael Vergeer for the Master of Science (MSc) program of the department Offshore and Dredging Engineering (ODE). The department results from interfaculty collaboration between the faculty of civil engineering (CEG) and the faculty of mechanical, maritime and materials engineering (3mE) at the Delft University of Technology (DUT).

A structural topology optimization study is carried out at IHC SAS-Hytop which is located in Alphen aan den Rijn, The Netherlands. The SAS-Hytop Corporation is a unit within the Royal IHC group which is specialized in designing and manufacturing tensioners, winches and pipe handling equipment for the offshore industry.

The goal of this master research is to investigate if topology optimization can be implemented in the design process at IHC. This methodology is applied to a device which is called a motion compensated gangway.

Author

Michael Christianus Jozef Vergeer
Student number: 4400348

Thesis committee

The thesis committee is comprised of:

Dr. Ir. S.A. Miedema	:	Associate professor of Dredging Engineering and Mechatronics at TU Delft.
Dr. Ir. M. Langelaar	:	Associate professor, structural optimization and mechanics.
Dr. Ir. R. Helmons	:	Postdoc / Researcher, Dredging Engineering at TU Delft, Faculty 3mE.
Dr. Ir. P. Kromwijk	:	Manager structural engineering department at IHC SAS-Hytop.

Confidential

An electronic version of this document is available at:
https://repository.tudelft.nl/

Acknowledgements

Before you lies the master thesis which is performed in the past 12 months. This research is performed by Michael Vergeer as final part of the master study Offshore and Dredging Engineering given at the Technical University in Delft. The basis of this report is a research which is performed at IHC SAS-Hytop about the structural optimization of a motion compensated gangway. The goal of this master thesis is to perform a structural optimization of a gangway boom for a motion compensated gangway by using topology optimization. The result of this optimization process can be used to determine if this design methodology can be applied at IHC. The research is done under the supervision of Technical University Delft in corporation with IHC SAS-Hytop. I was involved in researching and writing this report from September 2017 to September 2018.

This project was undertaken in corporation with my supervisor Mr. Peter Kromwijk who is the manager of the structural engineering department at IHC SAS-Hytop. The research question was formulated together with him. The research was very interesting and showed me the all the aspects related to gangway structures, topology optimization and finite element analysis. My supervisors Peter Kromwijk, Matthijs Langelaar and Sape Miedema were always available and prepared to answer my questions. Their feedback during the meetings was very useful and helped me to improve my research. I would like to thank Rudy Helmons for being part of the graduation committee.

I would like to thank my supervisors for their guidance and support during my graduation period. They helped me with formulating my research objective and to define the boundaries of my research. In particular, the author would like to thank Matthijs Langelaar for his guidance and feedback during the optimization process. He was very interested in the research and his comments contributed to the course of this research. I would also like to thank the company IHC for the opportunity to do this master thesis and to make use of their facilities. In addition I want to thank the companies Solidthinking, Limitstate and Altair which have generously provided software licences and support.

I hope this thesis will give rise to fruitful thoughts and inspires people to learn more about the application of topology optimization.

Michael Vergeer.

Alphen aan den Rijn, The Netherlands.
October 2, 2018.

Abstract

In the offshore industry, there is a growing demand for designing efficient, sustainable and competitive products. In order to fulfil the component requirements, a method named topology optimization can be applied. This is a mathematical design method which can be used in the early phases of the design process. During the last decade, topology optimization has grown to be a more accepted method to produce conceptual designs. At IHC there is an interest of finding and exploring topology optimization for their equipment development process. Therefore the possibilities and limitations of the method should be investigated thoroughly.

This master thesis covers the optimization process of a motion compensated gangway. A motion compensated gangway is a walkway which can be used to provide access from the transport vessel to the offshore structure. Its function is to transport people and cargo safely from the ship to the offshore structure or vice versa. It creates a safe, firm and stable connection between the floating vessel and the offshore structure. Nowadays, the development of the offshore equipment is performed by an iterative development process in which a design is proposed by the design engineer which is subsequently validated by a structural engineer. If the design does not satisfy the requirements then the design is adjusted and reanalysed by the structural engineer. This process often requires multiple iteration steps in order to obtain a feasible design which complies with all the stated requirements.

The goal of this master thesis is to determine to what extent topology optimization can be used in the design of a motion compensated gangway: Finding an optimized result in terms of weight and stiffness by using this mathematical method which satisfies all the requirements. The structural optimization is carried out with several commercial software packages which are compared by using a multi-criteria analysis. From the multi-criteria analysis it was found that the Hyperworks package is a capable tool for performing the optimization task. Although the user interface is more complicated compared to the other software packages, it allows the user to control every aspect of the optimization process. This enables the user to prevent several problems which are encountered by the other software packages and this results in more realistic and useful solutions. Also the software support was available for questions and help. This was not the case for the other software packages.

During the optimization process it has been found that there are essentially two stages in the optimization process. In the first stage, the topology or beam orientation of the structure is defined by the topology optimization process. In this part the concept of the design is generated. Variation of the optimization parameters was used in order to develop an efficient structure. The objective for the optimizer was to minimize the compliance of the structure for a certain volume fraction.

In the second stage, the dimensions of all the beams and elements are defined by performing a size optimization. A line model is generated which represents the orientation of the members in the structure. During the size optimization the shape and the dimensions of the members are defined in order to fulfil the objective. The objective is to minimize the mass of the structure while constraints are defined for the maximum allowable stresses in the members and the maximum vertical deflection of the structure. This post-processing step is required in order to obtain a feasible design. It must be noted that the optimization process is unable to account for the stability of the structure. The structural stability of the gangway was increased by performing a linear buckling analysis and by adapting the structure in order to reduce the buckling behaviour.

In the final step of the optimization process, a CAD drawing is generated. This model is analysed by using finite element analysis. This thesis presents the final design in which the topology optimised design yielded a weight reduction of 36,4 % compared to the current design. The weight was reduced from 13,08 ton to 8,31 ton, while still satisfying all the constraints.

Key words: *Optimization, Topology optimization, Shape optimization, Motion compensated gangway, Structural optimization, CAE driven development, SIMP, Optistruct, Size optimization, Stepwise optimization, Member sizing, design realisation, post-processing steps, Optimization process, Hyperworks.*

Table of Contents

1. Introduction

This master thesis project is performed at IHC SAS-Hytop in order to investigate the possibility of using topology and size optimization in the design process of a motion compensated gangway structure. This investigation will determine to what extent topology optimization can be used in the design process of IHC SAS-Hytop. In this chapter the background, problem definition and research objective are described together with the method of approach. The final part of this section provides an outline of the thesis.

1.1 The context

The offshore industry is a rapidly changing industry which relies greatly on the oil price. When the oil price is high, investments are made. Currently the oil price is relatively low and unstable and therefore investments are postponed or cancelled. This makes it difficult for the manufacturers of offshore equipment to gain orders. There is not enough work for all the companies in the market and this results in a lot of competition between the different manufacturers of offshore equipment.

At IHC a variety of offshore equipment is manufactured and developed. In the design process it is desired to find an optimum design for given circumstances. Often a trade-off must be made between different properties in order to obtain a design which satisfies all the requirements. For example, a weight reduction of a component often leads to a decrease of the stiffness of that component. A certain stiffness is required in order to satisfy the requirements of that component. It is very difficult to find the point where you have the minimum weight for a certain stiffness.

It is desired to find an optimized balanced design in the early phase of the product development to enable competitive lead times, to obtain a feasible design and to keep the costs as low as possible. It also reduces the risk for late design changes which are related to increased costs.

The large amount of competitors pushes the manufacturer to generate a design which differs from most other designs available in order to stay competitive. Nowadays efficiency is an import aspect in the offshore industry because the operational costs of a device are taken into consideration. The selection of component material and design is an important aspect in order to gain sustainable and competitive results.

This thesis will address the procedure for performing an optimization process by using topology optimization. It focuses on the structural optimization of a motion compensated gangway, which is a special device to transfer people safely from a vessel to an offshore structure. The different guidelines are investigated which determines the load cases to which the device is subjected. These load cases will form the boundary conditions for the optimization process. The goal is to compare different topology optimization software packages and to determine which program is the most suitable to solve this problem. The set-up and approach to use topology optimization software is given. The motion compensated gangway structure is used as an example for the optimization process.

1.2 Problem definition

Transfers from a ship to a fixed offshore platform are difficult for the following reason: the wave forces cause the ship to be continuously in motion while the fixed structure is almost static. The relative motions between the vessel and the structure makes it difficult to transfer people and cargo between the offshore structure and the moving vessel. Transferring people and cargo at open sea can result in dangerous situations for ships, platforms and crew. A motion compensated gangway is a device, which is able to transport people and cargo safely from a floating vessel to another offshore structure. The ship will remain stationary by its dynamic positioning system but it cannot compensate for roll, pitch and heave motions. The gangway will compensate the heave, roll and pitch motions by using hydraulic actuators. These “counter-movements” result in a safe and stable connection between the two objects, enabling the cargo and crew to be transferred and positioned without problems, even in rough seas. Marine access with a gangway is also more efficient, reliable and safer than other access methods like helicopter flights.

To enable a safe and efficient transfer of personnel and cargo from a ship to an offshore platform or wind turbine, IHC is designing a motion compensated gangway. This gangway is still under development and is currently in the design phase. To reduce the forces and moments in the system, the weight of the gangway must be optimized. Because the weight of the gangway is an important driver in the design of the complete system, a weight reduction reduces the required hydraulic power to operate the structure and it influences the deflection of the structure. The weight of the structure determines also the mass inertia of the structure and the deck loads. A reduction in the required hydraulic power allows the use of smaller parts which usually reduces the manufacturing costs of the device. Another important advantage is that it reduces the operational costs of the device. A weight reduction of the gangway design will result in a reduction of the pedestal and deck loads. These are all important parameters for the design of the motion compensated gangway and therefore the design of the gangway or boom must be optimized.

The current design method is an iterative process in which a design is proposed by a design engineer and is subsequently analysed with a finite element method by a structural engineer. If the design does not satisfy the requirements, the design is modified and improved by the designer and the process is repeated. This iterative process takes a lot of time and results in a sub-optimal design as multiple changes are added to the design with little overview of the design process. The use of topology optimization could decrease the lead time which results in lower costs.

1.3 Research objective

In consultation with the company supervisor, the main objective of this research is defined as:

Find the optimum weight reduction for the gangway structure in the design domain, for a given set of loads and boundary conditions while fulfilling the service constraints.

The optimality depends on minimizing the weight, while the service constraints could be given by its maximum displacements and stresses. The weight of the gangway structure must be reduced, taking into account the operational requirements and certification guidelines. The gangway must remain operational and must be designed according to these guidelines.

The main research revolves around the design approach. It starts from the design domain through the design realization process to the detailed design. The aim of this thesis is to provide a methodology to use topology optimization during the design process. This leads to the following sub-questions in this research:

- *Which software package is the most suitable for optimizing the gangway structure?*
- *Can the weight of the current design be reduced by using topology and size optimization?*
- *Is this methodology practically applicable in the industry?*

The research starts with the basic design of the gangway structure. The different loads and types of supports on this structure must be examined. These different loads and the types of support determine the boundary conditions which serve as input for the topology optimization process. Different commercial software packages will be compared and the most suitable method is chosen.

A topology optimization study will be performed and the results of this study will be analysed and the design will be adjusted if necessary. The design must meet all the specified requirements and guidelines. The goal of this optimization process is to reduce the weight of the gangway while maintaining its required stiffness. This weight reduction will lead to a reduction in the required hydraulic power and deck loads.

Important parameters are the amount of deflection of the gangway, the maximum stresses and the manufacturability; this drives the costs of the gangway structure. Also the buckling behaviour of the structure should be incorporated into the design. The optimization includes related subjects such as: the objective function, constraints and also so-called manufacturing constraints for both the topology and shape/size optimization. The load cases and boundary conditions which are relevant with respect to the optimization process are treated. The main task is to investigate to what extent topology optimization can be used in the design process of a motion compensated gangway. This includes the possibilities and limitations of the software and the areas in which this technology is applicable. The purpose of this master thesis is to gain insight and understanding in the use of topology optimization and to determine if it can be used to improve the design processes at IHC SAS-Hytop.

1.4 Method of approach

For determining the optimal design of the gangway, topology optimization will be used. Topology optimization is a mathematical method that optimizes the material layout within a given design space for a given set of loads, boundary conditions and constraints [20]. The goal of this method is to optimize the performance of the gangway in terms of stiffness and mass. The work is in this master thesis is divided into three steps.

The first step is to perform a literature study in which the technique of topology optimization and the working principle of the gangway are investigated. All the loads, boundary conditions and applicable guidelines needs to be analysed and determined. An overview is given of all the different topology methods available and it is determined which method is best suitable for optimizing the gangway.

After the literature study, the selected topology optimization method is investigated in order to understand the applied method. After the mathematical interpretation of this method is fully understood, the topology optimization can be performed. Different optimization software packages are used during this research. The most promising software package is used for the optimization of the gangway structure. The two parts of the gangway structure will be analysed separately, but the influence they have on each other will be taken into account.

The current gangway design will be analysed with the Finite Element Analysis (FEA) and hand calculations. The maximum inner and outer dimensions of the gangway will be used as design space. This design space will serve as input for the optimization tool. The reason for this approach is to gain insight in the behaviour of the structure and to help understanding the obtained results from the optimization tool. Step by step the complexity of the problem will be increased. The design from the topology optimization process will be validated with FEA.

When the results of the topology optimization satisfy the stated criteria, the third step can be performed. The third step consists of a size optimization in order to make the design producible. After this size optimization the final design is checked with FEA. The general outline of the master thesis is depicted in Figure 1.

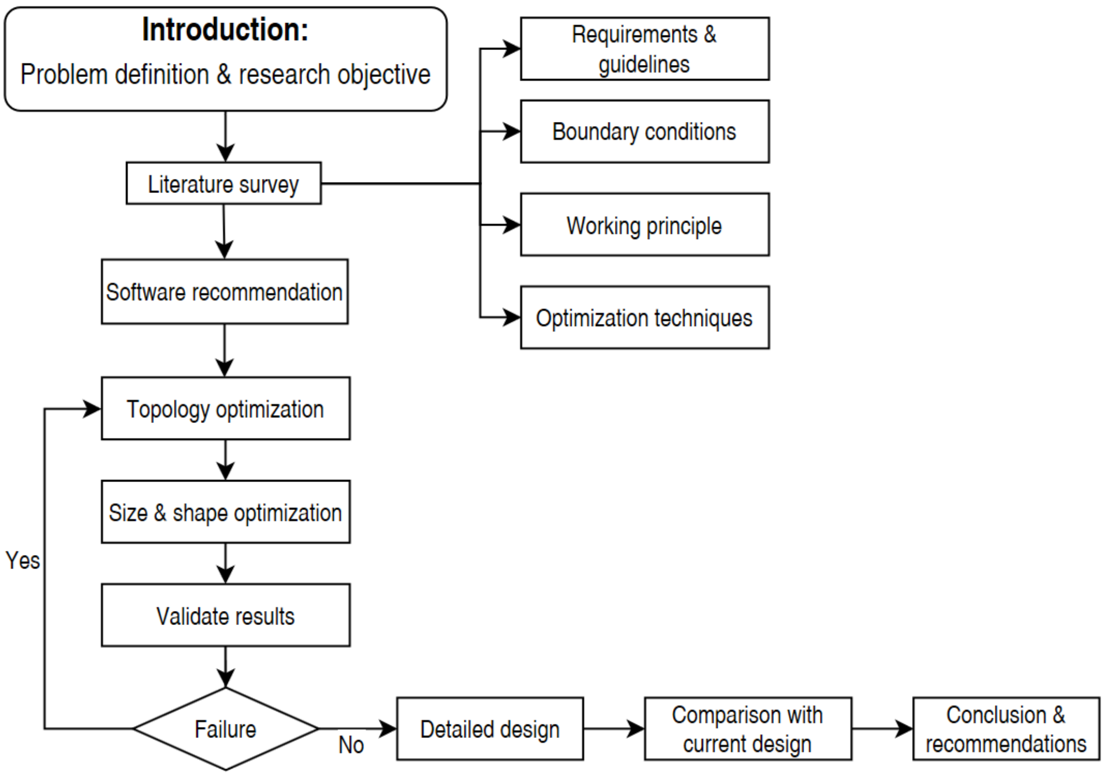


Figure 1: Workflow master thesis

Report outline

In order to give an overview of the project, a schematic outline of the remainder of the thesis is shown below.

Chapter 2	**Motion compensated gangways**: This chapter describes the working principle of a motion compensated gangway. The different types and the operational procedure of the gangway are described in this section.
Chapter 3	**Requirements and boundary conditions**: Contains information about the different requirements and guidelines which are applicable to the gangway structure. These requirements and guidelines determines the boundary conditions and input parameters for the optimization process.
Chapter 4	**Optimization methods**: The theory behind the optimization process is briefly described even as the different optimization techniques. Different types of structural optimization techniques are explained with some examples.
Chapter 5	**Recommendation optimization software**: This chapter describes the different software packages available. The preliminary results from the optimization process are shown and the final software package is chosen by using a multi-criteria analysis in which the different software packages are reviewed and compared to each other.
Chapter 6	**Design approach**: Describes the current design process and explains the structural analysis of the current design.
Chapter 7	**Topology optimization process**: Describes the design process by using topology optimization. The set-up and the results of the optimization process are shown in this chapter.
Chapter 8	**Design realization**: This chapter shows the steps which are performed in order to post-process the results from the optimization process.
Chapter 9	**Structural stability**: A buckling check must be performed in order to check the structural stability of the design. Modifications to the design will be performed if the structural stability of the structure must be increased.
Chapter 10	**Evaluation new design**: The new proposed design is validated by using FEA and compared to the current design.
Chapter 11	**Conclusion and discussion**: This chapter gives a review on the design process reconsidering the research goal. Some recommendations are given on how to proceed with the research.

2. Motion compensated gangways

This section of the report discusses the basics of a motion compensated gangway. First, the working principle of the motion compensated gangway is explained. Afterwards, the different types of motion compensation are explained. The purpose of this section is to introduce the reader to the need and working principle of a motion compensated gangway.

2.1 Definition of a motion compensated gangway

A motion compensated gangway is a walkway or gangway which can be used to provide access from the transport vessel to the offshore structure. The function of the gangway is to transport people and cargo safely from the ship to the offshore structure or vice versa. It creates a safe, firm and stable connection between the floating vessel and the offshore structure. The motion compensation function is used to compensate the vessel motions which cannot be compensated by the DP-system. This enables the system to work in harsher conditions.

Why do we need this?

Transfers from a ship to a fixed offshore platform are difficult for the following reason: The wave forces causes that the ship is continuously in motion while the fixed structure is almost static. The relative motions between the vessel and the structure makes it difficult to transfer people and cargo between the offshore structure and the moving vessel. Transferring people and cargo at open sea can result in dangerous situations for ships, platform and crew. The ship will remain stationary by its dynamic positioning system but it cannot be compensated for its roll, pitch and heave motions. These are the rotations around the X- and Y-axis and the translation in the Z-direction respectively. This motion compensated gangway will compensate the heave, roll and pitch motions by using hydraulic actuators. These “counter-movements” results in a safe and stable connection between two objects, enabling the cargo to be transferred and positioned without problems, even in rough seas. Therefore the accessibility of offshore wind turbines and offshore platforms can be increased significantly by using motion compensated gangways. This can be seen in Appendix: A-12. Offshore access methods in Table 70.

2.2 Ship and gangway motions

The gangway and the vessel have different motions, axis systems and control systems. The different compensation techniques of all the parts will be explained in this section.

Interface and orientation

For this, we start at the beginning by describing the different conventions. For a floating object at sea, the wave induced ship motions can be described by six degrees of freedom (DOF). The DOF is the number of independent motions of the body relative to a fixed frame of reference.

The vessel motions

For a free floating ship these are three translations and three rotations also known as:

- *Surge, translation in the X-direction.*
- *Sway, translation in the Y-direction.*
- *Heave, translation in the Z-direction.*

- *Roll, rotation around the X-axis.*
- *Pitch, rotation around the Y-axis.*
- *Yaw, rotation around the Z-axis.*

All these translations and rotations are shown in Figure 2.

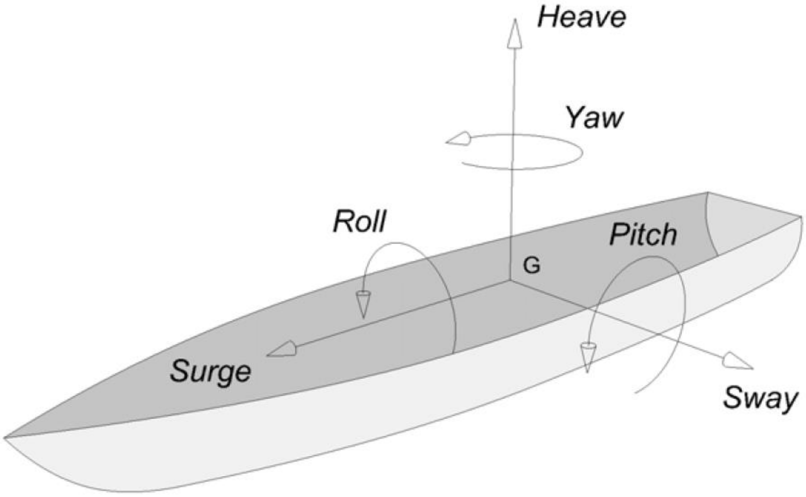


Figure 2: Six degrees of freedom of ship motions (Researchgate.net)

A ship can be equipped with a dynamic positioning (DP) system. This is a fully automatic system which enables the vessel to maintain its position or heading by use of its rudders and/or thrusters [23]. The system constantly measures the vessels surge, sway and yaw motion. By comparing it with the required position and heading, the system can calculate the amount of thrust and the orientation of the thrusters which is required to keep its position and/or heading.

Motions of the gangway

The translations in the X- and Y-directions (respectively surge and sway motion) and the rotation around the Z-axis (yaw motion), can be compensated by the dynamic positioning system. The heave, roll and pitch motion cannot be compensated by the DP system and needs to be compensated by the gangway itself. The gangway has a local coordinate system which is located at the hinge point.

The working principle of the motion compensated gangway can be compared with a crane. It can rotate around its own axis and its boom has a telescopic function and is hinged at its rotation axis.

These motions are also known as:

- Linear translation of the telescopic boom, *telescoping* motion.
- Rotation of the boom in the horizontal plane, known as *slewing*.
- Rotation of the boom in the vertical plane, known as *luffing*.

These motions are visualized in Figure 3.

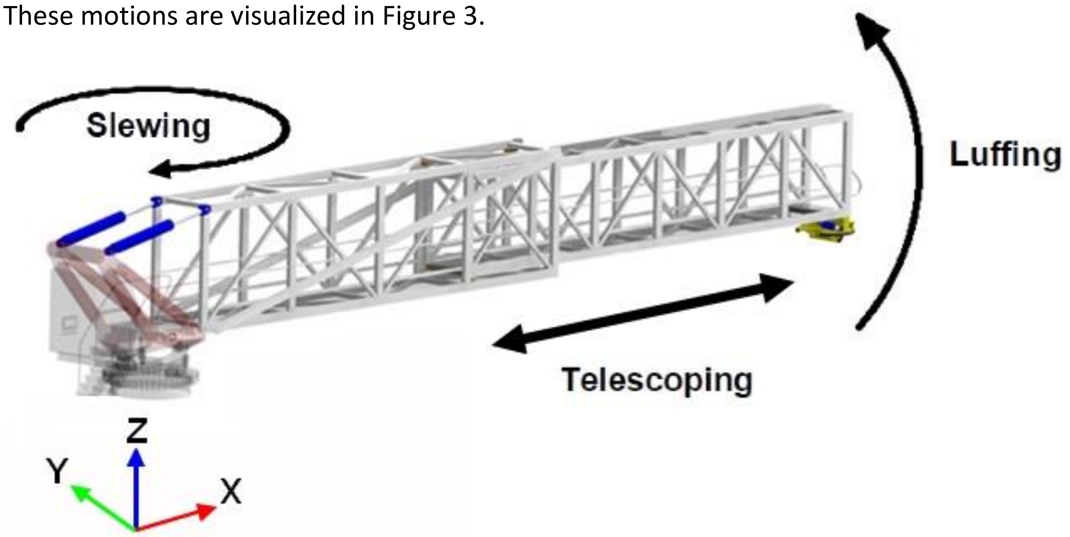


Figure 3: Motions of the gangway

All the gangway motions are defined in the local coordinate system. This local gangway coordinate system is shown in Figure 87. The slewing motion will rotate the gangway clockwise and counter clockwise. It is the rotation around the Z-axis. The luffing motion will move the tip up and down and this is the rotation around the Y-axis. The telescoping motion will extend or retract the gangway thus making it longer or shorter. The telescopic motion will define the length of the gangway in the X-direction. A combination of these three motions can be used to keep the tip of the gangway to a fixed point. The tip of the gangway is connected to the offshore structure and this will enable a smooth and safe connection to the offshore structure.

There are some concepts that uses motion compensation in more than three degrees of freedom. The advantage of these concepts is that not all the vessel rotations can be compensated with a three DOF design. Depending on the location of the gangway on the vessel, the pitch motion or the roll motion cannot be compensated. In most cases, the motion compensated gangway is located in the middle of the ship and the gangway will hang over the SB or PS as can be seen in Figure 87. When this is the case, the roll of the vessel can be compensated by changing the luffing angle and the length of the gangway, but the pitch motion of the vessel cannot be compensated by the gangway.

2.3 Types of motion compensation

The gangway can be equipped with motion compensation. A motion compensator is a device that decreases the undesirable effects of the relative motion between two objects. Motion compensation systems are widely used in marine environments in order to improve safety and increase efficiency. This technique increases the safety of the people passing the gangway or allows the operation of the gangway in more severe conditions. Two types of motion compensation are available: passive and active motion compensation.

2.3.1 Passive motion compensation

In this mode the actuators of the gangway are placed in free flow mode and the ship motions are not actively compensated but free to follow the movements of the ship. The engines are turned off and no operators are required. In this mode the gangway is allowed to absorb movements in all three directions through telescoping, slewing and luffing. In the passive motion compensation mode, the actuators acts as a spring device with a predefined, relatively low stiffness. In this mode the gangway is allowed to accommodate the relative motions between the vessel and the structure without making any use of external systems or equipment.

2.3.2 Active motion compensation

In this mode the vessel movements are compensated by the actuators from the gangway and this enables a smooth connection to the landing area. The motions of the vessel are measured by motion reference units (MRU's). This device measures the motions and accelerations of the vessel in six DOF. The output of the MRU will be coupled to the controller which controls the manifolds and subsequently will drive the active compensation components. A control algorithm determines the combination of response which is required to obtain the desired position. The function of the control algorithm is to determine the combination of luffing, slewing and telescoping motion which is required to maintain the desired position. The tip of the gangway is kept motionless with respect to the touchdown point. The luffing and slewing motions are compensated in the tip of the base frame. The X, Y, Z displacements are fully compensated in the tip of the gangway. This is achieved by a combination of luffing, slewing and telescoping the gangway. This will make the gangway tip motionless with respect to the offshore structure. The advantages of this mode are an increased connection window, gentler landings and safer connections.

In general, the system is often equipped with both types of motion compensation. Active motion compensation will be used for the connection procedure of the gangway to the offshore structure. When the connection is made, the system switches to passive motion compensation for efficient operation. Nowadays there is a wide variety of motion compensated gangways available. These systems can be divided into two categories. The first category compensates the gangway structure in all six degrees of freedom and therefore keeps the entire gangway motionless with respect to the offshore structure. An example of this type of motion compensated gangway is shown in Figure 4. This type of gangway is patented by Ampelmann and this concept is based on the working principle of a flight simulator, but now compensates the motions instead of generating them. The second category only compensates the gangway structure in three degrees of freedom and therefore only keeps the tip of the gangway motionless with respect to the offshore structure. For this design only three degrees of freedom needs to be actively controlled. An example of this type is shown in Figure 5.

Figure 4: Motion compensation in 6 DOF (Ampelmann.nl)

Figure 5: Motion compensation in 3 DOF (Motustech.no)

The design with motion compensation in three degrees of freedom is used by almost all manufacturers. The reason for this is that the reduction in degrees of freedom reduces the amount of required actuators. This reduces the complexity and cost price of the structure and the required amount of power to operate the structure. Even Ampelmann started providing this type of gangway.

For the Ampelmann principle, which compensates the motions in six degrees of freedom, the maximum allowable motion of the cylinder is influenced by the state of the other five cylinders. For example: If the platform is pitching, then the maximum allowable heave compensation is limited due to the cylinder extension for the pitch motion. This problem could be reduced by using cylinders with a larger stroke but this would increase the buckling behaviour of the cylinders. This would also increase the required power to operate the system.

For a three DOF design, the compensation of each degree of freedom is not influenced by the other degrees of freedom. This simplifies the design and increases the workspace of the device. A large workspace is preferred, because it will allow the system to operate under various conditions.

For a system it is important that it is not operating at the edge of its limitations. Because then the risk could arise that it would reach its geometric limit. This geometric limit is the configuration of the system in which one or more DOF are locked and the system is unable to reach the desired configuration. To guarantee safety, regulations are defined which must be satisfied by the device. These certification guidelines are made by ABS [24] or DNV-GL [25].

2.4 Working principle

First we start by explaining the different parts of the motion compensated gangway. All the different parts are shown in Figure 6.

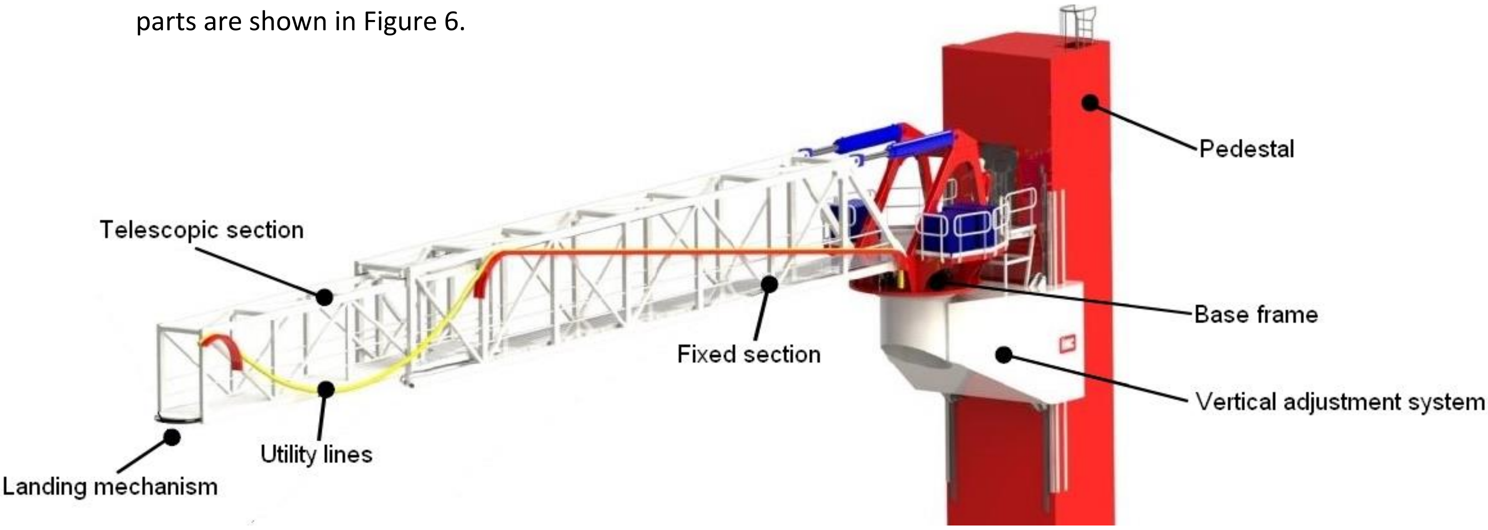


Figure 6: The motion compensated gangway *(Royal IHC)*

The first part that we consider is the pedestal or mast. It is a deck mounted pillar which allows the gangway to translate to the nominal operational height in order to keep the gangway as horizontal as possible. The pedestal transfers the forces from the gangway boom to the deck of the ship and serves as a foundation for the gangway structure.

The elevator is installed in the pedestal and its function is to transport the cargo and crew from the main deck to the gangway transfer deck level. The vertical height of the gangway can be adjusted by the vertical adjustment system. The pedestal mast is equipped with a rack and pinion system and therefore the vertical adjustment system is able to move vertically on the pedestal.

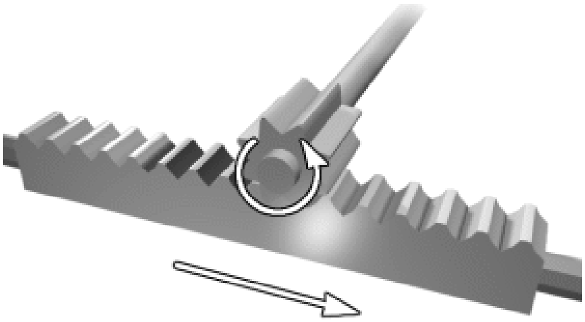


Figure 7: Rack and pinion system *(Wikimedia.org)*

The base frame is the steel structure between the pedestal and the gangway. It is the base structure to which the gangway main boom is attached. It transfers the forces and moments from the gangway boom to the vertical adjustment trolley. The base frame is integrated into the vertical adjustment system and consists of a slewing bearing which allows the gangway to rotate around its Z-axis (also known as the slewing motion). This mechanism allows the system to slew and deploy the gangway either to starboard (SB) or portside (PS). The gangway is hinged at the base frame which allows the luffing motion of the gangway. The base frame provides temporary storage for euro size pallets or for people to wait until they can safely pass the gangway.

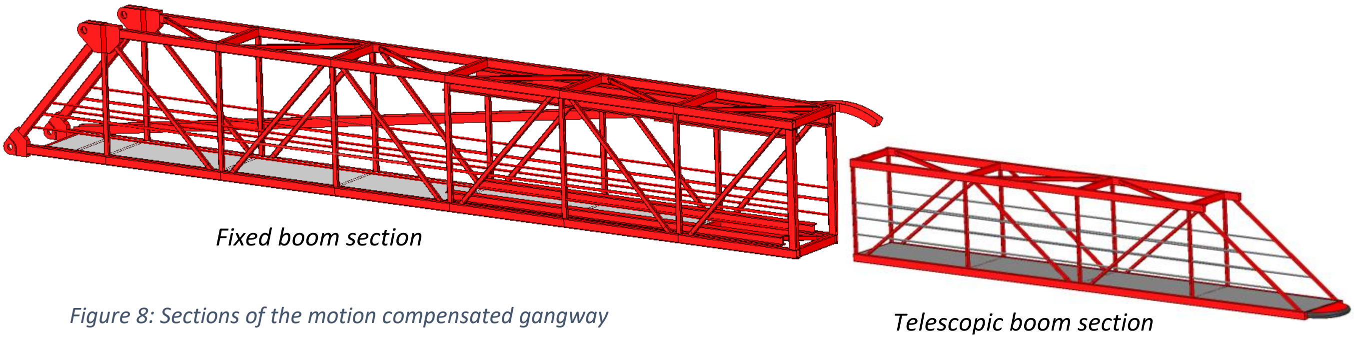


Figure 8: Sections of the motion compensated gangway

The gangway boom consists of two parts, the fixed part and a telescoping part as can be seen in Figure 8. The main body of the gangway is a welded lattice structure with one end attached to the base frame. It is the pathway through which the telescopic boom extends. The main boom is fitted with linear guidance rollers in order to guide the dynamic linear motion of the telescopic boom. A double ended hydraulic winch is used, around which the cable is reeved several times. By reeving the cable several times, the required stroke of the cylinder is decreased. An advantage of this is that the system is tensioned automatically by the winch, the controllability is increased and the influence of the cable wear and stretch is reduced. The working principle of this system is shown in Figure 9.

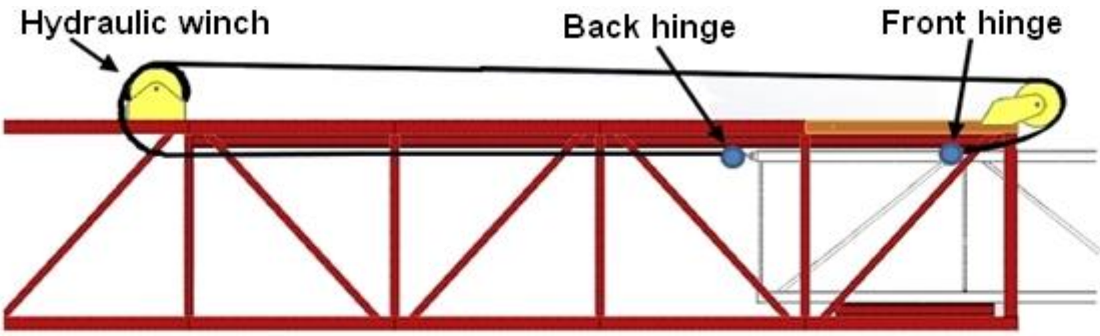


Figure 9: The gangway telescopic system *(Royal IHC)*

The linear guidance rollers are visualized in Figure 10. Two pairs of rollers are located on the fixed section of the gangway structure. Two pairs of rollers are located on the telescopic section of the gangway structure. The reason for this layout is that in this orientation, the roller bearings on the left side will move with the stroke of the bridge. The roller bearings on the right side are always located on the end of the fixed section. Therefore in this orientation, two sets of roller bearings will always be in contact with the gangway structure.

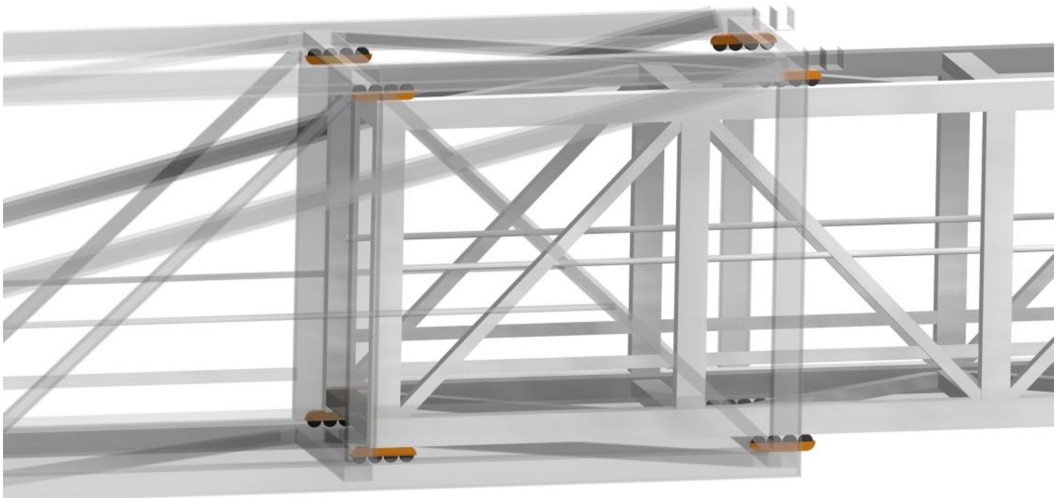


Figure 10: The linear guidance rollers

The landing mechanism is located at the tip of the telescoping boom. The function of the landing mechanism is to provide a safe and firm connection of the gangway with the offshore structure. There are different designs of landings mechanisms available:

-*Rubber bumper*: A rubber bumper located at the tip of the gangway structure is pushed with an adjustable force onto the offshore structure. In most cases it has a round shape to ensure no torsional loads are transferred to the gangway. The friction between the rubber bumper and the offshore structure enables the required vertical support at the tip of the gangway.

-*Landing cone*: A landing cone with a rotatable joint is located at the tip of the gangway. The rotatable joint makes it able to connect the gangway at different angles. Often the landing cone is locked to the offshore structure after deployment.

-*Gripper mechanism*: A mechanical gripper located at the gangway tip is connected to a static pole mounted on the offshore structure, or to the boat landing. This ensures a continuous connection of the gangway to the structure.

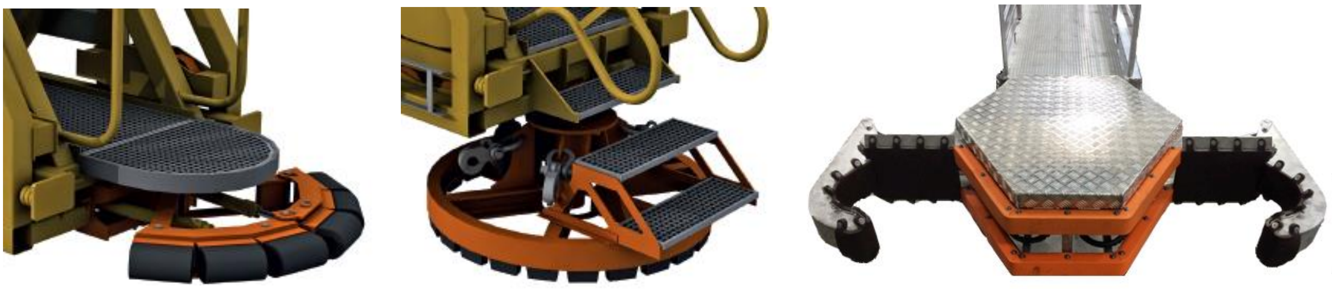


Figure 11: Different landing mechanisms for the gangway *(kenz-figee.com)*

3. Required and current guidelines

This chapter is concerned with the required guidelines available for offshore design. These guidelines will determine the boundary conditions for the optimization process. An overview of the most common guidelines is given. A recommendation is given for the set-up of the topology optimization.

3.1 Guideline requirements

There are different technical guidelines available for offshore design. For example the ABS or the DNV-GL guidelines. These guidelines ensure that offshore units are safe, efficient and reliable. Clients do often demand that the designed structure is certified according to these guidelines. When a product complies with the criteria stated in the guidelines, it is certificated. Product certification assists users to select quality products, which comply with the relevant European standards. It guarantees that the product which is sold, is identical to the tested product and that the products are fully tested according to all the relevant standards. The ABS guidelines are developed in the US and the DNV-GL guidelines originates from Norway and originated from the corporation between DNV and Germanischer Lloyd (GL) These institutes are independent bodies which examine the product and make a statement about its compliance. These guidelines help the manufacturer to certify their product for a relatively large market and to get their product accepted in many countries. It results in a simpler procedure for testing and introducing new products in many countries. The application of the DNV-guidelines will be explained in the next section. The ABS-guidelines are not often used in Europe and therefore can be found in appendix: A-7. The ABS-guidelines. The ABS guidelines are quite similar to the DNV guidelines.

3.2 The DNV-guidelines

For the design of motion compensated gangways, the DNV-GL has designed a special guideline named: *Certification of offshore gangways for personal transfer.* This document consist of all the requirements for designing and maintaining a motion compensated gangway. The most important design considerations will be discussed in this chapter. These regulations will influence the design of the gangway and need to be incorporated into the design in order to certificate the structure.

The material specifications

The materials for the gangways have a design temperature down to -20°C. The design temperature is the lowest acceptable service temperature. The used materials shall be adequately marked for identification. There are specially approved suppliers and manufacturers which can supply materials which are certified by the DNV. Certificates needs to be provided covering the specification and the chemical composition and mechanical properties of the material. An impact test needs to be performed at the operational temperature for the structural material. These test values shall show conformity with the approved specifications. The test specimens shall be taken from the products delivered.

The loading conditions

According to the guidelines, the gangway under consideration is a type 2 gangway. This means there is a controlled flow of people. The flow of people is regulated by means of an operator. The connection time is usually less than 24 hours. At least one end of the gangway is supported in the X, Y and Z-direction. In operation mode it is supported at both ends. The type of gangway determines the different load cases and acceptance criteria to which the gangway must be subjected. There are five general load combinations to be considered:

1. Normal working condition, gangway in operation mode. (Three different cases)
2. In uplift situation, deployment or retrieval case. (Two different cases)
3. Emergency lift off/disconnection.
4. Parked position.
5. Load test.

The load cases are shown in Appendix: A-6. Input parameters optimization process. The five load combinations result in eight different load cases. For every load case the principal loads, loads due to climatic effects and the loads due to the motions are defined. The principal loads consist of the weight of the components and the different live loads. Live loads are for example the personal on the gangway. The loads due to climatic effects consists of wind loads and green sea loads. The loads due to the motions originates from the accelerations of the vessel and the gangway itself.

The design wind velocity and pressure shall be based on the highest 3 second gust wind speed expected to occur at the gangway location. For the gangway, the following cases will be analysed as shown in Table 2. The wind velocities for the different loads conditions are given in Table 1.

Table 1: Wind velocities at different load conditions

Load condition	Wind velocity [m/s]
Operational wind speed	20 m/s
Deployment/retrieval wind speed	36 m/s
Transit/survival/parked wind speed	44 m/s

The permissible stresses with respect to yielding and buckling are given. The gangway structure must be designed to satisfy these criteria for all the load cases. The acceptance criteria consist of a safety factor for the elastic analysis of the permissible stresses or with respect to elastic buckling. This is shown in Table 3 and Table 4.

The safety of the structure shall be evaluated for the load combinations defined in Table 2. Each of these load cases must be checked with the most unfavourable position and direction of the forces. The strength calculations are based on accepted principles of structural strength and strength of materials. The verification of the safety may be based on the limit state method (LRFD) or on the working stress analysis (WSD) method.

Table 2: The load cases

	LC 1a	**LC 1b**	**LC 1c**	**LC 2a**	**LC 2b**	**LC 3**	**LC 4**	**LC 5**
	Normal working condition	Normal working condition	Normal working condition	Deployment/ retrieval	Deployment/ retrieval	Emergency disconnection	Parked/transit/ survival	Load test
Self-weight (G)	G	G x MOA	G x MOA	G x DF	G x (DF + MOA)	G x (DF + MOA)	G X MTA	G
Live load (LL)	LL	LL	120kg					Test load
	2 x LL	2 x LL x MOA	2 x LL x MOA					
Live load (LL) (applied at the tip of the gangway)						F≥350 kg		
						F x (DF_z + MOA)		
Bumper loads	100%	100%						
Centrifugal force				100%	100%			
Green sea loads							100%	
Wind loads		Operational wind speed	Operational wind speed		Deployment/ retrieval wind speed	Deployment/ retrieval wind speed	Parked/transit/ Survival wind speed	
Acceptance criteria	I	II	II	I	II	III	II	Maximum deflection

MOA = Maximum operational accelerations.
MTA = Maximum transit/parked accelerations.
G = Gangway self-weight and all installed equipment.
DF_y/DF_z = Dynamic factor to vertical/horizontal loads due to operational motions.
LL = Live load.

The vessel motions are dependent on the type of vessel on which the gangway is installed. It also depends on the location of the gangway on the supporting vessel. The MTA and MOA values can be obtained from the extreme values for the acceleration of the supporting vessel.

Below, the five generic load combinations are considered. The definition of each load case will be given in this section. The acceptance criteria of the load cases depends on the maximum permissible stresses which can be found in Table 3. The acceptance criteria for the test load case depends on the maximum deflection.

-Normal working condition
In the normal working condition the motion compensated gangway is in operational mode. This means that the gangway structure is able to transfer people from and to the offshore structure. For the normal working condition the gangway shall be designed for the most onerous of the following two scenarios:

1. The live load (LL) on the gangway shall be the maximum number of persons, including hand tools and luggage allowed on the gangway at the same time. The load will be applied at the most limiting location.

2. A load applied on the gangway tip which is equal to 120 kg when the gangway is in uplift/cantilever position at is maximum length.

The design load of the gangway shall be equal to two times the live load.

-Deployment or retrieval case
In this case the gangway is in uplift condition. No live loads are acting on the gangway structure. The principle loads on the gangway consists of the self-weight and additional weight acting on the gangway structure. The wind load and the horizontal and vertical loads due to the operational motions must be included. The centrifugal force based on the maximum angular velocity and the radius of the considered mass should be included in this situation.

-Emergency disconnection case
For the emergency disconnection case, the principal loads are applied. The principal loads consists of the loads from the self-weight of the gangway structure and the loads due to the live loads. The live load shall be applied at the tip of the gangway. The inertia forces shall be taken into account by multiplying the self-weight of the gangway with the sum of the dynamic factor DF_z and the maximum vertical operational acceleration. The loads due to the motion of the vessel shall be included. The gangway is in uplift/cantilever position at its maximum length (including the safety length). The live load on the gangway tip shall be at least 350 kg.

-Parked/transit position
In this mode the gangway is completely pulled-in and supported at the free-end in a cradle or bridge rest. The gangway is secured in a sea-fastening frame. In this case the principle loads consists of the self-weight of the gangway. The survival wind loads are applied and the maximum transit accelerations are included in this situation.

-Load test
This test case will be executed before a gangway is put into service. In this case the gangway will be extended to its maximum length and is supported in the vertical direction at both ends. A test load which is equal to 1.25 times the live load will be applied at the middle of the gangway. The acceptance criteria in this load case is the maximum deflection of the gangway structure.

Excessive yielding

The criteria for the check with respect to excessive yielding is given in Table 3. σ_y is the guaranteed minimum yield strength and defined as: $\sigma_y = \sigma_u \cdot 0.8$ in which σ_u is the ultimate yield strength of the material.

Table 3: Criteria for checking with respect to excessive yielding

Criteria for checking with respect to excessive yielding				
Method of verification		Acceptance criteria I	Acceptance criteria II	Acceptance criteria III
Safety factor	Elastic analysis	1.50	1.33	1.10
	Plastic analysis	1.69	1.51	1.25
Permissible stresses	Elastic analysis	$\sigma_y/1.50$	$\sigma_y/1.33$	$\sigma_y/1.10$

When the actual stresses on the structure are lower than the admissible stresses, then the criteria with respect to excessive yielding is satisfied. The required acceptance criteria depends on the method of verification and the load case.

Buckling criteria

The next criteria is to check the structure with respect to buckling. The principle is that the safety against buckling shall be the same as the required safety against the yield limit load being exceeded. This principle indicates that the safety factors for the elastic analysis of the permissible stresses should present the normal requirement. These acceptance criteria is based on the assumption that the loads are determined by recognized methods, taking possible effects of geometrical imperfections and initial stresses into account. But if the determination of the critical stresses or loads are uncertain, then the required factors for various types of buckling are given in Table 4. Elastic buckling in this case means that the elastic buckling strength does not exceed the yield strength.

Table 4: Safety factors with respect to buckling

Criteria for checking with respect to buckling			
Type of buckling	Acceptance criteria I	Acceptance criteria II	Acceptance criteria III
Elastic buckling	1.86	1.66	1.38
Elastic-plastic buckling	1.69	1.51	1.25

Maximum deflection of the gangway

The gangway will be tested on its maximum deflection by a bridge load test. The gangway will be tested at its maximum length, so when it is fully extended. A test load (TL) equal to 1.25 times the LL shall be applied at the middle of the gangway. For a gangway which is supported on both ends, the maximum deflection in the middle of the gangway may not exceed L/200. In which L is the length of the gangway. G is the self-weight of the gangway. For cantilever gangways, the maximum deflection at the gangway tip may not exceed L/100. The supporting conditions of the gangway depends on the type of landing mechanism that is used. The test shall not cause permanent deformation of the gangway.

Table 5: Deflection criteria gangway

Support condition		**Limit for δ_{max}**	**Limit for δ_2**
Gangway supported at both ends	G<2*TL	L/200	L/300
	G=2*TL		L/400
	G>2*TL		L/600
Cantilever gangway	G<2*TL	L/100	L/150
	G=2*TL		L/200
	G>2*TL		L/300

In Figure 12 a schematic overview of the gangway is given. In which δ_1 is the deflection of the gangway due to its self-weight and δ_2 is the deflection due to the applied test load. The limiting values for δ_2 and for δ_{max} can be found in Table 5.

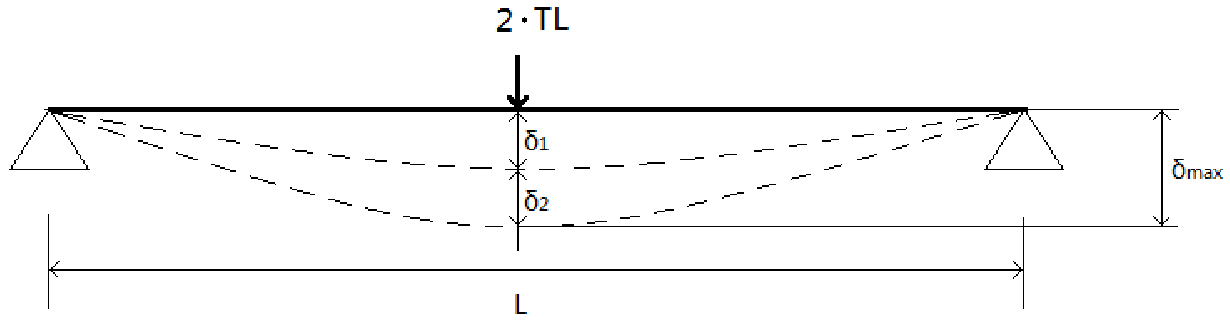


Figure 12: Schematic overview gangway

The gangway must be delivered with an operation manual which describes all the relevant aspects related to the gangway like: operation modes & limitations, redundancy, emergency procedures and maintenance routines.

Safety Length

For the telescoping gangways, the length of the gangway and the arrangement shall be such that there is a minimum free length or movement reserve available, beyond the gangway's maximum operational stroke. This ensures that the gangway is not operating at its geometric limit. This must hold for each direction. The reserve length can be calculated with equation *3.1* in which L is the maximum gangway length in meters.

$$Reserve\ length = (1 + (L - 20)/50) \tag{3.1}$$

Dimensions gangway

The dimensions of the gangway are defined as follows: The free internal walking height shall be a minimum of 2.1 meters, the clear internal width must be minimal 0.6 meters for a type 2 gangway but for the transport of cargo a larger width is preferred. Toe boards of at least 100 mm high shall be fitted on either side of the walkway. The gangway must be designed with handrails on both sides of at least 1 meter high. The stanchions shall be spaced not more than 1.5 meter apart and must have at least 3 courses. The opening below the lowest course of the handrails shall not exceed 230 mm. The other courses shall be not more than 380 mm apart. The handrails must have smooth surfaces. The edges must be rounded or be chamfered.

The operational angel of the gangway (luffing angle) may not exceed 10 degrees. 20 degrees is allowed if the gangway is fitted with enhanced slip resistance features. When the maximum luffing angle of the gangway is exceeded by 5 degrees, an alarm must go off.

3.3 Client requirements

Client requirements can be integrated in the design of the gangway structure. For the transfer of cargo over the gangway. It is preferred to increase the width of the gangway in order to move euro pallets from the vessel to the offshore structure. The pedestal can be equipped with an elevator, this enables a stepless access to the gangway and allows for an easy and smooth transfer of cargo and crew. The gangway can be provided with hose provisions for the transfer of grout, hydraulic power or chemicals. Another important aspect is the slewing angle of the gangway. Client prefers to have a large slewing angle so they are able to operate under all conditions. The available slewing angle depends on the design of the gangway and the deck-layout. At the tip of the gangway, a lifting winch can be placed and the gangway can be used as a lifting winch. A lifting winch will lead to an additional load case.

3.4 Boundary conditions

The structural design of the gangway structure is checked against DNVGL-ST-0358 Certification of offshore gangways for personnel transfer [25]. The gangway is classified as a structure and needs to be designed according to the following design criteria:

Permissible stresses:

- Case I: Gangway working without wind load σ_y / 1.50
- Case II: Gangway working with wind load σ_y / 1.33
- Case III: Gangway subjected to exceptional loadings σ_y / 1.10

The yield strength (σ_y) of the structure is based on the technical purchase requirements of the construction materials. Corresponding yield reduction is taken into account. The following load cases apply to the gangway structure as can be seen in Table 6. Some load cases are overruled by other cases, where this is applicable is mentioned in the last column. The load cases which are overruled will not be used in the analysis because they are already covered by the other load cases. These load cases will be used during the optimization and validation process and can be seen in Table 7.

Table 6: The load cases

Load case	Description	Self-weight	Live load	Tip-load	Bumper loads	centrifugal force	Wind loads	DNV case	Accel.	Overruled by LC
[-]	[-]	[N]	[N]	[N]	[N]	[N]	[N]	[-]	Hs [m]	[-]
$LC1_a$	Normal working condition	G	2 x LL	-	100%	-	-	Case I	0 - 3.5	LC1c
$LC1_b$	Normal working condition	G x MOA	2 x LL x MOA	-	100%	-	Operational wind load	Case II	0 - 3.5	LC1c
$LC1_c$	Normal working condition	G x MOA	2 x LL x MOA	-	-	-	Operational wind load	Case II	0 - 3.5	-
$LC2_a$	Deployment/ retrieval	G x Df	-	-	-	100%	-	Case I	0 - 3.5	LC2b
$LC2_b$	Deployment/ retrieval	G x (Df+MOA)	-	-	-	100%	retrieval wind load	Case II	0 - 3.5	-
LC3	Emergency disconnection	G x (Df+MOA)	-	LL	-	-	retrieval wind load	Case III	0 - 3.5	-
LC4	Parked/ survival	G x MTA	-	-	-	-	transit wind load	Case II	Survival	-
LC5	Load test	G	Test load	-	-	-	-	Max deflection	0 - 3.5	-

A reduction in the amount of load cases is preferred because it reduces the amount of effort which is required to prepare these load cases into the optimization and validation process. A reduction in load cases does also reduce the required solving time for the optimizer.

In order to reduce the amount of load cases for the optimization and validation of the gangway structure, it is chosen to change the acceptance criteria for some load cases. Load cases 1a,1b and 1c are quite similar, they all refer to the normal working condition. Load case 1a has the most strict acceptance criteria for the allowable stress, but the other two load cases includes the operational accelerations. Load case 1b does also include bumper loads. The landing tool for the gangway structure is a landing cone which does not require any bumper load. This means that no bumper loads are applicable during the load cases and therefore this value may be neglected. For load case 1c, It is chosen to adapt a safety factor of 1.5 in order to cover the first two load cases. Load case 1c will be subjected to DNV case I instead of case II which means that the allowable stress is reduced by 11.1% (From 1/1.33 to 1/1.5). The wind load is considered as a non-dominant load due to the fact that it acts on an open truss structure. Therefore the wind loads will be included in the validation of final design.

The two deployment and retrieval load cases are also quite similar. These load cases include the centrifugal forces acting on the gangway. From the calculation according to the DNV, it is calculated that the centrifugal force of the gangway is only 1300 N. This is quite low compared to the other loads acting on the structure. If we compare this to the reaction force due to the accelerations of the structure in the longitudinal direction, which is equal to 12502 N. Therefore this centrifugal force is only 6,9% of the force due to the longitudinal acceleration. Therefore we can conclude that this force can be neglected compared to the mass inertia of the structure. To reduce the deployment and retrieval load cases, it is chosen to subject load case 2b to an acceptance criteria of 1.5 in order to overrule load case 2a. This also means a reduction of 11.1% on the allowable stress.

The emergency disconnection load case is a standalone load case. This load case has lower acceptance criteria and is only subjected to a safety factor of 1.1 for the allowable stress. In this load case a tip load is applied which is equal to the live load acting on the structure. Therefore it is not certain if this load case is overruled or does overrule any load case and therefore it is chosen to include this load case in the optimization and validation of the gangway structure.

The final two load cases are the survival load case and the load test. The parked/survival load case is considered to be not the most severe load case. This is due to the fact the gangway in this situation is retracted and located in the middle of the ship. Also it is supported and secured on both ends. Therefore it is chosen to only analyse this load case during the FEA. The load test needs to be performed when both sections of the gangway structure are designed. Then the structure will be analysed using FEA to check if it satisfies the constraints. Also the deflection limits are incorporated into the design constraints of the other load cases. Therefore this load case should be overruled by the other load cases. With these assumptions and considerations it is possible to update Table 6 to obtain a reduced amount of load cases which apply to the gangway structure. The load case reduction is given in Table 7.

Table 7: Load case reduction

Load case	Description	Self-weight	Live load	Tip-load	centri-fugal force	Wind loads	DNV case	Accel.	Comments
[-]	[-]	[N]	[N]	[N]	[N]	[N]	[-]	Hs [m]	[-]
1c	Normal working condition	G x MOA	2 x LL x MOA	-	-	Operational wind load	Case I	0 - 3.5	
2b	Deployment/ retrieval	G x (Df+MOA)	-	-	100%	retrieval wind load	Case I	0 - 3.5	
3	Emergency disconnection	G x (Df+MOA)	-	LL	-	retrieval wind load	Case III	0 - 3.5	
4	Parked/ survival	G x MTA	-	-	-	transit wind load	Case II	Survival	Only FEA & hand calc.
5	Load test	G	Test load	-	-	-	Max deflection	0 - 3.5	Only FEA & hand calc.

From this table we can conclude that five load cases must be analysed. Three load cases are overruled by other load cases. Load cases 4 and 5 will only be considered during the validation of the design. Load case 1c, 2b and 3 will be included in the optimization process as with the validation of the design. For all the optimization steps, wind load is considered to be neglected, the wind loads will only be included in the validation of the detailed design. The calculation of the values which corresponds to the abbreviations in Table 6 can be found in A-5. Design calculations gangway. Also the load cases are extensively described in this section.

4. Optimization methods

In this chapter, an explanation is given of the general mathematical concepts used for structural topology optimization. The conventional design process is presented which is followed by an explanation of the different topology optimization methods. At last, an overview of the different topology optimization software packages is given with some examples of topology optimization.

4.1 The definition of topology optimization

Before the optimization methods are explained, the definition of optimization is given [35].

Optimization is defined as:
"The procedure used to make a design as effective or functional as possible" [27].

Topology is an example of an optimization technique and is defined as:
"The study of certain properties that do not change as the geometric figures or space undergo continuous deformation"

And finally structural optimization can be defined as:
"The rational establishment of structural design that is best of all possible designs within a prescribed objective and a given set of limitations"

With all these definitions, structural topology optimization can be defined as:
"A mathematical process whereby the location and number of voids within a structure is defined given a prescribed set of limitations and a clear objective"

Structural optimization consists of a process of determining the best material distribution within a volume domain, to safely transmit or support the applied loading conditions. The principle of optimization is a method to find the best possible solution under certain circumstances [55]. For structural optimization, this is the optimal distribution of the material that satisfies the given requirements.

4.2 Types of structural optimization

There are different types of structural optimization. These are for example size- and shape optimization. Size optimization defines the ideal component parameters, like the cross-section dimensions and thickness to determine the ideal size and thickness of a design based on the stated criteria [28]. This method is suitable for a design, where the size of the structure is required while all the other aspects of the structure are already known. Shape optimization determines the optimal shape of a structure while satisfy given constraints. The form or contour of the part inside the structural domain is determined. Shape optimization can manipulate the shape of the boundaries, but it cannot create new boundaries or change the connectivity within the structural domain. Topology optimization is the most general form of structural optimization. Topology optimization determines the material and connectivity in the design domain. It determines the number and sizes of holes in the design domain. This procedure consists of a topology optimization which is followed by shape optimization and finished with sizing optimization.

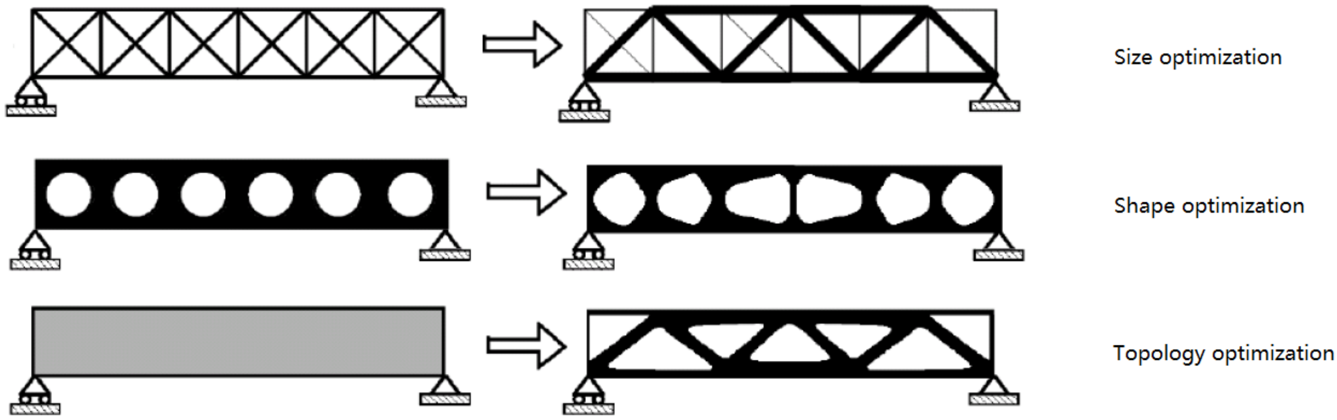


Figure 13: Types of structural optimization *(www.sciencedirect.com)*

In size and shape optimization, the dimensions and form of the structure can have any value between the boundaries. When the shape and size of the structure are unknown, topology optimization can be used.

4.3 Topology optimization

Topology optimization is a mathematical method that optimizes material layout within a given design space for a given set of loads, boundary conditions and constraints [20]. Topology optimization has two major distinctive features [28]:

> -The elastic property of the material is a function of a certain variable which can vary on the complete design domain.
> -The material can be removed from the design domain.

Currently there are several topology optimization techniques available which can be divided into two main categories [28].

1. Density-based methods.
2. Heuristic or intuitive methods.

The *density-based method* is an indirect method of optimization. This method satisfies a set of constraints which determines the behaviour of the structure. This method is suitable for design problems with a large number of design variables.

Heuristic methods are derived from intuition, observation or biologic systems. These methods cannot always guarantee optimality. Examples of heuristic optimization methods are: Fully stressed design, Computer-aided optimization, soft kill option, Evolutionary structural optimization (ESO) and bidirectional ESO. Due to the efficiency of the heuristic methods, will these methods not be analysed or applied in this master thesis. Also these methods have no mathematical substantiation. Therefore effort will be spend into the optimality criteria methods for topology optimization which are often included in commercial software.

In general, the topology optimization is carried out in three steps. First a design domain is specified and the boundary conditions and loads are assigned to this design domain. The design domain is discretized in a mesh for the FEA. Secondly, the design evaluations takes place by calculating the governing state equations. In structural optimization the governing equation is often the stiffness equation. The result of the design evaluation is used to calculate the sensitivity of each element and is decided if they contribute in minimizing the objective. In the third step the sensitivities of the objective with respect to the design variable is used in a search algorithm. These three steps are repeated until a certain convergence criterion is met.

The most common objective for structural engineering it to reduce the compliance and therefore increasing the stiffness of the structure. The most used method for topology optimization is the density-based method. A popular density-based optimization method is called the Solid Isotropic Materials with Penalization (SIMP) method. This method works with a discretized domain, consisting of small mesh elements. The density of each element is related to the stiffness of the material. The algorithm determines if a mesh element needs to be a void or must contain material. Structural optimization is the optimization process which enables the best design within a prescribed objective given a set of limitations. Many of the optimization processes deal with continuous functions for which variable calculus is used to find the location of the optimal solution. This is achieved by searching for the location of an extrema in a function. In the conventional density-based approach there are density variables assigned to every mesh element and therefore the Young's modulus becomes a function of the local density. So the stiffness becomes a function of the local density. Unfortunately this would lead to an infinitely fine porous microstructure and this is impractical. To avoid this problem, the solution can be restricted to pure solid/void designs. This means that there can be material in a certain location or there can be no material in that location. This restriction will transform the problem into a discrete problem. This makes it difficult to solve this optimization problem because the optimum solution cannot be found using variable calculus.

4.3.1 Conditions for optimality

An optimal design is an abstract definition. In what context is it an optimal design? The criteria for optimality must be considered carefully. Different conditions for optimality are [35]:

- *Fully stressed design*: The situation where all elements in a design utilize their full strength.
- *Minimum compliance*: Design for the minimum deformation/deflection of a structure.
- *Minimum weight:* Design for the minimum weight of a structure for a certain load condition.

The most common solution is to design for minimum compliance. For a given amount of material, the stiffest structure is found. The structural optimization problem can be formulated by an objective function, design variables and state variables.

The goal of an optimization problem is to determine the design variables which minimize or maximize the objective while satisfying all the constraints. This objective could be for example the stiffness or volume of a structure. The solution of the optimization process depends on a particular set of design variables which needs to be expressed with a numerical value. These design variables are variables by which the design problem is parameterized. Given as x_1 till x_n in this example. The objective is the quantity that has to be minimized or maximized and this is usually denoted by the cost or objective function $f(x)$. The cost function is a function of the design variables. The constraints are the conditions that have to be satisfied. The optimization problem is most often formulated as a minimization or maximization of the cost function subjected to constraints. This is expressed as [55]:

$$\text{Find } X = \begin{Bmatrix} x_1 \\ x_2 \\ \dots \\ x_n \end{Bmatrix} \text{ Which minimizes } f(x) \tag{1.1}$$

$$\text{Subject to} \quad K(x)U = F(x) \quad \} \text{ Equilibrium constraint} \tag{1.2}$$

$$\left. \begin{cases} g_i(x) \leq 0, i = 1,2,3, \dots, m \\ h_j(x) = 0, j = 1,2,3, \dots, n \end{cases} \right\} \text{Constraints}$$

Where X is the vector containing design parameters (x_n) and $f(x)$ is the cost or objective function. The objective function is subjected to the governing state equation and constraints. In structural optimization the state equation is the stiffness equation. The goal of the optimization process is to find the design parameters x for which the objective function $f(x)$ is minimized. The functions $g_i(x)$ and $h_j(x)$ are called the inequality constraint functions and the equality constraint function respectively and they define the constraints of the problem. For example the minimum and maximum values for the density or the maximum available volume in the design domain. This is called a constrained optimization problem. The optimization problems are typically solved using an iterative algorithm.

An example of this iterative algorithm is given in Figure 14. First we have a simulation model, commonly a numerical model. The input for this model are the design variables and constraints. The output of this model is called the response. These responses are used as an input for the optimizer. Also the derivatives of these responses are desired, because they provide information on the response of the output as a function of the input. These derivatives are called the design sensitivities. The optimizer determines the change in the design variables which are used as new input value for the model. The optimization algorithm changes the design based on these design variables. This cycle may be repeated many times before the solution convergences to the optimal solution. This can make the optimization process time consuming and therefore quite expensive.

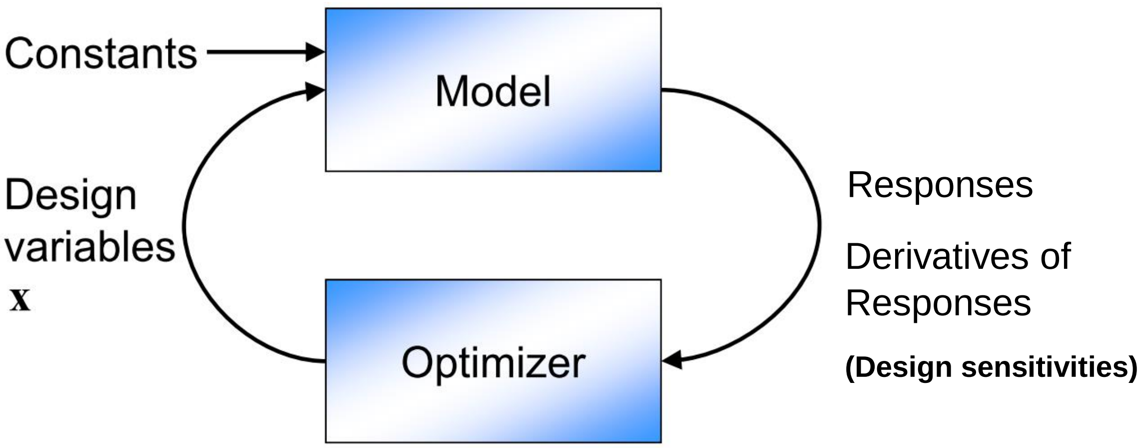


Figure 14: The iterative process of an optimization problem.

4.3.2 The SIMP method

There are different methods available for topology optimization. The SIMP method will be explained in this chapter. *SIMP* stands for: ***S****olid* ***I****sotropic* ***M****aterial with* ***P****enalization*. This method can be used for generating an optimal design for minimum compliance.

The optimization problem can be solved by allowing the material to take any value between zero and one. This is called relaxation of the problem and this allows the use of gradient based optimization methods. Relaxation means that the discrete problems are converted into equivalent continuous problems. The penalization is used to recover the continuous problem into a discrete solution. SIMP is the most common method to relax the problem. Relaxation is a modelling strategy that approximates the difficult problem to a nearby problem which is easier to solve. In order to obtain a discrete function, the values are penalized with a penalization factor. In this case the penalization consists of raising the values of the density to a power of the penalization factor. This will tend the behaviour of the continuous function towards the behaviour of the discrete function. But this penalization method is not able to reproduce the perfect discrete structure. The SIMP approach uses penalization to make intermediate densities unattractive. Because they have no physical significance in structural optimization. It forces the design into a solid/void solution and lowers the stiffness/ weight ratio for intermediate densities as can be seen in Figure 15. [30]

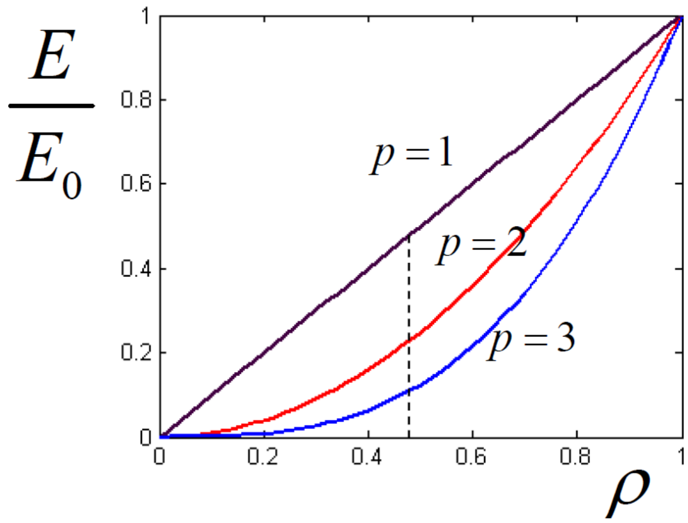


Figure 15: Stiffness-weight ratio for different values of p. (M.Langelaar)

SIMP is an interpolation scheme which allows the topology problem to be converted into a sizing problem on a fixed domain. The sizing problem has a large amount of design variables. The process of the SIMP method is depicted in Figure 16. It starts with an initial guess which distributes the material over the solution domain. This can be for example a homogeneous distribution of the material. The solution domain is the area in which the solution can be found. There is a finite amount of material which can be used in the solution domain. This solution domain is specified by the user and is in the most cases just a simple shape in which the solution can be found. The ratio of the material to the solution space is called the volume fraction.

An optimal material distribution must be found in the available design domain. In mathematical terms this can be written as: $\Omega_{mat} \subset \Omega$. In which Ω is the available design domain. The design variable x is represented by the density vector ρ. This vector contains the densities of all elements denoted as ρ_e. The local stiffness tensor E can be formulated by relating the density to the original elastic modulus of the design material.

$$E(\rho) = \rho E^0 \qquad (2.1)$$

$$\rho_e = \begin{cases} 1\ if\ e \in \Omega_{mat} \\ 0\ if\ e \in \Omega \,/\, \Omega_{mat} \end{cases}$$

Also a volume constraint is applied to the problem, due to the fact that the volume of the solution must be smaller or equal to the volume of the design domain. This results in the following constraint.

$$\int_{\Omega} \rho d\Omega = Vol(\Omega_{mat}) \leq V_s \qquad (2.2)$$

For topology optimization with the SIMP interpolation method, two objectives can be minimized. The compliance or the volume of the structure can be minimized. The stiffness of a structure can be maximized in order to minimize its compliance and the volume of a structure can be minimized in order to obtain the lowest weight of the structure. The compliance is defined as the equivalent strain energy in the FEA which enables maximum stiffness when it is minimized. A state function constraint can be for example the displacement in a certain direction.

The objective function is the function which has to be minimized in the topology optimization. Generally this is the compliance of the structure. A constraint on the usable volume is applied on the structure. The compliance of a structure is related to the forces on the structure and stiffness of the structure. The force vector of a finite element is given by:

$$F = Ku \tag{2.3}$$

In which F is the force vector, K is the global stiffness matrix and u is the displacement vector. The compliance is the inverse of the stiffness. So $C = 1/k$ and with some basic algebra the equation can be rewritten and the compliance is given by:

$$C = F^t u \tag{2.4}$$

Equation *2.3* can be transposed to obtain the transposed force vector and this can be substituted in equation *2.4* and this results in:

$$C = u^t K u \tag{2.5}$$

The global stiffness matrix K is not transposed because it is a symmetric matrix and therefore this operation has no effect on the matrix. The SIMP method uses only one design variable per mesh element. This design variable is an artificial design density which varies between 0 and 1. The volume of each mesh element must be multiplied by this artificial density to produce its actual volume. So this means when the artificial density is equal to zero, the actual volume is zero and therefore the mesh element does not exist. This means that the actual volume of the design domain is given by:

$$V = \sum_{e=1}^{N} v_e \cdot \rho_e \tag{2.6}$$

In which N is the total number of mesh elements in the design domain and ρ_e is the artificial density of the e[th] element. v_e is the volume of a mesh element and V is the total volume of the design domain and consists of the summation of the volumes of all the mesh elements. The SIMP method places a penalty on the density when it is multiplied to the elastic modulus of the element. This will force the solution into a discrete problem.

The elastic modulus E_e can be written in terms of the artificial density as:

$$E_e = \rho_e^p \cdot E_e^0 \tag{2.7}$$

In which E_e^0 is the original elastic modulus of the material and E_e is the new artificial elastic modulus of the e[th] mesh element. P is the penalisation factor to convert the solution to a discrete solution. A penalization factor of P≥3 is recommended for materials with a Poisson ration of 0,3 [28] [56]. The value for the artificial density is in the range (0,1]. Now the assumption can be made that the global stiffness matrix K_e is a function of the relative density ρ_e^p and this results in:

$$K_e = \rho_e^p \cdot K^0 \tag{2.8}$$

In which K^0 is the local stiffness matrix for an element with a relative density of 1. Now the total compliance can be written as:

$$C(\rho_e) = \sum_{e=1}^{N} (u_e)^t K_e u_e \tag{2.9}$$

u_e is the local displacement vector. By taking the summation of the compliance over the complete structure, the total compliance of the structure is obtained.

The next step is to run an iterative process until a complete discrete design is generated. The iteratively process is formulated as [28][33]:

$$Minimize: c(\rho_e) = \{F\}^T(u) \tag{2.10}$$

$$Subject\ to: [\sum_{e=1}^{N} \rho_e^P K_e]\{u\} = \{F\} \tag{2.11}$$

$$\sum_{e=1}^{N} v_e \rho_e \leq \overline{V_s} \quad \Big\} \text{Volume constraint} \tag{2.12}$$

$$\left.\begin{aligned} 0 < \rho_{min} \leq \rho_e \leq 1: e = 1,2, \dots, N \\ P = 1,2, \dots, P_{max}: P_{max} > 3 \end{aligned}\right\} \text{Constraints}$$

In which ρ_e is the artificial density which can vary between zero and one. If $\rho_e = 0$, the mesh element is considered to be a void and if $\rho_e = 1$ the mesh element is filled with material. N is the number of mesh elements. F is the load vector and u is the displacement vector. V_s is the original volume of the design space and K_e is the (global level) element stiffness matrix. A volume constraint is added to the iterative process to prevent the optimized structure from ending up with the full design volume as a result when searching for its maximum structural stiffness. A lower bound on the design variables has been applied to avoid singularity of the stiffness matrix. First the optimum is calculated for P=1 and this value is increased until a complete discrete design is generated.

Another possibility is to minimize the volume of the structure in order to obtain the lowest weight of the structure. This can be written as:

$$V(\rho) = \sum_{e=1}^{N} \rho_e^p V_e^0 \tag{2.13}$$

In which V_e^0 is the initial volume of the mesh element. A constraint for the maximum displacement or effective stress is required in order to prevent the optimization from minimizing the weight by removing all material from the design domain. The optimization process is performed with respect to this objective function and constraints.

The optimization algorithm structure consists of the following steps [34]:

-It starts with an initial design, this can be for example a homogeneous distribution of the material. Then the iterative process starts: This initial design is entered into a finite element method to compute the displacements and strains. Sensitivity analysis is used to find the derivatives. The derivatives are used to find the displacement field, which is given implicitly in terms of the design variables through the equilibrium equation.

-The next step is the low pass filtering. This method is used to ensure the mesh independency. The mesh independency is the ability of a method to converge to the final solution irrespective of the mesh that is used.

-The SIMP optimization method can use different methods for solving the optimization problem. The optimization problem can be solved with the use of Lagrange multipliers or the method of moving asymptotes. The optimization problem is converted from a continuous problem with constraints to another problem without any constraints. This problem has extrema at the points where the original function had extrema as well but also has extrema at the location where the original problem coincides with a constraint. Now the problem can be solved with normal variable calculus.

-The final step is to update the design variable and to check if there is convergence. If there is no convergence, the iteration process continues, otherwise the results are plotted.

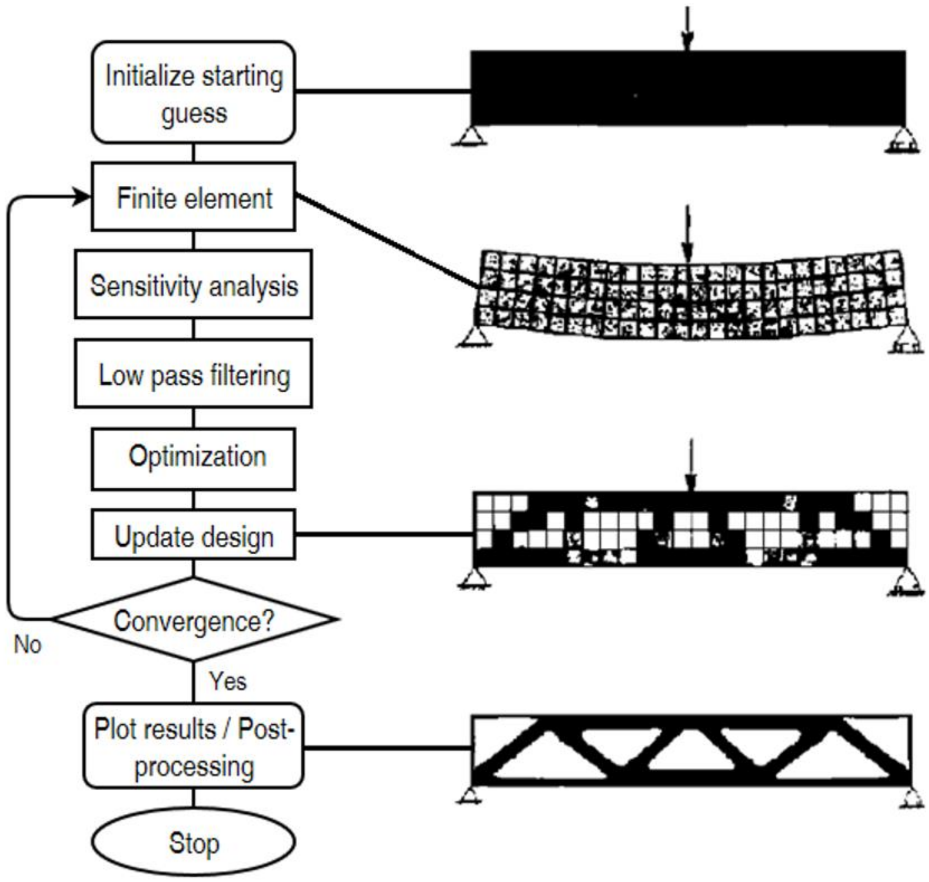


Figure 16: Optimization process flowchart

4.3.3 Checkerboard effect

A common problem that arises by using the density-based method is a phenomena called the checkerboard effect. This is the tendency for elements to mesh together into a checkerboard pattern. With this problem a checkerboard pattern is formed in which the meshing elements have alternating densities between zero and one. For many types of mesh elements this is a stable configuration in the FEA but this solution has no physical meaning. This solution is often stiffer compared to the normal material distribution, therefore it is an attractive solution for the optimizer. This effect can be suppressed by using a higher-order finite element method or using mesh refinement. A disadvantage of this is that it increases the computational effort which is required. Also a filter can be used which smoothens the density after each iteration. This is visualized in Figure 17. The density of the mesh element is than averaged with the adjacent elements. A density slope control can be applied to restrict the local gradient of the element values to a certain value. These last two methods can also be used to prevent mesh-dependency by enforcing a minimum member size.

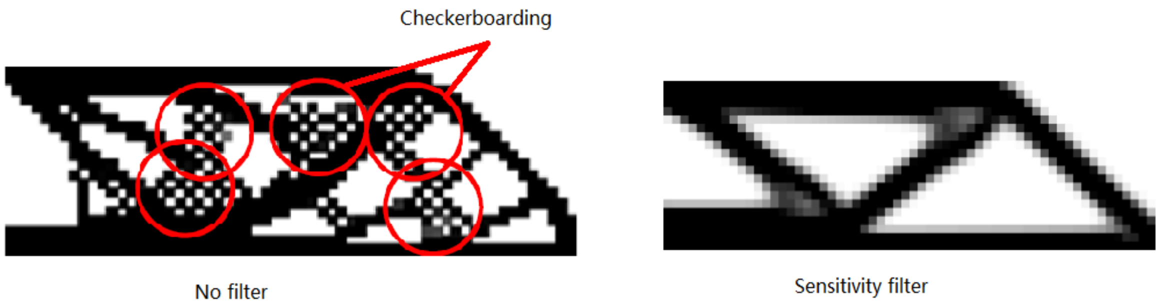


Figure 17: The checkerboard phenomena and the effect of a sensitivity filter

4.3.4 Basic equations

The residuals of the governing equation of a given system are [40]:

$$R_k\big(x_j; u_k(x_j)\big) = 0 \tag{3.1}$$

Where R_k is the residual of the governing equation, x_j are the independent design variables. u_k are the state variables that depend on the independent variables through the solution of the governing equations. So this means that the value of u_k depends on the state of the system. The number of equations must be equal to the number of state variables. Any perturbation in the variables of this system of equations must result in no variation of the residuals, if the governing equations are satisfied. This can be written as:

$$\delta R_k = \frac{\partial R_k}{\partial x_j}\delta x_j + \frac{\partial R_k}{\partial u_k}\delta u_k = 0 \tag{3.2}$$

There is a variation due to the change in design variables and due to the change in the state vector. The total derivative du_k/dx_j can be obtained by dividing the equation by δx_j.

$$\frac{\partial R_k}{\partial x_j} + \frac{\partial R_k}{\partial u_k}\frac{du_k}{dx_j} = 0 \tag{3.3}$$

The objective function h_i depends also on both x_j and u_k and therefore the total variation of the objective function is:

$$\delta h_i = \frac{\partial h_i}{\partial x_j}\delta x_j + \frac{\partial h_i}{\partial u_k}\delta u_k \tag{3.4}$$

This equation can be divided by δx_j to get an alternative form:

$$\frac{dh_i}{dx_j} = \frac{\partial h_i}{\partial x_j} + \frac{\partial h_i}{\partial u_k}\frac{du_k}{dx_j} \tag{3.5}$$

The term on the left-hand side is the derivative of the response with respect to the design variable x_j. The first term on the right-hand side determines the response of the system which depends directly on the design variable. The second term determines the response due to the change in the state of the system. The adjoint and the direct approach can be used to determine the state sensitivity of the system.

4.3.5 The direct approach.

The direct approach first calculates the total variation of the state variables u_k, by solving the differentiated governing equation for du_k/dx_j. This is the total derivative of the state variable with respect to a given design variable. This means solving a linear system of equations.

The differentiation of the state equation with respect to the design variables x_j gives [39]:

$$K(x_j)u(x_j) = f(x_j) \quad \rightarrow \quad \frac{\partial(Ku)}{\partial x_j} = \frac{\partial f}{\partial x_j} \tag{4.1}$$

This can be rewritten by using the product rule:

$$\frac{\partial K}{\partial x_j}u + \frac{\partial u}{\partial x_j}K = \frac{\partial f}{\partial x_j} \tag{4.2}$$

Now this equation can be rewritten in terms of $\partial u/\partial x_j$ and solved with respect to this value. This result can be used for calculating the gradient of the objective function.

$$\frac{du}{dx_j} = \frac{\partial u}{\partial x_j} = K^{-1}\left(\frac{\partial f}{\partial x_j} - \frac{\partial K}{\partial x_j}u\right) \tag{4.3}$$

One back-substitution is needed for every design variable. This makes it not attractive for many design variables. An alternative solution to obtain these results is by using the adjoint method.

4.3.6 The adjoint approach

The adjoint method is available in continuous or discrete form. The difference between the continuous and discrete form is the order of operations [41]. Both methods starts with the partial differential equation. The continuous method first calculates the adjoint operation and then discretizes the partial differential equation. For the discrete method the order is reversed as can be seen in Figure 18.

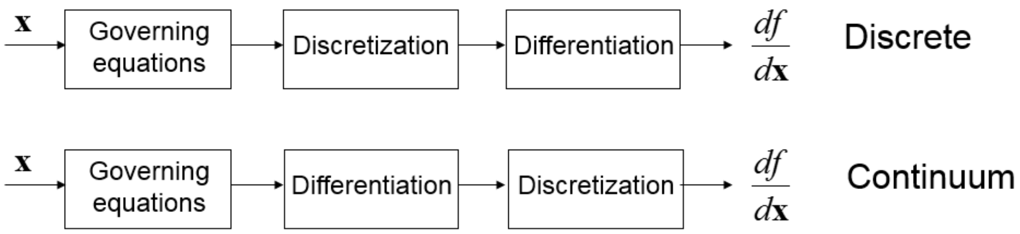


Figure 18: The discrete and continuum form

The starting point for calculating the adjoint sensitivities is the augmented response. This is the response where we are interested in. It is augmented in the same way that you can form a Lagrangian and it is given as [30][42]:

$$h^* = h(u(x); x) + \underbrace{\lambda_i^T(f(x) - K(x)u(x))}_{=0 \; for \; all \; x} \tag{5.1}$$

In which λ_i^T is the Lagrange multiplier. This is an arbitrary value because it only consider variations for which the governing equation is satisfied. The last term of the equation contains a constraint and this term is always equal to zero to provide an equilibrium state. Now it is possible to differentiate this equation with respect to its design variables to obtain:

$$\frac{dh_i}{dx_j} = \frac{dh_i^*}{dx_j} = \frac{\partial h_i}{\partial x_j} + \frac{\partial h_i}{\partial u}\frac{du}{dx_j} + \lambda_i^T\left(\frac{\partial f}{\partial x_j} - \frac{\partial K}{\partial x_j}u - K\frac{du}{dx_j}\right) \tag{5.2}$$

In which du/dx_j is the state sensitivity, which appears twice in this equation. This equation can be rewritten in terms of the state sensitivity in order to obtain:

$$\frac{dh_i}{dx_j} = \frac{dh_i^*}{dx_j} = \frac{\partial h_i}{\partial x_j} + \lambda_i^T\left(\frac{\partial f}{\partial x_j} - \frac{\partial K}{\partial x_j}u\right) + \left(\frac{\partial h_i}{\partial u} - \lambda_i^T K\right)\frac{du}{dx_j} \tag{5.3}$$

To avoid the computation of the state vector derivative du/dx_j, λ_i is chosen in such a way that this state vector vanishes from the equation. The value of the LaGrange multiplier is chosen in such a way, that the term with which it is multiplied is equal to zero. To achieve this, the term between the brackets must be set equal to zero:

$$\frac{\partial h_i}{\partial u} - \lambda_i^T K = 0 \tag{5.4}$$

When we switch one term to the other side of the equation to obtain:

$$\lambda_i^T K = \frac{\partial h_i}{\partial u} \tag{5.5}$$

Now is it possible to multiply both sides with K^{-1} in order to isolate λ_i^T:

$$\lambda_i^T = \frac{\partial h_i}{\partial u}K^{-1} \tag{5.6}$$

Then we make use of the property $(AB)^T = B^T A^T$.

$$\lambda_i = K^{-T} \left\{\frac{\partial h_i}{\partial u}\right\}^T \tag{5.7}$$

This result can be substituted into equation *5.3* to make the state sensitivity vanish from the equation. So the last term drop out due to a proper selection of the Lagrange multiplier.

$$\frac{dh_i}{dx_j} = \frac{\partial h_i}{\partial \mathrm{x_j}} + \lambda_i^T \left(\frac{\partial f}{\partial \mathrm{x_j}} - \frac{\partial K}{\partial \mathrm{x_j}} u\right) \tag{5.8}$$

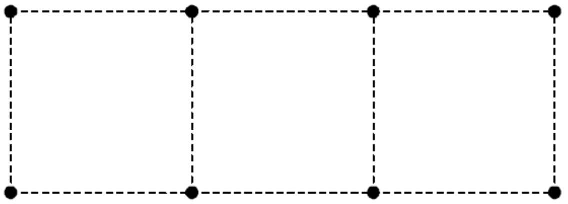
So the final result is:

$$\frac{dh_i}{dx_j} = \frac{\partial h_i}{\partial \mathrm{x_j}} + \underbrace{\frac{\partial h_i}{\partial u} K^{-1}}_{\lambda_i^T} \left(\frac{\partial f}{\partial \mathrm{x_j}} - \frac{\partial K}{\partial \mathrm{x_j}} u\right) \tag{5.9}$$

This means that there is only one back-substitution per response. This is an attractive method in the case of many design variables (j) and few responses (i).

4.4 Ground structure method

The most commonly used method for optimizing truss structures is the ***G**round **S**tructure **M**ethod* ***(GSM).*** This method starts with an array or set of nodes that are interconnected with linear members. At some of these nodes forces or support conditions are applied. The method starts with generating a fixed grid of joints and adding bars in all the possible connections between the nodal joints. This is visualized in Figure 19 and this is called the “Ground structure”.



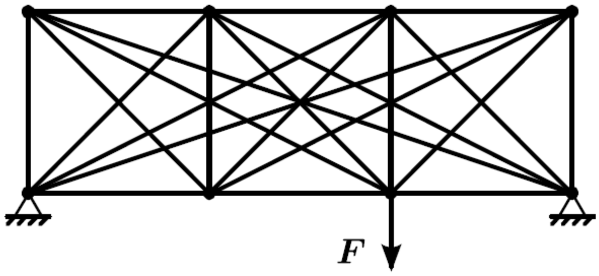


Figure 19: Array of joints and the ground structure

The ground structure consists of all the potential bars in the design domain and each bar connects two of the nodal points. Each bar is a potential structural member and the cross-sectional area of the bars is used as a design variable. The bars can be removed from the structure by setting the cross-sectional area of the bar equal to zero. This will set the stiffness of the member equal to zero and therefore it has no longer a physical meaning. The number of joints is not used as a design variable. Each member in the structure is sized in an iterative process considering the applied forces and constraints and therefore this problem can be seen as a size optimization but it has the effect of a topology optimization. The optimization process consist of systematically removing inefficient members from the initial ground structure until the objective function is minimized and the optimal solution is found. The task of this method is to find an optimal truss structure that satisfies all the load and support conditions. This is achieved when objective function is minimized.

Now we assume that n is the amount of nodal points in the design domain and that m is the amount of connections. Let a_i and l_i denote the cross-sectional area and length of bar i. All the bars are made from linear elastic material with young’s modulus E_i. Then the total volume of the truss becomes [54]:

$$V = \sum_{i=1}^{m} a_i l_i \qquad \text{With } i = 1, \dots, m \tag{6.1}$$

Which is the summation of the volumes of all the bar elements. We can simplify the bar volumes with $t_i = a_i l_i$.

The static equilibrium is expressed as:

$$F = Bq \tag{6.2}$$

In which q and F are respectively the member forces and the nodal force vectors. If $q > 0$ then the member is in tension, otherwise it is in compression. The nodal force vector contains the forces in each degree of freedom.

The matrix B is the geometric matrix which contains the bar direction cosines. This matrix translates the member forces to the nodal forces. According to Christensen & Klarbring the local stiffness matrix of a bar element in a truss structure is given by [31]:

$$K_i^0 = \frac{E_i}{l_i}\begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix} \tag{6.3}$$

Where $S = \sin(\theta_i)$ and $c = \cos(\theta_i)$ and θ_i is the angle between the bar and the horizontal plane. The matrix K_i^0 represents the element stiffness matrix for bar i per unit area.

$$K_i(t_i) = \frac{t_i}{l_i}K_i^0 = a_i K_i^0 \tag{6.4}$$

The stiffness matrix K of the complete truss can be written as:

$$K(t_i) = \sum_{i=1}^{m} K_i \tag{6.5}$$

In which K_i is the local element stiffness. The minimum compliance for a truss structure given a volume of material can be formulated by:

$$Minimize: c(u,t) = \{F\}^T(u) \tag{6.6}$$

$$Subject\ to: [\sum_{i=1}^{m} t_i K_i]\{u\} = \{F\} \tag{6.7}$$

$$\sum_{i=1}^{m} t_i = V \tag{6.8}$$

$$t_i \geq 0 : i = 1,2,\dots,m$$

The design variable t_i is the control variable and the displacement u_i is the state variable. The zero lower bound on the variables t_i indicates that the bars of the ground structure can be removed.

5. Recommendation optimization software

In this chapter the most important software is described which is used during this master thesis. Three software packages are used to solve the problem. These are Limitstate:form, which uses the ground structure approach, Inspire from Solidthinking and Optistruct from Altair. These software packages discretize the domain and generate a design. A multi-criteria analysis is used to determine which software package is most suitable for performing a size optimization.

5.1 Topology optimization software

Nowadays a wide variety of commercial software is available for structural topology optimization. These optimization packages are often integrated into existing FE software like Ansys, Comsol, Nastran, Hyperworks, Tosca and Abaqus. Other software packages are especially designed for topology optimization like Solidthinking Inspire and Genesis.

At this moment 33 different software packages have been identified that cope with topology optimization [37]. Besides commercial software there are also open source packages available like: TOPOpt and Topostruct. These are educational tools which are commonly only suitable for simple loading conditions and do not support additional analysis like: vibration analysis, shape optimization or smoothening [36]. Most of the open source software is designed for 2D topology optimization but can also be extended to 3D problems. The commercial design software packages are well developed and are able to solve the structural design problems in 3D by using their own FEA modules. They also have well documented capabilities and support. Commercial software is often equipped with additional analysing tools for the post-processing of the results. For structural analysis with isotropic materials, the SIMP method is commonly used in commercial and educational software.

The different optimization solvers have different capabilities which make them more suitable to handle different problems. Some of the software packages are designed to produce fully functional designs which do not require any post-processing while other packages are more suitable to visualize load paths for educational purposes. All the commercial software includes eigenvalue analysis and optimization, shape optimization, smoothening and the use of manufacturing constraints. The performance of the structural optimization depends on the required computational time and the quality of the optimum.

Three optimization software packages will be used in this master thesis. These are: Inspire from Solidthinking, Limitstate:form from Limitstate3D and Optistruct from Altair Engineering. Inspire and Optistruct utilizes the SIMP optimization technique where the Limitstate software uses the ground structure approach. The method for topology optimization in Optistruct depends on the problem which is defined, it can either be the SIMP method or the homogenization method [57][58]. The advantage of the software from Solidthinking is that is has a clear graphical interface with intuitive controls. It is equipped with quick and easy geometry editing tools and is provided with automatic meshing tools. This lowers the pre-processing time. The software from Limitstate is suitable for optimizing a truss design. The advantages of this program is that it has a clear user interface, quick analysis and developed for minimum-weight truss design. This program uses different optimization techniques and are therefore assumed to generate different designs compared to the other software packages.

5.1.1 LimitstateForm

Limitstate is the solver used for structural optimization. This software utilizes a mathematical method to automatically identify an efficient arrangement of structural members for a specified design problem. It is incorporated in the ANSYS Spaceclaim direct modelling software. It uses a discrete representation of the problem, where nodes are divided within a prescribed design domain and they define the end and begin point of the potential members. The software determines the optimal arrangements to carry the different loads with the minimum volume of material in order to satisfy the stress constraints or deflection limits. It starts with an initial layout optimization phase in which the minimal weight layout of the members is determined using the predefined nodal positions. Secondly, the solution is improved by a geometry optimization in which the position of the nodes is adjusted in order to improve the solution. Finally an automatic filter is conducted on the structure in order to remove any member from the structure to satisfy the user defined criteria.

The workflow of the software is as follows: First a design domain is specified with the type of material and the different loadings and support conditions acting on it. After this, the design domain is discretized with nodes. Each node is connected to all the other nodes using potential members. The nodal resolution determines the efficiency of the design. In the layout optimization the software identifies the optimum group of members which are required to resist the applied loading. The geometry optimization adjusts the position of the joints in the optimized structure in order to refine the solution and to increase the efficiency. A set of optimization parameters is available for assigning the type of profile, minimum thickness and filter and fabrication options. After the iteration process, the solution can be edited by using the edit functionality of the software. This provides a number of tools for modifying the structural layout of the design. Nodes and members can be added or removed from the design and the efficiency of the structure is reported. Additional checks can be performed such as: stress, deflection, frequency or buckling checks. The sizing of the members can be adjusted in order to reduce the stresses in the design. The final step is to generate the members or an unioned solution of the CAD geometry for further FEA validation. This FEA can be performed with ANSYS Workbench.

5.1.2 Solidthinking Inspire

The company Solidthinking developed a software package which is called: “Inspire”. This software uses a continuous representation of the design problem and uses a density based approach in order to solve this problem. Therefore this approach differs from the ground structure approach. It all starts with a defined design domain. This is a volume in which a certain material is specified. The loading and boundary conditions are applied to this design domain. This design domain is discretized into a mesh. This is means that the design domain is divided into a large amount of small elements. The properties and the response of the material is considered linear over these small elements. This method satisfies a set of constraints which determines the behaviour of the structure. In Inspire the structure can be optimized in terms of mass or stiffness. The density of each element is related to the stiffness of the material. An internal algorithm determines the required density of each mesh element. A penalization function is used to convert the solution into a discrete solution which determines if a mesh element needs to be a void or must contain material. Inspire is capable of performing a range of different analysis such as static and buckling analyses. Different types of loads such as point loads, moments, pressures and gravitational loads can be applied. The software is capable of dealing with angular velocities, temperatures, point-masses and displacements constraints. Different connection can be applied such as fasteners, joints, spot welds and contacts. By the set-up of a new model, different support conditions can be applied such as: hinged, fixed or sliding contacts. Shape control ensures a symmetrical shape of to design and reduces the required computational time. Al these options makes it possible to accurately define the input for the optimization process. The output of the optimization process can be smoothened to enables a smooth design for further analysis.

5.1.3 Altair Hyperworks Optistruct

Altair Engineering has developed a software package which is called Hyperworks. Hyperworks is a Multiphysics CAE platform consisting of the following software packages: Hypermesh, Optistruct and HyperView. Hypermesh is the finite element pre-processer which is used to discretize the CAD model and to define the material properties, boundary conditions, loads and optimization objective in order to prepare the model for the optimization process. Hypermesh is used to define the optimization. From Hypermesh, the problem is exported and processed by Optistruct. Optistruct is a structural analysis solver for solving linear and non-linear problems. This solver will solve the problem which is defined by using Hypermesh. The results from Optistruct can be post-processed in HyperView which allows the user to visualize the results. The workflow of this software can be found in Figure 20.

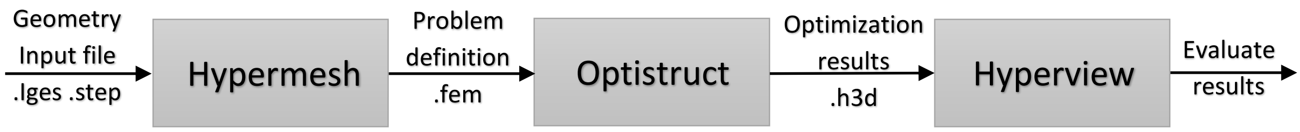


Figure 20: Workflow in Hyperworks

Optistruct allows the user to control every aspect of the optimization process. The solver allows for performing static, model and buckling analysis and optimization. This software is capable for performing thermal analysis and optimization. In addition of topology optimization, this software can perform size, free size, topography and shape optimization. Another interesting feature is the multi-model optimization. In which the design domain can be divided in multiple separate design domains with different properties to be optimized for various objectives. This allows for different manufacturing constraints and materials to be impressed upon discrete regions of the part.

5.2 Results topology software packages

Three different tools are used for the optimization of the gangway structure. The input values are described in the appendix section A-6. Input parameters optimization process. The results from the software packages and the rating of each program is described in this section. This preliminary optimization process is conducted in order to investigate which software package is the most suitable for optimizing the gangway structure. The software which is best performing, is used for further analysis. The software is rated on the required solving time, the final objective function value and the amount of user-friendliness.

Optistruct

The *advantages* of Optistruct is that is has a low calculation time compared to inspire. It also allows to abort the optimization process if necessary and to review the results so far or to review the results during the optimization process. Another advantage is that the optimization process can be restarted if it was interrupted. The calculation time can be determined manually by setting a maximum amount of iterations allowable and by setting an objective tolerance. If one of these two values is satisfied, then the optimization process is terminated. Also a convergence graph is generated which shows the convergence rate of the solution. It has numerous meshing options which allows for mesh refinements. Optistruct provides the possibility to choose among a large variety of objective functions and constraints.

Different types of objectives can be specified and it has abilities to create complex load cases. There is an extensive tutorial library available and there is an active support forum which supports users of solving their optimization problems.

The *disadvantages* is that the pre-processing time is quite long and that the interface is quite complicated. It takes some time to get familiar with this software. The units are not automatically tracked and are determined by the input values of the CAD model. Several steps needs to be performed in order to extract the optimized geometry.

The best result which is obtained by using Optistruct can be found in Figure 21.

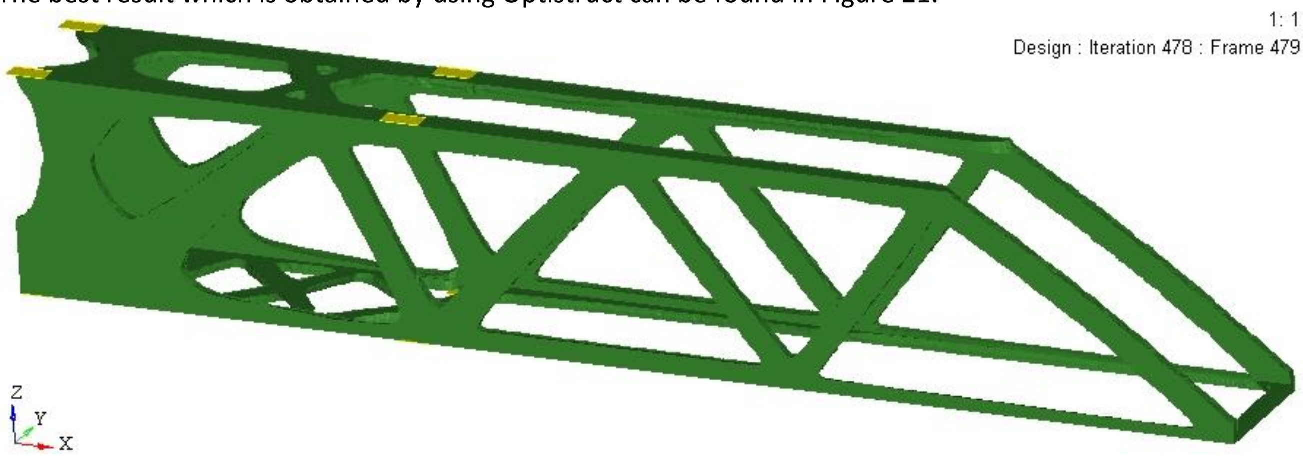


Figure 21: Optistruct Optimization results

Inspire

The *advantages* of Inspire is that it offers a simplified graphical interface for modelling and optimization. The software allows for generating or editing the geometry. The finite element mesh is created and refined automatically, which reduces the required pre-processing time.

The *disadvantages* of Inspire is that it does not support any shape optimization. The solver run time is quite long compared to the other software packages. Another disadvantage is the low control aspect of the optimization. Inspire contains lots of black box functionalities in which the program determines the control parameters for the optimization process and the user is not able to change these parameters. Therefore the user cannot influence the optimization process. Inspire only allows optimization for minimum mass or maximum stiffness. The loading tools do not allow for load cases with the same complexity as with Optistruct. The simplified interface and the automatic meshing function reduces the capabilities of this software package. Another disadvantage is that this software does not allow the use of multiple design domains or a disconnected design domain. The software crashes quite often which results in the loss of data. The best result which is obtained by using Inspire is shown in Figure 22.

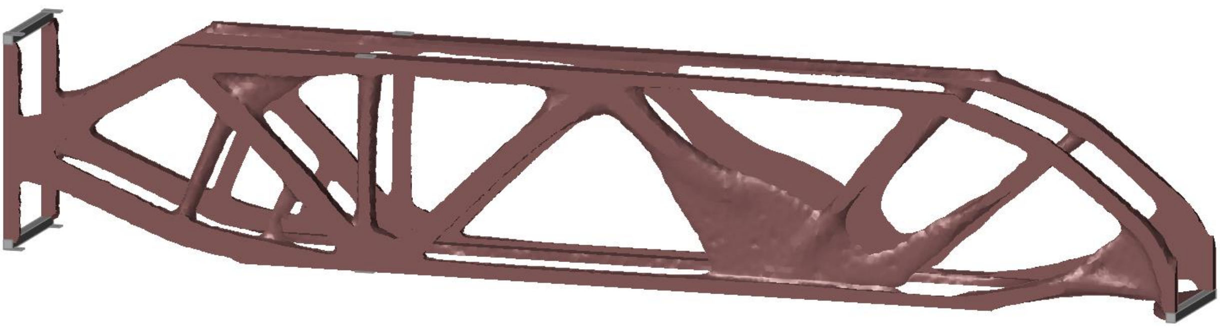


Figure 22: Inspire optimization result

Limitstate

The advantages of Limitstate is that the solving time is very short compared to the other optimization software packages. This is due to the fact that this software is solving a discrete problem instead of a continuous problem. It has an easy to use interface with the ability to refine the design solutions. The structural efficiency can be determined globally. This software is specially designed for designing truss structures. After the layout optimization, a geometry optimization can be performed, in which the optimal location of the nodes is determined. Multiple design parameters are available in order to influence and steer the optimization process. However not all these design parameters are applied during the optimization process or are just simply bypassed.

The disadvantages of this software is that the nodal resolution, the arrangements of the nodes in the design domain, is automatically determined. Only five different resolutions can be chosen for a certain design domain or a custom amount of nodes can be specified. The software locates automatically nodes on the corners of the domain, even if these edges and corners are very close together. This is huge limitation of the capability of the software. Another disadvantage is that the solver often fails without giving any cause of the error. This makes it difficult for the user to solve or prevent this error. The editing tool allows the user to move, add or remove nodes or members from the system. This can be useful but it is an attractive feature in order to generate a design which is appealing to the user but not very efficient at all. The loading tools are very basic and only allows for simple loading conditions. Another huge disadvantage of this software package is that the optimizer often generate beams which are located outside of the design domain. This results in an infeasible design because the design has violated the user defined constraints.

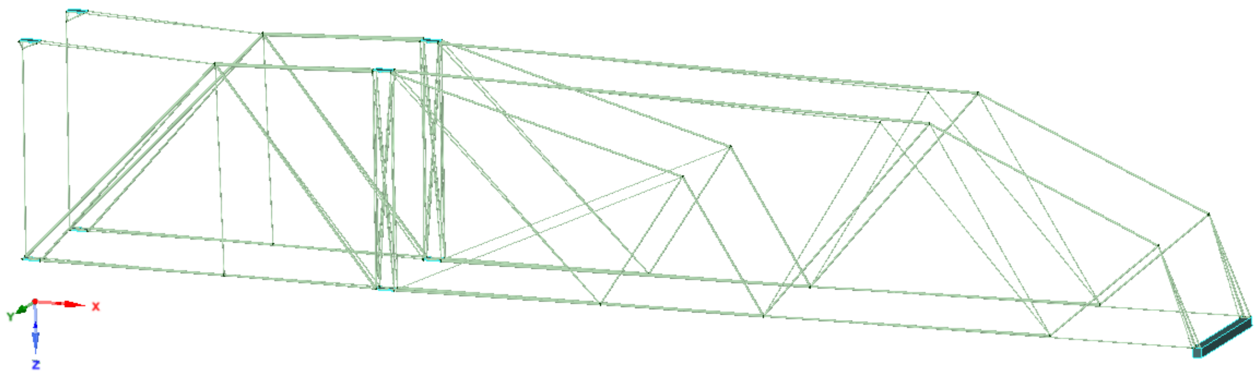


Figure 23: Limitstate:form optimization results

5.3 Multi-criteria analysis

The objectives and constraints define the requirements for the topology optimization. The software package should be able to optimize the parts with respect to mass or volume, while offering constraints on the stress or displacement. The software must also include some form of training or support which allows that the user is able to achieve the goals of the project within its deadlines. Support is required to help the user when the software is misunderstood or malfunctioning and problems are encountered. Additionally it would be ideal if the software has other functions besides only topology optimization. For the comparison and rating of the different software packages a Multi-criteria analysis (MCA) is performed. This method is used for solving decision making problems which involves multiple criteria. It is used to determine the most promising alternative from the set of available solutions. The software packages will be used to determine the most promising alternative from the set of available solutions. Factors that determine the overall functionality of the software have been identified. The software will be rated according boundary conditions stated in Table 8.

Table 8: Boundary conditions

Boundary condition	Description
Solving time	The amount of time which is required in order to solve the problem to obtain satisfactory results.
Pre-processing time	The amount of time and the associated steps which are required to set-up the topology optimization.
Objective convergence	The final value of the objective function after optimization.
Export options/compatibility	The allowable import and export types which are supported by the software.
Additional options	Additional features and options which are provided by the software.
Post-processing	The amount of time and steps which are required for the post-processing of the results.
Support	The training availability and technical support which is provided by the software.
Computational requirements	The system requirements for running the software.
Price	The price of the software.
Usability	The required time and effort which is required in order to understand and operate the software.

The boundary conditions are given a weight factor. These weighting factors are normalized by 1 and therefore each factor is divided by the sum of the factors. Each design software package has been rated with a score ranging from -2 to 2 for each boundary condition. This indicates to which extent the software satisfies this criteria. A score of -2 means that the software does not comply with the stated boundary criteria, a score of -1 means that it does not comply to a certain extent and zero is a neutral value. For the positive values just the opposite holds. The scores for these design criteria are multiplied and summed up for the different software packages. A large positive score means that the software is suitable for the problem and has favourable characteristics. A negative score means that the software is not suitable for solving the problem. The software with the highest score, will be used for optimizing the gangway structure. The weighting factors are given in Table 9.

Table 9: The weighting factors

	Solving time	Pre-processing time	Objective convergence	Export compatibility	Additional options	Post-processing	Support	Computational requirements	Price	Usability	SUM	Weight factor
Solving time	-	1	0	1	1	1	0	1	1	0	6	0.13
Pre-processing time	0	-	0	1	1	1	0	1	1	0	5	0.11
Objective convergence	1	1	-	1	1	1	0	1	1	0	7	0.16
Export options/compatibility	0	0	0	-	0	0	0	1	1	0	2	0.04
Additional options	0	0	0	1	-	1	0	1	1	0	4	0.09
Post-processing	0	0	0	1	0	-	0	1	1	0	3	0.07
Support	1	1	1	1	1	1	-	1	1	0	8	0.18
Computational requirements	0	0	0	0	0	0	0	-	0	0	0	0
Price	0	0	0	0	0	0	0	1	-	0	1	0.02
Usability	1	1	1	1	1	1	1	1	1	-	9	0.20
										Total	45	

These weighting factors are used for the MCA. The values in the MCA are multiplied by these weighting factors and the summation of the values in the table results in the total score.

Table 10: The Multi-criteria analysis

	Solving time	Pre-processing time	Objective convergence	Export compatibility	Additional options	Post-processing	Support	Computational requirements	Price	Usability	**Total score**
Optistruct	1	-1	2	2	2	2	2	1	-1	0	**1.067**
Inspire	-2	2	1	1	-1	1	-2	-1	1	2	**0.2**
Limitstate	2	2	1	-1	-2	-2	-1	2	-1	2	**0.311**

From the MCA of the three software packages, it is clear that Optistruct is recommended for solving this specific problem. It has the highest total score in the MCA. Although the user interface is more complicated compared to the other software packages, it allows the user to control every aspect of the optimization. This enables the user to prevent several problems which are encountered by the other software packages. This results in more realistic and useful solutions.

Solving time

The solving time is crucial for testing the effects of the boundary conditions and design parameters. The solving time in Limitstate is in the order of seconds, while with Optistruct it is in the order of minutes or hours. For inspire it can be in the order of hours or even days.

Pre-processing time

The pre-processing time in Optistruct is more complicated and time consuming compared to the other software packages. The longer pre-processing time is offset by the faster solving time. For the other programs, the optimization process can be started within a few steps.

The objective convergence

The objective convergence can be chosen manually with Optistruct, while it is fixed with the other software packages. Also the optimization process can be terminated even when the convergence criteria is not satisfied. Then the program shows the final iteration step.

Export compatibility and post-processing
The export compatibility in Limitstate is very poor, the direct link which should export the model to Ansys Workbench is not working and only a line model without any member dimensions can be exported. Which is quite useless because a size optimization is then still required in order to obtain the beam dimensions. With inspire, a .STL file can be exported with contains the shape of the optimization results. This result can only be used for 3D-printing or additional post-processing steps are required. With Optistruct it is possible to import the results from the optimization process into Hypermesh and it is possible to perform a FEA. These results can also be used in order to obtain a line model for further post-processing. Multiple file extensions are available for exporting the results. Another solution is to perform a shape optimization in order to define the beam dimensions. This is an additional feature which is a preferred post-processing step in the optimization process.

Additional options
Another additional option is to incorporate multiple constraints and responses. Which gives insight and guidance to the optimization process. These options often lack at the other software packages. Where most functions are hidden within the software or are simply just not available. This simplifies the use of the software but it reduces the capabilities of the software.

Software support
The software support of Altair is very active and prepared to help solving your problem. When problems are encountered during the use of the software, specialists are available which are willing to solve your problem. There is an active forum and lots of tutorials are available. The other two software packages also provides tutorials in order to get familiar with the software, unfortunately they don't have any active support which helps the user when the software is misunderstood or not working.

Computational requirements
The computational requirements are quite similar for all the software packages. The only disadvantage of Inspire is that it generates larges sized files which requires a lot of disk space. Even when a run has failed it stores the failed run on the hard drive.

The price
The software of Optistruct is the most expensive, because it requires the complete Hyperworks package in order to operate. Inspire is therefore a little bit cheaper due to the fact that all the functions are incorporated one single software package. Limitstate form is also quite expensive and for this amount of money you only receive a small plugin for Ansys Spaceclaim. So this means that also an Ansys Spaceclaim and Ansys Workbench licence is required in order to review and validate your results.

The usability
The final criteria is the usability of the software. Limitstate and Inspire both have a user friendly user manual and software interface. It is very easy to set-up and perform an optimization task. With Optistruct this is less straight forward. Lots of post-processing steps are required in order to perform an optimization task. This requires some patience and effort from the user in order to get familiar with all these steps. These steps gives the user more insight and influence on the optimization process but makes it harder for the user to operate the software. The overall opinion is that the software is suitable for the optimization and analysis and is a good supporting tool for the design engineer.

6. Design approach

This chapter explains the different design processes. A description of the current component development process is given as well as the design approach which uses topology optimization. This optimization method will be applied during this master thesis. A finite element analysis of the current gangway design is performed.

6.1 Conventional design process

A conventional design process consists often of a trial-error approach or an iterative-intuitive process. This approach is shown in Figure 24. This is an iterative process where a design is proposed by the design-engineer. This design depends on the designer's knowledge, experience and understanding of the problem. This design is analysed and evaluated by a structural engineer using FEA and/or hand calculations. These analyses will often be complemented by mechanical tests on a prototype. After this analysis, modifications to the design can be made to improve the design performance or to satisfy unfulfilled requirements. These modifications are often made in an intuitive way. After these improvements the new design is analysed and the complete process is repeated. This iterative process will be repeated until all the design criteria are met. This iterative process can take a lot of time and will result in an (sub)optimal design because multiple changes are added to each other.

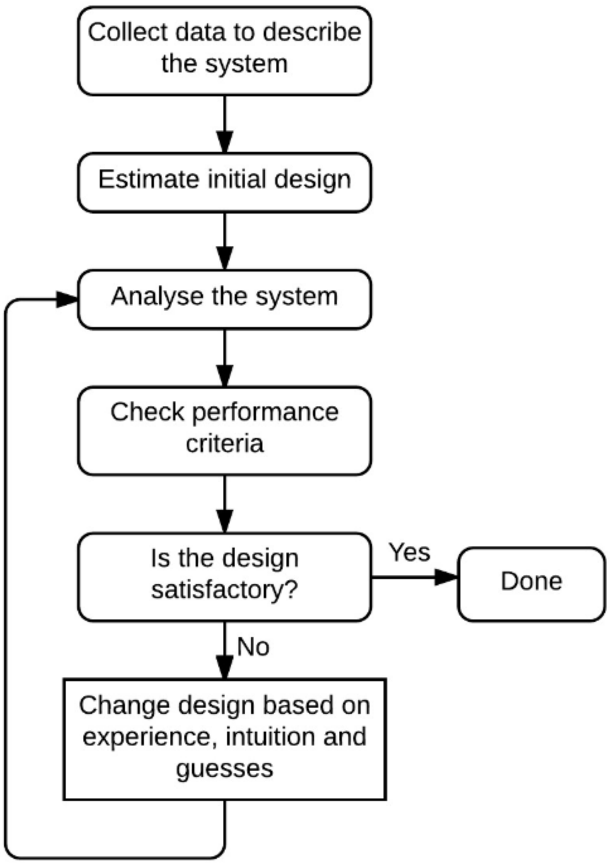


Figure 24: Conventional design process

During the last decade, commercial topology optimization software has been developed rapidly. The idea of this software is that the designer and engineer are both involved in the initial stage of the design. This will generate an optimum design concept. Because the design and verification iteration steps are combined into one program. This will enhance the validation of the design and reduce the amount of time required.

6.2 Topology optimization process

The topology optimization process starts with defining a component design volume or design space. This is a three dimensional space, in which the design may exist. Rigid bodies are included or connected to this design domain. These are parts or regions in the design domain which are not available for topology optimization. This can be for example bearing houses, connection points or other load areas. These rigid bodies are connected to or located in the design domain and the loads or support conditions can be applied to these rigid bodies. The shape and volume of these rigid bodies will not change during the optimization process. The design space will be used to transfer the loads between the rigid bodies to obtain a static structure which satisfies all the constraints.

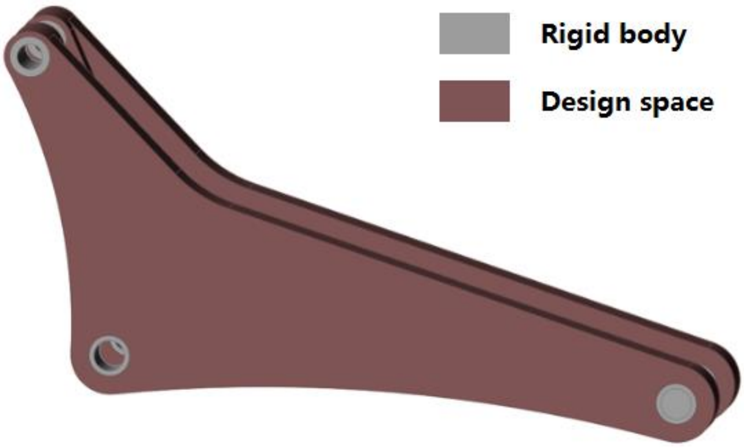


Figure 25: Topology optimization inputs

Optimization goals are defined such as a *minimum weight* or *maximum stiffness* while simultaneously certain requirements are fulfilled like: target weight, stiffness, stress or Eigen frequency. The topology optimization process generates a design that fulfils these requirements. For the optimization objective for minimizing the mass, a stress safety factor needs to be specified. This factor determines the ratio of the yield stress and the actual stress in the design. This factor determines the maximum permissible stresses in the design. For a design with the maximum stiffness, a mass target needs to be defined. This means that the mass of this final design has the maximum stiffness for this specified mass. If this value is not specified, the optimization process will use all the available material to maximize the stiffness and therefore the design space will not change. When all the optimization input parameters are defined, the optimization process can be started. A finite element model is constructed based on the design volume. The design volume is discretized with mesh elements. Different load cases are applied and included in the topology optimization process. The FEA determines the stresses in the design and adjusts the design according to this.

The topology optimization generates a design with an irregular surface due to that fact that it eliminates mesh elements during the optimization process. The optimized design can be converted to a more smooth structure. After this step, the smoothened design is redesigned with respect to manufacturing constraints. This realization step is crucial for generating a design which can be designed with the desired manufacturing method. The final design is interpreted and reviewed by a structural engineer in order to check if it still satisfies the design criteria. The topology optimization process is visualized in Figure 26.

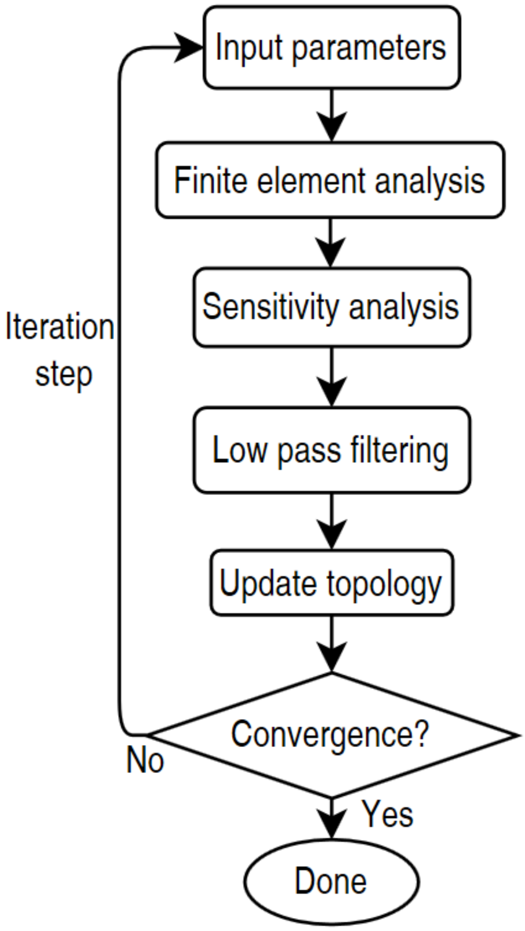


Figure 26: The topology optimization design process

6.3 Application to the gangway structure

In this chapter the performance of the current design of the gangway structure is presented and evaluated. The current gangway design is evaluated and validated with FEA. The results of this analysis will be used to compare the new design with the current design with respect to performance and properties.

6.3.1 The current gangway design

The current gangway design is a truss like structure which consists of two parts made out of S355 steel. It consists of a truss structure made of rectangular shaped tubes which are welded together. The structure is supported by two hinges and two hydraulic actuators located at the origin of the structure. The boundary conditions 3 and 4 fixates the translation in X, Y, and Z-direction and only allows a rotation around the Z-axis. Boundary conditions 1 and 2 are fixed in the X, Y and Z-direction and only allows a rotation around the Z and X-axis. This hinge point may shift in the XY-Plane due to the stroke and orientation of the actuators. The boundary conditions are visualized in Figure 27. The main boom is connected to the base frame which has a fixed connection with the pedestal. The pedestal is fixed to the deck. The second part of the gangway design is the telescoping part. This part is also a truss like structure which is also made from S355 steel. This part is able to move in the longitudinal direction. The telescopic part is equipped with roller bearings which allows this telescopic part to move in the longitudinal direction. These roller bearings are constrained by frictionless contacts, this means that frictionless sliding may occur between the two contacting surfaces but that the roller bearings can only transfer compression forces and separates if put under tension. The end of this telescopic part is supported in the vertical direction in the operational conditions and is free in the emergency disconnection case. The current component weight based on the CAD model and hand calculations is 2038 kg for the telescopic boom and 11000 kg for the fixed boom. De reason for this difference in weight is due to the sizing of these two separate sections and especially due to the difference of the thickness of the members in the structure.

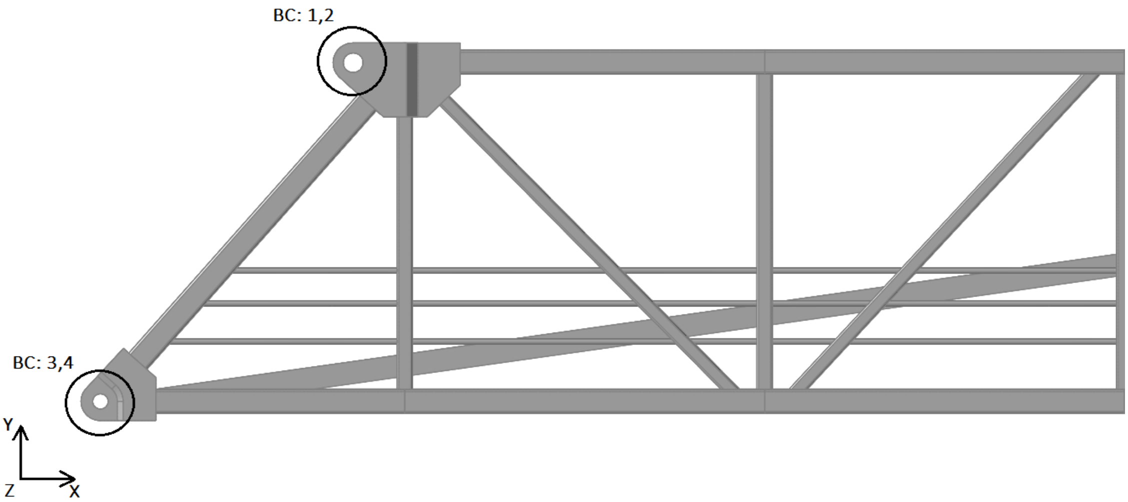


Figure 27: The boundary conditions in the design domain

The stiffness of the current design and the requirements are presented in Table 11. As can be seen, all the requirements are fulfilled. An effective stress plot of the emergency disconnection condition is shown in Figure 32. Which is considered as the most severe load case. The allowable stress level is equal to the yield stress divided by a safety factor of 1.5. Also the maximum displacements are given for this load case. The maximum allowable displacements are given by the DNV-guidelines.

Table 11: Displacements of the current design

Support condition	Location	Limiting value	actual value
Gangway supported at both ends	Middle of the gangway	L/200 = 165 mm	14,21 mm
Cantilever gangway	Gangway tip	L/100 = 330 mm	73,56 mm

6.3.2 The finite element model.

The finite element model is constructed in order to analyse and to check the current design. The model is mainly built with solid elements with a combination of tetra solid and square shaped elements. The finite element grid consists of 155036 elements with a global element size of 50 mm. Mesh refinements are applied at the hinge points and at the connecting joints of the diagonal members. No solid shell elements are used due to numerical errors which were encountered during the analysis.

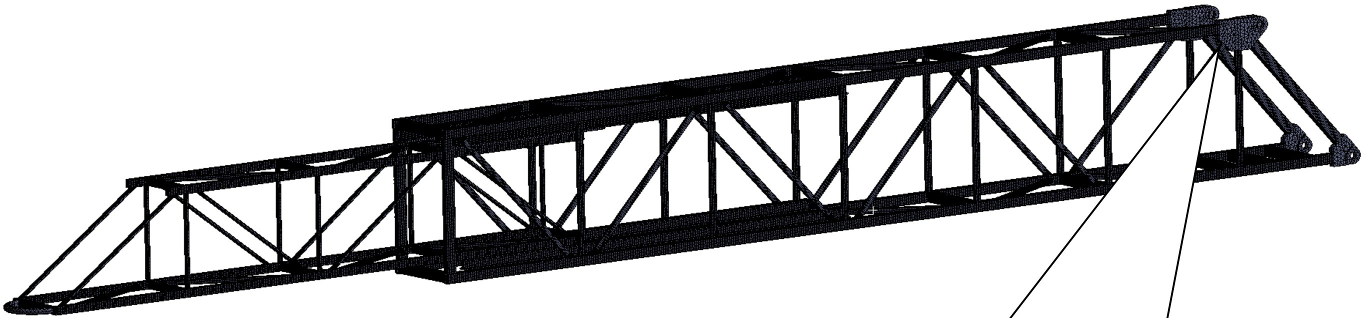


Figure 28: The applied mesh in the finite element model

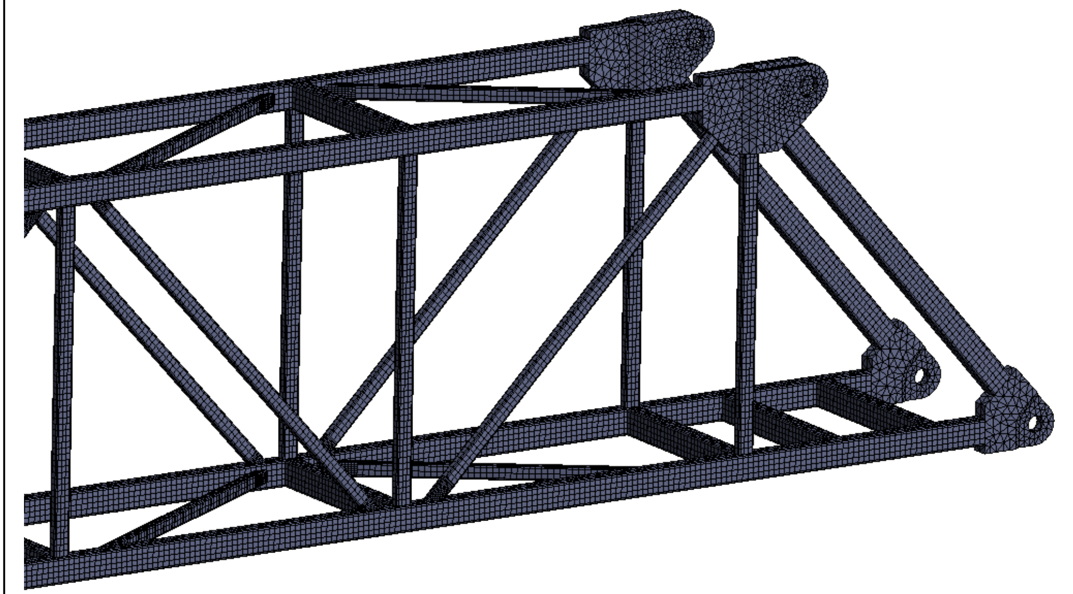


Figure 29: A detail view of the applied mesh

The loads and boundary conditions are applied to this model. The locations of these loads are presented in Figure 30.

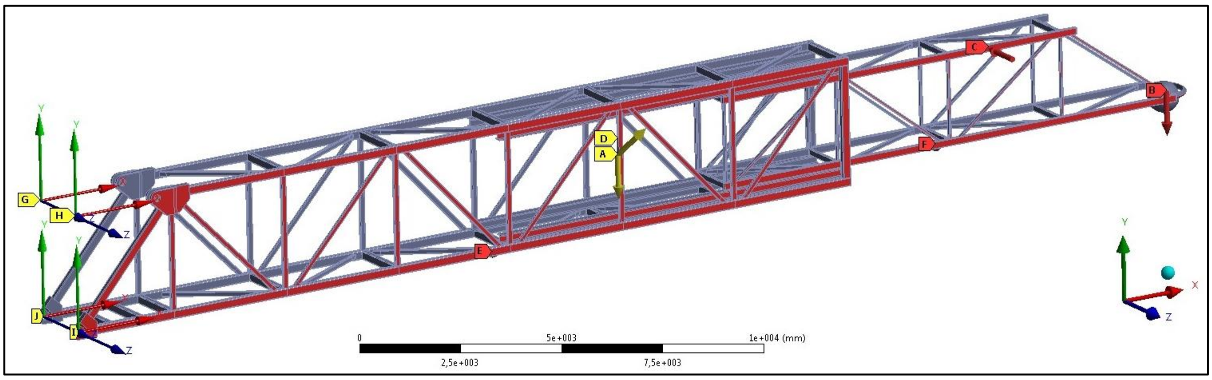


Figure 30: The finite element model

The situation which is visualized in the figure above is for the emergency disconnection case. This load case is considered as the most severe load case. All the other load cases can be found in appendix: A-16. The structure is constrained by means of remote displacement supports at the hinges and at the cylinder brackets. For the hinges every DOF except the axial rotation is constrained. For the cylinders every DOF is constrained except for the rotation around the cylinder hinge and the rotation around the cylinder rod. The maximum vertical deflection for the emergency disconnection case is shown in Figure 31. The maximum deflection in the Y-direction is equal to 73,56 mm.

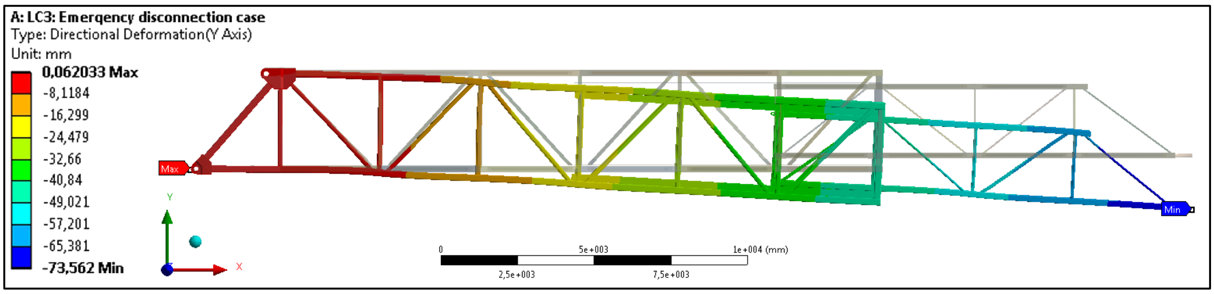


Figure 31: Directional deformation, Y-direction, scale = 24

The overall von-mises stresses are shown in Figure 32.

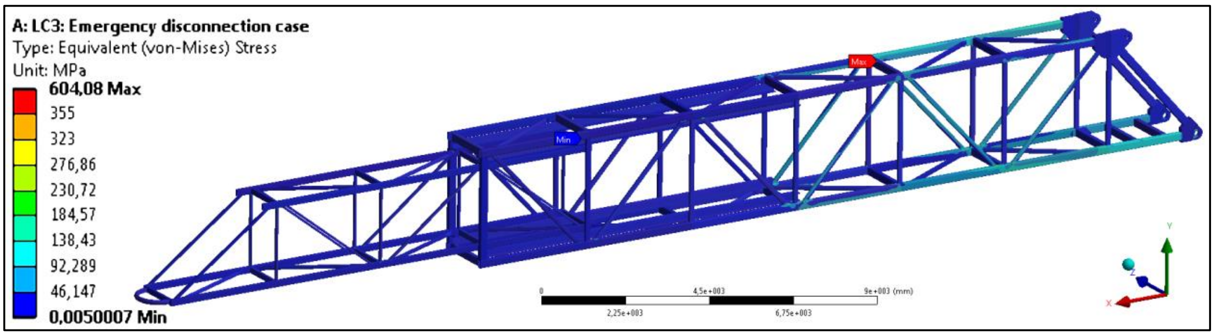


Figure 32: Overall Von Mises stresses

The general stress level is lower than 355 MPa / 1.1 = 323 MPa and is therefore considered acceptable. The plots with detail views of the peak stress areas shows the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and only single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria.

The results of the current design are summarized in Table 12. These values will be used for the comparison with the new design. This will enable to visualize improvements of the design parameters for the optimized design.

Table 12: Specifications gangway structure

Complete gangway structure	
Mass [kg]	13080 kg
Maximum displacement [mm]	73.56 mm
Maximum stress [MPa]	184 MPa

7. Topology optimization process

This chapter is concerned with the steps required for performing an optimization task. The different optimization parameters needs to be defined and specified. The objective function, constraints and the design space must be defined. At last, the results from the optimization process are presented.

7.1 Optimization set-up

The first step in the optimization process is to define the input parameters for the optimization. The following input parameters must be defined in the optimization process:

- *Design domain:* The space dimensions and shape in which the design process may take place.
- *Boundary conditions:* Constraints and load conditions. This consists of zero displacements conditions, regions where the structure is fixed in certain degrees. Load conditions where the structure is affected by external forces. These are characterized by their position and magnitude and the self-body forces, for example the self-weight.
- *The material properties:* The material properties determines the stresses and deflections in the design and therefore influences the design process. It is assumed that the material is working in its elastic range to make the structure physically feasible.
- *The mesh:* The type, number and the dimensions of the elements which are used to mesh to design domain. This parameter determines the required computational effort and the numerical error in the simulation.
- *The objective:* The objective function is the goal of the optimization process. The objective is the response function which must be optimized. The response is a function of the design variables and will be further explained in section 7.2 Responses, objective and constraints.

The optimization process starts with defining a design space and a design material. The design domain is the admissible volume for the design space in which the solution can be found. The support conditions and loads acting on the structure must be applied on the design domain. Different combinations of loads and support conditions can be incorporated into load cases. Constraints can be defined, which are certain characteristics which must be satisfied during the optimization process.

7.2 Responses, objective and constraints

Several steps are required before an optimization process can be performed. The steps which are required are described in this section. The required steps for performing an optimization are:

-Import the geometry and discretize the model.
-Define the design variables and the constraints on the design variables.
-Define the responses that will be used for an objective or as a constraint.
-Define the optimization objective.
-Set constraints on the responses.

A discretized CAD model is always required when performing a topology optimization. The discretized FE model should strive to capture the behaviour of the modelled component. This is achieved by representing the model with varying sizes of elements. A trade-off must be made between the mesh size and the required computational time. The discretization of the model must ensure sufficient accuracy of the output results.

Responses

Before an objective or a constraint can be defined, it is important to define the responses. A response is a numerical measure of a design variable due to the input on the model. These responses can be used as an objective function or as a constraint. The most important properties can be found in Table 13 & Table 14.

Table 13: Optimization responses from design

RESPONSES FROM DESIGN	
MASS	Actual mass of the design
VOLUME	Actual volume of the design
MASS FRACTION	The ratio of mass to the mass of the original design domain
VOLUME FRACTION	The ratio of volume to the volume of the original design domain

Table 14: Optimization responses from the load cases

RESPONSES FROM LOAD CASES	
WEIGHTED COMPLIANCE	The weighted sum of compliances from the different load cases
WEIGHTED EIGENFREQUENCIES	The weighted sum of Eigen frequencies from the different load cases
STATIC DISPLACEMENT	The static displacement at one or more locations
STRESS	The maximum global stress in the design

Also geometric constraints can be defined such as symmetry constraints, extrusion constraints and draw direction constraints. Symmetry constraints will generate a design which is symmetric across a user defined plane or around an axis, which is cyclic symmetric. The extrusion constraint will generate a design which has a fixed cross section in a specified direction. The draw direction constrain generates a design which has no cavities in one direction. This constraint is required for casting purposes. Symmetry constrains will reduce the required optimization time and the shape of the design. All these constraints will influence the shape of the final design.

For setting up a topology optimization, it is required to specify a design variable and an objective function. These parameters can be specified in any order before the optimization process is started.

Design variables

The design variables are used to specify which elements or which parts of the model are subjected to the optimization process. This design variable can be subjected to geometric, stress or member size constraints. Only solid or shell elements can be used as a design variable. The design elements needs to be distinguished from the non-design element in order to assign the different properties.

Design constraints
Beside of the optimization constraints, additional constraints can be added. These additional constraints can be defined in the same dialog as the design variables. Geometric, stress and member sizing constraints can be applied to the model. The global stress constraint will only deal with the global stress and will not suppress the peak stresses in the design. Minimum member sizing is recommended due to the fact that it suppresses the checkerboard effect. A minimum member size of three times the mesh size is recommended. The maximum member sizing constraints are not applied due to the fact that these constraints are not directional. This means that if this constraint is satisfied in one direction, it will not be applied in another direction. Due to the thickness of the design space, this constrain has no effect on the optimization process and is therefore neglected.

Table 15: Examples design constraints

DESIGN CONSTRAINTS	
COMPLIANCE	The measure of strain energy in the structure
VOLUME/MASS	Minimum or maximum volume/mass of the design
STRESS	Maximum stress in the elements
EIGENFREQUENCY	Minimum or maximum Eigen frequency of the structure

Objective function
One objective is required in order to execute the optimization. This objective is defined as maximizing or minimizing a certain response. Additionally, upper and lower bound can be defined on the responses. Only one objective function is allowed for the optimization.

7.3 Design space
The design space is the volume in which material may be placed and attached. It defines the geometric restrictions for the optimizer. To illustrate how topology optimization can be used to optimize the gangway, it is chosen to assign the design domain as the maximum dimensions of the current design. This means that the design space of the optimization is equal to a rectangular tube with the outer dimensions of the gangway structure and inner dimensions as specified by the guidelines. This enables the generation of a design which has the same outer and inner dimensions as the current design. This makes it easy to compare the optimized design to the current design. It also enables to replace the current design by the new proposed design from the topology optimization.

The design domain for the telescopic part is visualized in Figure 33. The rectangular tube has inner dimensions of 2.1 by 1.4 meter and has a total length of 13 meter. This rectangular tube is supported by four roller bearings in each load case. These supporting wheels have a contact area of 0.1 by 0.2 meter. The grey transparent part is the design domain and the blue parts belongs to the non-design domain.

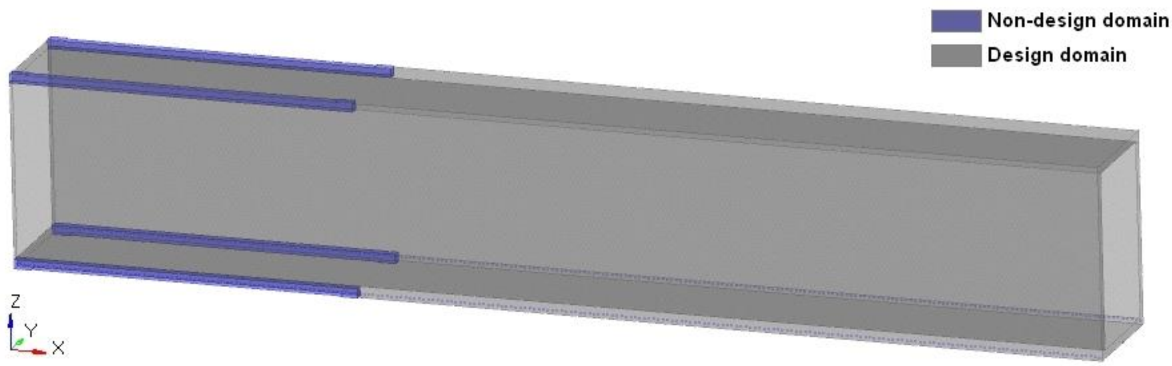


Figure 33: Design space telescopic section

A solution with a mass of less than 5% of the original design space, is able to replace the current design without any increase in weight. This means that small beams are required in order to satisfy this objective. Therefore small sized elements are required in order to find this solution. This makes the topology optimization process a computationally expensive procedure. This will limit the allowable mesh size even as the maximum amount of mesh elements which are allowed for the student version of Optistruct.

The limitation on the mesh size and the amount of mesh elements, will generate problems for the optimizer to find a design with thin walled beams. It is expected that solid beams are used in the optimized design. This will make the design more expensive from a material point of view and this will make the design more sensitive for buckling failure. Due to the fact that buckling is not incorporated in the topology optimization process, it is chosen that the final result of the topology optimization process is used as an input for the post-processing process. It is expected that the optimizer will produce a solution which is not direct suitable for replacing the current design and therefore some post-processing steps are required.

Since the optimizer is unable to find the optimal cross sections, it is chosen to not focus on the performance and mass of the optimized result. The obtained results from the optimization process will serve as an input for the beam orientation in the final design. The result from the optimization process determines the layout and orientation of the beams. The sizing and the profile type of the beams will be determined in the section Design realization.

7.4 Set-Up boundary conditions

The boundary conditions are an important factor for the results of the optimization process. In this section the boundary conditions are defined which are applied during the topology optimization process. According to Table 7, three load cases are incorporated in the optimization process:

-LC1c: Normal operational condition.
-LC2b: Deployment/retrieval condition.
-LC3: Emergency disconnection.

According to Table 6, five load cases needs to be analysed. It is chosen to only use three load cases for the optimization process. Due to the fact that the remaining two load cases consists of load cases in which both sections must be considered. The loads and masses used in this section are calculated in A-5. Design calculations gangway and A-6. Input parameters optimization process.

The structure is constrained by means of load collectors. Load collectors are generated in which the support conditions and load conditions are generated respectively. A load collector contains the forces or support conditions per load case. Optistruct uses constraints which restricts certain DOFS. The DOFS 123456 refer to the translations and rotations with respect to the X, Y and Z-axis. The coordinate system is used to define the boundary conditions and loadings. The local coordinate system is defined at the centre at the origin of the gangway, with the X-axis pointing in the longitudinal direction of the design domain and the Z-axis pointing in the upward direction, according to the right hand rule.

7.4.1 Point masses

During the optimization process it is chosen to set the density of the design material equal to zero. The masses of the gangway structure are incorporated in the design by means of point masses. These point masses are equally distributed over the length of the design domain and they represent the mass of the current design and the additional masses of the structure. The reason for this choice is that this will result in a more realistic weight distribution of the model. The final result from the topology optimization will consist of solid members which will have a significant mass. Gravitational and operational accelerations will act on the structure and therefore this mass has a large mass inertia and this will affect the design of the gangway significantly. Therefore it is chosen to neglect the mass of the design domain and to incorporate the mass of the structure by using point masses.

The mass of telescopic part of the gangway is equal to 2038 kg and the additional masses are estimated at 845 kg. The mass of the landing tool is equal to 400 kg. The mass of the landing tool is applied as a point mass of 0.4 tonnes located in the middle of the tip of the gangway. This is the location of the red dot, as can be seen in Figure 34. The gangway mass and the additional masses are added together and divided over the nodes of the two main beams located at the bottom of the gangway. These are denoted by the white nodes in Figure 34. These point masses are applied for all load cases.

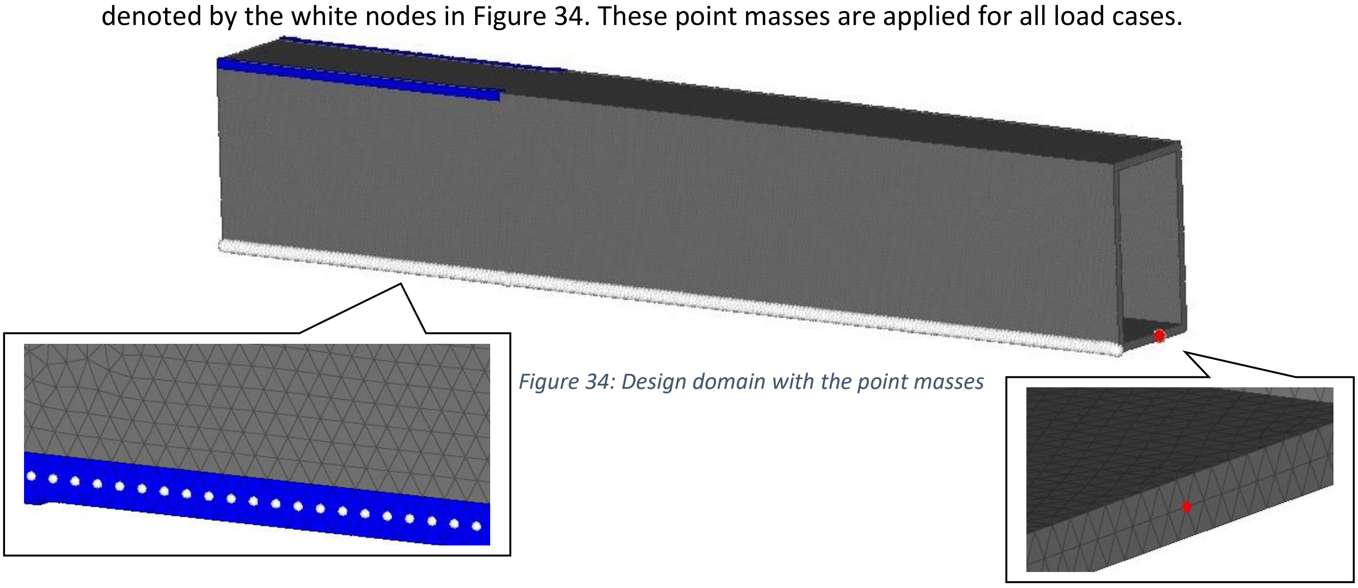


Figure 34: Design domain with the point masses

7.4.2 LC1c: Normal operational condition

First the *operational situation* is considered. In this situation the gangway is supported on both sides and the structure is able to transfer people and cargo from and to the offshore structure. In this condition, the structure is supported by the following displacement constraints: At the hinge point for connecting the cable for the linear motion, a displacement constraint is added which constrains the translation in the X-direction. This constraint is applied at the middle of the origin of the gangway structure at location E as can be seen in Figure 36. The landing mechanism is located at the tip of the gangway. To incorporate the behaviour of the entire gangway, it is chosen to apply the support reaction force at the location of the landing mechanism. This reaction force B_Y is applied as a distributed force over all the nodes located at the tip of the gangway. This force is acting in the positive Z-direction. The magnitude of this force is equal to 89788 N and this force is divided over 26 nodes. The substantiation of this method can be found in A-6. Input parameters optimization process.

The supporting roller bearings are located at the top and bottom of the gangway system. These roller bearings constrains the displacements in the Y- and Z-direction and only allows a rotation around the Y-axis. These constrains are applied at the locations of the active roller bearings. These boundary conditions are incorporated into the optimization process and therefore two load collectors are defined with the corresponding support conditions and loads.

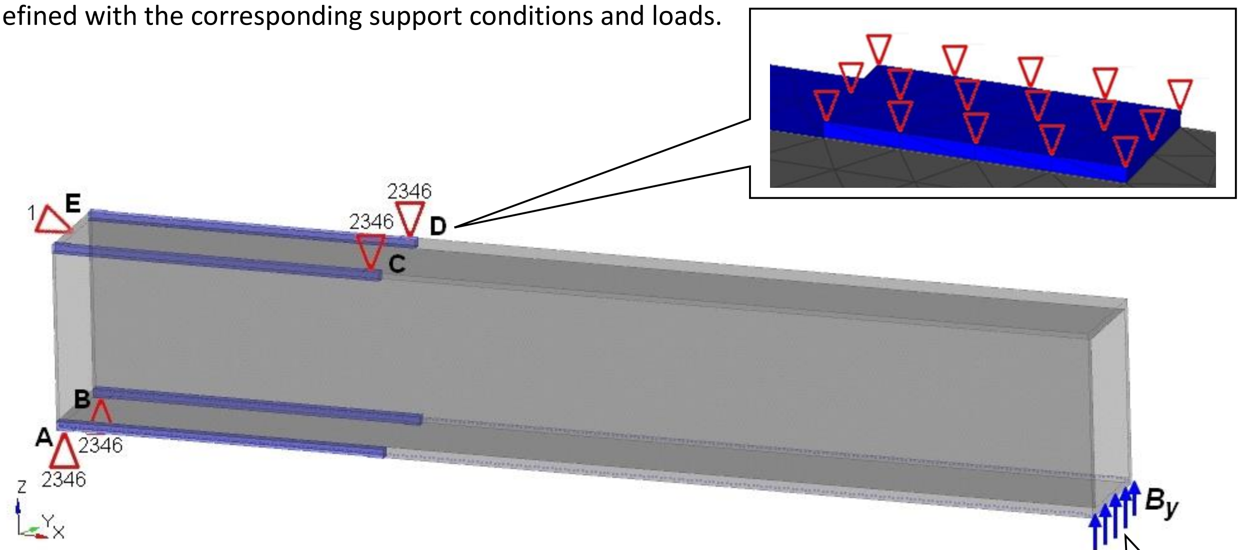
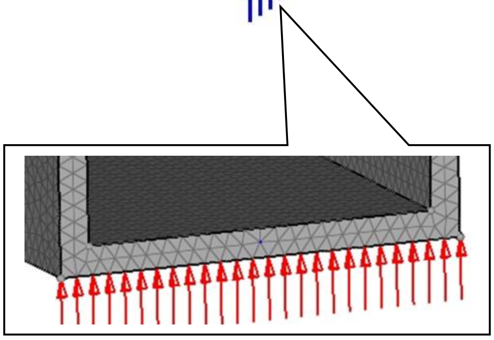


Figure 35: Boundary conditions for LC1c

The displacement constraint in point E is acting on 5 nodes as can be seen in Figure 36. This point represents the cable connection for the cable which drives the linear motion of the telescopic section.



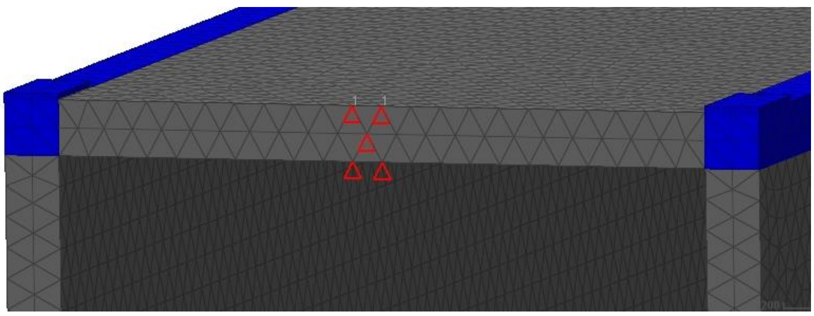


Figure 36: Displacement constraint at hinge point

An overview of the boundary conditions for the operational conditions are given in Table 16. The contact areas of the design and non-design domain consists of bonded contacts. This means that these parts are fixed to each other.

Table 16: Boundary conditions for LC1c

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC1c	Nodes	E	0	Free	Free	Free	Free	Free
	Surface	A,B,C,D	Free	0	0	0	Free	0

In this next part, the applied forces and accelerations are defined. A standard earth gravity is applied to the geometry. Additionally, the operational accelerations are included in the model. The accelerations for the operational load case is given in Table 46 which results in four additional acceleration combinations. The vertical gravity acceleration is included in these accelerations, assuming that this is always the most dominant direction. These accelerations work in the opposite direction and therefore the resultant force can be added together. These accelerations can be incorporated into Hyperworks by specifying the accelerations as a factor of the gravity acceleration. In this situation orthogonal components are defined which are a factor of the vertical gravity acceleration. The factors in Table 17 are multiplied with the gravity acceleration in order to obtain the acceleration components.

Table 17: Operational accelerations for LC1

Acceleration combinations				
	Additional load cases			
Load case	**LC1Ca**	**LC1Cb**	**LC1Cc**	**LC1Cd**
Gravity acceleration [mm/s²]	11350			
Longitudinal	-0.12511	0.12511	0.12511	-0.12511
Transversal	0.074	-0.074	0.074	-0.074
Vertical	-1	-1	-1	-1

7.4.3 LC2b: Deployment/retrieval condition.

In *deployment/retrieval condition* the gangway is in uplift condition. No live loads are acting on the gangway structure. The principle loads on the gangway consists of the self-weight and the additional weights which are acting on the gangway structure. The horizontal and vertical loads due to the operational motions must be included. The centrifugal force based on the maximum angular velocity and the radius of the considered mass should be included in this situation. The centrifugal force is visualized in Figure 37 and is only acting on a single node. The supporting roller bearings are located at the top and bottom of the gangway system. These roller bearings constrains the displacements in the Y and Z-direction and only allows a rotation around the Y-axis. These constrains are applied at the locations of the active roller bearings. The displacement constraint at the hinge point is defined in exactly the same way as in in the normal operational condition and can be seen in Figure 36.

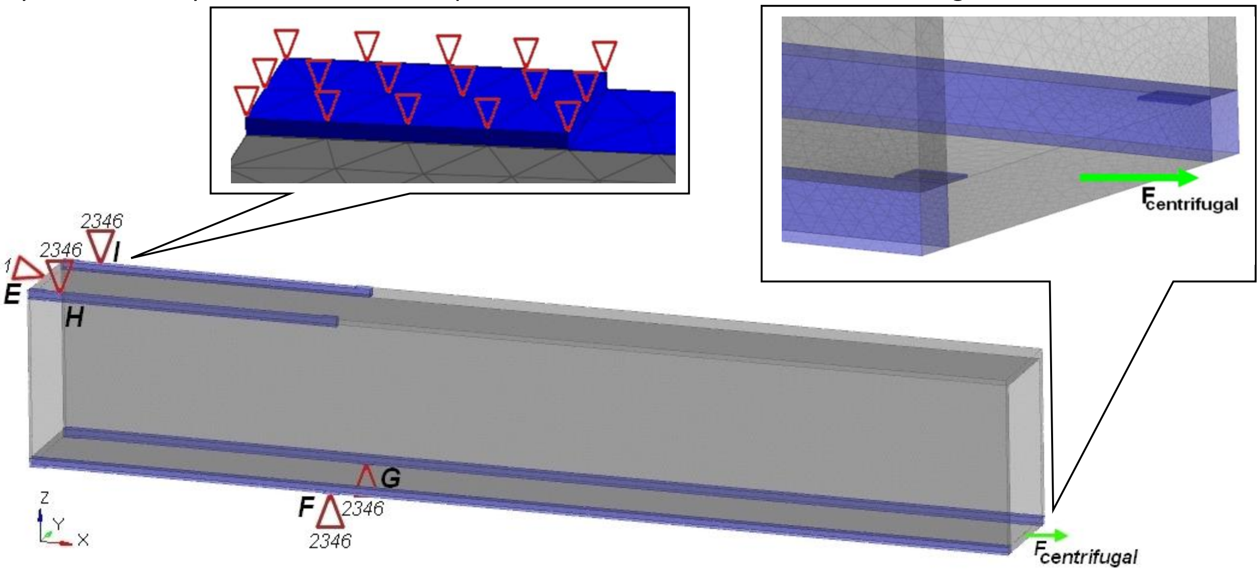


Figure 37: Boundary conditions for LC2b

An overview of the boundary conditions for the operational conditions are given in Table 18.

Table 18: Boundary conditions for LC2b

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC3	Nodes	E	0	Free	Free	Free	Free	Free
	Surface	F,G,H,I	Free	0	0	0	Free	0

A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the deployment retrieval case, dynamic factors are applied. These dynamic factors are specified by the DNV and are given in Table 27. The accelerations are defined as a factor of the gravity acceleration. In this situation orthogonal components are defined which are a factor of the vertical gravity acceleration. These dynamic factors combined with the gravity acceleration results in a gravity acceleration of -10791 mm/s^2.

7.4.4 LC3: Emergency disconnection condition.

In the *emergency disconnection* case, the gangway is in uplift position. Therefore the tip of the gangway is not supported by the landing tool. The live load is applied at the tip of the gangway structure which has a magnitude of 10800 N. The applied tip load is visualized in Figure 38 and is only acting on a single node. The supporting roller bearings are located at the top and bottom of the gangway system. These roller bearings constrains the displacements in the Y and Z-direction and only allows a rotation around the Y-axis. These constrains are applied at the locations of the active roller bearings. The displacement constraint at the hinge point is defined in exactly the same way as in in the normal operational condition and can be seen in Figure 36.

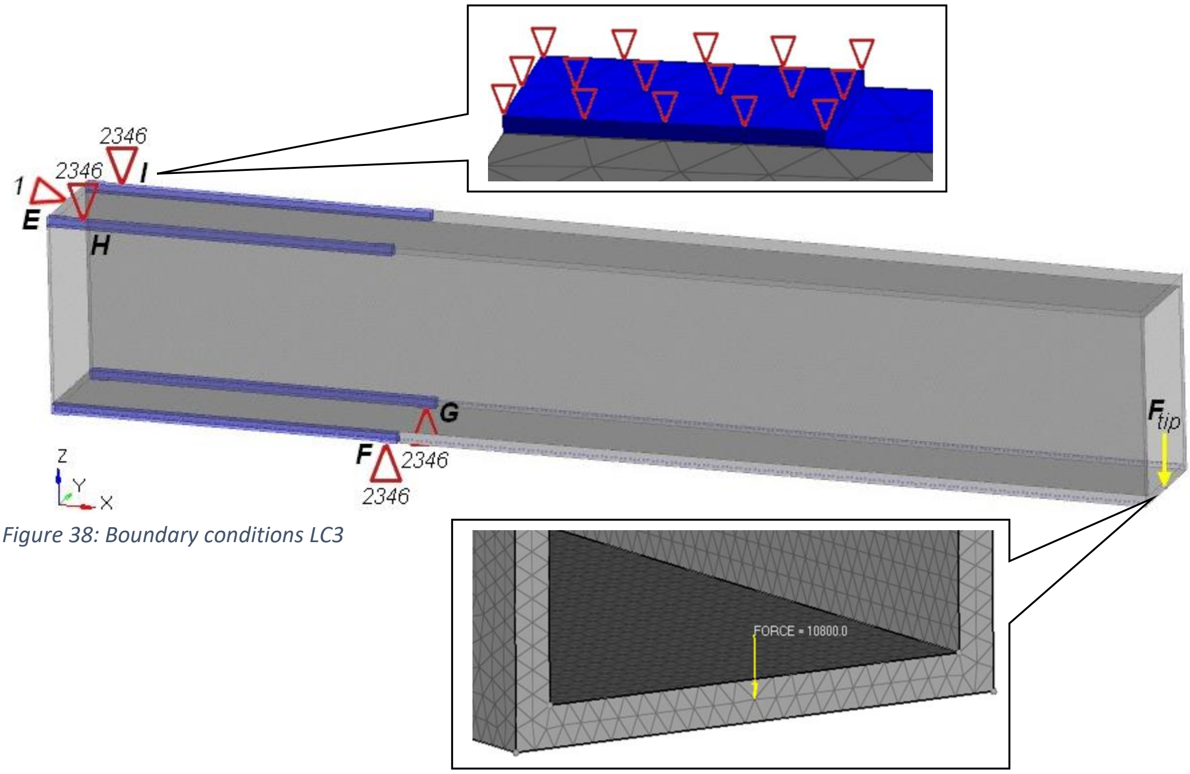


Figure 38: Boundary conditions LC3

Table 19: Boundary conditions for LC3

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC3	Nodes	E	0	Free	Free	Free	Free	Free
	Surface	F,G,H,I	Free	0	0	0	Free	0

A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the emergency disconnection case, dynamic factors are applied. These dynamic factors are specified by the DNV and are given in Table 27. The accelerations are defined as a factor of the gravity acceleration. In this situation orthogonal components are defined which are a factor of the vertical gravity acceleration. These dynamic factors combined with the vessel accelerations results in the acceleration combinations stated in Table 20.

Table 20: Acceleration combinations LC3

Acceleration combinations [mm/s²]				
	Additional load cases			
Load case	**LC3a**	**LC3b**	**LC3c**	**LC3d**
Gravity acceleration [mm/s²]	12450			
Longitudinal	-0.2024	0.2024	0.2024	-0.2024
Transversal	0.0675	-0.0675	0.0675	-0.0675
Vertical	-1	-1	-1	-1

7.5 Additional optimization parameters

The optimization control parameters allows the user to set the control parameters for the optimization process. This will override the default settings for the optimization process and allows the user to customize the optimization process for each specific design problem. During this research, the effect of the optimization parameters on the output of the optimization process is investigated. The method for testing these control parameters was OFAT (One Factor A Time). This means that only one parameter was changed at a time to investigate the effect on the output. Different control parameters are analysed and explained in this section.

DESMAX

The optimization parameter *DESMAX* is used to determine the maximum amount of iterations allowable for which the solver can run before the optimization process is terminated. The default setting of DESMAX is 30. For quick analysis of the optimization process, a low value for this parameter is suggested. This will reduce the required simulation time but it will decrease the quality of the final result. This effect is due to the non-converged results. This control parameter was investigated further with larger values in order to determine the effects on the optimization results. For higher values, the result converges until the objective tolerance was reached. This resulted in an increased design volume and a decreased weight compared to the optimization run with less iterations. This optimization parameter was set to a large value to ensure convergence of the objective.

OBJTOL

This control parameter is the relative convergence criteria of the objective function. It describes the difference between two consecutively iteration steps. It describes the similarity between two successive iterations in a row. When this convergence criteria is below the stated value of the OBJTOL, then the optimization process is terminated. A reduction of this parameter results in an increase of the amount of iterations required to reach the converge objective which increases the simulation time. During the optimization process, the value of the OBJTOL parameter reduced in order to investigate the effect on the final result. This resulted in an increase of the design volume and a reduction in the design weight. The design volume change was asymptotic with respect to the amount of iterations.

MINDENS

The *MINDENS* parameter is used to control the minimum element density which is allowed for the mesh elements in the optimization. It was interesting to investigate the effect of this parameter on the design volume and weight. Lowering the value of this parameter resulted in an increase of the design volume. For low values of this parameter, the design volume converged at a larger increase in design volume. The weight of the design reduces considerably and therefore a lower value was obtained for the optimization process. According to the Hyperworks manual, very low values increases the risk of numerical problems. The default setting is 0.01.

MATINIT

This parameter determines the element density for the initial design space. The element density is homogenously distributed over the initial design domain. The investigation of this parameter showed the following behaviour: When the default value was applied, then the design volume decreased during the initial iterations and increased in later iterations. At the end of the optimization process it converges at an increased design volume compared to the starting point of the optimization. The reason for the phenomena is due to the fact that the high initial density will help to satisfy the stiffness constraints but when the objective is to minimize the mass, then the optimizer will decrease the design volume. Therefore for optimizing a design with a minimum mass, it is recommended to attain a low value for the initial design space density. In this case the stiffness constraints are not fulfilled and the design volume is increasing during the iterations until the stiffness constraints are satisfied.

DISCRETE
The parameter is used to control the penalization factor for the SIMP method. This parameter will suppress the intermediate densities in the solution. Therefore by increasing this value, a more discrete solution will be found. The penalization technique in Optistruct is the “power law representation of elastic properties” which is extensively described in section 4.3.2 The SIMP method. The value of this parameter determines the penalization factor. An increase of this parameter results in a decrease of the amount of elements which have a density between 0 and 1. Increasing this parameter will result is a more discrete solution, this means that the distinction between solid and void elements becomes more clear. A large value of this parameter will result in a structure with a large proportion of elements with high densities. This parameter can be varied between 0 (default) and 4. Larger values leads to unintuitive structures with bad properties.

MINDIM
This parameter determines the minimum member size of any member in the structure. It will prevent the generation of small beams in the optimized design. This parameter can be used to control the beam size in order to generate an optimized topology which satisfies certain manufacturing constraints. By increasing this constraint it will generate a less optimal but more realizable structure. The benefits of increasing the MINDIM parameter is that it generates a mesh independent solution and that the checkerboarding effect is suppressed. The disadvantage is that more iterations are required to reach convergence and that the solution will lose optimality.

VOLFRAC
The parameter *volfrac* constraints the allowable volume fraction of the initial design domain that shall be used in the optimized structure. This parameter is can be defined as the objective or as a constraint where the upper and/or lower bounds are specified. Lowering this value means that the allowable volume is decreased which means more strict boundary conditions for the optimizer and therefore more iterations are required in order to reach convergence.

Overview optimization parameters
The following parameters are used during the optimization process. These parameters are determined by a trial and error approach, or by investigating similar optimization processes.

Table 21: Design parameters

Parameter	*Value*
Desmax	1000
Mindim	500
Matinit	0.200
Mindens	0.010
Discrete	3.000
Checker	1
MMcheck	1
Objtol	0.00005
Shapeopt	2
Optmeth	Dual

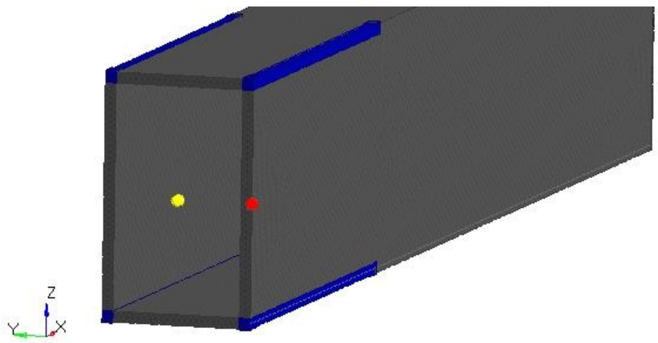


Figure 39: mirror nodes

Due to the complexity and relation of the two sections, it is decided to analyse the two sections of the gangway separately. The loads and masses of both parts will be related to each other. A symmetric plane could be applied to the mid-section of the tube. This would reduce the amount of computational effort which is required and ensures a complete symmetric structure at the X-Z plane. This mirror plane is defined by two nodes: An anchor node, which defines the location of the mirror plane and a first grid node, which defines the orientation of the mirror plane. This type of pattern grouping requires that the anchor point and the first point be defined. A vector from the anchor point to the first point is normal to the plane of symmetry. In Figure 39, the anchor node is defined in yellow and the first grid is shown in red. These two nodes define the mirror plane in the X-Z plane.

7.6 Topology optimization results.

Before the optimization task is performed, it is recommended to perform a finite element analysis. This step allows the user to check the applied boundary conditions for the different load cases and therefore to validate a proper set-up of the optimization. Also the response of the design domain on the loads and supports can give an indication where material may be placed by the optimizer.

For the optimization process, it is chosen to vary the allowable volume for the optimization process. It starts with an allowable volume domain of 30%, which was lowered gradually to 20 %. Lowering this constraint means that the optimizer must perform more iterations in order to reach the convergence criteria. This is due to the fact that it is more difficult to find a solution with these more severe boundary conditions. This means that the simulation time which is required to solve the problem is also increased. The most promising results will be used for the size optimization.

7.6.1 Telescopic section

All the parameters described in the section: Topology optimization process are applied in the optimization process. After 99 iterations the following result was obtained as can be seen in Figure 40. This was achieved for an allowable volume fraction of 20%. An increase in the volume fraction did not change the structure significantly. The contour plot shows the element densities of the design. Red means that the mesh element has a density of 1 and therefore fully consists of material. Blue means that the mesh element only consists of only 1% material. The other colours are showing the intermediate densities, varying between 1% and 100% material.

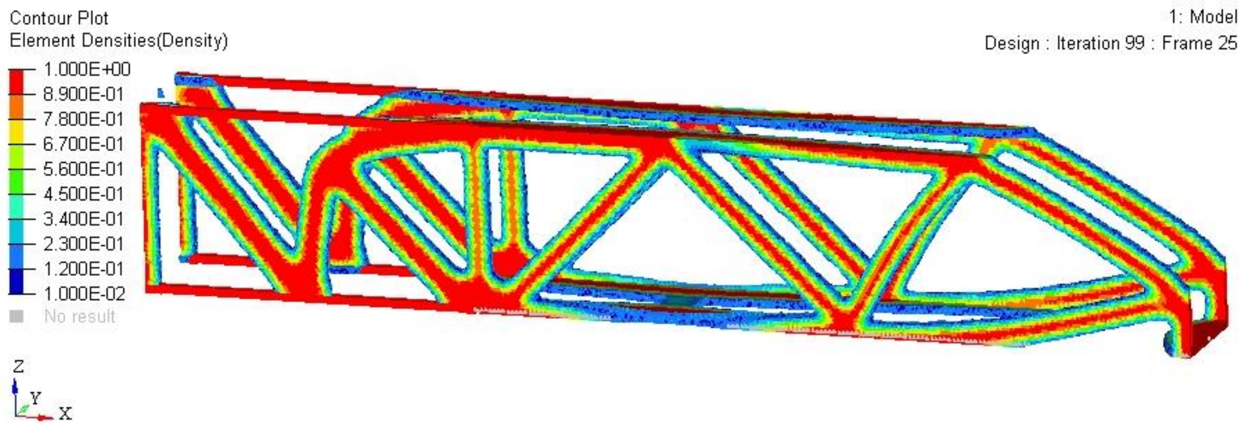


Figure 40: Optimization results with the corresponding densities

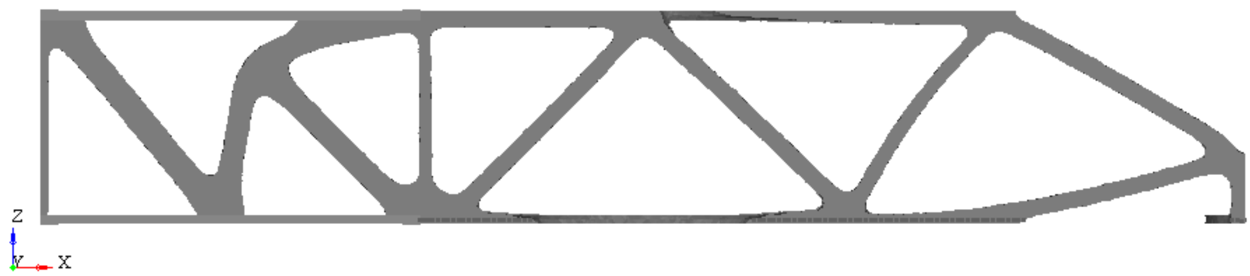


Figure 41: Side view of the optimization results

From a structural point of view this does not look like an efficient structure. The tip of the gangway is very susceptible for buckling and the curved beams are not efficient. This design will probably fail due to buckling. Also no horizontal stiffeners are generated between the two sides of the design, while operational accelerations are incorporated into this optimization. The material at the location of the cable connection is disappeared, which would suggest that this support condition is unnecessary. This result is obtained by using proper boundary conditions which shows the expected response during the FEA check. The results from the topology optimization are sometimes difficult to interpret and quite often the result is not intuitive. The results from topology optimization will most often tend to be quite organic looking and therefore not something that a manufacturing engineer would approve to put into production. The beam surfaces are still somewhat undefined due to the fuzzy transition from low to high densities. It consists of solid beams which are often slightly curved. Therefore these results are not applicable in reality and should be interpreted into something that is possible to manufacture. However, this result shows a clear orientation of beams and shows the load paths which transfers the load from the tip of the telescopic section towards its support conditions.

The result from the optimization process will only serve as an indication for the final design. The post-processing steps or the design realization process is used to interpret and convert the results from the topology optimization in order to obtain a feasible design. In the detailed optimization the design is redesigned by using size and shape optimization. In consultation with the company supervisor, it is chosen check the results from this optimization run, but also to use the optimization results which are obtained with the first test run. This model has been obtained by using a different set-up of boundary conditions but this resulted in a different design. This result can be seen in Figure 42. With this approach a comparison between the different beam layouts can be given.

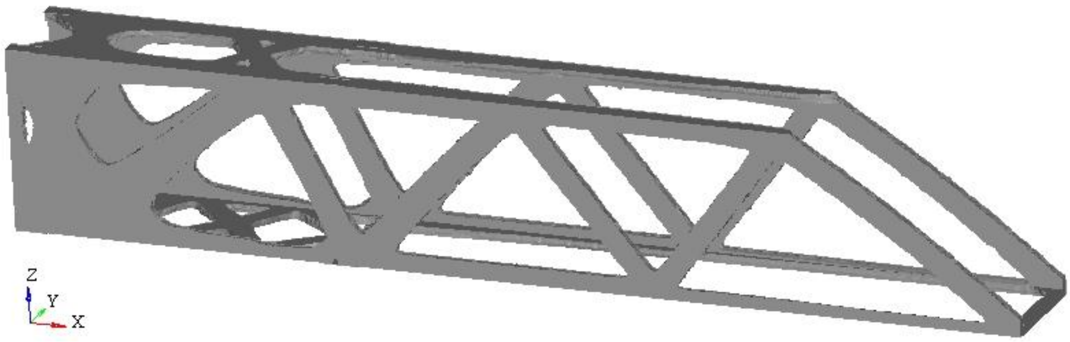


Figure 42: Optimization result of the first run

7.6.2 Fixed section

In this section the results for the topology optimization of the fixed section are given. The set-up and approach of this optimization process is identical to the optimization process of the telescopic section. For this reason it is chosen to show only the results of this optimization process. An extensive description of the set-up and approach of the topology optimization is given in appendix: A-17. Topology optimization: Fixed Part.

After 381 iterations the optimization process was terminated manually, due to the low convergence rate. The following result was obtained as can be seen in Figure 43. This was achieved for an allowable volume fraction of 20%. An increase in the volume fraction did not change the structure significantly.

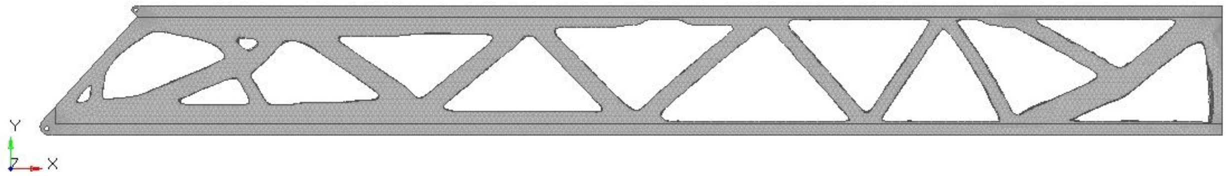


Figure 43: Results topology optimization fixed part

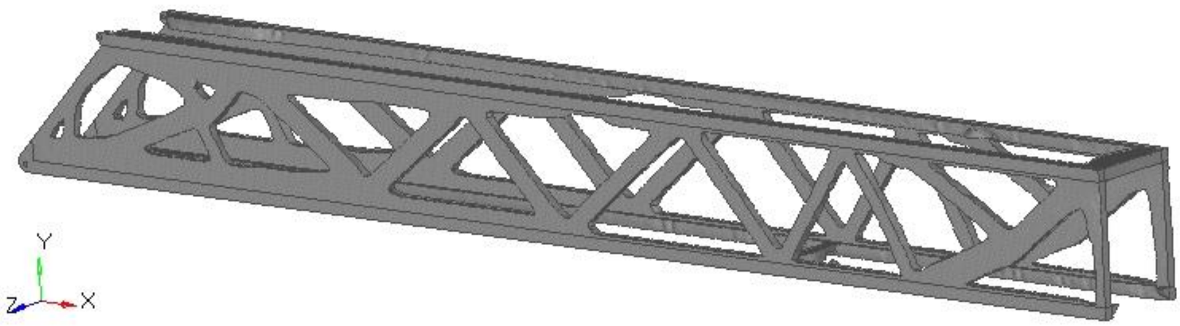


Figure 44: Results size optimization fixed part in isometric view

The result shows a structure which has the characteristics of a truss structure. The main longitudinal members are connected by means of diagonal bars. Some side stiffeners are generated which connects the two sides of the gangway structure. These are generated due to the operational accelerations. A symmetry plane was applied in order to obtain a symmetric structure in the X-Y plane. The minimum member size constraint enforces a design which consists of well-defined beams.

The result from the optimization process will only serve as an indication for the final design. The design realization is used to interpret and convert the results from the topology optimization in order to obtain a feasible design. In the detailed optimization the design is redesigned by using size and shape optimization. This model is converted to a line model in appendix: A-18. Size optimization: Fixed part. The results of the size optimization are shown in Chapter 8.

8. Design realization

This chapter is concerned with all the post-processing steps which are performed in order to obtain a feasible design. The approach, set-up and description of all steps are given. At last, the results from the design realisations are presented.

8.1 Post-processing steps

The results from the topology optimization often have an organic shape with a non-smooth surface depending on the mesh size. Often this design is difficult to manufacture and can only be realized by using 3D printing. This is not a realistic solution for manufacturing a gangway structure. The beam surfaces are irregular and the structure is modelled as a solid casted part without any connection points. The obtained results are therefore not applicable in reality and therefore should be converted into a design which is suitable for common manufacturing methods. The concept obtained from the topology optimization is often not an optimal structure. It has an optimal topology but often the units are not well dimensioned. In the design realization the results from the topology optimization are converted and changed by the size optimization in order to produce an optimal solution in terms of dimensions. Interpreting the results from the topology optimization is a difficult task and requires experience and knowledge of other aspects such as manufacturability and buckling behaviour. Therefore, the results from the topology optimization serves as a guideline for the design engineer, which can use these results for designing the detailed design of the structure.

The realization is needed in order to perform a fair comparison with the current design and to obtain a feasible design. Therefore the goal is to generate a realizable suggestion from the topology optimization result. It is not recommended to use solid beams because the material in the centre of the profile does not contribute to the torsion and bending stiffness. The buckling behaviour of the structure could be decreased by using hollow profiles. Therefore it is not optimal to use solid beam elements. Hollow profiles are also preferred due to their availability and practical implementation. To get the optimum dimensions of the beams, a size optimization will be performed.

The following steps needs to be performed in order to set-up the optimization:

-Step 1: Open the topology optimization results and define temporary nodes.
-Step 2: Define linear lines between the temporary nodes.
-Step 3: Define the beam element cross-sections.
-Step 4: Define the material properties.
-Step 5: Define element properties and assign it to elements.
-Step 6: Mesh the model.
-Step 7: Assign element properties to mesh elements.
-Step 8: Create load collector and define constraints and loads.
-Step 9: Define the load cases.
-Optional: perform an analysis
-Step 10: Define the design variables.
-Step 11: Define the generic relationship between the design variables.
-Step 12: Define the required responses.
-Step 13: Define the design constraints.
-Step 14: Define the objective
-Step 15: Run the optimization.

An extensive step by step description can be found in appendix: A-15. Set-up size optimization. In this section, a brief description of the set-up is given. Due to the fact that the results of the topology were dissimilar and often not intuitive from an engineering point of view, it is chosen to apply the size optimization for two different designs which were obtained from the topology optimization process. Also the size optimization will be applied to the current design. The design with the lowest weight will be used as input for the optimization of the fixed section. The set-up for the size optimization of the first design will be explained in this section. The procedure is exactly the same for the two other designs, but for these two designs, only the results of the size optimization will be shown. The first step in the design realisation, is to convert the topology model to a line model. The line model will represent the orientation and location of the 1D beam elements. 1D beam elements are first order elements which are generated between two nodes to model axial forces, shear forces and bending and torsion moments. The three line models which are used as input parameters for the size optimization are shown in Figure 45, Figure 46 and Figure 47 respectively.

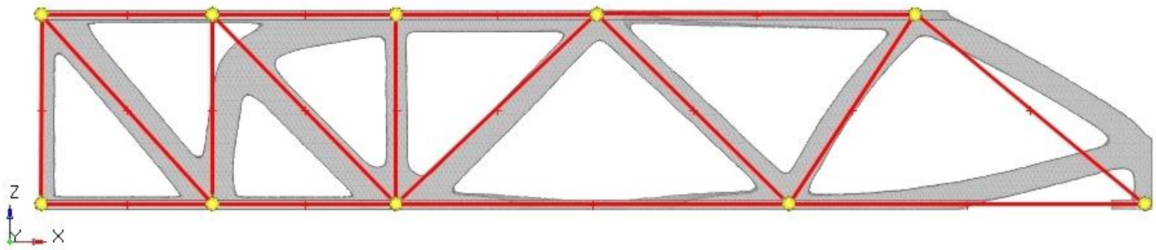


Figure 45: Nodal configuration topology 1

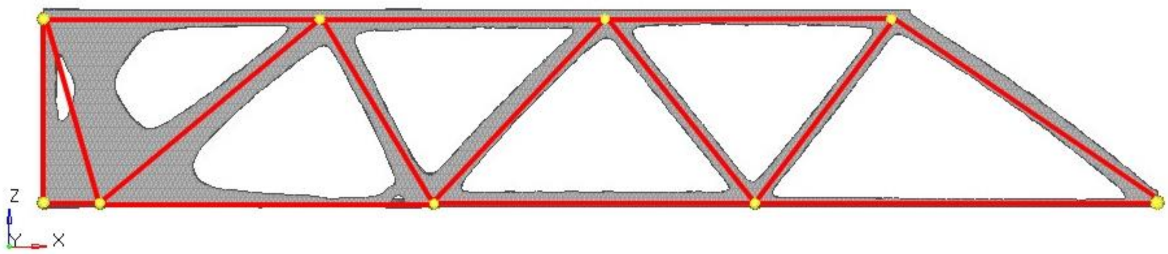


Figure 46: Nodal configuration topology 2

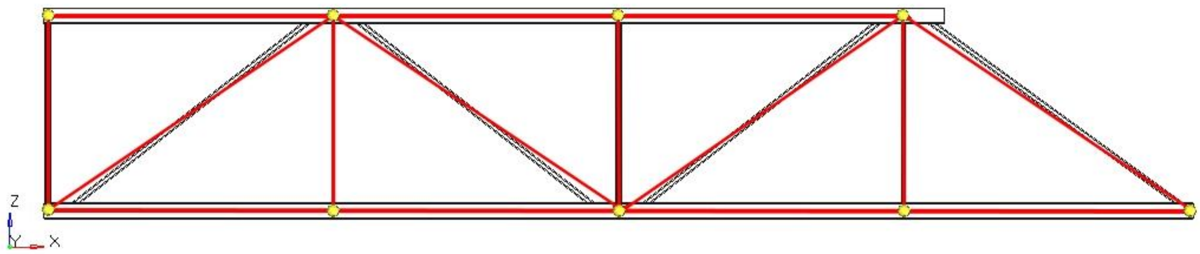


Figure 47: Nodal configuration current design

The result of the topology optimization is imported into Hypermesh by using OSSmooth. The model can be imported by selecting the model and the corresponding result file. Temporary nodes are defined on the intersection point of the beams as can be seen in Figure 48.

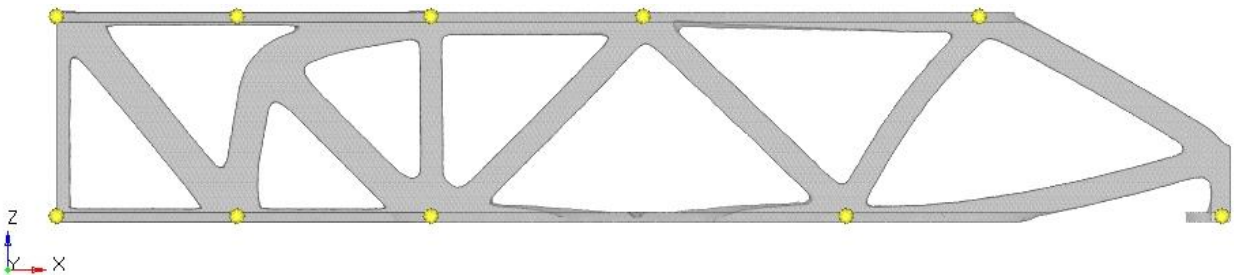


Figure 48: Temporary nodes in the design domain

The nodal coordinates are defined manually and created on the intersection points of the beam elements in the centre of the beam. This can be seen in Figure 49.

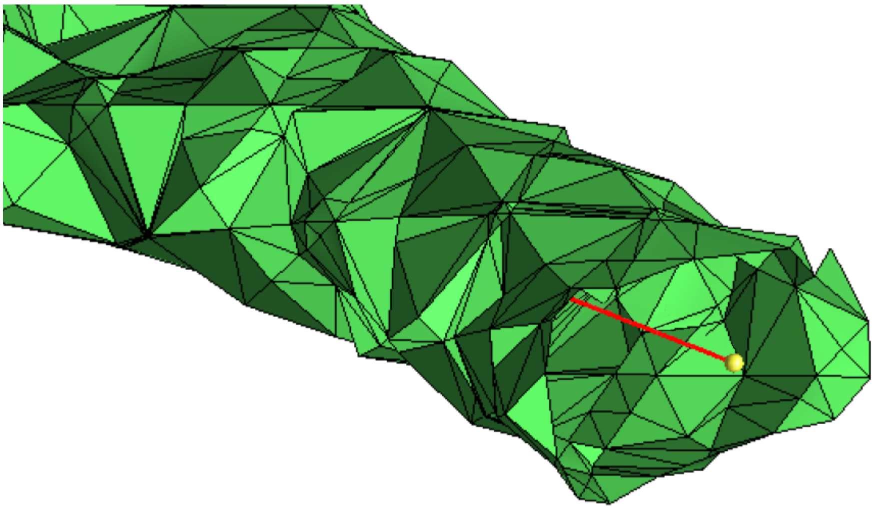


Figure 49: Nodal coordinates in centre of the beam

These temporary nodes are interconnected by linear lines, which represent the orientation and direction of the beams. The beams have rigid connections at the nodes. The line model can be seen in Figure 50.

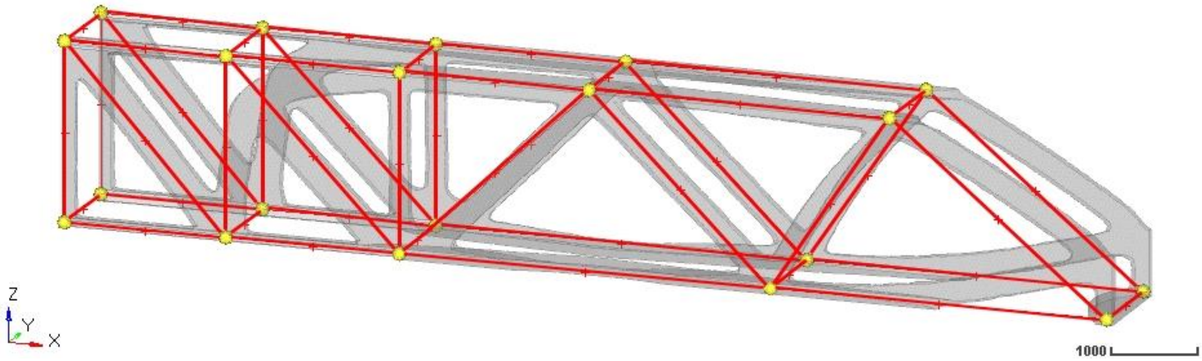


Figure 50: Nodal configuration connected by linear lines

The grey transparent shell corresponds with the topology optimization results. The yellow points are the temporary nodes which are located at the midpoint of the cross-section at the intersection points of the beams. The red lines define the beam axial centrelines. Different cross-sections are defined which can be related to the different line elements. A circular hollow section is chosen for the diagonal and vertical elements and for all the horizontal elements a rectangular tube section is chosen. The reason for this is that the rectangular cross-section allows for attaching the supports for the linear telescopic movement. These are for example the sliding pads or roller bearings. Another advantage is that it allows a more ease connection with the circular tubes. This is preferred for the manufacturing process. The cross-sections are defined by using Hyperbeam, which is a build-in tool in Hyperworks to create cross-sections. The gangway structure will be made out of steel tubes and therefore a material collector is defined with the properties of S355 structural steel. The assumed primary structure material is S355 or similar, with the properties according to Table 22.

Table 22: Properties S355 structural steel

Property	**Magnitude**
Yield strength	355 MPa
Modulus of elasticity	200 GPa
Poisson's ratio	0.3
Density	7850 kg/m^3

Now the gangway structure will be partitioned into different domains. Each beam elements and its symmetric adjacent element is assigned to a property. This allows independent size optimization of all the different beams in the design domain. For each property, the corresponding cross-section and material is selected. These properties will be assigned to the mesh elements in a later stage. The properties are related to the model according to element numbering visualized in Figure 51 and Figure 52. It is chosen to have a constant cross section for the longitudinal beams in the structure. This will enhance the manufacturability of the structure.

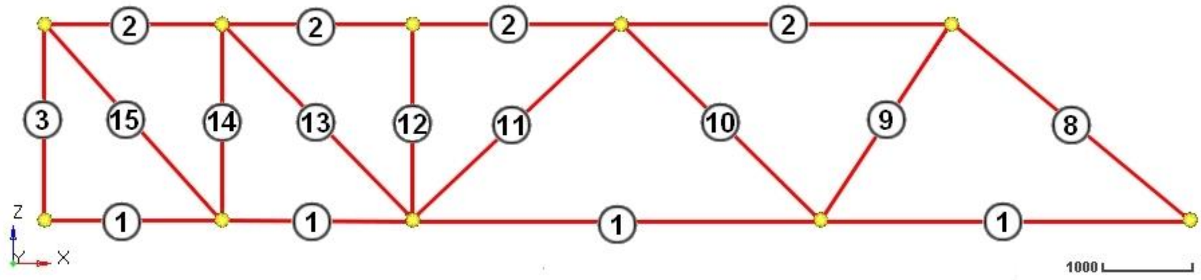


Figure 51: Property element relation

In Figure 52, the isometric view of the line model is given. All the lines which can be mirrored according to the Z-X Plane are assigned to the same property. This is done in order to obtain a complete symmetric structure in the Z-X plane.

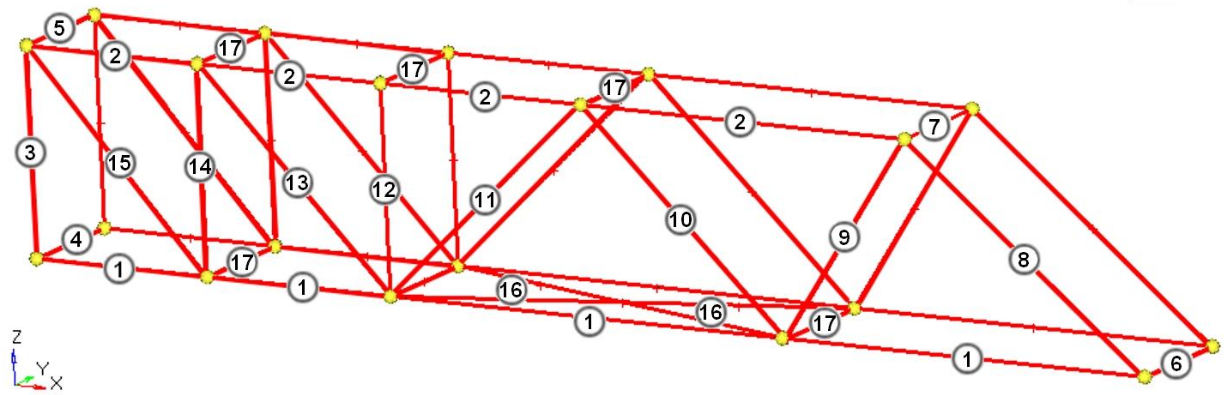


Figure 52: Property element relation isometric view

The element properties are given in Table 23, in which the materials and the beam sections are given.

Table 23: Beam element properties

Beam elements	**Material**	**Beam section**
1 - 7	S355 Steel	Hollow rectangular
8 - 17	S355 Steel	Hollow circular

The model is meshed with 1D line elements with an element size of 4 mm. The total amount of elements is equal to 29932 .This mesh size is validated by performing a mesh convergence check, in which the mesh size is reduced until the response is converged. This mesh convergence check is shown in appendix A-11. The orientation of the elements needs to be specified and the different properties are assigned to the different elements.

8.2 Boundary conditions

The gangway structure is subjected to several load cases according to Table 7. The following load cases are incorporated into the sizing optimization process:

-LC1C: *Normal operational conditions.*
-LC2B: *Deployment / retrieval load case.*
-LC3: *Emergency disconnection.*

According to Table 6, five load cases needs to be analysed. It is chosen to only use three load cases for the optimization process. The other load cases will be analysed during the FEA. An extensive description of the load cases is given in A-6. Input parameters optimization process. The structure is constrained by means of load collectors. Load collectors are generated in which the support conditions and load conditions are generated respectively. A load collector contains the forces or support conditions per load case. A description of the loads and support conditions per load case is given in the following sections.

8.2.1 LC1c: Normal operational condition

First the operational situation is considered. In this situation the gangway is in normal operational condition and therefore it is supported on both sides. The structure is able to transfer people and cargo from and to the offshore structure. The structure is supported by the following displacement constraints: At the hinge point for connecting the cable for the linear motion, a displacement constraint is added which constrains the translation in the X-direction. This constraint is applied at node 671 as can be seen in Figure 53. The landing mechanism is located at the tip of the gangway. This system constrains the displacements in the X, Y and Z-direction and only allows rotation around the Y and Z-axis. This boundary condition is not applied, instead of this boundary condition, it is chosen to apply the reaction force at this point. This is due to the fact that the live load is acting in the middle of the total gangway and that the mass of the fixed section is also partially supported by this boundary condition. This is explained in A-6. Input parameters optimization process. These boundary conditions are incorporated into the optimization process and therefore two load collectors are defined with the corresponding support conditions and loads.

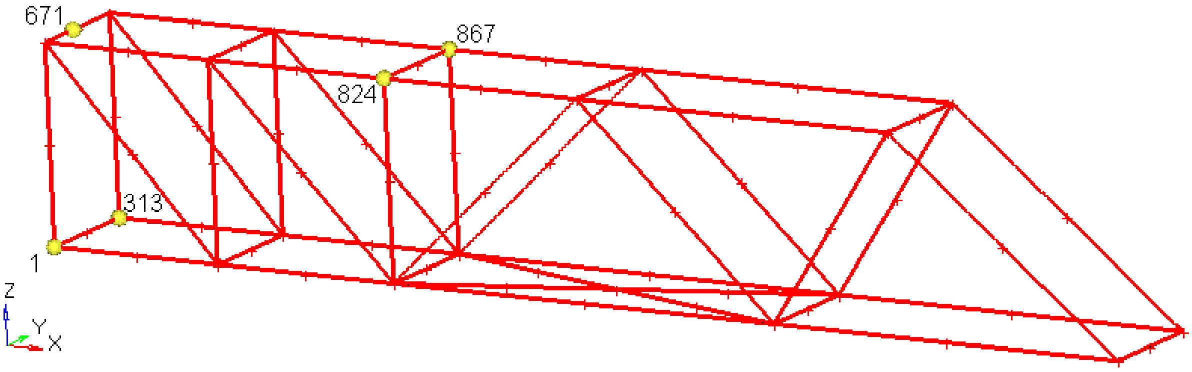


Figure 53: Constrained nodes for LC1c

An overview of the boundary conditions for the operational conditions is given in Table 24.

Table 24: Boundary conditions for $LC1_c$

	Node	*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
$LC1_c$	1	Free	0	0	0°	Free	0°
	313	Free	0	0	0°	Free	0°
	824	Free	0	0	0°	Free	0°
	867	Free	0	0	0°	Free	0°
	671	0	Free	Free	Free	Free	Free

In this next part, the applied forces and accelerations are defined. A standard earth gravity is applied to the geometry. Additionally, the operational accelerations are included in the model. These accelerations acts parallel or normal to the gangway deck. The downward (vertical) acceleration is considered normal to the deck. The longitudinal and transversal accelerations are considered parallel to the gangway deck and are perpendicular to each other. The vertical gravity acceleration is equal to 9810 mm/s^2. The accelerations for the different load cases are given in Table 25 which results in four additional load cases for the normal operational condition. The vertical gravity acceleration is included in these accelerations, assuming that this is always the most dominant direction due to the fact that these accelerations work in opposite direction and therefore the resultant force can be added together.

Table 25: Operational accelerations

Acceleration combinations [mm/s^2]				
	$LC1C_A$	**$LC1C_B$**	**$LC1C_C$**	**$LC1C_D$**
Longitudinal	-1420	1420	1420	-1420
Transversal	840	-840	840	-840
Vertical	-11350	-11350	-11350	-11350

In addition to the self-mass of the gangway structure, the gangway boom will be loaded by additional components such as flooring, hand rails and landing tool. For the structural optimization of the beams it is important to incorporate these masses into the load collector. The extensive calculation of these additional masses can be found in appendix: A-5. Design calculations gangway.

The additional loads acting on gangway boom are estimated as:

Point mass	Stated mass
Roller mass (R_m)	600 kg
Flooring and hand rails (F_m)	1000 kg
Telescoping adjustment system (A_{sm})	1000 kg
Intermediate platform (P_m)	500 kg
Utility lines and supporting structure (U_m)	1000 kg
Landing/connection tool (L_m)	400 kg

These values are split between the fixed and telescopic section according to the length ratio of the two parts. This results in a total additional mass of 845 kg for the telescopic part.

$$additional\ mass\ telesc. = \frac{F_m}{Total\ length} \cdot cantileverd\ length + \frac{U_m}{Total\ length} \cdot cantileverd\ length + Lm$$

These additional masses are represented by point masses. These point masses are distributed equally over the members 1 till 4 and their symmetric equivalent. The mass of the landing tool is located at the tip of the gangway structure and is equal to 0.4 tonnes. Therefore a point mass located at node 275 is defined as can be seen in Figure 54.

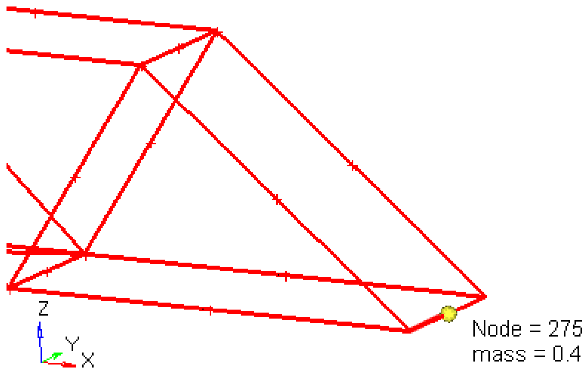


Figure 54: Point mass gangway tip

The reaction force from the support condition is applied as a distributed force at the location of the support as is specified in A-6. Input parameters optimization process. For the normal operational conditions, a safety factor of 2 is applied according to the DNV. The applied load is visualized in Figure 55. The reaction force *Fr* is divided over the amount of nodes of the supporting beam.

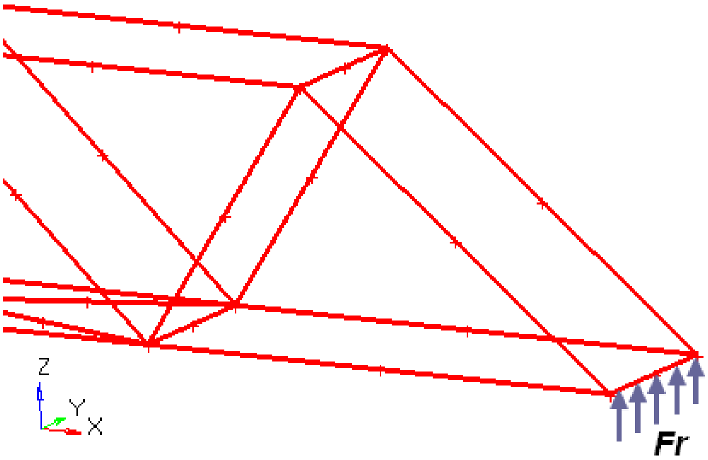


Figure 55: The reaction load acting on the gangway

The wind load is not applied in this load case. The wind force can be calculated if the exposed surface area is known. In the optimization process, the lateral surface area changes and therefore it is chosen to apply the wind force during the final FEA. It is assumed that the wind load has a relative low impact on the structure due to the fact that it acts on an open truss structure. The retrieval wind velocity is specified by the DNV, which is converted into a pressure. For the operational condition, the wind pressure is equal to *245 Pa*. The wind direction is orientated in the opposite direction of the lateral acceleration, which results in the most unfavourable situation. This can be the positive or negative Y-direction. The lateral acceleration of the gangway structure is equal to the acceleration in the Y-direction.

During the normal operational conditions, the gangway is subjected to additional constraints. According to the DNV, the safety factor regarding the yield stress is equal to 1.5. The means that the member stresses may not exceed the limiting value of 237 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. In the normal operational condition, the deflection of the structure may not exceed a value of 65mm. This constraint is applied to the nodes 16, 46, 260, 275 and 290 may not exceed a vertical deflection of 65mm. These nodes are visualized in Figure 56.

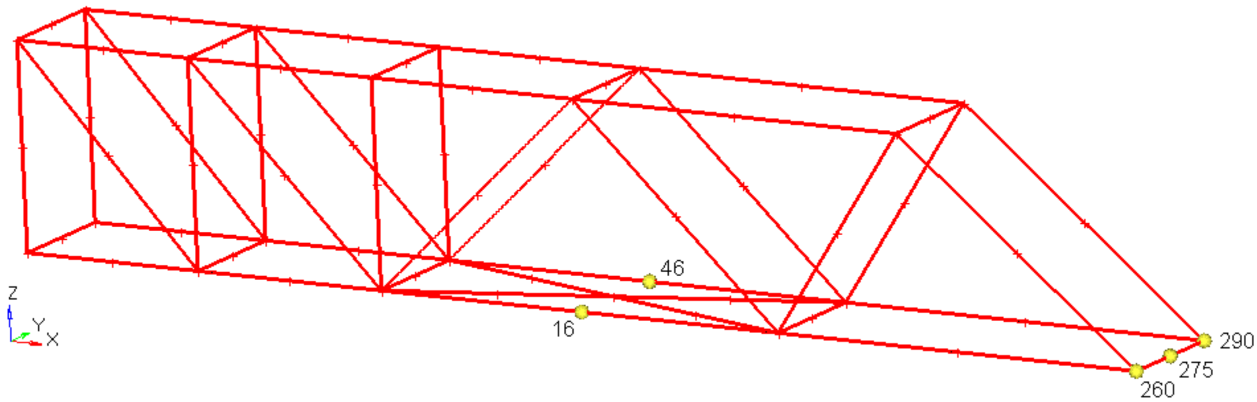


Figure 56: Deflection limits LC1c

8.2.2. LC2b: Deployment / retrieval condition.

In this situation, the gangway is fully extracted and in uplift position. The tip of the gangway is not supported by the landing tool. Only the self-weight and the additional weights are acting on the structure and no live loads are considered. The load due to the operational motions and the wind loads are included in this situation.

In this situation the gangway is supported by the following boundary conditions: A displacement constraint is added at node 671 which constrains the translation in the X-direction. This constraint is due to the cable which drives the telescopic motion of the gangway. The nodes 615, 658, 2185 and 2186 are the roller bearings or sliding pads which supports the gangway in the Z and Y-direction. The location of these nodes can be found in Figure 57 and an overview is given in Table 26.

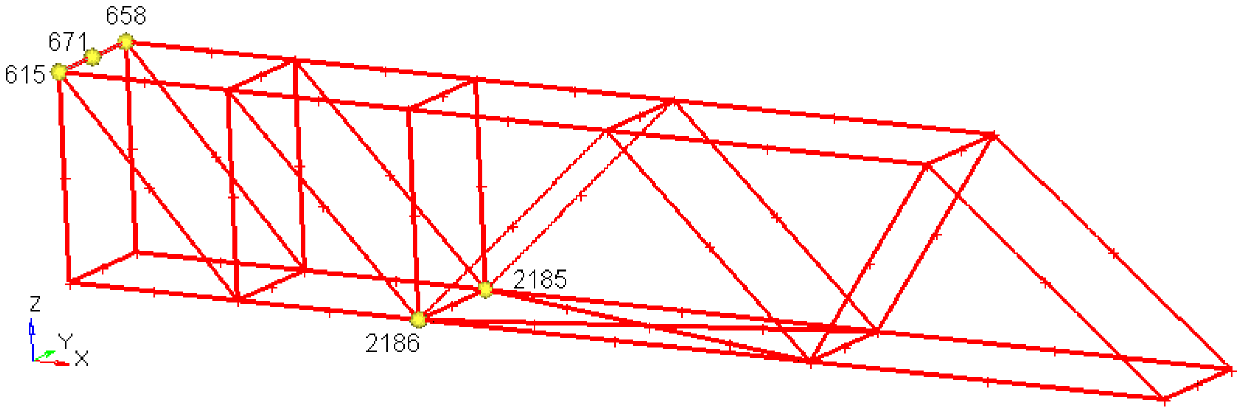


Figure 57: Boundary conditions for LC2$_b$

Table 26: Overview boundary conditions LC2$_b$

	Node	*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC2$_b$	615	Free	0	0	0°	Free	0°
	658	Free	0	0	0°	Free	0°
	2185	Free	0	0	0°	Free	0°
	2186	Free	0	0	0°	Free	0°
	671	0	Free	Free	Free	Free	Free

A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the deployment & retrieval condition and in the emergency disconnection case, dynamic factors are applied. These dynamic factors are specified by the DNV and are given in Table 27.

Table 27: Dynamic factors

Direction	Dynamic factor
	[-]
X	1
Y	1,1
Z	1,1

These dynamic factors and the standard earth gravity can be included in the accelerations which results in the values given in Table 28. This results in four additional load cases for the deployment or retrieval of the gangway structure.

Table 28: Deployment / retrieval accelerations.

Acceleration combinations [mm/s^2]				
	$LC2B_A$	$LC2B_B$	$LC2B_C$	$LC2B_D$
Longitudinal	-1420	1420	1420	-1420
Transversal	840	-840	840	-840
Vertical	-12450	-12450	-12450	-12450

In appendix: A-5. Design calculations gangway, an approximation has been made for the centrifugal force. It is estimated that the centrifugal force is equal to 1300 N which is applied as a concentrated force located at the tip of the gangway. This force is acting in the longitudinal direction as be seen in Figure 58.

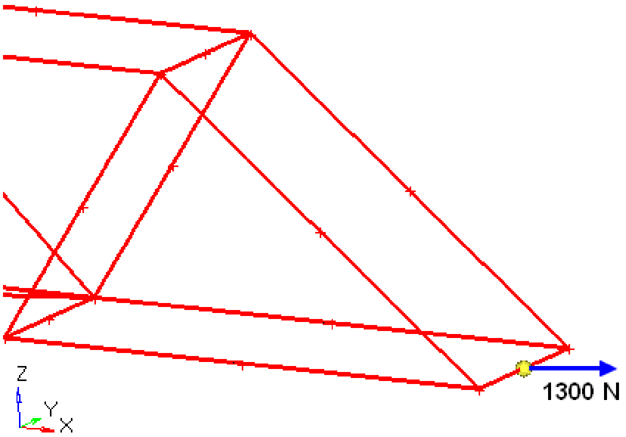


Figure 58: Centrifugal force applied to tip

Also wind loads needs to be considered during the deployment and retrieval load case. In this situation the wind velocity is equal to 36 m/s. This wind velocity is converted to a wind pressure which is multiplied to the lateral surface area. This corresponds to an additional side pressure of *794 Pa* which acts in the opposite direction as the lateral acceleration. This wind load is not included in the optimization process but will be applied and checked during the final FEA analysis.

During the deployment and retrieval condition, the gangway is subjected to following additional constraints. According to the DNV, the safety factor regarding the yield stress is equal to 1.5, which corresponds to acceptance criteria I. The means that the member stresses may not exceed the limiting value of 237 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. For the gangway in cantilevered condition, the nodes located at the tip of the gangway may not exceed a vertical deflection of 130 mm. These nodes are visualized in Figure 59.

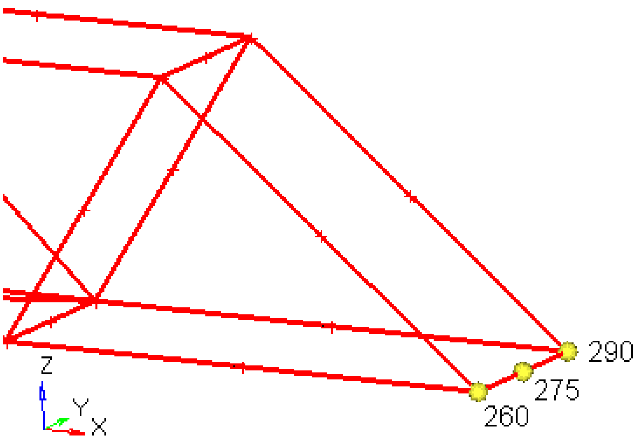


Figure 59: Deflection limit for LC2b and LC3

8.2.3. LC3: Emergency disconnection condition.

In the emergency disconnection case, the gangway is in uplift position. Therefore the tip of the gangway is not supported by the landing tool. A live load is located at the tip of the gangway. In this situation the gangway is supported by the boundary conditions which are identical to the stated boundary conditions in paragraph 8.2.2. LC2b: Deployment / retrieval condition.

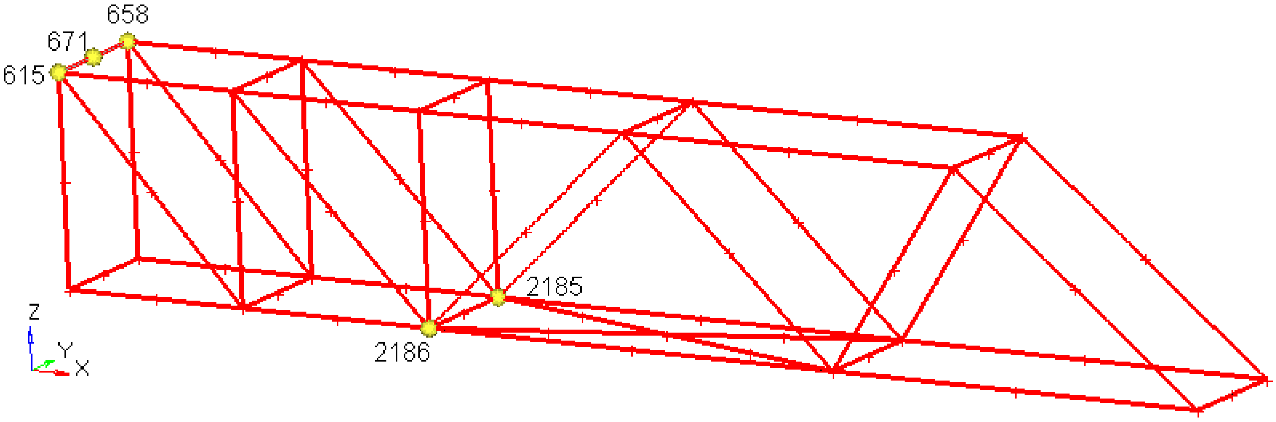


Figure 60: Constrained nodes for LC3

Table 29: Boundary conditions for LC3

	Node	*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC3	615	Free	0	0	0°	Free	0°
	658	Free	0	0	0°	Free	0°
	2185	Free	0	0	0°	Free	0°
	2186	Free	0	0	0°	Free	0°
	671	0	Free	Free	Free	Free	Free

A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the emergency disconnection case, dynamic factor are applied. These dynamic factors are specified by the DNV and are given in Table 27. These dynamic factors combined with the vessel accelerations result in the acceleration combinations in Table 30.

Table 30: Emergency disconnection accelerations

Acceleration combinations [mm/s²]				
	$LC3_A$	**$LC3_B$**	**$LC3_C$**	**$LC3_D$**
Longitudinal	-1420	1420	1420	-1420
Transversal	1940	-1940	1940	-1940
Vertical	-12450	-12450	-12450	-12450

The applied point masses are identical to the point masses as stated in the previous section. The live load is applied at the tip of the gangway without the use of a safety factor. The load is applied as a single point force at node 275 with a magnitude of 10800N. The applied tip load is shown in Figure 61.

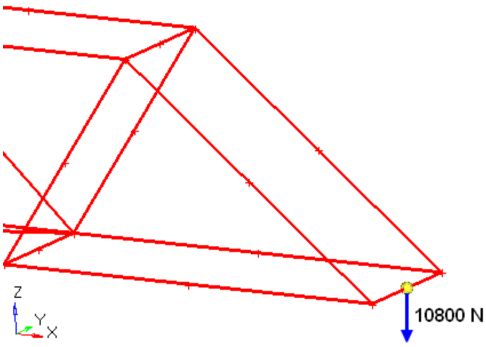


Figure 61: Tip load applied to the model

A retrieval wind load is acting on the structure during the emergency disconnection case. In this situation the wind velocity is equal to 36 m/s. This wind velocity is converted to a wind pressure which is multiplied to the lateral surface area. This corresponds to an additional side pressure of *794 Pa* which acts to the lateral surface area of the beam section. This pressure acts in the opposite direction as the lateral acceleration which is the most conservative case. This wind load is not applied during the size optimization and it will be included in the FEA of the final design.

During the emergency disconnection condition, the gangway is subjected to the following additional constraints. According to the DNV, in this condition the design is subjected to acceptance criteria III, which corresponds to a safety factor of 1.1 regarding to the yield stress. The means that the member stresses may not exceed the limiting value of 323 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. For the gangway in cantilevered condition, the node located at middle of the tip of the gangway may not exceed a vertical deflection of 130 mm.

In het emergency disconnection case, the gangway is in cantilevered condition and is slowly retracted while a live load is acting on the gangway tip. When the gangway is retracting, the support conditions located at the bottom of the gangway structure, are moving into the positive X-direction. Due to this linear motion, the horizontal distance between the support condition increases and the momentum which is generated by the live load is decreased. This is favourable but still it is important to check the situations between a full extended and a complete retracted gangway. Therefore it is chosen to add four additional load cases in which the support conditions are shifted to an intermediate and unfavourable condition. The locations for the support condition related to each load case is shown in Figure 62.

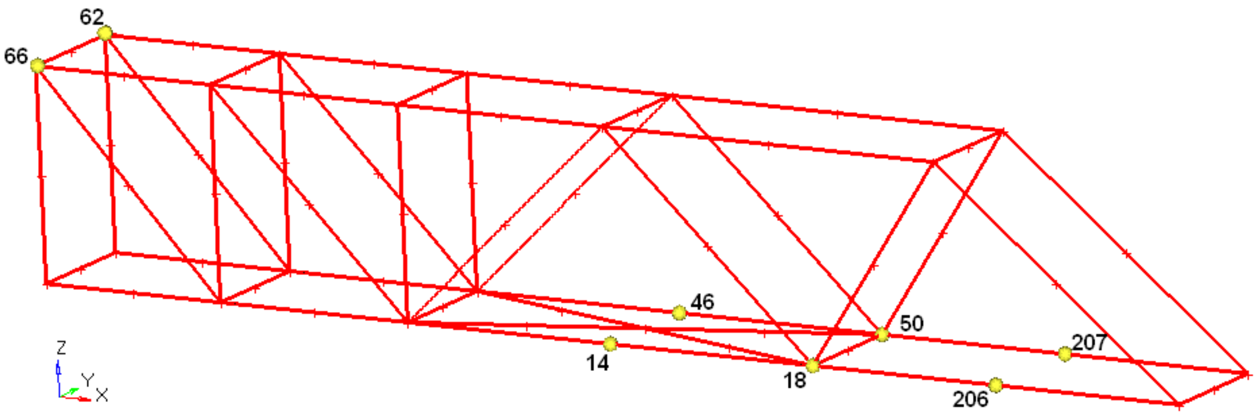


Figure 62: Intermediate load cases

An overview of the intermediate load cases is given in Table 31: Intermediate load cases. All these load cases are constrained in the X-direction for node 671. The subscript 1, 2 and 3 at each load case indicate the group of nodes to which the constraints are applied. This leads to four additional load cases which are subjected to all the conditions described in this section.

Table 31: Intermediate load cases

Load case	*Nodes*	*X component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
$LC3_1$	62, 66, 14, 46	Free	0	0	0°	Free	0°
$LC3_2$	62, 66, 18, 50	Free	0	0	0°	Free	0°
$LC3_3$	62, 66, 206, 207	Free	0	0	0°	Free	0°

8.3 Size optimization process

Design variables are required for the size optimization of the profiles. For the hollow circular tube, two design variables needs to be defined. The inner and outer radius of the tube. The inner and outer radius of the cross-sections will be varied within user defined lower and upper bounds respectively. Therefore design variables must be defined for each cross-section which defines the upper and lower bounds. For the rectangular cross section three different design variables needs to be defined. The height of the profile, the width of the profile and the wall thickness.

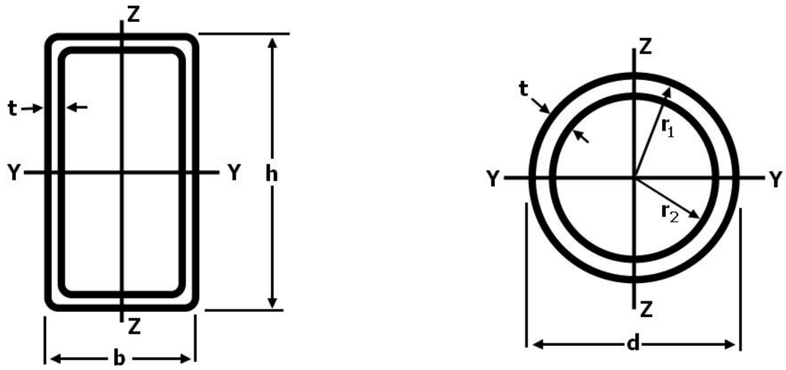


Figure 63: Cross-section design variables

An overview of these parameters is given in Table 32. A design parameter is a characteristic which determines a particular dimension of the profile. A design variable is an independent parameter which is varied during the optimization. The allowable range for the design variables needs to be specified. In here the upper and lower bounds of the design variables can be defined. An initial value must be specified which serves as a starting point for the optimizer. The amount of design variables can be reduced by relating the design parameters to the design variable. It is important to notice that the diameter ranges must be defined for each different element in the structure in order to obtain independent sizing of the elements. Only one design parameter must be defined for each profile due to the fact the all the other design parameters can be related to this design variable.

Table 32: Design variables

Profile	***Design parameter***	***Symbol***	***Relation***
Circular hollow section (CHS)	Inner radius	r_2	$Inner\ radius\ r_2 = \frac{311}{331} r_1 \approx 0.9\ \cdot r1$
	Outer radius	r_1	$Design\ variable\ r_1$
	Wall thickness	t	$Wall\ thickness\ t = r_1 - r_2$
Rectangular hollow section (RHS)	Outer height	h	$Design\ variable\ h$
	Outer width	b	$External\ width, b = \frac{149}{308} h\ \approx 0.5 \cdot h$
	Wall thickness	t	$Wall\ thickness, t \geq \frac{5}{308} h$

To relate all these design variables together, generic relationships needs to be defined between the design variables. This relates the design variables to the properties of the model. In these generic relationships, the property of the design variable is linked to a corresponding property of the model. It determines for example the outer radius of certain tube elements and the range in which it may vary. This step needs to be performed for each design variable and this can be reduced by generating a function or a relation for the other design parameters.

The next step is to define discrete design parameters. This will enforce discrete changes of the design parameters in specified steps instead of a continuous variation of the values. The incrementation values are defined for which the design variable may vary. This can be for example in steps of: 0.1, 0.5

or 1 mm etc. This will enable more realistic dimensions for the design parameters. The discrete design parameters must be included into the design variables.
The final step in the size optimization is to define the design equations. These design equation relate the design parameters to the design variables. The substantiation of these relations can be found in A-10. Cross section classifications. The reason for applying these design equation is to reduce the amount of pre-processing work for the optimization process but most of all, to incorporate additional constraints on the design variables which are not automatically included in the optimization.

For example: the optimizer does not include buckling constraints and therefore, for a constant cross-section area, the optimizer will increase the outer dimensions of the profile in order to maximize the torsion resistance and bending stiffness. This behaviour can be explained easily. A cross-section can have many solutions for a constant cross-section area. The topology optimizer is striving to enable a design which is only subjected to tension and compression forces, but this solution will still be exposed to bending and torsion. The tension and compression forces only depend on the cross-section area of the members, but the torsion and bending resistance depends on the second moment of inertia, which is determined by the distance of the material from the neutral axis. Therefore, the optimizer increases the dimensions of the cross-section at the expense of the wall thickness. It maximizes the dimension of the profile and afterwards it determines the required wall thickness to obtain the required cross-section area. The disadvantage of this phenomena is that it makes the structure vulnerable for local buckling. To avoid this, design equations are defined which relate the dimensions of the profiles to the wall thickness. This defined the minimum required wall thickness to prevent local buckling. Now the functional relationship can be defined for the different design parameters. The relations are given in Table 32 in column 4. When the design relations are defined, then the equation must be related to the existing design variables.

In order to obtain a design realizable, it is chosen to restrict the optimizer to use member sizes which are available and commonly used for structural applications. Which means that the optimizer is not able to vary the member sizes within a range, but it is restricted to fixed values of members sizes which are obtained from the supplier manual. When this is incorporated into the optimizer, the results from the size optimization can directly be used and does not be converted to the available member sizes. The following restrictions according to Table 33 are incorporated into the optimizer.

Table 33: Discrete values for the optimizer

Member type	**Design variable**	**Defined by**	**Range [mm]**
CHS	R1 = outer radius	Varied by the optimizer	24.15, 25.5, 27, 28.5, 30.15, 31.75, 35, 38.05, 41.25, 44.45, 50.8, 54, 57.15, 60.5, 63.5
	R2 = Inner radius	$R2 = 0.9 \cdot r1$	Determined by R2
RHS	H = profile height	Varied by optimizer	80, 90, 100, 120, 140, 160, 180, 200
	B = profile with	$B = 0.5 \cdot H$	Determined by H
	T = wall thickness	Discrete values	4, 5, 6.3, 8, 10, 12

The optimizer is allowed to vary the design variables according to this table. This will increase the objective function but it results in a design which consists of members which are commonly used.

The optimizer requires responses from the model. A response is a numerical measure of design variable due to the input on the model. These responses can be used for the objective function or constraints. The responses which are defined for size optimization are: mass, displacements and members stresses. In this situation, it is chosen to constrain the member stresses for each load case. These allowable member stresses are defined by the DNV for each load case. Static stress responses are required from each member in order to define the stress constraints. Displacement constraints are defined for the nodes specified in section 8.2. Responses from the vertical deflection of these nodes are required in order to apply constrains to these nodes. The final step in the set-up of the optimization is to define the objective. For the optimization of the gangway structure it is preferred to reduce the mass of the structure for a certain required stiffness. Therefore the objective of the optimization is set to minimize the mass of the structure. An overview of the responses is given in Table 34.

Table 34: Overview responses

Responses	Obtained from	Applied for
Mass	Total structure	Objective function
Static stress	Per defined member	Stress constraint
Displacements	Per node	Displacement constraint

8.4 Size optimization results

The results from the size optimization are shown in this section. The set-up of the size optimization was identical for the three different line models. The objective of this size optimization is to minimize the mass of the structure while satisfying all the design constraints. This means that the structure is constrained by the maximum member stress and vertical deflection of the structure.

8.4.1. Topology 1: Size optimization

First the result from the first size optimization is given. After 10 iterations, the limit on the amount of iterations was reached and the solution was converged and all the constraints were satisfied. This means that no stress or displacement constraints were violated. The total mass of the structure is estimated at 2.15 tonnes which is including the applied point masses. Without the point masses the mass of the structure is equal to 910 kg.

Table 35: Results size optimization for design 1

Size optimization results: Result 1.		
		Constraint satisfied
Structure mass	**0.91 ton**	-
Point masses	1.24 ton	-
Maximum stress LC1	237,0 MPa	237,0 ≤ 237 Yes, within the tolerance range
Maximum stress LC2b/LC3	272,5 MPa	272,5 < 323 Yes, below the limiting value
Maximum deflection LC1	61,9 mm	61,9 < 65, Yes, constraint satisfied
Maximum deflection LC2b/LC3	-15,1 mm	-15,0 < -130, Yes, constraint satisfied

From the obtained results we can conclude that the structure is stress driven and not displacement driven. This means that the structure is restricted by the maximum stress in the members and not by the maximum deflection of the structure. The von mises stress plot from LC1Ca is shown in Figure 64. Notice that the members in red are highly stressed but that only one single mesh element is stressed by 237 MPa and that all the other mesh element are below this stress.

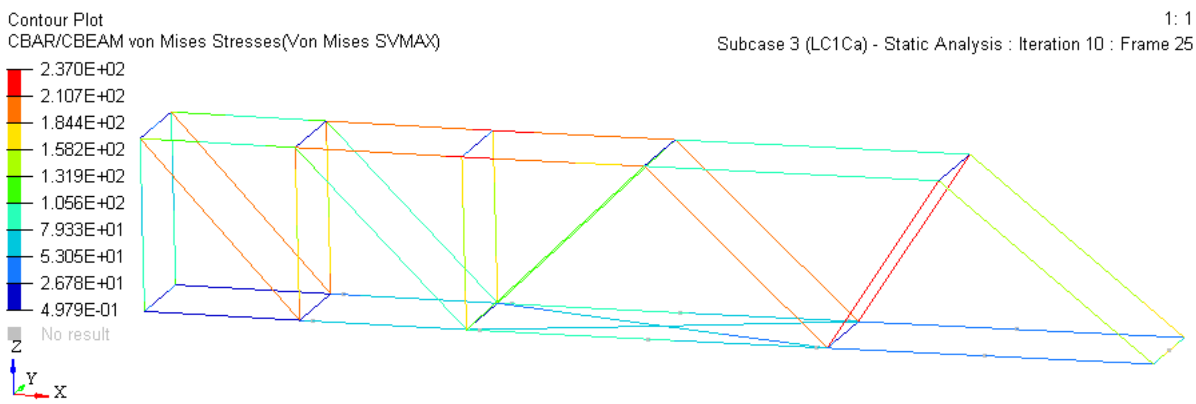


Figure 64: Von Mises stresses result 1

The maximum vertical displacement occurs during LC1Ca, Which is the normal operational condition in which a live load is acting in the middle of the total length of the gangway. The maximum displacement is 61.9 mm. This is a positive value due to the fact that the reaction force is working in the positive Z-direction in this load case. This means that in the normal conditions the origin of the section has is vertical displacement of -61.9 mm due to the reaction forces of the fixed part.

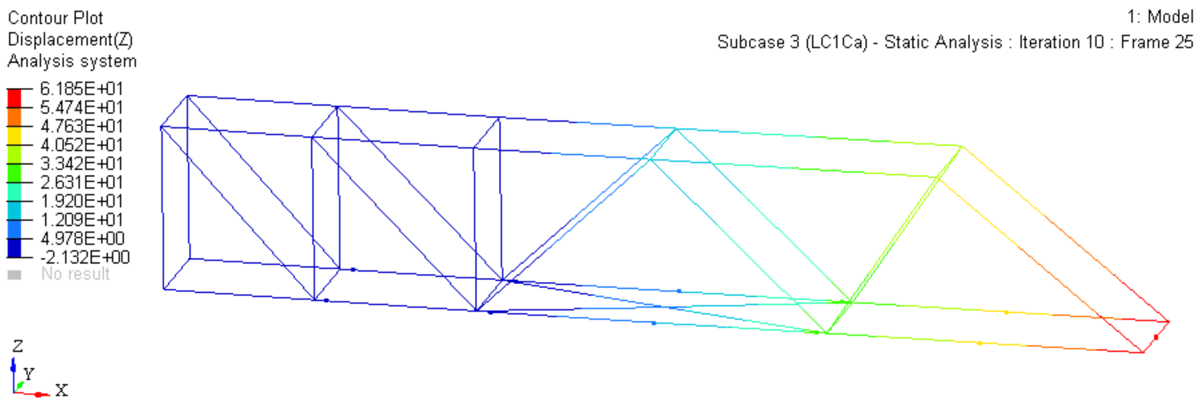


Figure 65: Displacement plot result 1

8.4.2. Topology 2: Size optimization

The size optimization was performed for result 2 which was also obtained during the topology optimization process. This result was obtained by using a different set of boundary conditions. This result is subjected to the identical boundary conditions and loads which are specified in section 8.2.

After 5 iterations, the optimization process was terminated by the optimizer and it resulted in a feasible design. The optimizer had reached its convergence criteria. The structural mass at the final iteration was equal to 1.65 ton, which is considerably higher compared to the previous result. An overview of this result is shown in Table 36.

Table 36: Results size optimization design 2.

Size optimization results: Result 2.		
		Constraint satisfied
Structure mass	**1.65 ton**	-
Point masses	1.24 ton	-
Maximum stress LC1	228,5 MPa	228,5 < 237 Yes, within the tolerance range
Maximum stress LC2b/LC3	272,1 MPa	272,1 < 323 Yes, within the tolerance range
Maximum deflection LC1	57,0 mm	57,0 < 65, Yes constraint satisfied
Maximum deflection LC2b/LC3	-21.5 mm	-21,5 < -130, Yes constraint satisfied

Now the Von Mises stress can be shown for LC3 intermediate situation which has the highest stresses. This plot can be used to visualize which parts and members of the structure are subjected to high stresses. The stress plot for this design is shown in Figure 66.

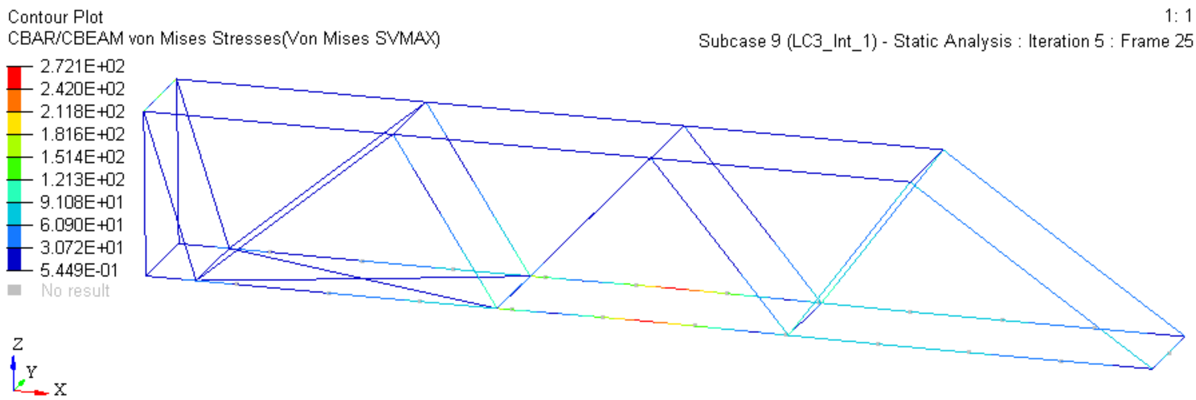


Figure 66: The von Mises stresses of result 2

From this graph it is clear that the horizontal members in the middle of the top sections are subjected to high stresses. The reason for these high stresses is because the members are loaded by bending which is an inefficient way of transferring loads. A horizontal stiffener could be applied in order to reduce these loads and to improve this design.

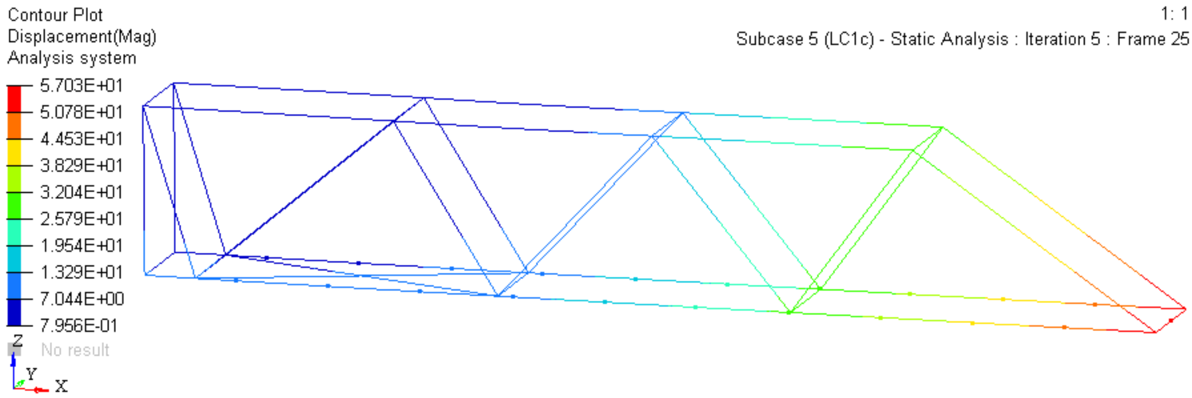


Figure 67: Displacements result 2

8.4.3. Topology 3: Size optimization

Finally, the size optimization was performed for result 3. This is an interesting one, because this consists of the beam orientation of the current design. If this optimization is successfully, then this would prove that the topology of this design is well defined. This line model is subjected to the same boundary conditions as mentioned in section 8.2.

After 5 iterations, the optimization process was terminated by the optimizer and it results in a feasible design. The structural mass at the final iteration was equal to 1.69 ton, which is almost the same as for result 2. But it is still considerably higher compared to the first result. An overview of the result is shown in Table 37.

Table 37: Result sizing optimization result 3

Size optimization results: Result 3.		
		Constraint satisfied
Structure mass	**1.69 ton**	-
Point masses	1.24 ton	-
Maximum stress LC1	228,9 MPa	228,9 < 237 Yes, within the tolerance range
Maximum stress LC2b/LC3	279,9 MPa	279,9 < 323 Yes, within the tolerance range
Maximum deflection LC1	51,3 mm	57,0 < 65, Yes, constraint satisfied
Maximum deflection LC2b/LC3	-31.7 mm	-31,7 < -130, Yes, constraint satisfied

Also for this design, it is possible to visualize the Von Mises stresses in order to estimate the location of the high stresses.

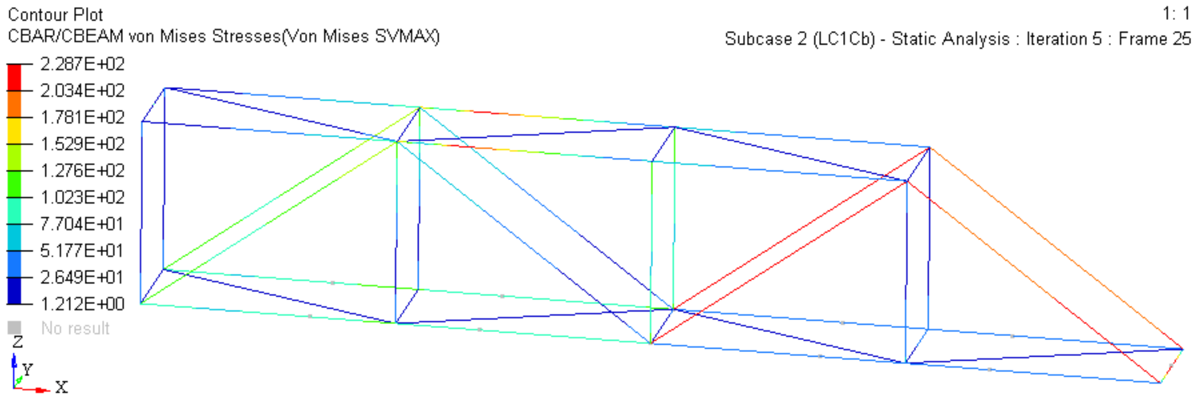


Figure 68: Von Mises stresses of result 3

From Figure 68, we can conclude that the high stresses in the members is located in the same region as with result 2. In this situation, the high stressed members are also subjected to bending stresses which are the causes the high stresses. Also 4 members are highly stressed, which means that the optimizer has reduced the member size till it approaches the stress limit. This is done in order to minimize the total weight of the structure. The location of the vertical stiffeners is wrong and therefore bending stresses are generated in the structure.

The maximum displacements are shown in Figure 69.

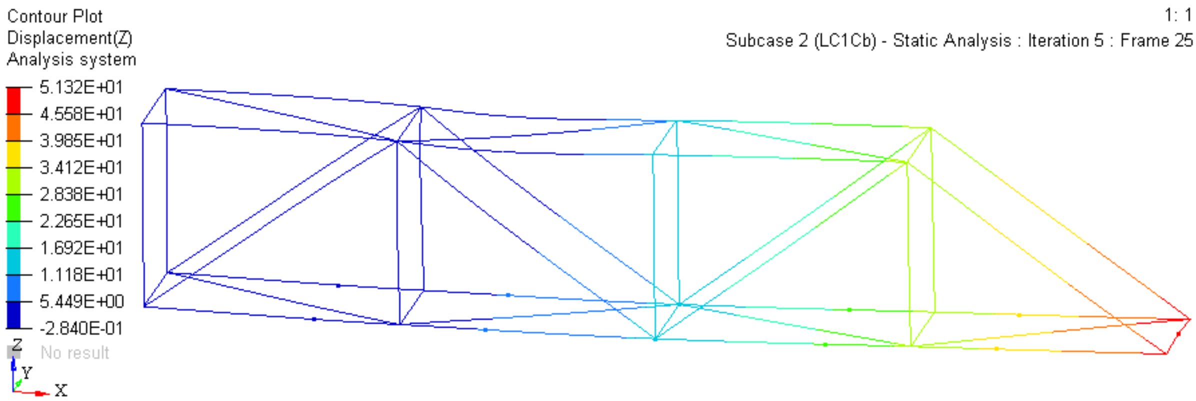


Figure 69: Displacements of result 3

8.4.4. Size optimization: Fixed part

In this section the results for the size optimization of the fixed part are given. The approach and set-up of this size optimization is similar as described above and extensively described in A-18. Size optimization: Fixed part.

After 6 iterations, the optimization was converged and it resulted in a feasible design. This means that all the constraints are satisfied. The highest stresses were found for the normal operational condition, in which the stress limit is reached. This means that the structure is stress driven and not displacement driven. The maximum vertical deflection is 86.11 mm which is at the tip of the fixed section during the emergency disconnection case. The structural mass at the final iteration was equal to 7.45 Ton. This is the mass of the steel structure. An overview of the results is shown in Table 38.

Table 38: Result sizing optimization fixed part.

Size optimization results: Fixed part			
		LC	**Constraint satisfied**
Structure mass	**7.45 ton**	-	-
Point masses	3.255 ton	-	-
Maximum stress	236,3 MPa	LC1Ca	236,0 < 237, Yes, constraint satisfied
Maximum deflection LC1c	39,69 mm	L1Cc	39,69 < 240, Yes, constraint satisfied
Maximum deflection LC2b/ LC3	-86,11 mm	LC3a	-86,11 < -240, Yes, constraint satisfied

Also for this design, it is possible to visualize the Von Mises stresses in order to estimate the location of the high stresses. The von-Mises stresses are visualized in Figure 70.

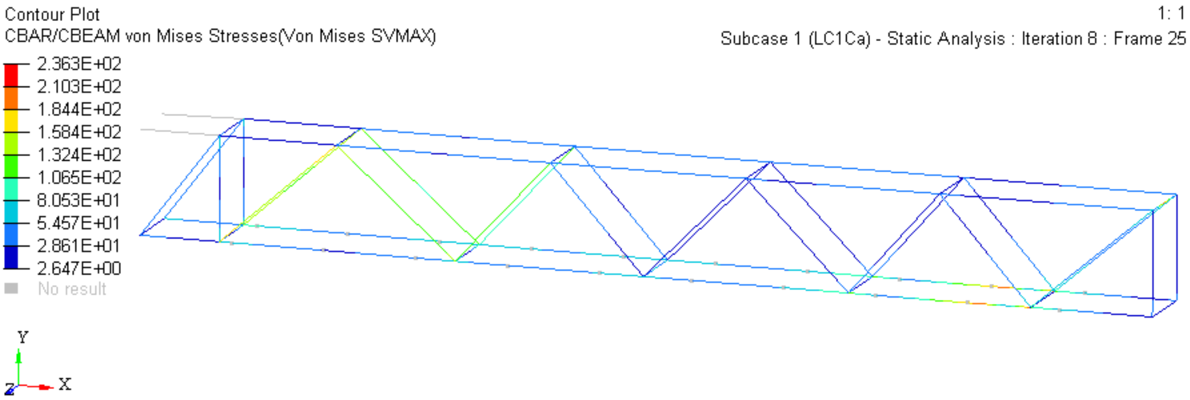


Figure 70: Von-Mises stresses in the fixed section

The high stresses are caused due to the reaction forces of the roller bearings.

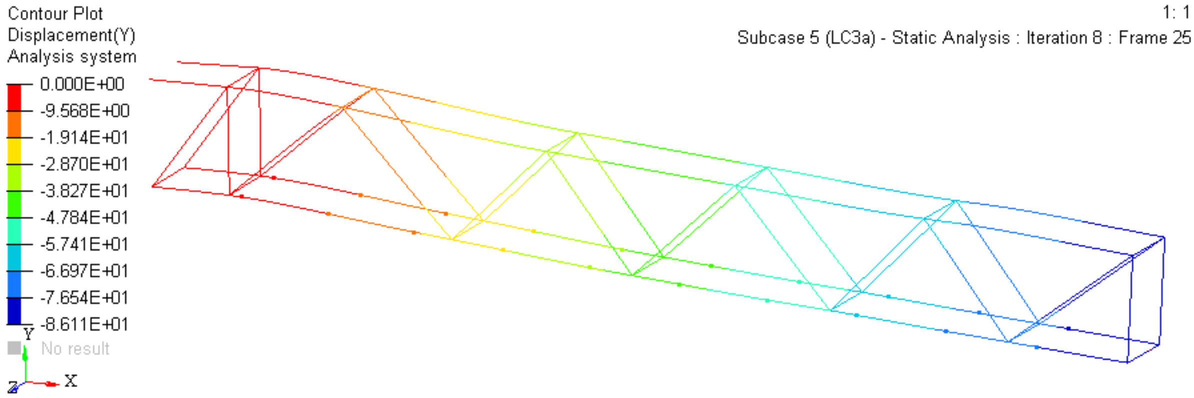


Figure 71: Vertical displacement fixed section

Now that the new mass of both sections is calculated it is possible to perform an iteration step. This means that the reaction forces are recalculated including the new masses of the structure. This will lead to a reduction of the reaction forces acting on fixed section of the gangway. Also the reaction forces from the support reactions will be reduced. The procedure will be exactly the same with as only difference the applied reaction forces. This will reduce the mass of the structure even further.

9. Structural stability

This chapter is concerned with the stability of the design. Buckling of a structure is a dangerous phenomenon which leads to catastrophic failure. A buckling check must be performed in order to check the structural stability of the design. Modifications to the current design will be performed if the structural stability of the structure must be increased.

9.1 Buckling of a structure

Buckling is a dangerous phenomenon. Buckling leads to catastrophic failure even if the stresses are below the yield limit. Buckling of a structure can be divided between three main types of buckling: Local buckling, global buckling and torsional buckling. Local buckling is the failure mode in which the structure fails due to deformation of the cross section, for example wrinkling of the profile. In this situation the member fails before the full strength of the beam is utilized because a flange or section of the beam has buckled first. Global buckling is a buckling mode where the member deforms with no deformation of its cross-sectional shape. Distortional buckling is the buckling mode which is related to shear flow. [63]

The local buckling behaviour of the structure was incorporated in the size optimization of the structure. Therefore this type of buckling is not investigated any further. Long and slender structures like a gangway structure can be susceptible to global buckling. A Linear buckling analysis is performed in order to verify the structural stability of the gangway structure. Buckling is not incorporated into the topology optimization process and therefore the structure should be checked against buckling. It is expected that the structure must be adapted and improved in order to reduce the global buckling behaviour. The outcome of a linear buckling analysis are the eigenvalues or critical multipliers. When all the applied loads are multiplied with these factors, then this load will cause a stability failure in a perfect system.[60] A perfect system means that no imperfections are taken into account.

The main use of the linear buckling analysis is to check if buckling occurs. When the obtained eigenvalues are lower than 1, this means that the structure is certainly unstable and will fail due to buckling. When the values are above 1 this does not mean that the structure is stable, but it gives an estimate how close the structure is to stability failure. Also the use of buckling analysis will visualize the regions which have stability issues. This information can be used to adapt the design in order to increase the structural stability. The disadvantages of linear buckling analysis is that it cannot take material or geometrical nonlinearity into account and it cannot take imperfections into account. In a real structure, imperfections and nonlinear behaviour keep the system from achieving this theoretical buckling strength, leading Eigenvalue analysis to over-predict the buckling load.[61]

9.2 Linear buckling analysis: Telescopic part

A linear buckling analysis is performed with Ansys Workbench. For each load case, the first 10 positive eigenvalues are calculated. Negative eigenvalues are disabled, due to the fact that these are often not possible in real problems. Negative eigenvalues means that all the load are reversed. This cannot be the case for the earth gravity which is always acting in the same direction. The same holds for the live load which is always located on the top of the gangway. Therefore it is chosen to disable negative eigenvalues in the analysis.

An eigenvalue buckling check is performed for the five different load cases. Two load cases originates from condition in which the gangway is in retracting condition. Also the additional masses and the wind loads are incorporated into this analysis. The linear buckling analyses is performed by using a simplified beam model. The boundary conditions and applied loads are identical to those described in section 8.2 Boundary conditions. 10 positive eigenvalues are obtained for each load case and the negative eigenvalues are checked for the situations where the accelerations are neglected. Only the first 5 eigenvalues are shown for convenience. An overview of the result is shown in Table 39.

Table 39: Critical load multipliers.

Mode	Load multipliers				
	LC1c: Normal operation	LC2b: Deployment /retrieval	LC3c: Emergency disconnection	LC3c: Intermediate 1	LC3c: Intermediate 2
1	0,88	9,32	7,08	6,81	7,75
2	1,25	9,50	7,28	6,88	11,32
3	1,31	10,20	7,71	7,49	12,03
4	1,33	10,49	7,85	7,60	12,87
5	1,37	11,24	8,62	8,46	13,60

From the results we can conclude that the structure is unstable for the normal operational condition. The first buckling mode is shown in Figure 72, which is the lowest buckling factor.

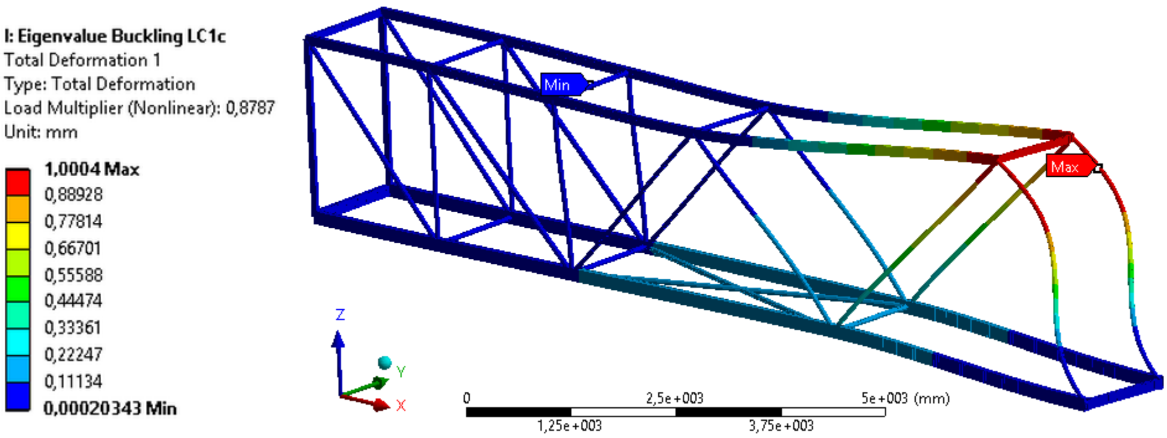


Figure 72: First buckling mode in operational condition

The value shown in red is lower than 1, which means that the structure will certainly buckle under this condition. The results from the buckling analysis show that not only a single member, but the complete structure will buckle under these conditions. The members will buckle sideways due to the fact that there are no stiffeners placed in this direction and therefore the stiffness in this direction is relatively low. The most efficient way to increase the load multipliers is to reduce the effective length of the column. Since the effective length factor K is already fixed, it is chosen to reduce the unsupported length of the column. This means that stiffeners will be applied to the current design in order to decrease the buckling length in order to increase the structural stability of the structure. Another option is to increase the moment of inertia of the cross sections of the columns. This option is less efficient but it will also increase the structural stability of the gangway structure. A combinations of these two options will be used to increase the buckling factor for the gangway structure. This disadvantage of this solution that it will introduce more mass to the gangway structure while the goal of this thesis is to optimize the gangway structure in terms of weight and stiffness. These extra stiffeners are required in order to keep the structure safe and operational.

Stepwise, stiffeners are added to the structure in order to increase the eigenvalues of the structure. For conservative approach SAS requires a minimum safety factor of 3 on buckling as evaluated by FEA. In literature, it is referred to apply a factor of safety of at least 3 for buckling loads.[62] The guidelines do not specify a value for the safety factor, instead they specify a safety factor on the yield load limit being exceeded. After a few iteration runs, the buckling factor of the structure was increased to 5,24 while the mass of the structure was increased from 1,042 to 1,309 ton. The starting value of the mass is a little bit higher compared to the solution of the size optimization. This difference is due to the fact that the CHS are chosen from the supplier manual in which the wall thickness is not always exact equal to 10% of the member diameter, as is specified in the size optimization. Therefore the wall thickness can be larger which results in a larger mass. The addition of extra members and increased member sizes results in an increase of 267 kg. Which is equal to an increase of about 25%. This is a large increase in weight of the structure but this is required in order to obtain a stable structure. The weight of this final design is still lower compared to the weight of the current design, which is equal to 2038 kg. The different load multiplier for the different load cases are shown in Table 40.

Table 40: Load multipliers for the telescopic section

	Load multipliers				
Mode	LC1c: Normal operation	LC2b: Deployment /retrieval	LC3c: Emergency disconnection	LC3c: Intermediate 1	LC3c: Intermediate 2
1	5,24	9,08	6,45	7,90	10,63
2	5,37	9,26	6,58	8,00	11,27
3	5,56	12,49	7,32	8,75	12,02
4	5,64	12,70	7,45	8,80	12,76
5	5,72	12,79	8,07	9,16	17,21

The buckling check shows that the buckling safety factor is over the required factor of 3 (as per SAS requirement) and is therefore considered as acceptable. The minimum load multiplier of 5,24 means, that the structure will buckle (under ideal conditions), when all the loads which are acting on the structure are multiplied with this factor. In reality the structure will buckle at a lower load due to imperfections. Additionally a non-linear buckling analysis could be performed in order to take the non-linearity's and imperfections into account. The additional bars which are added to this telescopic section are shown in Figure 73.

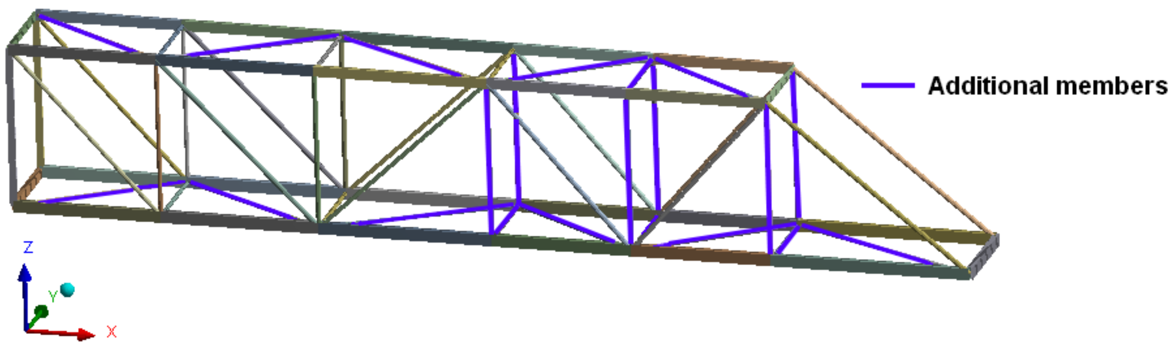


Figure 73: Additional members to increase the structural stability

9.3 Linear buckling analysis: Fixed part

The next step is to perform a buckling check for the fixed part of the gangway structure. The same procedure is performed for this section of the gangway structure: A line model is imported into Ansys Workbench and all the profile dimensions are assigned to the different beams in the line model. The dimensions of the profiles are determined by the size optimization process. All the different load cases are incorporated into the analysis. The applied boundary conditions are defined in appendix: A-18. Size optimization: Fixed part. The two sections of the gangway structure are analysed separately. The reaction forces from the telescopic part are used as input parameter for the analysis of the fixed part. The linking of the reaction forces and input loads is done by using a parameter set.

The first 10 positive eigenvalues are calculated. An eigenvalue buckling check is performed for the different load cases. One load case originates from condition in which the gangway is retracting. The linear buckling analyses is performed by using a simplified beam model. All the results from the buckling check are shown in Table 41.

Table 41: Critical load multipliers for the fixed section

	Load multipliers			
Mode	LC1c: Normal operation	LC2b: Deployment /retrieval	LC3: Emergency disconnection	LC3c: Intermediate 1
1	2,24	1,29	1,18	3,03
2	2,29	1,32	1,21	3,08
3	3,37	1,82	1,54	4,56
4	4,34	2,00	1,76	5,02
5	4,61	2,05	1,79	5,17

The linear buckling analysis shows that the structure is susceptible for buckling. All the load multipliers are larger than 1, but in some load cases the load multipliers are close to one which means that the structure is close to stability failure. These values are denoted in orange. The first buckling mode for the deployment retrieval load case can be seen in Figure 74.

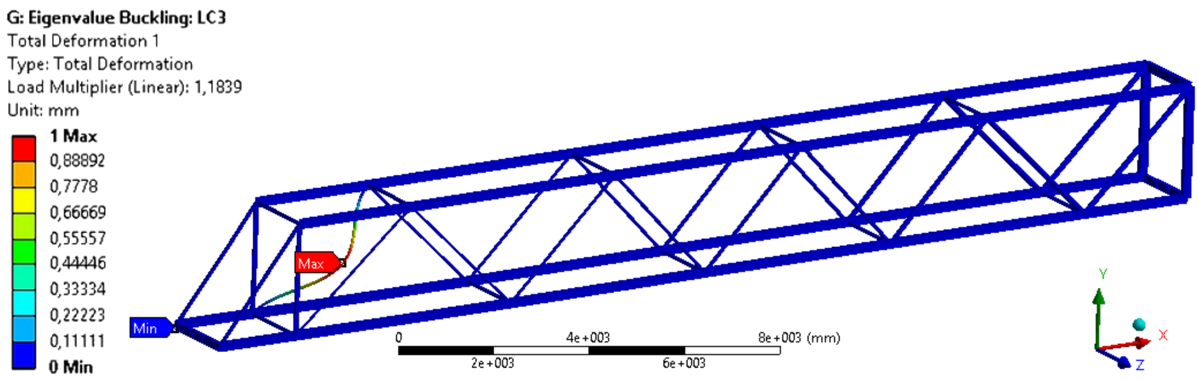


Figure 74: First buckling mode fixed section

The goal of the buckling analysis is to determine the regions which will have stability issues in order to increase the structural stability of the structure. The total mass of the current structure is equal to 4,6 ton. Additional members will be added to the structure in order to increase the structural stability of the structure. This will increase the structural stability of the structure but it also will increase the total mass of the structure. The new design with the additional members are shown in Figure 75. The mass of this reinforced design is equal to 4,94 ton. Which means an increase of 7,4%. This seems a lot but this is still less than 50% of the current design. Also the structural stability of the structure was increased significantly as can be seen in Table 42.

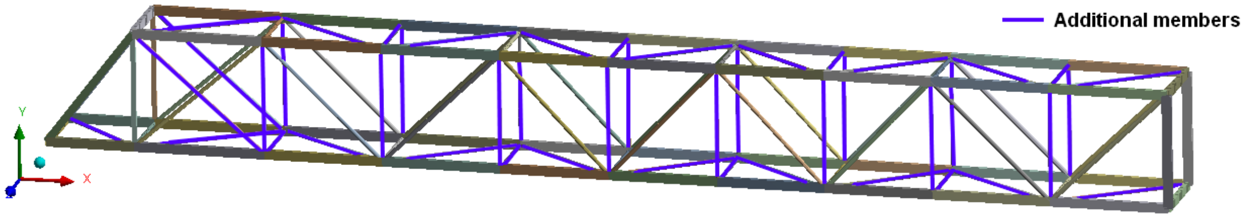


Figure 75: Additional members for the fixed design

Table 42: Load multipliers for the fixed section

	Load multipliers			
Mode	LC1c: Normal operation	LC2b: Deployment /retrieval	LC3c: Emergency disconnection	LC3c: Intermediate 1
1	11,44	7,84	6,05	6,43
2	12,08	8,63	6,65	6,93
3	12,80	8,70	6,91	7,10
4	13,93	9,36	7,76	7,49
5	13,52	10,05	8,16	7,60

10.Evaluation new design

This chapter is concerned with the evaluation of the final design. The gangway design will be checked by use of FEA in order to check if it satisfies all the criteria. A comparison is made between the current design and the new design which is obtained by using topology and size optimization.

10.1 Evaluation steps.

The first step in the evaluation of the design is the set-up of the FEA analysis. The 3D line model is converted to a CAD model by using Solidworks and is subsequently imported into Ansys Workbench. The next step is to apply all the boundary conditions and loads to the model. These boundary conditions and loads for the different load cases are identical to the boundary conditions and loads specified in section 8.2 Boundary conditions. The deflection and maximum stresses in the structure are analysed in order to check if they satisfy the acceptance criteria.

10.2 The finite element analysis.

The new gangway design is a truss structure which consists out of two sections made out of S355 structural steel. These sections consists of circular and rectangular hollow tubes which are welded together. The structure is supported by two hinges and two hydraulic actuators located at the base of the structure. The tip of the gangway is supported by the landing tool. The two sections of the gangway are connected by means of no separation contact and joints. For convenience, only the results of the two most severe load cases are shown in this section. All the other load cases are also evaluated but not reported because these load cases are not leading. All the load cases are explained and shown in appendix: A-16. Finite element analysis.

10.2.1 Normal operational condition.

The first situation which is visualized in Figure 76 is for the normal operational condition. The structure is constrained by means of remote displacement supports at the hinges and the tip of the gangway. For the hinges every DOF except the axial rotation is constrained. For the cylinders every DOF is constrained except for the rotation around Y and Z-axis.

The maximum vertical deflection for the operational condition is shown in Figure 76 . The maximum deflection in the Y-direction is equal to -34,44 mm.

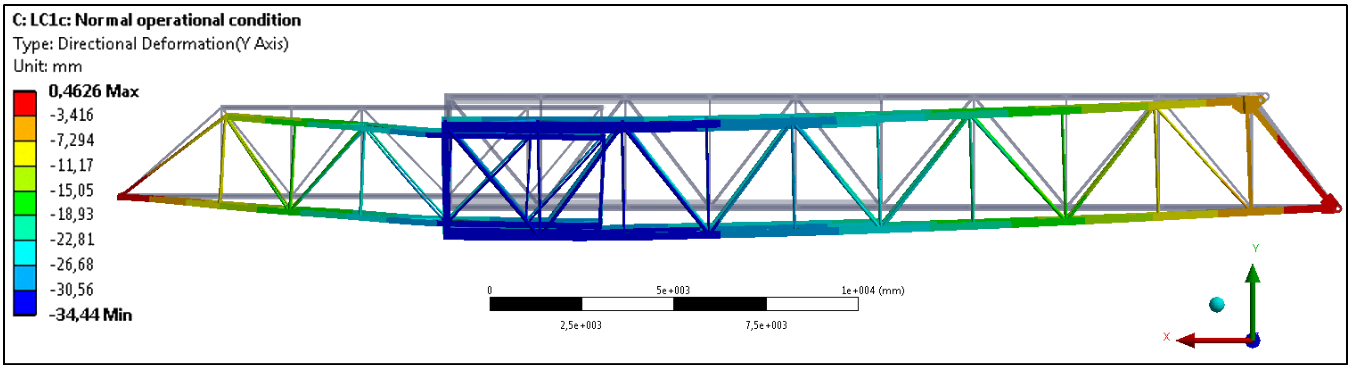


Figure 76: Vertical deflections for the operational condition, Scale = 24

Another design criteria are the maximum stresses in the design. The maximum von mises stresses are shown in Figure 77.

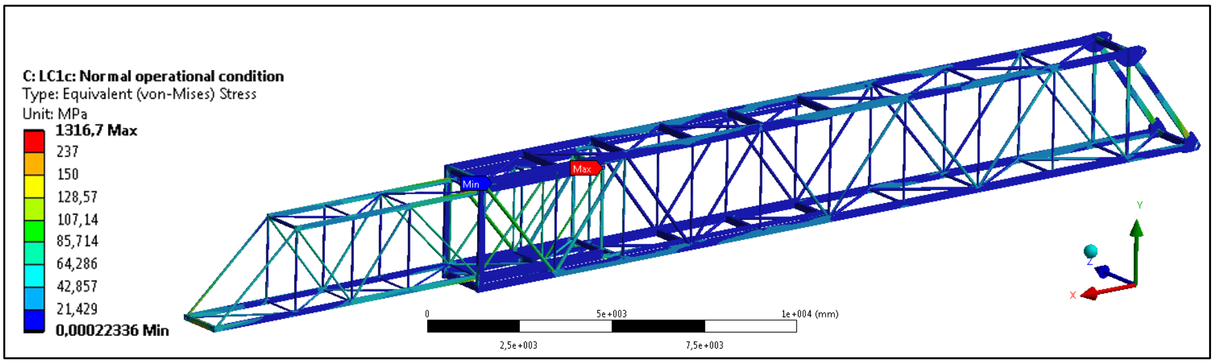


Figure 77: The maximum stresses for the operational condition

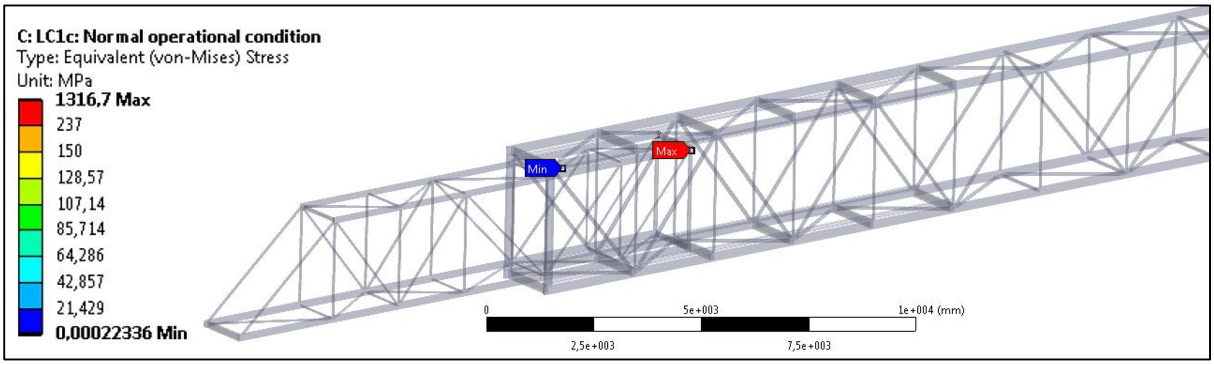


Figure 78: The stresses over 237 MPa

Now the high stress locations are investigated in detail. A detail view is shown in Figure 79.

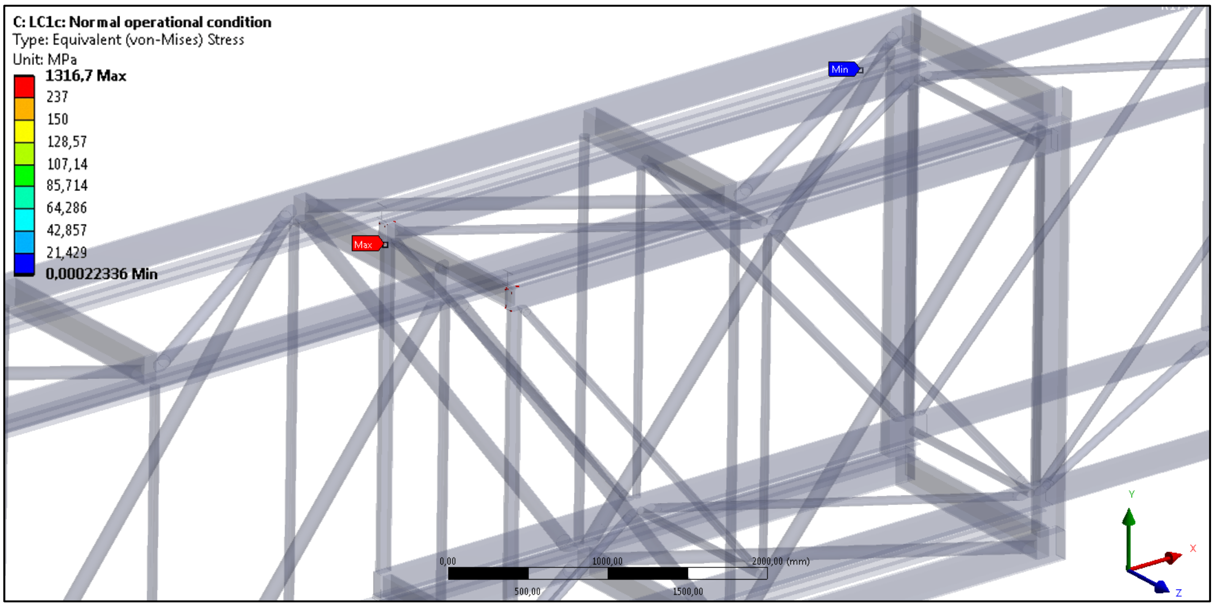


Figure 79: Detail view of the highest stress locations

The maximum stress level is equal to 201 MPa. This means that the general stress level is lower than 355 MPa / 1.5 = 237 MPa and is therefore considered acceptable. The plots with detail views of the peak stress areas shows the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and only single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria.

10.2.1 Emergency disconnection condition

The next situation which is visualized in the emergency disconnection case. This load case is considered as the most severe load case. Here the structure is constrained by means of remote displacement supports at the hinges and at the cylinder brackets. For the hinges every DOF except the axial rotation is constrained. For the cylinders every DOF is constrained except for the rotation around the cylinder hinge and the rotation around the cylinder rod. The extensive description of this load case is shown in appendix: A-16. Finite element analysis. This section only presents the results of the FEA.

The maximum vertical deflection for the emergency disconnection case is shown in Figure 80. The maximum deflection in the Y-direction is equal to -101,5 mm.

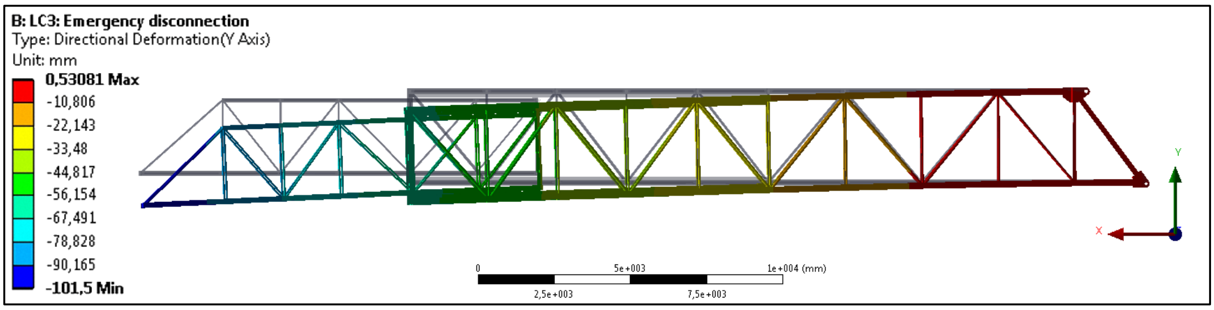


Figure 80: Vertical deflections for the emergency disconnection condition, Scale = 10

The overall von-Mises stresses are shown in Figure 81.

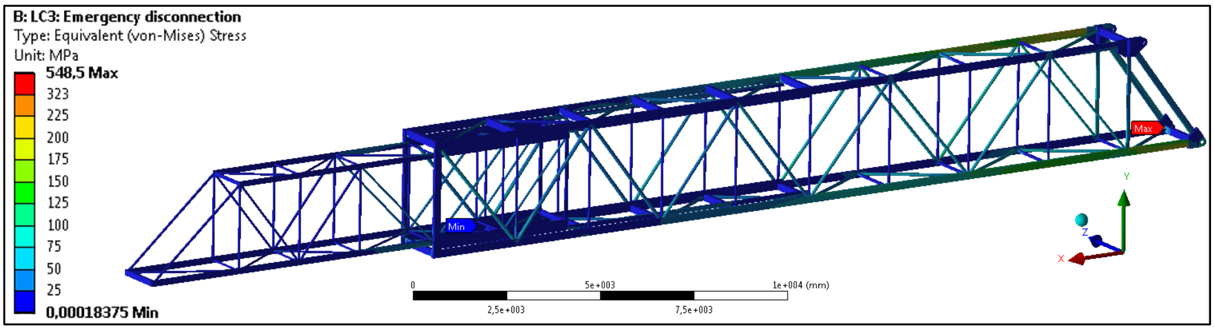


Figure 81: The maximum stresses for the emergency disconnection condition

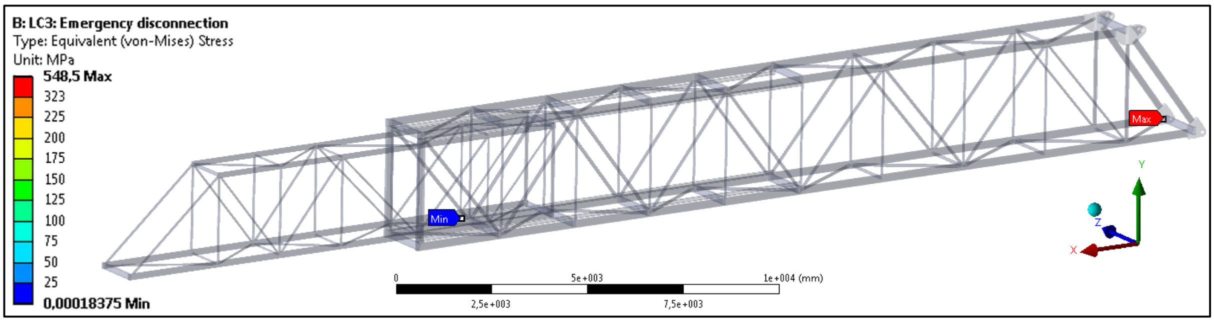


Figure 82: Stresses over 323 MPa

Now the high stress locations are investigated in detail. A detail view is shown in Figure 83.

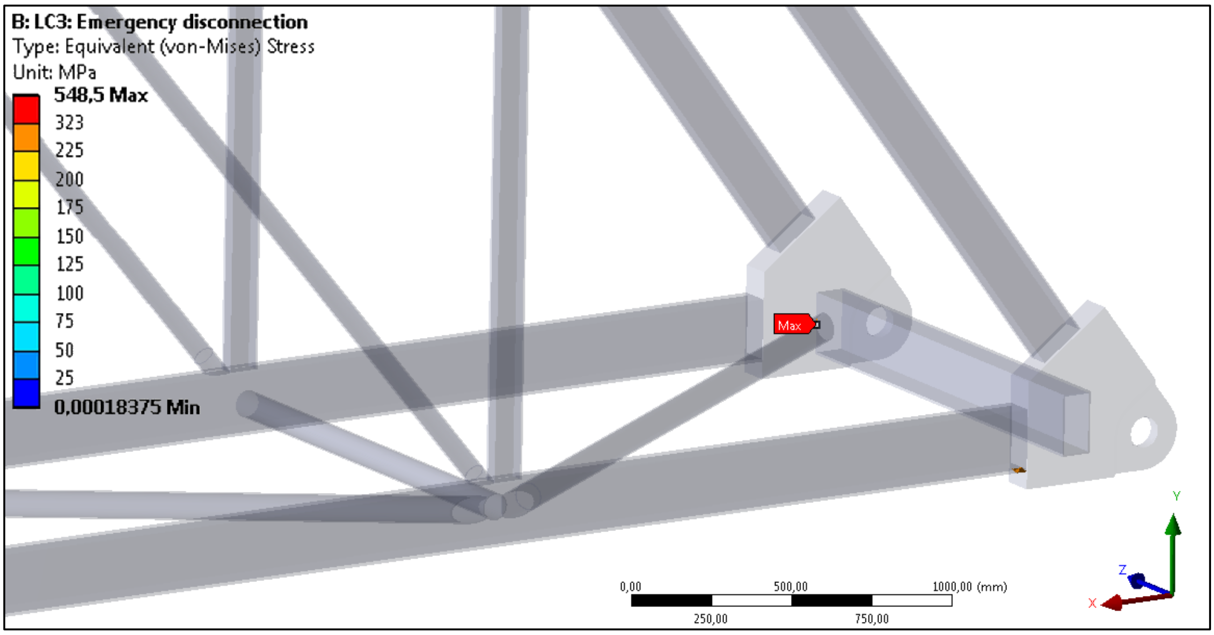


Figure 83: Detail view of the highest stress locations.

The maximum stress in the model is equal to 237 MPa. This means that the general stress level is lower than 355 MPa / 1.1 = 323 MPa and is therefore considered acceptable. Also in this plot there are values reported which exceeds the allowable stress level. However, these stress peaks occur at edges and sharp corner and are only a single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria.

An overview of the results of the finite element analysis is shown in Table 43.

Table 43: Results finite element analysis

Load case	Maximum deflection [mm]	Allowable deflection [mm]	Maximum stress [MPa]	Allowable stress [MPa]
LC1c: Operational condition	-34,44	-165	201	237
LC2b: Deployment/retrieval	-77,98	-330	159	237
LC3: Emergency disconnection	-101,5	-330	237	323
LC5: Load test	-27,63	-165	195	[-]

10.3 Comparison of the results.

Now that the new design has been evaluated, a comparison can be made between the current and the optimized design. Both gangways are designed according to the DNV guidelines and according to the specifications of SAS. First the telescopic sections are shown of the current and optimized design.

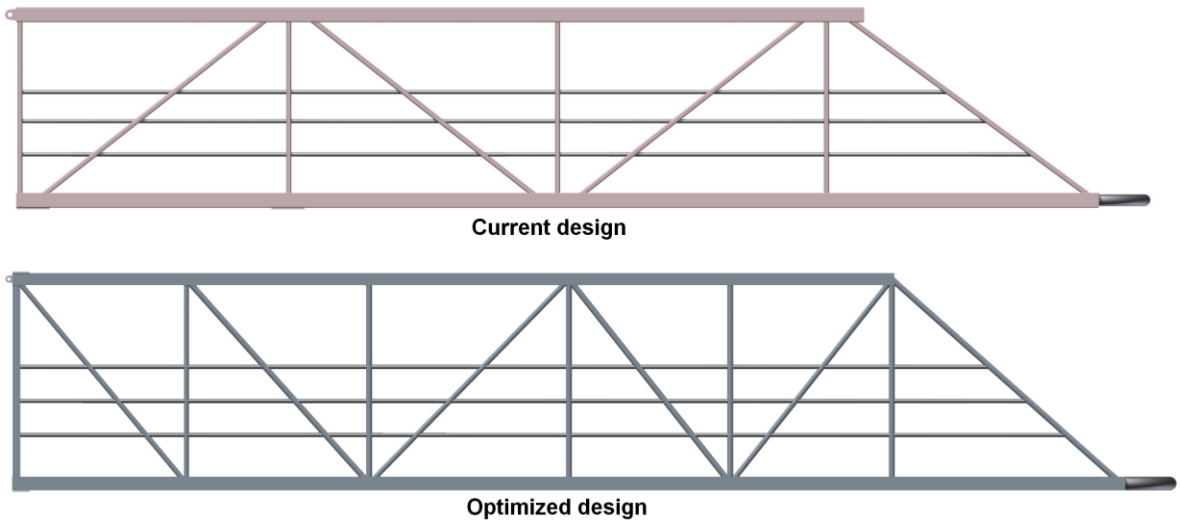


Figure 84: The telescopic sections of both designs

From Figure 84 , it can be seen that the main differences between the two sections is the amount of members in the structure. The current design consist of 36 members while in the new model the amount of members was increased to 52. Also the size and the location of the members is different for the two designs. The structural mass of the telescopic section was reduced by 34,7% from a mass of 2038 kg to a mass of 1330 kg. It can also be noted that the angles between the members are smaller in the optimized design. Now a comparison can be made between the two fixed sections.

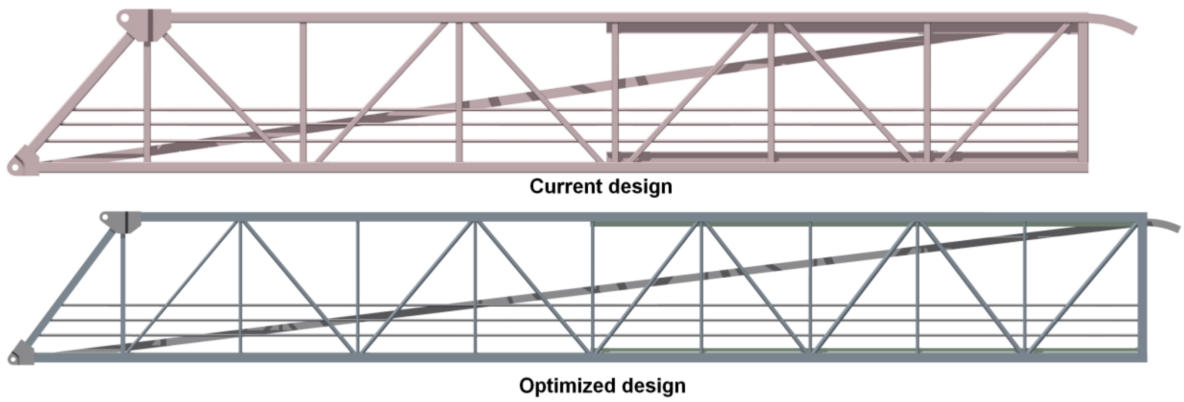


Figure 85: The fixed sections of both designs

For the fixed section of the gangway the same holds. The initial design consists of 59 members which was increased to 84 members in the optimized design. The average member size was reduced and this resulted in a weight reduction of 36,8%. The mass of this section was reduced from a mass of 11042 kg to a mass of 6981 kg. It can also be seen that the new design is increased in length. This was required in order to satisfy the requirement of SAS to reach a maximum length of 33 meters with an overlap of 4 meter of both sections. For the current design this was not possible and this design could only reach a total length of 32 meters.

An assembly can be made from the two sections to obtain the complete gangway structure. Now the current and the optimized design are showed side by side in order to determine the differences in the structure. Both designs are shown in figure 86.

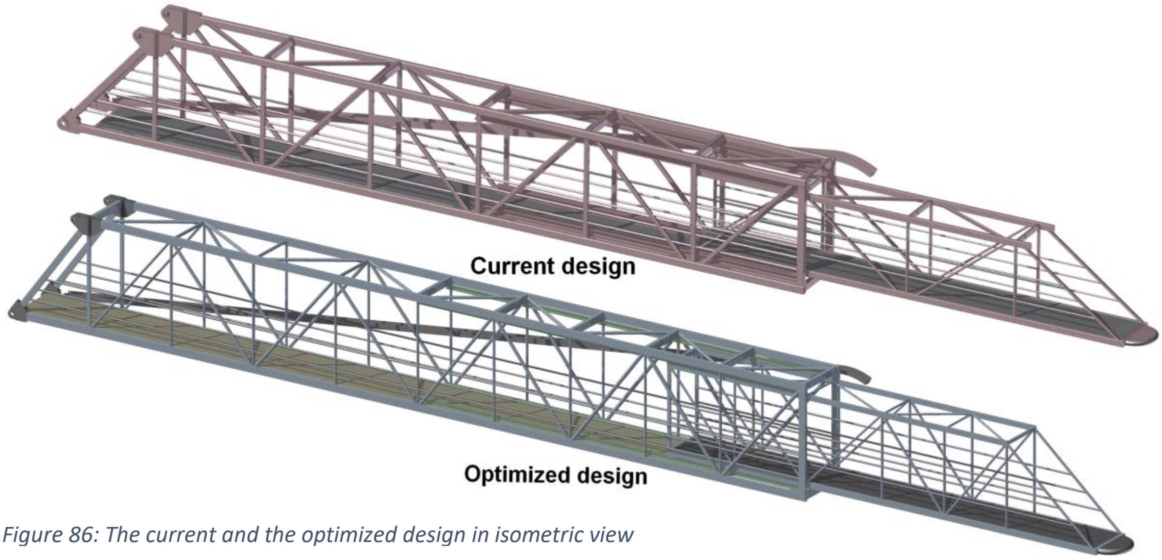


Figure 86: The current and the optimized design in isometric view

The first thing that can be noticed is the difference between the amount of members which is used. The base structure of the current design consists of 95 members while the new design consists of 134 members. These additional members will increase the stiffness of the structure and reduce the buckling length of the bars. The disadvantage of these additional members is that they are related to increased manufacturing costs. Also the length of the optimized design is equal to 33011 mm while the length of the current design is equal to 31900 mm. The different properties of the two designs are shown in Table 44.

Table 44: Comparison gangway designs

Gangway design	Current design	Optimized design	Difference
Structural mass	13,08 Ton	8,31 Ton	4,71 Ton
Total mass	21,6 Ton	14,40 Ton	7,2 Ton
Maximum stress	184 MPa	237 MPa	53 MPa
Maximum deflection	73,65 mm	91,48 mm	17,83 mm
Number of members	95	134	39
Maximum internal height	2065 mm	2275 mm	-
Maximum internal width	1250 mm	1220 mm	-
Total extracted length	31900 mm	33011 mm	-

The structural mass is the mass of the complete gangway structure which is obtained from the topology and shape optimization. This is the mass of all the supporting members in the structure. The total mass of the structure is the mass of the complete gangway structure with the flooring, railing, I-profiles and the landing platform attached to it. The table above shows some interesting results. The structural mass of traditional design has been reduced by 36,4% which is quite significant. This weight reduction can be justified by the fact that the wall thickness of the tubes is reduced compared to the wall thicknesses which are used in the traditional design. The same holds for the total mass of the structure. The increase in mass for the traditional design is quite large due to the fact that four thick I-beams are added to the structure. These four profiles add a significant amount of mass to the structure. The same holds for the railings which have a wall thickness of 5mm instead of 3 mm in the optimized design. These small changes can lead to a large weight reduction in the final design. Another important aspect is the free internal height of the structure. From the traditional design this value does not comply with the DNV requirement which require an internal height of at least 2100 mm.

11.Conclusion and recommendations.

The entire topology optimisation design approach of a motion compensated gangway has been discussed in this thesis. The goal of the optimization process was to minimise the weight of the structure while maintaining its stiffness and strength. Post-processing steps are performed in order to ensure the stability of the structure and to make it feasible.

11.1 The conclusion

The goal of this master thesis was to investigate if topology optimization can be used in the design process at IHC. The aim is to determine to what extent this method can be implemented practically or efficiently in order to reduce the leading time and the results. As example this method is applied to a device which is called a motion compensated gangway. This was formulated by the following research question:

Find the optimum weight reduction for the gangway structure in the design domain, for a given set of loads and boundary conditions while fulfilling the service constraints.

In this concluding chapter, the research questions are answered as a result of this thesis research. The main research revolves around the design approach. It starts from the design domain through the design realization process to the detailed design. The aim of this thesis is to provide a methodology to use topology optimization during the design process. This leads to the following sub-questions in this research:

- *Which software package is the most suitable for optimizing the gangway structure?*

From the multi-criteria analysis, it was shown that the optimization software package Optistruct from Altair Hyperworks, is the most suitable software package for performing the optimization task. Despite the fact that the user interface is more complicated compared to the other software packages, it allows the user to control every aspect of the optimization. This enables the user to prevent several problems which are encountered by the other software packages and it results in more realistic and useful solutions. The pre-processing steps in Optistruct are more complicated and time consuming compared to the other software packages. The longer pre-processing time is offset by the faster solving time. Another additional option is to incorporate multiple constraints and responses. Which gives insight and guidance to the optimization process. These options often lack at the other software packages. The software support of Altair is very active and prepared to help solving your problem, there is an active forum and lots of tutorials are available. The other two software packages also provides tutorials in order to get familiar with the software, unfortunately they don't have any active support which helps the user when the software is misunderstood or malfunctioning. The other two software packages are developed recently, which means that they are relatively new and can contain bugs or errors. Optistruct is the most mature software package. The overall opinion is that the software is suitable for the optimization and analysis and is a good supporting tool for the design engineer.

- *Can the weight of the current design be reduced by using topology and size optimization?*

Yes, from the finite element analysis it can be concluded that the current design is very stiff and that the maximum vertical displacement of the structure is very low compared to the maximum allowable displacement which are stated by the guidelines. This means that the weight of the structure can be reduced at the expense of the maximum deflection or allowable stresses in the structure. Topology optimization is used to determine the load paths in the design domain and a size optimization is used to define the dimensions for all the members. An important aspect is to check the structural stability of the structure due to the fact that this is not incorporated in the topology and size optimization process. Therefore a buckling check is performed in order to check and ensure the structural stability. The two parts are optimized separately and related to each other. Regarding the weight reduction objective, the topology optimised design yielded a mass reduction of 36,4% compared to the current design. The mass was reduced from 13,08 ton to 8,31 ton, while still satisfying all the constraints. This means that this methodology can be used for the structural optimization of a gangway structure. The mass of the gangway structure was reduced significantly. The combination of topology and size optimization resulted in the weight reduction for the gangway design.

- Is this methodology practically applicable in the industry?
The use of topology optimization can be implemented in the design process. It is not the ultimate solution for designing a structure, but this method can be used as a guide in the early phases of the design process. The use of topology optimization will not result in a one-step solution. The results from the topology optimization process are often difficult to interpret and quite often the result is not intuitive. The results from topology optimization will most often tend to be quite organic looking and therefore not suitable for manufacturing. Therefore the results are not applicable in reality and should be interpreted into something which can be manufactured. This requires a lot of post-processing steps. The results from the optimization process will only serve as an indication for the final design. The post-processing steps or the design realization process is used to interpret and convert the results from the topology optimization in order to obtain a feasible design. Many software packages are necessary to get from the design domain to a feasible design. During this project: HyperMesh, Ansys Workbench, SolidWorks and Spaceclaim were used to achieve the presented results. A common bottleneck is the output format that results from the topology optimization step. This file is build up as a surface consisting of many small triangles and not as a geometry, it is a tedious process to adjust the topology optimisation results for manufacturing. When topology optimization is applied, it is crucial to define a suitable design domain with properly defined boundary conditions in order to obtain a usable solution. The interpretation of the optimization results is a difficult task and requires knowledge of structural engineering and manufacturability. Also knowledge and experience about the tools and the mathematical methods is required in order to understand the behaviour ad meaning of the different functions in the software. The overall conclusion is that topology optimization is a good supporting tool for the design engineer.

-Find the optimum weight reduction for the gangway structure in the design domain, for a given set of loads and boundary conditions while fulfilling the service constraints.
Optimum is a difficult definition. In this situation, the optimality depends on minimizing the weight, while the service constraints could be given by its maximum displacements and/or allowable stresses. The weight of the gangway structure must be reduced, taking into account the operational requirements and certification guidelines. The gangway must remain operational and must be designed according to these guidelines. From the results of the optimization process and the analysis it can be concluded that a weight reduction is achieved by using this methodology. The weight of the new design has been reduced significantly while all the remaining requirements are met. The new design has a larger displacement of the current design and also the average stresses a larger compared to the base-case design. Still these values are within the limits which are specified by the guidelines. The total mass of the structure was reduced by 33,6 % from 21,6 tonnes to 14,4 tonnes. While the maximum vertical deflection increases from 73,56mm to 91,48 mm. The average stress increases from 184 MPa to 237 MPa. However these values are still within the specified limits and therefore acceptable.

11.2 Recommendations

This research is only a brief set-up of a design approach for optimizing and manufacturing parts with topology and size optimization. As time and effort is limited there are many aspects of this work that can be improved upon and extended on many fronts and some of the more interesting paths to take are:

- The structural stability of this new design is only checked by using a linear buckling analysis. This method is linear and cannot take into account second order bending and similar effects. Therefore a linear buckling analysis overestimates the maximum capacity of the columns. For a more accurate calculation of the structural stability of the gangway, a non-linear buckling analysis is recommended.

- An Eigen frequency analysis should be performed in order to check the range of discrete frequencies at which the system is prone to vibrate. This analysis shall be performed in order to ascertain that a periodic excitation does not cause a resonance that may lead to excessive stresses. It is important to check if this Eigen frequency of the structure is not in the range of the wave frequencies, which could lead to resonance of the structure which can lead to failure of the structure.

- An additional study in which the effect of size optimization is compared to the use of topology optimization and size optimization. An interesting aspect would be the comparison between the leading time and the final result.

- The new design consists of a truss structure in which multiple members are connected to a single joint. This resulted in multiple branches at a single joint which is unfavourable from a manufacturing point of view. Therefore it is suggested to change the angle of the diagonal members in order to translate the connection point of these members. This will enhance the manufacturing process of the truss structure.

- This structure is optimized in terms of weight and stiffness. No time is spend in the design of the telescopic movement system and other which are required for the operation of the gangway structure. These parts needs to be defined in order to obtain a fully working design. Only the load bearing capacity of the structure is investigated and optimized during this research.

- No material variations are considered during this research. An interesting approach would be the variation of different materials in order to achieve even a larger weight reduction of the gangway structure. From the results it can be seen that the structure is not stress driven and not displacement driven, but the structural stability is the limiting factor. Therefore a change in design material can have significant influence on the weight of the structure.

This research was conducted with a lot of manual work. No customization and adaptation of the software have been made and there was no prior experience with the use of Optistruct and the other software packages. The result of this is that it required a lot of time to get familiar with the software and to define a methodology to obtain a feasible design. A Lot of time has been spent on trial and error methods of problem solving. Therefore five steps should be followed in order to make the use of topology optimization much easier and to obtain satisfactory results.

1. Start with a simple design volume and apply the boundary conditions to surfaces or volumes which are not part of the design domain. Perform an analysis before running an optimization task to ensure that the system is in equilibrium and well defined.

2. Perform the optimization task without any accelerations or manufacturing constraints. Often the gravitational accelerations should be neglected due to the large mass of the volume domain. Investigate the different parameters of the optimization software. Make use of these different parameters in order to facilitate the optimization process. Check the common mistakes in section: A-19. Optimization pitfalls

3. Get designers involved. Designers should be involved when interpret topology results and realizing them. Discuss and purpose a manufacturing method with the design engineer. This is a difficult part and this can have a significant influence on the final result. With the help of a design engineer the realization step will be more efficient with enhanced results.

4. Use size and shape optimization in order to define the dimensions of the structure. This post-processing step will determine the dimensions of the structure which are required for the CAE and verify the manufacturing feasibility to a person with expertise in this area.

5. Check your final result by performing a finite element analysis and hand-calculations. These hand calculations are important in order to validate the FEA. If the exact solution is not known, hand calculations can give an estimate of the obtained results. Validation and verification of a finite element model is a critical step in any analysis. This allows the user to determine if his obtained results are correct. Lots of information can be found about this subject. [59]

The use of topology optimization requires experience and knowledge about the tools along with engineering intuition. It will not provide a clear solution after just pushing the button. For new users of topology optimization it is recommended to start with a small design space and simple and a few load cases. With this the designer will gain experience with the use of topology optimization and can use it as a tool in their daily work. Topology optimization shall be used in the early stage of the design process in which a concept design and load pads are more valuable than a detailed design. By implementing topology optimization early, great knowledge of the load paths and weaknesses in the structure is received early and will help the designer to make good decisions. If topology optimization is introduced in a late stage of the design phase, the influence of previously made decisions about the structures dimensions and geometry will affect the gain of using topology optimization. Topology optimization will serve as tool which provides input to the designers. No recommendations can be made for what parameters to for a general structure and how to relate them to what is desirable to be accomplished by the final design.

Literature list

[1] Wind Europe. (2017). *The European offshore wind industry*. Retrieved from https://windeurope.org/about-wind/statistics/offshore/european-offshore-wind-industry-key-trends-and-statistics-2016/

[2] Rademakers, L. U. C., Braam, H., & Obdam, T. S. (2007, December 6). Estimating costs of operation & maintenance for offshore wind farms. *ECN*, pp. 3-5.

[3] Rademakers, L. U. C., Braam, H., Zaaijer, M. B., & Van Brussel, G. J. W. (2003, Januari 1). Estimating costs of operation & maintenance for offshore wind farms. *ECN*, pp. 1-5.

[4] Holthuijsen, Leo H. (2007). *Waves in oceanic and coastal waters*. Cambridge, United Kingdom: Cambridge University Press.

[5] Gitter, R. (2008). Selection of structural alloys. *Design of aluminium structures*, *1*, 1-32.

[6] Mirzendehdel, A., & Suresh, K. (2017). *A Hands-On Introduction to Topology Optimization* (Rev. ed.). London, United kingdom: CreateSpace Independent Publishing Platform.

[7] Offshore & marine technology. (2013, November 13). Corrosion protection. *Offshore & marine technology*, *2013*(6), 72-74.

[8] Davis, J. R. (2001). *Alloying: Understanding the Basics*. Russell Township, America: ASM international.

[9] European Aluminium Association. (2002). *The aluminium automotive manual*. Retrieved from: https://european-aluminium.eu/media/1533/aam-materials-3-designation-system.pdf

[10] D. J. Cerda Salzmann. Ampelmann: development of the access system for offshore wind turbines. BOXPress, Oisterwijk: 2010.

[11] G. Van Bussel, "Offshore wind energy, the reliability dilemma," in *Proceedings of the* International Conference, pp. 2–6, 2002.

[12] J. Carroll, A. McDonald, and D. McMillan, "Failure rate, repair time and unscheduled O&M cost analysis of offshore wind turbines," *Wind Energy*, 2015.

[13] L. Dai, "Safe and efficient operation and maintenance of offshore wind farms," 2014.

[14] G. Hassan, "A guide to UK offshore wind operations and maintenance," *Scottish Enterprise* and The Crown Estate, 2013.

[15] Rademakers, L. Braam, H., Zaaijer, M, & Van Brussel, G. J. W. ASSESSMENT AND OPTIMISATION OF OPERATION AND MAINTENANCE OF OFFSHORE WIND TURBINES, pp. 1-5.

[16] Dowell, J., Zitrou, A., Walls, L., Bedford, T., & Infield, D. *Analysis of Wind and Wave Data to Assess Maintenance Access to* (Dowell 2013). Retrieved from: https://www.strath.ac.uk/media/departments/eee/iee/windenergydtc/publications/Dowell2013b.pdf

[17] 4coffshore, "Wind farm service vessels (wfsvs) - an analysis of supply and demand," 2015.

[18] Windpowermonthly. (2017, July 5). [TEN of the Biggest Turbines Offshore]. Retrieved November 3, 2017, from: https://www.windpowermonthly.com/10-biggest-turbines

[19] Energy Information Administration. (2016, May 12). world energy consumption by 2040. Retrieved from: https://www.eia.gov/todayinenergy/detail.php?id=26kg\

[20] Wikipedia. (2017, October 9). Topology optimization. Retrieved from: https://en.wikipedia.org/wiki/Topology_optimization

[21] D. C. Salzmann, J. Prezzi, S. ten Haaf, S. Groenteman, et al., “Walk to work offshore using motion compensated gangways,” in OTC Brasil, Offshore Technology Conference, 2015.

[22] U.S. Energy information administration. (2017). *Levelized Cost and Levelized Avoided Cost of New Generation Resources in the Annual Energy Outlook 2017*. Retrieved from: http://www.eia.gov/forecasts/aeo/pdf/electricity_generation.pdf

[23] Willemse, C. A., Grimmelius, H. T., & Tjallema, A. R. (Eds.). (2008). *Dictaat Dynamic positioning* (Rev. ed.). Delft, The Netherlands: Technical University Delft.

[24] American bureau for shipping. (2016 August). *OFFSHORE ACCESS GANGWAYS*. Retrieved from: https://www.safety4sea.com/wp-content/uploads/2016/08/ABS-Offshore-Access-Gangways-Guide-2016_08.pdf

[25] DNV-GL. (2017 September). cer*tification of offshore gangways for personal transfer*. Retrieved from: https://rules.dnvgl.com/docs/pdf/DNVGL/ST/2015-12/DNVGL-ST-0358.pdf

[26] Wikipedia. (2017, November 1). Active Heave Compensation. Retrieved from: https://en.wikipedia.org/wiki/Active_heave_compensation

[27] Dictionary.com. (2017). Optimization. Retrieved from: http://www.dictionary.com/browse/optimization

[28] Querin, O., Victoria, M, Gordoa, C, Ansola, Rubn., & Martí, P (2017). *Topology Design Methods for Structural Optimization* (Rev. ed.). Leeds, United Kingdom: Academic Press.

[29] Osher, S., Sethian, J.A. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations J. Comput. Phys. 79: 12{49, 1988.

[30] Langelaar, M. ,Technical University Delft. (2017). [Course ME46060: Engineering optimization] [Lecture slides]. Retrieved from www.blackboard.tudelft.nl[1]

[31] Christensen, P, & Klarbring, A. (2009). An introduction to structural optimization. Netherlands: Springer.

[32] Deaton, J. & V.Grandhi, R. (2013). A survey of structural and multidisciplinary continuum topology optimization. *Structural multidisciplinary optimization*, *1*, 1-7.

[33] Bendsoe, M. P., & Sigmund, O. (2004). *Topology optimization: Theory, methods and applications* (2nd ed.). Berlin, Germany: Springer.

[34] Gunwant, D., & Misra, A. (2012). International journal of advances in engineering & technology. *Topology optimization of continuum structures using optimality criterion approach in Ansys*, *5*(1), 472-473.

[35] Renold, W. (2006). *Investigation into structural topology optimization problem formulations*. Retrieved from: http://www.lulu.com/shop/william-renold/simp-and-eso-topology-optimization-methods-investigation-into-structural-optimization-problem-formulations/hardcover/product-11919833.html

[36] Ferguson, I., & Pennsylvania state university. (2015). *Using topology optimization to improve design for additive manufacture*. Retrieved from: https://etda.libraries.psu.edu/files/final_submissions/11113

[37] Topology Optimization Guide. (2017). Software list topology optimization. Retrieved from: http://www.topology-opt.com/software-list/

[38] Bradley, A. (2013). *PDE-constrained optimization and the adjoint method*. Retrieved from: http://cs.stanford.edu/~ambrad/adjoint_tutorial.pdf

[39] Kollmann, M. (2010). *Sensitivity analysis: The direct and Adjoint method*. Retrieved from:

[1] Source from Blackboard TUD (not freely accessible)

http://www.numa.uni-linz.ac.at/Teaching/Diplom/Finished/kollmann-dipl.pdf

[40] Martins, J. (n. *Sensitivity analysis: Multidisciplinary design Optimization* (Stanford University). Retrieved from: http://aero-comlab.stanford.edu/jmartins/aa222/aa222sa.pdf

[41] Verboom, J. M., Technical university Delft. (2017). *Design and additive manufacturing of manifolds for Navier-stokes flow* (Master Thesis). Retrieved from: https://repository.tudelft.nl/islandora/object/uuid:c843e8a0-3041-40db-a6b0-d4558a5fd1dd?collection=education

[42] Wein, F. (2008, August). Topology optimization using the SIMP method [Lecture slides]. Retrieved from: http://sites.google.com/site/fabianwein/simp_presentation.pdf

[43] DNV-GL. (2013 June). cer*tification for lifting appliances appendix A*. Retrieved from: https://rules.dnvgl.com/docs/pdf/DNV/stdcert/2011-10/Standard2-22.pdf

[44] Costi, D., Torricelli, E., Splendi, L., & Pettazzoni, M. (2011, July 6). Optimization methodology for an automotive hood substructure. *World congress on engineering*, *2011*(1), 1-4. Retrieved from: http://www.iaeng.org/publication/WCE2011/WCE2011_pp2390-2393.pdf

[45] Tomlin, M., & Meyer, J. (2011). *Topology optimization of an additive layer manufactured aerospace part* (7th Altair CAE technology conference). Retrieved from: https://www.additivemanufacturing.media/cdn/cms/uploadedFiles/Topology-Optimization-of-an-Additive-Layer-Manufactured-Aerospace-Part.pdf

[46] Ampelmann. (2010). The history. Retrieved December 13, 2017, Retrieved from: https://www.ampelmann.nl/story

[47] Chen, J., Young, B., & Uy, B. (2006). *Behavior of high strength structural steel at elevated temperatures* (University of Wollongong). Retrieved from: http://ro.uow.edu.au/cgi/viewcontent.cgi?article=1345&context=engpapers

[48] Air liquide. AISI-SAE classifications of steel [Press release]. Retrieved December 20, 2017, from: http://blueshield.ca/en/docs/pdf/techInfo/DataBook/E-Fsection218to228.pdf

[49] Composites world. (2014, January 4). Offshore Applications: The Future Is Now. Retrieved September 20, 2017, from: https://www.compositesworld.com/articles/offshore-applications-the-future-is-now

[50] Suresh Babu, M., Baksi, S., Srikanth, G., & Biswas, S. (2016, August 2). Composites for Offshore Applications. Retrieved October 15, 2017, from: https://www.compositesworld.com/articles/offshore-applications-the-future-is-now

[51] Wieloch, D., & Taslim, I. (2004). Driving million dollar saving at ford using topology optimization. In ORG. Altair Engineering (Ed.), *Altair Engineering technology* (pp. 1-7). Retrieved from: https://altairhyperworks.in/ResourceLibrary.aspx?keywords=ford+bracket

[52] Royal IHC. SOV T60-18 Service operation vessel [Dataset]. Geraadpleegd op 8 december 2017, from: https://www.google.nl/url?sa=t&rct=j&q=&esrc=s&source=web&cd=1&ved=0ahUKEwjxyY7Jip3YAhUjAsAKHbE8DAcQFggnMAA

[53] Lammey, M. (2017, December 20). Ampelmann hit with HSE notice after worker injured on gangway. Retrieved January 3, 2018, from: https://www.energyvoice.com/oilandgas/north-sea/159512/ampelmann-hit-hse-notice-worker-injured-gangway/

[54] Smith, C. J., Gilbert, M., Todd, I., & Derguti, F. (2016). Application of layout optimization to the design of additively. *Structural multidisc optimization*, *1*, 1297-1313. Retrieved from: https://link.springer.com/article/10.1007/s00158-016-1426-1

[55] Singiresu, S. (1996). *Engineering Optimization | Theory and Practice* (3rd ed.). Hoboken, America: John Wiley & Sons.

[56] Bendsøe, M., & Sigmund, O. (1999). Material interpolation schemes in topology optimization. Archive of Applied Mechanics, 69(9-10), 635-654. Retrieved from: https://doi.org/10.1007/s004190050248

[57] H. Thomas, M. Zhou, and U. Schramm. Issues of commercial optimization software development. Struct. Multidisc. Optim., 23:97{110, 2002.

[58] Altair Engineering Inc. Optistruct 14.0 help files, 2017.

[59] D. Cook, Robert, S, Malkus, David, E. Plesha, Michael, E & J. Witt, Robert. (z.d.). *Concepts and applications of finite element analysis* (4e ed.). Wisconsin, America: Wiley.

[60] Skotny, L. (n.d.). Explanation of linear buckling. Retrieved September 2, 2018, from: https://enterfea.com/linear-buckling-explained/

[61] Timoshenko, S., & Gere, J. (2009). *Theory of elastic stability* (2nd ed.). New York, America: Dover publications.

[62] Akin, J. E. (Ed.). (n.d.). *Buckling analysis*. Houston, Texas: Rice university.

[63] Romeijn, A. ,Technical University Delft. (2015). [Course MT2432: Sterkteleer voor schepen] [Lecture slides]. Retrieved from: www.blackboard.tudelft.nl[2]

[2] Source from Blackboard TUD (not freely accessible)

Appendices.

Table of contents

A-1. *Nomenclature*

Symbol	Explanation	Unit
%	Percentage	-
α	Angle between wind direction and exposed surface	°
ρ	Density	kg/m^3
ρ_e	Artificial density	kg/m^3
θ_i	Angle between the element and the horizontal plane	°
λ_i	Lagrange multiplier	-
M	Friction coefficient	-
δ	Deflection	m
δ_{max}	Maximum deflection	m
δ_1	Deflection due to self-weight	m
δ_2	Deflection due to applied test-load	m
°	Degrees	°
°c	Degrees Celsius	°C
σ	Normal stress	N/m^2
σ_y	Guaranteed minimum yield strength	N/m^2
σ_u	Ultimate strength of material	N/m^2
a_i	Cross sectional area of element i	m^2
B	Element direction matrix	-
C	Compliance	m/N
E_e^0	Original elastic modulus	N/m^2
E_e	Artificial elastic modulus	N/m^2
F	Force vector	N
F_r	Friction force	N
F_n	Normal force	N
$f(x)$	Costs or objective function	-
G	Self-weight	N
$g_i(x)$	Inequality constraint	-
h_i	Objective function	-
H_s	Significant wave height	m
$H_j(x)$	Equality constraint	-
K	Stiffness matrix	N/m
K_e	Global stiffness matrix	N/m
K^0	Local stiffness matrix	N/m
l_i	Length element i	m
m	Number of elements in the structure	-
N	Number of mesh elements	-
Q	Nodal force vector	N
R_k	Residual of the governing equation	-
S_c	Stress coefficient	-
t_i	Volume element i	m^3
T_z	Mean zero-crossing period	s
u	Displacement	m
u_k	State variables	-
U_e	Local displacement vector	m
v_e	Volume mesh element	m^3
V	Design volume	m^3
V_e	Volume mesh element	m^3
V_s	Original volume design space	m^3
x_j	Independent design variables	-
X	Vector containing design parameters	-
x_n	Design parameters	-

A-2. List of abbreviations.

Abbreviations	Explanation
ABS	American bureau of shipping
AHC	Active Heave Compensation
ALS	Accidental limit states
AMC	Active motion compensation
ASD	Allowable stress design
COG	Centre of gravity
CF	Centrifugal force
CTV	Crew Transfer Vessel
DNV GL	Det Norske Veritas Germanischer Lloyd
DOF	Degrees of Freedom
DP	Dynamic positioning
EIA	Energy Information Administration
FEA	Finite element analysis
FLS	Fatigue limit state
Ge	Gripper end
GRP	Glass reinforced plastics
GSL	Green sea loads
HAZ	Heat Affected Zone
HPu	Hydraulic power unit
Hs	Significant wave height
IHC	Industriële Handels Combinatie
LC	Load case
LCC	Landing cone connection
LF	Landing foot
LL	Live load
LRFD	Load and resistance factor design
MCP	Motion compensated pedestal
MRU	Motion reference unit
O&M	Operation and maintenance
PMC	Passive motion compensation
PS	Portside
RB	Rubber bumper
RE	Roller end
SB	Starboard
SOV	Service Operation Vessel
SIMP	Solid isotropic material with penalization
SLS	Serviceability limit state
SW	Self-weight
TBC	Table of contents
TL	Test load
TpS	Tilted pedestal
ULS	Ultimate limit states
WSD	Works stress design

A-3. List of figures

A-4. List of tables

A-5. Design calculations gangway

The different load conditions for the gangway structure depends on the different load situations. The DNV-guidelines demands that the gangway structure complies with the load cases stated in

In this appendix all the different input parameters and load cases are calculated and justified. The motion operational accelerations are dependent of the vessel on which the gangway structure will be installed, as well on the specific location of the gangway structure on the supporting vessel.

Vessel characteristics

For the design of the gangway structure, the data from the SOV-T60-18B vessel is used as a reference. This is a service operation vessel (SOV) designed by IHC and the gangway system will be installed on this vessel. This vessel has DP2 operating capabilities. The vessel data can be found in the following tables [52]:

Table 45: Vessel dimensions

SOV-T60-18B	
Dimension [-]	Length [m]
Length	82
width	18
Draught	5

Table 46: Vessel accelerations

SOV-T60-18B accelerations [m/s^2]		
Direction [-]	Operational	Deployment/retrieval
Longitudinal	0.84	0.84
Transversal	1.42	1.42
Vertical	1.54	1.54

The accelerations acts parallel or normal to the vessel deck. The downward (vertical) acceleration is considered normal to the deck. The longitudinal and transversal accelerations are considered parallel to the deck and are perpendicular to each other. The vertical gravity acceleration is equal to 9,81 m/s^2. There are three different conditions considered. The maximum operational condition is the operational limit of the gangway structure. The motion compensated gangway must keep a steady connection with the offshore structure taking into account the maximum acceptable operational motions. The limitation of the structural design and the hydraulic equipment must be taken into account. The deployment or retrieval condition consists of the start-up or shut-down procedure of the gangway. The limitations for this condition are equal to the operational condition. In the maximum survival mode the device is in non-operational condition. The gangway structure is secured in a sea-fastening frame and the ship is in transit. The accelerations in survival condition are often equal to the maximum survival conditions of the supporting vessel. At this stage of the research they are still unknown.

The gangway structure will be designed according to the DNVGL-ST-0358 Certification of offshore gangways for personnel transfer. These guidelines specify the following dynamic factors according to Table 47.

Table 47: Dynamic factors according to the DNV

Minimum dynamic factors (Df)	
Horizontal (Df_y)	1.10
Vertical (Df_z)	1.10

These accelerations and dynamic factors must be incorporated into the load cases. For the load cases, it is assumed that the accelerations can work in the positive and negative direction. The accelerations are applied in combination with standard earth gravity which is always acting in the negative vertical direction. It is assumed that the influence of the acceleration in the longitudinal direction of the gangway structure is small and therefore the maximum value is assumed in order to reduce the amount of load cases. Also only the negative vertical acceleration is considered due to the fact that it will be added up to the gravity acceleration instead of counteracting it. Therefore it is assumed to be the most severe situation. Now it is possible to generate a table with all the different acceleration combinations. For the emergency disconnection load case, the dynamic factors are included as specified by the DNV.

Different coordinates systems are defined for the vessel and for the gangway structure. For the vessel, the positive X-direction is defined from the stern to the bow, parallel to the ship's axis. The positive Y-direction is from starboard to portside and the positive Z-direction is from the deck pointing in the upwards direction. For the gangway structure, the positive X-direction is defined in the longitudinal direction of the gangway. The Y-direction is perpendicular to this axis according to the right hand rule and the Z-axis is in the positive upward direction parallel to the gangway deck. The difference between these two coordinate systems is visualized in Figure 87.

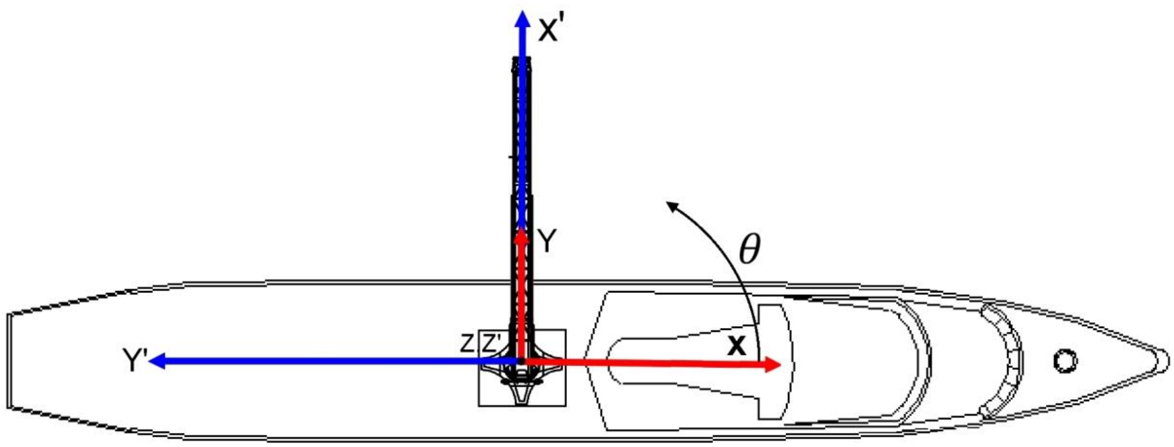


Figure 87: Local and global coordinate systems

The coordinate system in red is related to the ship coordinate system. The coordinate system in blue is the local gangway coordinate system which is denoted with X', Y' and Z'. This means that when the gangway is in operational conditions, then the gangway structure is fully extended and in horizontal position. It points horizontal perpendicular outwards of the vessel. This means that the local gangway coordinate systems is rotated with an angle of 90 degrees along the Z-axis with respect to the vessel coordinate system. This means that the accelerations for the vessel in the longitudinal direction is equal to the transversal accelerations in the local gangway coordinate system. The same holds for the accelerations in the transversal direction which is equal to the longitudinal direction in the local gangway coordinate system.

Table 48: Acceleration combinations in the local gangway coordinate system

Acceleration combinations [m/s^2]								
					LC 3$_A$	**LC 3$_B$**	**LC 3$_C$**	**LC 3$_D$**
	LC 1C$_A$	**LC 1C$_B$**	**LC 1C$_C$**	**LC 1C$_D$**	**LC 2B$_a$**	**LC 2B$_b$**	**LC 2B$_c$**	**LC 2B$_d$**
Longitudinal	1.42	1.42	-1.42	-1.42	2.52	2.52	-2.52	-2.52
Transversal	0.84	-0.84	0.84	-0.84	0.84	-0.84	0.84	-0.84
Vertical	-11.35	-11.35	-11.35	-11.35	-12.45	-12.45	-12.45	-12.45

Matrix 1, can be used to convert the acceleration vectors from the vessel coordinate system to the local gangway coordinate system. In this matrix, θ represents the rotation angle around the Z-axis in the counter clockwise direction. For an angle of 90°, this results in matrix 2.

Matrix 1: $\begin{bmatrix} cos(\theta) & \sin(\theta) & 0 \\ -sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$ For $\theta = 90°$, this results in Matrix 2: $\begin{bmatrix} 0 & 1 & 0 \\ -1 & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix}$

This results in four acceleration combinations per load case. For the two load cases this results in eight different load combinations which must be analysed. These accelerations will be used as input parameters for the topology optimization. The standard earth gravity is included in the vertical accelerations. The total vertical acceleration is equal to the sum of the gravity acceleration and the vertical vessel acceleration.

Loads on the gangway

The self-weight (G) of the gangway depends on the design of the gangway. The total volume of the structure and the material density determines the total weight of the gangway structure. In the beginning of the optimization process, this weight is equal to the volume of the design space multiplied by the density of the construction material times the gravity acceleration constant. The optimization process will reduce the required volume and this results in a weight reduction. The self-weight of the structure will be included in the FEA analysis in order to satisfy the acceptance criteria. For the size optimization. It is chosen to use the weight of the current gangway design in order to capture the behaviour of the combined structure.

The live load represents the allowable number of persons on the gangway at one moment. It is defined as the sum of the cargo capacity, trolley mass and the operator mass multiplied by the number of persons allowed on the gangway. This results in formula C.1.

$$LL = \left(cargo_{capacity} + Trolley_{mass} + Operator_{mass}\right) \cdot g \cdot N_{persons} \quad (C.1)$$

We assume that only one person is allowed on the gangway during operation. The mass of one operator is equal to 100 kg. The cargo capacity is equal to 300 kg and the trolley mass is equal to 700 kg. When we fill this in the equation above we obtain a total live load of 10800 N.

$$LL = (300 + 700 + 100) \cdot 9.81 \cdot 1 = 10800\ N \quad (C.2)$$

The worst live load condition is chosen considering the maximum bending moment acting on the structure. For the normal condition it will be placed in the middle of the gangway when it is at its maximum length. A safety factor of 2 is applied during this condition. For the cantilevered condition it is placed at the tip of the gangway. In this case the maximum moment on the supported side of the gangway will be considered. The DNV guidelines demands that the gangway structure must be able to handle double the live load.

Wind loads

The gangway structure is subjected to wind loads. The wind load normal to a flat surface can be calculated with equation C.3 according to DNV [43].

$$P = A \cdot q \cdot C \cdot \sin(\alpha) \tag{C.3}$$

Where:

$P = Wind\ force\ [N]$
$A = Exposed\ area\ [m^2]$
$q = Air\ velocity\ pressure\ [pa]$
$C = Average\ pressure\ coefficient\ for\ the\ exposed\ surface\ [-]$
$\alpha = The\ angle\ between\ the\ wind\ direction\ and\ the\ exposed\ surface\ [°]$

The value of the air velocity pressure can be calculated with equation C.4.

$$q = \frac{\rho \cdot v^2}{2} \tag{C.4}$$

In which:

$\rho = Mass\ density\ of\ air\ [1.225\ kg/m^3]$
$v = wind\ velocity\ [m/s]$

Now the value of the air velocity pressure can be calculated for the three different load conditions which are specified by the DNV. Also the wind force can be calculated if the exposed surface area is known. This value is unknown until the optimization process is performed. Therefore it is chosen to wind pressure into the sizing optimization. A safety factor of 1.2 is used to accommodate for additional surface areas like: protective fences, railings, additional equipment and auxiliary lines. The values in Table 49 are obtained by using equation C.4.

Table 49: The wind velocities & pressures

Load condition	Wind velocity [m/s]	Wind pressure [pa]
Operational wind speed	20	245
Deployment/retrieval wind speed	36	794
Transit/survival/parked wind speed	44	1186

The average pressure coefficient C, can be obtained from the DNV-standard 2.22 guidelines [43]. For a flat sided section, the coefficient has a value of 2. An angle of 90 degrees is chosen because this is the most limiting condition. Now the approximated lateral surface area of the current design is equal to 5.51 m2 for the telescopic boom and 20.2 m2 for the fixed boom. These values can be multiplied with a safety factor 1.2 and inserted into equation C.3. to calculate the wind load on the gangway boom. The results of these calculations are given in Table 50.

Table 50: The Wind forces

Load condition	Wind velocity [m/s]	Wind force [N]	
		Main boom	Telescopic part
Operational wind speed	20	11878	3240
Deployment/retrieval wind speed	36	38493	10500
Transit/survival/parked wind speed	44	57497	15684

Bumper loads

The gangway can be equipped with a rubber bumper at the tip as a landing mechanism. This rubber bumper is pushed against the offshore structure and is required to keep the gangway structure into position and to provide a safe and firm connection with the offshore structure. A bumper load is required to generate the required sliding friction between the offshore structure and the gangway tip. Friction is the resistance to motion when two surfaces slide over each other. The static friction is the value of the limiting friction just before slipping occurs. The required bumper load can be calculated with equation C.5.

$$F_r = \mu \cdot F_n \tag{C.5}$$

In which: $F_r = The\ total\ friction\ force.$
$\mu = The\ static\ friction\ coefficient.$
$F_n = The\ normal\ force\ or\ equivalent\ bumper\ load.$

The total required friction force can be calculated with equation C.6. The friction force consists of the self-weight of the gangway plus two times the live load acting on the middle of the gangway. The factor two comes from the fact that the gangway structure is designed to handle two times the live load. Both loads must be multiplied with the maximum vertical acceleration of the supporting vessel in order to cope for the dynamic behaviour of the vessel. The assumption is made that the COG is in the middle of the gangway. Also the live load is acting at the middle of the gangway as specified by DNV [25], this results in a symmetrical load distribution over the support conditions. The static friction coefficients are given in Table 51 in which the lowest values are shown. For the calculation of the bumper load a friction coefficient of 0.45 is used in order to be conservative. The self-weight of the current design is used as a starting value due to the fact that weight of the optimized gangway structure is still unknown at this stage of the research. This bumper load should only be included in case the gangway is equipped with a bumper as a landing mechanism.

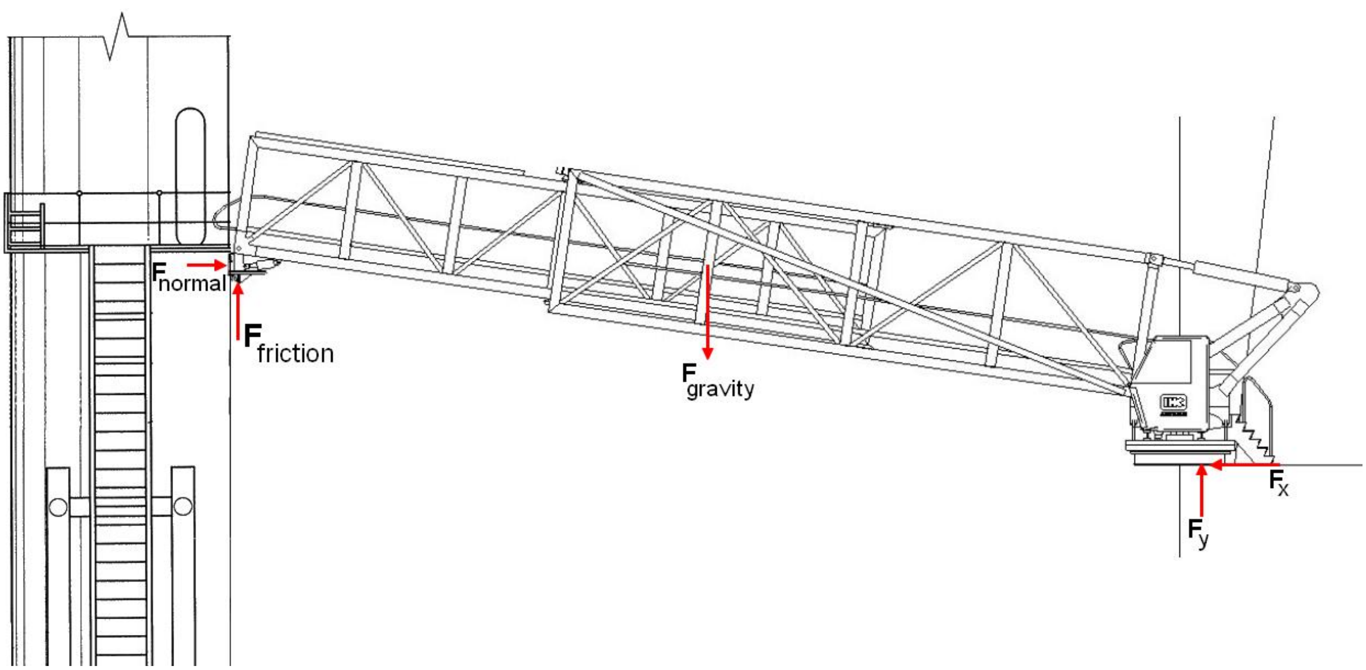


Figure 88: schematization of the friction force and bumper loads

The total required friction force can be calculated with formula C.6.

$$F_r = 0.5 \cdot (MOA_z + G) \cdot (F_{gravity} + 2 \cdot LL) \tag{C.6}$$

In which:
MOA_z = The vertical operation acceleration $[m/s^2]$
G = Earth gravity accelerations $[m/s^2]$
$F_{gravity}$ = Selfmass of the gangway structure $[Kg]$
LL = The live load $[Kg]$

Table 51: Static friction coefficients

Material	Static friction coefficient (μ_s)
Rubber on concrete (wet)	0.45
Rubber on concrete (dry)	0.6
Rubber on steel	0.6

The live load (LL) is equal to 10800 N. The maximum heave acceleration is equal to 1.54 m/s2. These values can be substituted in equation C.6. in order to calculate the total required friction force. The required bumper load can be calculated with equation C.5. The results for the total required friction force and the required bumper load are given in Table 52. The total required bumper load should only be applied in the case when the gangway structure is provided with a rubber bumper as a landing mechanism. Otherwise is does not have to be included in the analysis. When all the parameters are substituted in equation C.6, then the values in Table 52 are obtained.

Table 52: The load parameters

Parameter	Load [kN]
Total required friction force	1118
Required bumper load	2485

Additional masses

The total mass of the current design of the gangway structure is equal to 13038 kg. In addition to the self-mass of the gangway structure, the gangway boom will be loaded by additional components such as:

- -Rm = rollers mass (300 kg)
- -Fm= Flooring and hand rails (1000 kg)
- -ASm= Telescopic adjustment system (1000 kg)
- -Pm = Intermediate platform, if applied (500 kg)
- -Um = Utility lines and supporting structures (1000 kg)
- -Lm = landing/connection tool (400 kg)

These masses are obtained from similar structures or by approximation. The masses are split between the fixed and telescopic section according to the length ratio of the two parts.

$$additional\ mass\ fixed = R_m + \frac{F_m}{Total\ length} \cdot fixed\ boom\ length + AS_m + P_m + \frac{U_m}{Total\ length} \cdot fixed\ boom\ length \tag{C.7}$$

$$additional\ mass\ telesc. = \frac{F_m}{Total\ length} \cdot cantileverd\ length + \frac{U_m}{Total\ length} \cdot cantileverd\ length \tag{C.8}$$

When the values are substituted in the equations above, this results in a total additional mass of 3255 kg for the fixed part and 845 kg for the telescopic part. At the telescopic part, the mass of the landing tool is located at the tip of the structure. The total additional mass is equal to 4500 kg as can be seen in Table 53.

Table 53: Additional masses gangway structure

Additional masses	
Fixed section	*3255 kg*
Telescopic section	*845 kg*
Tip of the gangway	*400 kg*

Centrifugal force
The radial/centrifugal force can be determined on the basis of the maximum angular velocity and the radius to the considered mass. According to the DNV, equation C.9. may be used to calculate the centrifugal force.

$$CF\ (kg) = \left(\frac{G_m}{1000}\right) \cdot (n^2 \cdot r) \tag{C.9}$$

In which: $G_m = gangway\ mass\ [kg]$
$r = radius/distance\ from\ the\ revolving\ axis\ to\ the\ gangway\ COG\ [m]$
$n = revolutions\ per\ minute\ [\text{min}^{-1}]$

The revolutions per minute depends on the slewing speed of the gangway. The centrifugal force will act on the tip of the gangway and is in the longitudinal direction of the gangway structure. The maximum slewing speed is equal to 3.6°/s, which is equivalent to 0.6 rpm. The maximum luffing speeds is equal to 4°/s, which is equivalent to 0.67 rpm. The distance from the revolving axis to the centre of gravity is equal to 16.5m which is half of the maximum extended length. The self-weight of the current gangway design including additional masses is equal to 17538 kg. The substitution of these values leads to a total centrifugal force of 129.9 kg-force which is equivalent to approximately 1300 N. This is a relatively small value and this is due to the low slewing speed. Still, this value will be included in the size optimization.

Green sea loads
According to DNV, the green sea loads must be incorporated during the parked/survival conditions. These loads will be neglected in the optimization due to the fact that the gangway is be located in the middle of the ship and the gangway will be fixed in a sea fastening frame during transit conditions. The gangway consist of an open truss type gangway structure which is suitable to cope with green sea loads. Therefore it is assumed that green sea loads have no significant impact on the gangway structure. This load case will not be included in the optimization process.

Overview results

An overview of all the calculated loads are given in Table 54. These values comply with the symbols from the load cases in Table 2 and will be used as input parameters for the optimization process. Only five load cases will be analysed due to the fact that these load cases overrule the other load cases. The values for the wind pressure and combined accelerations are given in Table 48 and Table 49 respectively.

Table 54: Results calculations

Symbol	Description		Value [N]
G	Self-weight current design	Fixed part	110000
		Telescopic part	20380
LL	Live Load		10800
BL	Bumper Load		2485000
CF	Centrifugal force		1300
AM,f	Additional mass fixed section		32550
AM,t	Additional mass middle telescopic section		8450
$AM_{,Lt}$	Additional mass tip telescopic section		4000

These values will be used for the set-up and definition of the boundary conditions of the load cases. In the next section, the support conditions, load and the relation between the two gangway sections is defined.

A-6. Input parameters optimization process

The basic design of the motion compensated gangway has the following main specifications as shown in Table 55. The gangway structure must be designed according to these specifications.

Table 55: Gangway specifications

Gangway specifications	Value
Retracted gangway length (Including safety length)	25 m
Extended gangway length (maximum length)	33 m
Maximum stroke	9 m
Minimum overlap sections	4 m
Length fixed section	24m
Length telescoping section	13m
Internal free width	0.9 m
Internal free height	2.1 m
Slewing angle	-135° to +135°
Maximum slewing speed	3.6°/s
Maximum luffing range	-16° to +16°
Maximum luffing speed	4°/s
Maximum telescoping speed	2.5 m/s

Now all the required parameters for the load cases are calculated or approximated. The different load cases are analysed and it is determined which load cases overrule the other load cases. This resulted in a set of five load cases which needs to be analysed. Three load cases will be included into the optimization process. These load cases will be specified in this section.

Now the next step is to determine how to optimize both sections of the gangway structure separately and therefore how to relate them. The gangway structure consists of two separate sections in order to perform the telescopic motion. The gangway structure is equipped with roller bearings to allow the telescoping boom section to freely move in the longitudinal direction. Due to the fact that these are two different structures with different loads and boundary conditions, it is chosen to analyse these parts separately. A schematic view of the different boom sections of the gangway is shown in Figure 89.

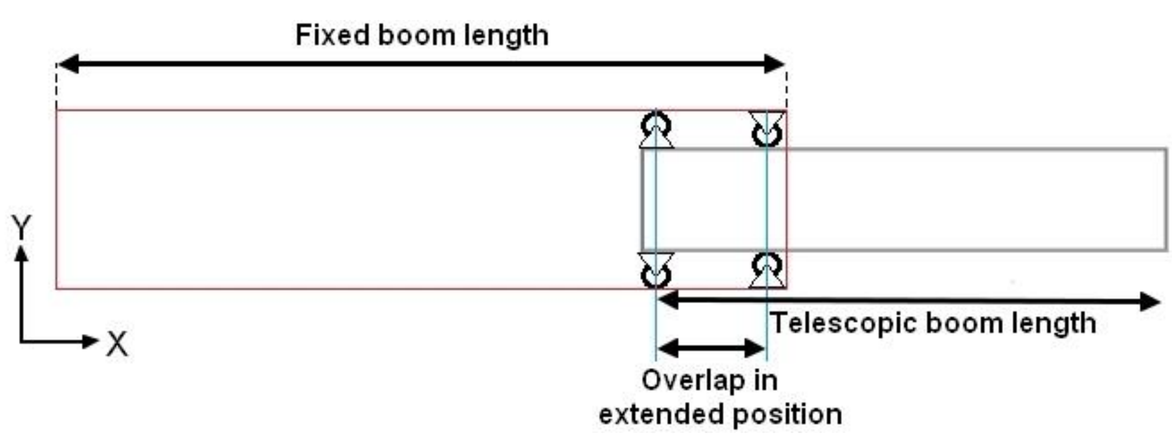


Figure 89: Boom sections

During each load case, the different sections of the gangway are supported by different boundary conditions. An overview of the different support conditions for each load case are given in Table 56. During these load cases, the situation of the gangway can be divided into two categories:

- Gangway in cantilevered condition.
- Gangway in double supported condition.

The first situation refers to the load cases 2_b and 3, in which the gangway is double hinged at the origin of the fixed section and the tip of the gangway is unsupported. This means that the gangway is in cantilevered condition. The second condition refers to load cases 1_c, 4 and 5. In which the gangway is supported on both ends by means of hinged connections.

Table 56: Different support conditions

Load case	Description	Fixed boom section		Telescoping boom section	
		S.C. left	**S.C. right**	**S.C. left**	**S.C. right**
1_c	Normal working condition	Hinged	Roller bearing	Roller bearing	Hinged
2_b	Deployment/retrieval	Double Hinged	Unsupported	Roller bearing	Unsupported
3	Emergency disconnection	Double Hinged	Unsupported	Roller bearing	Unsupported
4	Parked/survival	Hinged	Roller bearing	Roller bearing	Hinged
5	Load test	Hinged	Roller bearing	Roller bearing	Hinged

S.C = Support condition

The fixed part of the gangway structure is equipped with roller bearings to allow the telescoping boom section to freely move in the longitudinal direction, as can be seen in Figure 90. The support condition on the right side of the telescoping part depends on the type of landing mechanism that is used. For this case we assume a landing cone which can be approximated by a hinged connection.

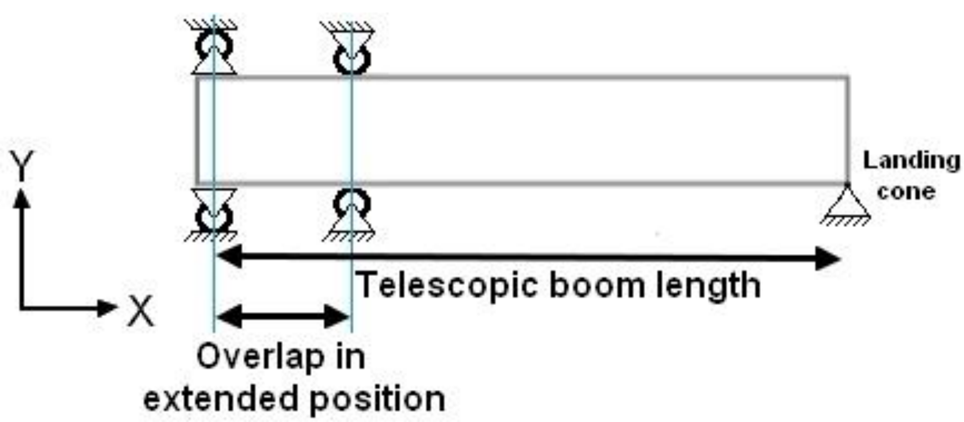


Figure 90: Support condition telescopic part in operational conditions

The roller bearings exert vertical forces which generates a resulting moment on the gangway which results in a connection between the two separate sections. The contact zone between the roller bearings and gangway section are related to high stresses. Therefore it is important to model these loads precisely. The next step is to elaborate the two load conditions and to define the relation between the two separate sections.

The gangway in double supported condition.

In this case, the gangway is fully extended and supported at both ends by hinged connections. The live load (LL) is acting in the middle of the gangway with a safety factor of 2. The loading conditions can be visualized with the free body diagram given in Figure 91. The self-weight and the additional masses are acting in the middle of the two sections. The additional load due to the landing tool is located at the gangway tip. The live load is acting in the middle of the total length of the gangway, which is the most unfavourable condition. The hydraulic luffing cylinder is not working in this condition and therefore the left side of the gangway has only one support condition.

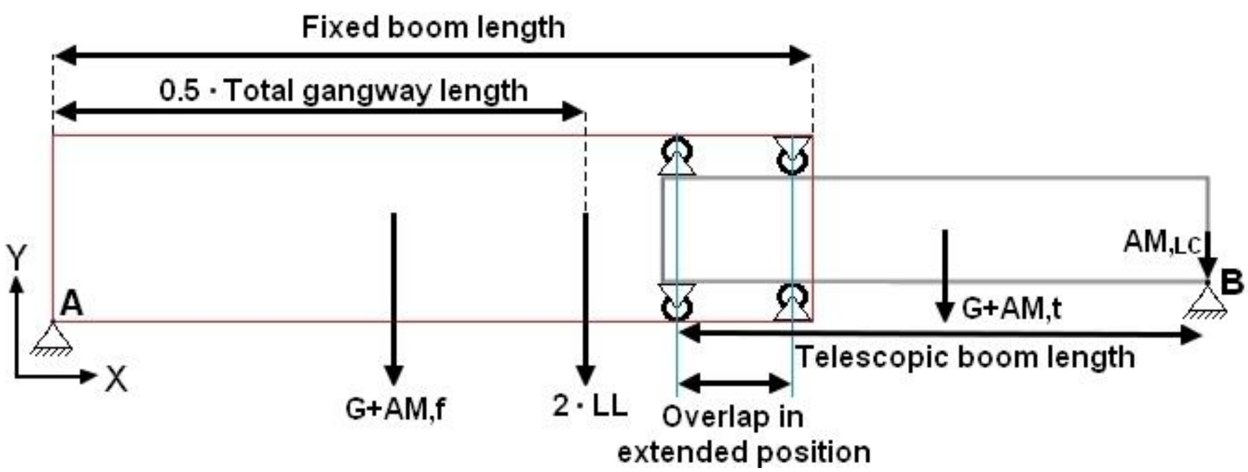


Figure 91: The gangway in normal working condition

In this situation, the complete structure is statically determinate. Therefore it is possible to calculate the support reactions for the two hinged connections. The magnitude of these support reaction determines the behaviour of the structure and will be used in the optimization process. The free body diagram of the telescopic section is showed in Figure 93. The left side of the telescopic boom section is supported by four roller bearings, which are able to exert a moment on the structure. These roller bearings exert vertical forces on the gangway structure. The right side is supported by a landing cone, which is simulated as a hinge.

In order to capture the behaviour of the total structure, the reaction forces acting on the support conditions are required. To determine these reaction forces, an equilibrium of moments is applied. The reaction forces in the points A and B are required for the input of the optimization process. All the dimensions and the forces acting on the structure are given in Figure 92.

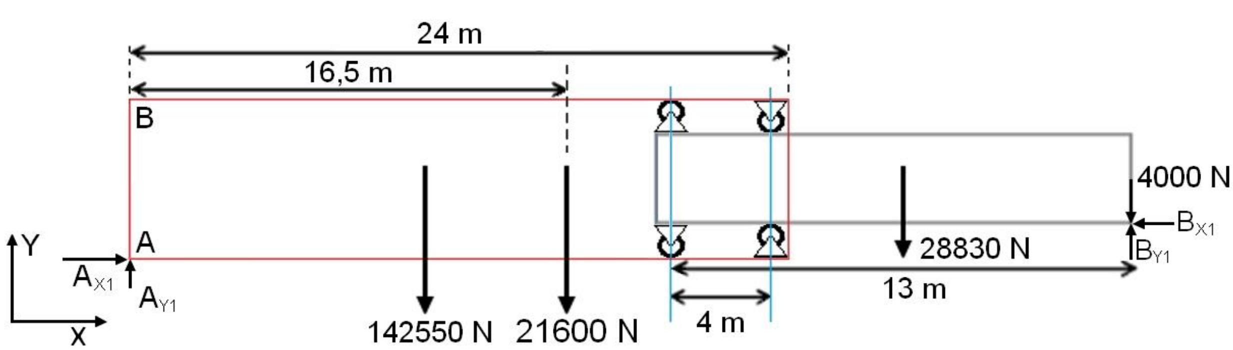


Figure 92: Free body diagram for the normal operational condition

For convenience, the results from the calculations are shown again in Table 57.

Table 57: Results calculations

Symbol	Description	Value [N]
$G_{1,2}$	G_1= Self-weight fixed part	110000
	G_2= Self-weight telescopic part	20380
LL	Live Load	10800
CF	Centrifugal force	1300
AM,f	Additional mass fixed section	32550
AM,t	Additional mass middle telescopic section	8450
$AM_{,Lc}$	Additional mass tip telescopic section	4000

The reaction forces are obtained by using the moment equilibrium principle. It is chosen to set the moment around point A equal to zero for an equilibrium state. Because the sum of moments about any given point is equal to zero for a static situation. For the sign convention it is chosen that clockwise moments are positive and counter-clockwise moments are negative. Therefore the moment in point A due to the forces acting on the gangway is equal to:

$$\circlearrowright_+ \sum\nolimits_{M_a} = \left(G_1 + Am_f\right) \cdot 12 + 2 \cdot LL \cdot 16.5 + (G_2 + Am_t) \cdot \left(24 - 4 + \frac{13}{2}\right) + 33 \cdot Am_{lc} - 33 \cdot B_{y1} = 0 \qquad \text{(D.1)}$$

This equation can be simplified to:

$$\circlearrowright_+ \sum\nolimits_{M_a} = \left(G_1 + Am_f\right) \cdot 12 + 2 \cdot LL \cdot 16.5 + (G_2 + Am_t) \cdot (26.5) + 33 \cdot Am_{lc} = 33 \cdot B_{y1} \qquad \text{(D.2)}$$

Now we can substitute the values from Table 57Table 54 into this equation and flip the equation.

$$33 \cdot B_{y1} = (110000 + \ 32550) \cdot 12 + 2 \cdot 10800 \cdot 16.5 + (20380 + 8450) \cdot (26.5) + 33 \ \cdot (4000) \qquad \text{(D.3)}$$

We can write this out and solve for B_{y1} to obtain the vertical support reaction for support B.

$$B_{y1} = \frac{2962995}{33} = \mathbf{89787.73\ N} \qquad \text{(D.4)}$$

Now the support reaction for point A can be determined by applying vertical equilibrium. For a static structure, the sum of the forces in the Y-direction must be equal to zero. So the sum of the forces in the vertical direction is equal to:

$$\uparrow_+ \sum F_y = F_{Ay} - \left(G_1 + AM_f\right) - 2 \cdot LL \ - (G_2 + AM_t) - AM_{lc} + B_{y1} = 0 \qquad \text{(D.5)}$$

The next step is to substitute all the values in equation D.5.

$$\uparrow_+ \sum F_y = B_{y1} - (110000 + 32550) - 2 \cdot 10800 \ - (20380 + 8450) - 4000 + 89787.73 = 0 \qquad \text{(D.6)}$$

Now all the terms of B_{y1} is switched to the other side of the equation and all the values are multiplied with a factor -1.

$$A_{y1} = 142550 + 21600 + 28830 + 4000 - 89787.73 \qquad \text{(D.7)}$$

$$A_{y1} = \mathbf{107192.27\ N} \qquad \text{(D.8)}$$

Now that the vertical reaction forces in the supports A and B are known, it is possible to apply them in opposite direction. The reason for this choice is that this allows to incorporate the behaviour of the two sections separately, which can be used during the optimization. In the optimization process, the roller bearings are modelled as a roller support, while the vertical support reaction force is applied to the location of the hinges. These are at the tip and the origin of the gangway. This results in the set-up as can be seen in Figure 93 and Figure 94 respectively. The right side of the telescopic section is loaded by the reaction force B_{Y1}. The right side of this section is supported by the two roller bearings.

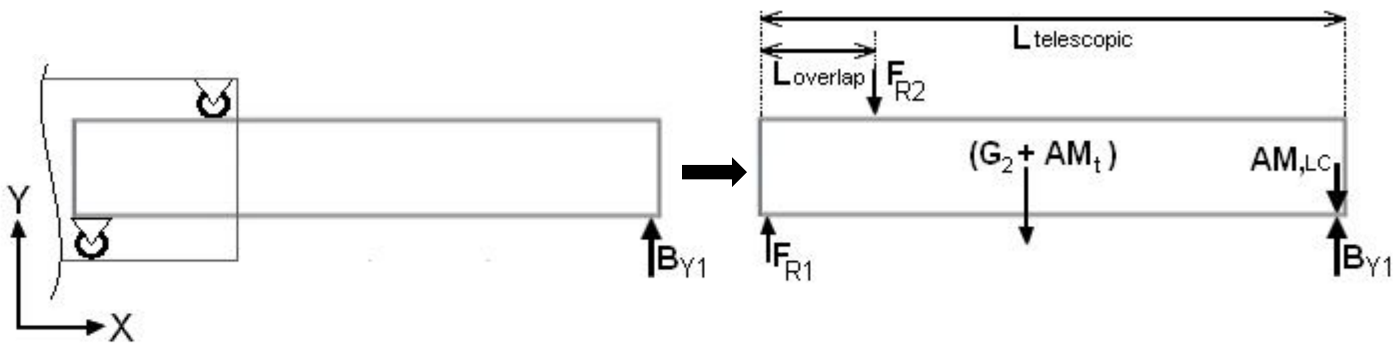


Figure 93: Free body diagram of the telescopic section.

For the fixed boom section the loading conditions are visualized in Figure 94. The left side of the boom section is loaded by the vertical reaction force A_{Y1}. The right side of the gangway is supported by two roller bearings. This support conditions are able to apply a vertical force to the gangway, which results in a counteracting moment. This moment should be equal but opposite to the moment in the telescopic section. This ensures an equilibrium condition and therefore a static situation.

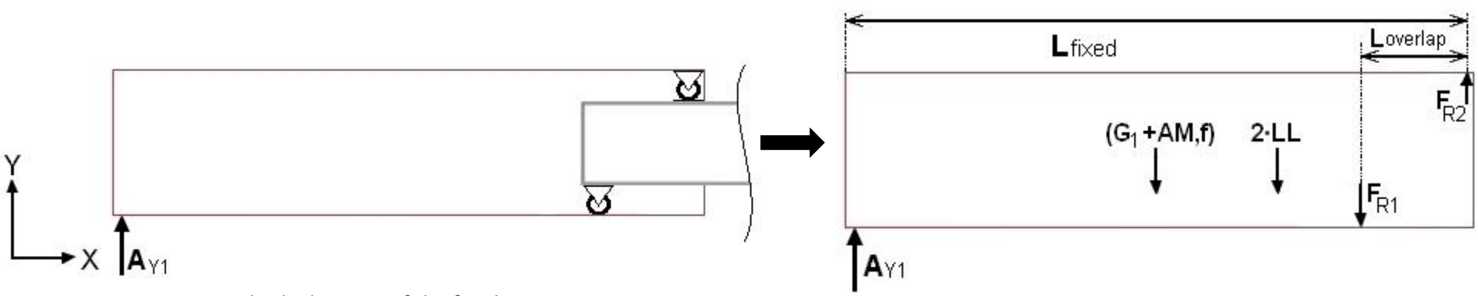


Figure 94:Free body diagram of the fixed section

The reaction forces F_{R1} and F_{R2} can be calculated manually in order to check if the internal forces are equal to zero. These two forces should counteract each other to obey Newtons third law. First we start with the fixed section of the gangway to calculate the reaction forces.

Therefore we can write for the fixed section:

$$\circlearrowright_{+} \sum_{M_{F_{R2}}} = A_{Y1} \cdot \mathrm{L_{fixed}} - (G_1 + AM_f) \cdot (\frac{L_{fixed}}{2}) - 2 \cdot LL \cdot (\mathrm{L_{Fixed}} - \frac{L_{Gangway}}{2}) - 4 \cdot F_{R1} = 0 \quad \text{(D.9)}$$

$$\circlearrowright_{+} \sum_{M_{F_{R2}}} = 107192.27 \cdot 24 - (142550) \cdot (\frac{24}{2}) - 2 \cdot 10800 \cdot (24 - \frac{33}{2}) - 4 \cdot F_{R1} = 0 \quad \text{(D.10)}$$

$$F_{R1} = \mathbf{175003,62\ N} \quad \text{(D.11)}$$

Now we can obtain F_{R1} by applying the vertical equilibrium equation.

$$\uparrow_+ \sum\nolimits_{F_Y} = A_{Y1} - \left(G_1 + AM_f\right) - 2 \cdot LL - F_{R1} + F_{R2} = 0 \tag{D.12}$$

$$\uparrow_+ \sum\nolimits_{F_Y} = 107192{,}27 - 142550 - 21600 - 175003{,}62 + F_{R2} = 0 \tag{D.13}$$

$$F_{R2} = \mathbf{231961{,}35} \tag{D.14}$$

Now it is possible to perform the same procedure for the telescopic section:

$$\circlearrowright_+ \sum\nolimits_{M_{F_{R1}}} = (AM_{LC} - B_{Y1}) \cdot \mathrm{L_{telescopic}} + (G_2 + AM_t) \cdot \left(\frac{L_{telescopic}}{2}\right) - F_{R_2} \cdot L_{overlap} = 0 \tag{D.15}$$

$$\circlearrowright_+ \sum\nolimits_{M_{F_{R1}}} = (4000 - 89787{,}73) \cdot 13 + 28830 \cdot \left(\frac{13}{2}\right) + F_{R_2} \cdot 4 = 0 \tag{D.16}$$

$$F_{R2} = \mathbf{231961.37\ N} \tag{D.17}$$

Now we can obtain F_{R1} by applying the vertical equilibrium equation.

$$\uparrow_+ \sum\nolimits_{F_Y} = F_{R1} - F_{R2} - (G_1 + AM_f) - AM_{LC} + B_{Y1} = 0 \tag{D.18}$$

$$\uparrow_+ \sum\nolimits_{F_Y} = F_{R1} - 231962{,}37 - 28830 - 4000 + 89788 = 0 \tag{D.19}$$

$$F_{R1} = \mathbf{175003{,}64\ N} \tag{D.20}$$

From these results we can conclude that the boundary conditions are well defined and the structure is internally stable. An overview of the calculated forces is shown in Table 58.

Table 58: Results calculation

Force	Description	magnitude [N]
$\mathbf{A_{Y1}}$	Vertical support reaction in point A	1071973
$\mathbf{B_{Y1}}$	Vertical support reaction in point B	89788
$\mathbf{F_{R1}}$	Vertical support reaction force roller 1	175004
$\mathbf{F_{R2}}$	Vertical support reaction force roller 2	231962

The gangway in cantilevered condition

For emergency disconnection the situation in Figure 95 holds. The right side of the gangway boom is unsupported and therefore the weight of the gangway structure must be translated towards the support conditions at the origin of the gangway boom. The LL is located at the gangway tip.

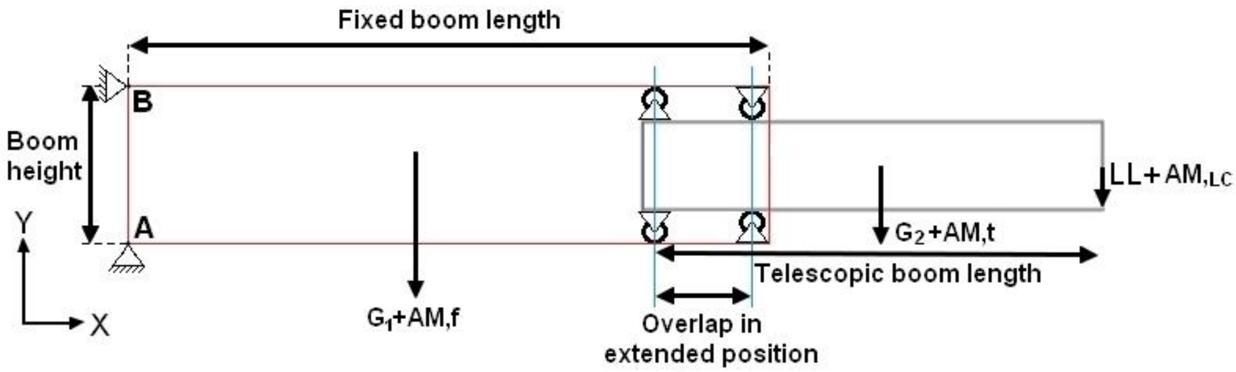


Figure 95: The emergency disconnection condition

The free body diagram of the telescopic part is shown in Figure 97. The beam is supported by two roller bearings. The live load is acting on the tip of the gangway structure and also the self-weight of the telescopic section is taken into account. The dimensions and the magnitude of the loads are visualized in Figure 96.

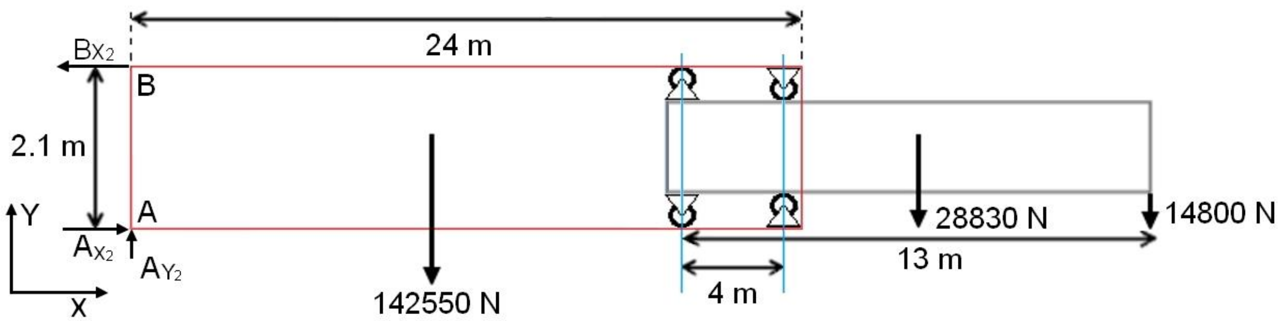


Figure 96: Free body diagram for the emergency disconnection case

The reaction forces in the supports A and B can be calculated by using the moment equilibrium equation. It is chosen to set the moment around point A equal to zero for an equilibrium state. Because the sum of moments about any given point is equal to zero for a static situation. For the sign convention it is chosen that clockwise moments are positive and counter-clockwise moments are negative. Therefore the moment in point A due to the forces acting on the gangway is equal to:

$$\circlearrowright_{+} \sum\nolimits_{M_a} = \left(G_1 + Am_f\right) \cdot 12 + \left(G_2 + Am_t\right) \cdot \left(24 - 4 + \frac{13}{2}\right) + \left(LL + AM_{lc}\right) \cdot 33 - 2{,}1 \cdot B_{X_2} = 0 \quad (D.21)$$

The values from Table 57 can be substituted into the equation and this results in:

$$\circlearrowright_{+} \sum\nolimits_{M_a} = (142550) \cdot 12 + 28830 \cdot 26{,}5 + 33 \cdot 14800 = 2{,}1 \cdot B_{x_2} \quad (D.22)$$

This formula can be solved for B_{x_2} in order to obtain the horizontal reaction force of support B.

$$B_{x_2} = \mathbf{1410950\ N} \quad (D.23)$$

The horizontal reaction force in support A can be determined by using the horizontal equilibrium equation. For a static situation, the sum of the forces in horizontal direction must be equal to zero. Therefore we can write:

$$\rightarrow_{+} \sum_{F_X} = -B_{x_2} + A_{x_2} = 0 \tag{D.24}$$

Now we can substitute the value of $B_{x_2}$and solve for A_{x_2}.

$$A_{x_2} = \mathbf{1410950}\ \boldsymbol{N} \tag{D.25}$$

The vertical equilibrium equation can be applied to obtain the vertical reaction force in support A:

$$\uparrow_{+} \sum_{F_Y} = A_{Y_2} - (G_1 + AM_f) - (G_2 + AM_t) - (LL + AM_{Lc}) = 0 \tag{D.26}$$

Now we can subtitude all the values into this equation and solve for A_{Y_2}.

$$A_{Y_2} = \mathbf{186180}\ \boldsymbol{N} \tag{D.27}$$

Now all the support reaction forces for the emergency disconnection case are known, it is possible to calculate the reaction forces of the roller bearings $F_{R_1}$and F_{R_2}respectively.

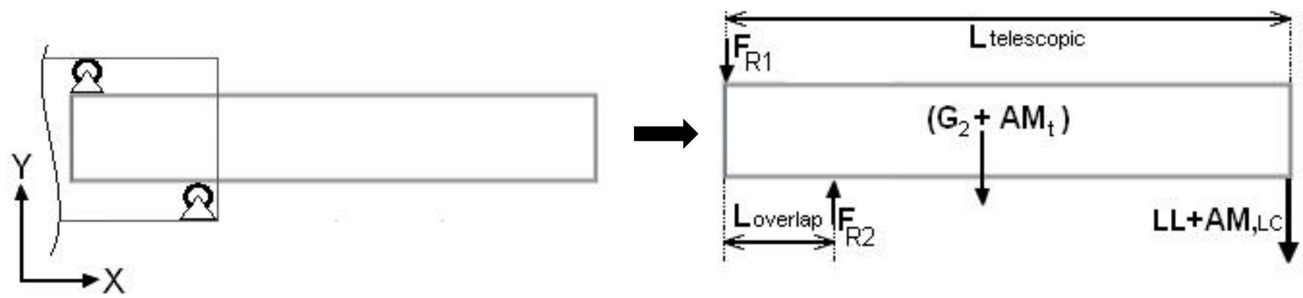


Figure 97: The free body diagram for the telescopic section

The reaction forces can be calculated by using the moment equilibrium. The moment equilibrium is set to zero at the location of the reaction force F_{R_1}.

$$\circlearrowleft_{+} \sum_{M_{FR_1}} = -F_{R_2} \cdot 4 + (G_2 + Am_t) \cdot \left(\frac{L_{telescopic}}{2}\right) + (LL + Am_{lc}) \cdot L_{telescopic} = 0 \tag{D.28}$$

Now we can substitute all the values and solve for F_{R_2}.

$$\circlearrowleft_{+} \sum_{M_{FR_1}} = -4F_{R_2} + (28830) \cdot \left(\frac{13}{2}\right) + (10800 + 4000) \cdot 13 = 0 \tag{D.29}$$

$$F_{R_2} = \mathbf{94948, 75}\ \boldsymbol{N} \tag{D.30}$$

To obtain the value of F_{R_1}, we use the vertical equilibrium equation.

$$\uparrow_{+} \sum_{F_Y} = -F_{R_1} + F_{R_2} - (G_2 + AM_t) - (LL + AM_{Lc}) = 0 \tag{D.31}$$

Substitute the values and solve for F_{R_1} gives:

$$-F_{R_1} + 94948{,}75 - (20380 + 8450) - (10800 + 4000) = 0 \tag{D.32}$$

$$F_{R_1} = \mathbf{51318{,}75\ N} \tag{D.33}$$

In Table 59, an overview is given of all the calculated forces.

Table 59: Calculated forces gangway in cantilevered condition

Force	Description	magnitude [N]
$\mathbf{A_{X2}}$	Vertical support reaction in point A	1410950
$\mathbf{A_{Y2}}$	Vertical support reaction in point A	186180
$\mathbf{B_{X2}}$	Vertical support reaction in point B	1410950
$\mathbf{F_{R1}}$	Vertical support reaction force roller 1	51319
$\mathbf{F_{R2}}$	Vertical support reaction force roller 2	94949

For the deployment and retrieval situation, the gangway is not loaded by a live load located at the tip. Instead of this load, a centrifugal force is acting at the tip which is directed in the longitudinal direction. The reaction forces are not calculated for this load condition, due to the fact that this force is acting in the non-dominant direction. Also it is eight times smaller than the live load. Also the lever arm of the live load to the support conditions is much larger compared to the lever arm of the centrifugal force. Therefore it is considered that these support reactions are lower compared to the emergency disconnection case.

A-7. The ABS-guidelines

The ABS-guidelines differs in a small extent from the DNV-guidelines. The differences between these two guidelines will be covered in this chapter. The ABS guidelines makes also a distinction between type 1 and type 2 gangways and between passive and active controlled gangways. The ABS guidelines have specified minimum thicknesses for solid and hollow sections which are critically stressed. The values are shown in Table 60.

Table 60: Minimum thickness according to the ABS-guidelines

Type of stress	Type of section	Minimum thickness
Critically stressed	Solid	6 mm
Critically stressed	Hollow	4 mm
Not-critically stressed	Hollow or solid	4 mm

Special protective coatings are to be applied at structural members of offshore access gangways where the thickness is less than 6 mm. In case of an aluminium construction the corrosion protection must be adequate to deal with marine conditions.

The materials must follow the international standards for the design and fabrication. The use of aluminium alloys will be considered upon submission of the proposed specification for the alloy and the method of fabrication.

Load conditions

The load conditions according to the ABS-guidelines are quite similar to the load conditions of the DNV-guidelines. For a type II gangway, when the gangway is supported on both ends, the minimum design load must be equal to two times the maximum number of persons including their carry-on equipment on the gangway. The live load is concentrated at the middle span.

For a gangway designed to be supported at both ends, when it is extended to its maximum operational length, a test load that is equal to the maximum design live loads (not less than 2.4kN multiplied by a dynamic amplification factor) is to be applied in the middle of the gangway. The dynamic amplification factor is not to be taken less than 1.25. For emergency lift-off conditions, a minimum live load of 3.5 kN must be applied on the gangway tip. For the ice and snow loads, a uniformly distributed load of 490 N/m2 is considered if applicable.

The following loads are considered according to the ABS guidelines: Dead loads, live loads, motion-induced loads, functional loads, wind, ice and snow loads, impact loads, accidental loads and miscellaneous loads. These loads are considered for different loads cases which can be found in Table 61. This results in 10 different load cases in which X indicates that the load is to be applied in the loading conditions.

The dead loads are the self-weight of the gangway including all the equipment which is attached to it. The live load of the gangway is the load on the gangway due to the people on the gangway. This is depending on the number of persons allowed on the gangway. The motion-induced loads are produced by the motions of the vessel on which the gangway is installed and the relative motions between vessels or units when the gangway is deployed. The functional loads are loads due to the dynamic effects of lifting, lowering, slewing, telescoping and landing the gangway. The wind, ice and snow loads are additional loads due to the wind, ice of snow acting on the gangway. The impact loads results from green water. It is considered that there will be no direct wave loading on the gangway during the operational condition. The accidental loads are caused by: collision, dropped objects, fire or blasts. Accidental loads are to be determined on the bases of risk assessment. Finally the miscellaneous loads include the loads resulting from tie-downs or lashing used to secure the gangway in its stowed positions for severe storm and transit conditions.

Table 61: Load cases according to the ABS-Guidelines

Design conditions		**Dead loads**	**Live loads**	**Motion-induced loads**	**Functional loads**	**Wind loads**	**Ice and snow loads**	**Impact loads**	**Accident-al loads**	**Misc. loads**
Operating conditions	*Deployment*	X		X	X	Transit wind				X
	Operating static	X	X				X			X
	Operating combined	X	X	X	X	X	X			X
	Retrieval	X		X	X	Transit/ storm wind				
	Unexpected lift-off	X	X	X	X	X				
Transit condition		X		X		Transit wind	X	X		X
Severe storm condition		X		X		Storm wind	X	X		X
Accidental conditions	*Emergency lift-off*	X	X	X	X	X				X
	Damaged	X	X	X	X	X			X	X
	Impact	X	X	X	X	X		X		

Strength assessment

The different parts of the structure needs to be assessed using linear elastic methods to determine the adequacy of the structure. The structural strength assessment is shown in Table 62.

Table 62: Structural strength assessment.

		Yielding check	Buckling check	Ultimate strength check	Fatigue check
Local structures	Plating	X	X	X	-
	Stiffeners	X	X	X	-
Primary structural members		X	X	X	X
Hull interface structure		X	X	X	X

The design acceptance criteria are concerned with four limit states as follows:

- Accidental limit states (ALS) to verify the survival of the structure when subjected to anticipated accidental and damaged conditions.
- Ultimate limit states (ULS) to resist yielding, buckling and ultimate strength.
- Fatigue limit state (FLS) to resist fracture from cyclic load effects.
- Serviceability Limit State (SLS) to address the structural deflections of the gangway.

The application of the allowable stress assessment (ASD) criteria requires the determination of representative allowable stresses for individual components. Allowable stresses are not to be exceeded for the type of component and loading condition being considered. The allowable stress coefficient S_c determines the amount of the minimum yield stress of the material which may be applied for different types of stressed sections. The calculation of the buckling strength and fatigue assessment criteria is in the same way as the DNV requirements.

The deflection

The serviceability limit state depends on the maximum deflection of the gangway. The relative deflection of the gangway, δ_{max}, in operating condition, may not exceed the following criteria:

$\delta_{max} \leq L/200$ for a gangway designed with both ends supported

In which L is the design length of the gangway and δ_{max} is the maximum relative vertical or lateral deflection. Two test cases are analysed, with and without personnel carried in uplift position. The test load is to be applied at the extended end of the gangway.

A-8. Overview different manufacturers

This chapter gives an overview of the different manufacturers of motion compensated gangway structures. The differences between these manufacturers is shown in Table 63. In total there are 16 different manufacturers which produce motion compensated gangways. This is excluding IHC which is developing a motion compensated gangway at this moment. There is some difference between the different manufacturers in the compensation techniques and the specifications. The Ampelmann types which uses the gough-stewart platform are not included in this overview.

Table 63: Overview different manufacturers

Company Name	***Transfer system***	***Operational conditions (Hs)***	***Material gangway***	***Gangway length [m]***	***Compensation technique***	***Landing mechanism***	***Company location***
Ampelmann	S-type	3.0 m	Aluminium	25	AMC	RB	Netherland
Barge master	BM 4.5	4.5 m	Unknown	25	MCP	Unknown	Netherland
DL-marine	G25	Unknown	Steel	25	PMC	LCC	China
Ferri	AHC 38	4.0 m	Steel	32	PMC	RB	Spain
F-E-T	PTB	Unknown	Steel	50	AMC	LCC	America
Houlder	PTS	3.0 m	Aluminium	23	AMC	LF	England
Kenz	GW-15/25	3.0 m	Unknown	25	AMC	LCC or RB	Netherland
Kongsberg	K-walk	3.5 m	Steel	24	TPS	Unknown	Norway
Motus	MWW GW	3.5 m	Aluminium	30	AMC	LCC	Norway
Ossbit	Maxaccess	Unknown	Unknown	25	AMC	GE or LF	England
Van Aalst	Safeway	3.5 m	Unknown	28.5	TPS	LCC or RB	Netherland
SMST	xl-standard	Unknown	Steel	36	AMC	LF or RB	Netherland
SubC	OAS	Unknown	Aluminium	25	AMC	Unknown	Denmark
Tensa	marine WG	3.0 m	Aluminium	12	AHC	RE or GE	Norway
Uptime	GW 23,4 m	3.5 m	Aluminium	23.4	AMC	LCC or RB	Norway
Zbridge	Zbridge	4.5 m	Aluminium	Unknown	TPS	RB	Netherland

AMC= Active motion compensation
MCP= Motion compensated pedestal
PMC= Passive motion compensation
TPS= Tilted pedestal in combination with active motion compensation
RB= Rubber bumper
LCC= Landing cone connection
LF= Landing foot
RE= Roller end
GE= Gripper end

A-9. Benchmark problem

In order to verify the proposed approach and to investigate its stability, a benchmark problem is performed. The results of the proposed approach are compared with the results of these benchmark solutions found in previous work. For optimization problems, it is common to validate the set-up of your model by performing a benchmark test. Standard test problems are available which are commonly used for this purpose. The example that is used for this problem is the ten bar plane truss which is shown in Figure 98. This example is often used as benchmark problem in structural optimization.

10-bar plane truss

This benchmark problem is frequently found in literature which is related to plane truss optimization. This structure consists of 6 nodes which are connected by 9 bar elements. The structure is supported by two hinges with a vertical distance of 9.144 meters. The horizontal and vertical spacing between the nodes is equal to 9.144 meters. At the two free nodes (node 2 and 4), a vertical load of 444,82 kN is applied. The deflection of the four free nodes is constrained to 50,8mm (2 Inch) in the positive and negative direction. The members are made out of aluminium with a Young's modulus equal to 68.9 GPa and a specific mass of 2770 kg/m^3. The allowable minimum area for the cross-section of the members is equal to 64,5 mm^2. The element stresses are limited to 172.37 MPa for compression and tension. Buckling is ignored in this situation. The dimensions, node numbers, element numbers and loads acting on this structure are shown in Figure 98. The properties for the single load case are shown in Table 64. Ten design variables are considered for the optimization, namely for the cross section area of each bar element.

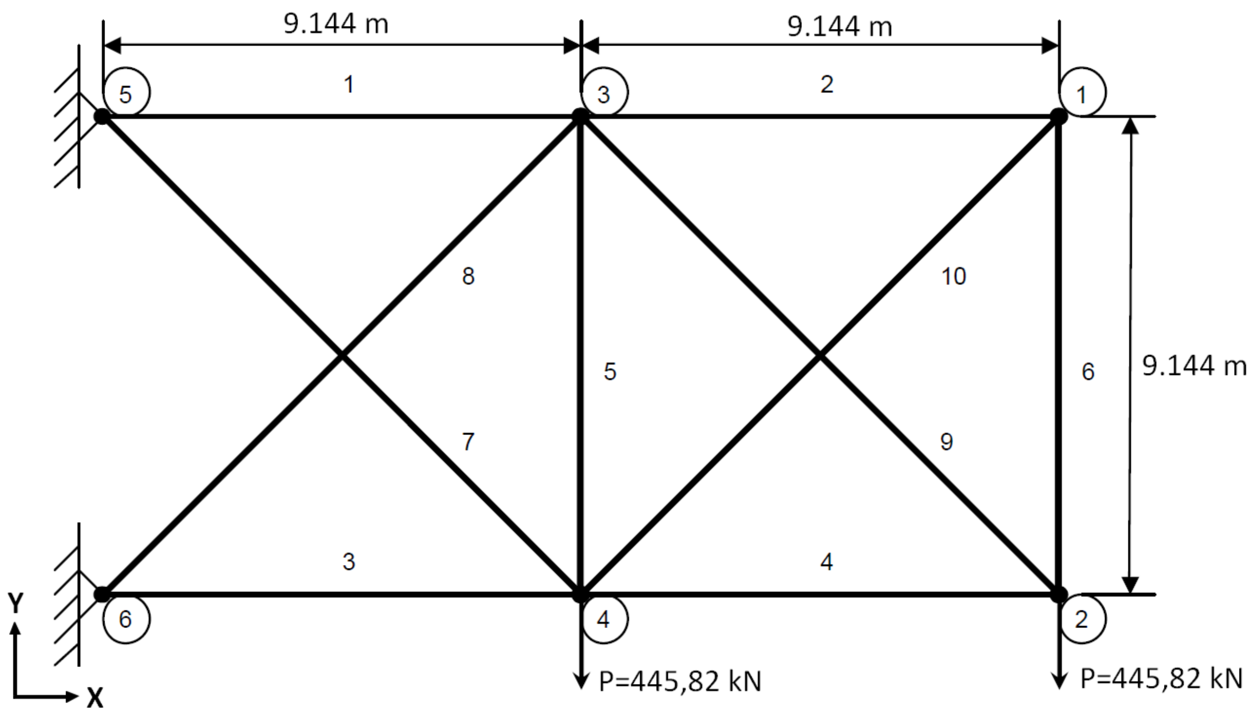


Figure 98: 10-bar truss

Table 64: Input parameters benchmark problem

Property	Value	Dimension
Horizontal spacing	9144	mm
Vertical spacing	9144	mm
Load P	444820	N
Deflection constraint	50.8	mm
Young's modulus	68900	MPa
Density	2.77E-9	T/mm^3
Element stresses	172,387	MPa
Min cross sectional area	64.52	mm^2
Max cross sectional area	120932	mm^2

The units in Table 64 are chosen according to the unit consistency table which is prescribed by Hyperworks. The 10-bar truss structure is optimized for minimization of mass for a single load case and stress constraints applied to every member. Displacement constraints are applied to the unsupported nodes of the structure. In this benchmark problem, the cross sectional areas of the elements are the design variables for the optimization.

The objective of the optimization is to minimize the total mass of the structure. Nodes 1 till 4 are constrained by a vertical displacement of 50.8mm in the positive and negative direction. The allowable stresses in the members are constrained up to 172.37 MPa. This means that the tension and compressive stresses may not exceed this value. The responses of the structure are the member stresses, the vertical displacements of the nodes 1 till 4 and the total mass of the structure.

Set-up in Optistruct
This problem is defined in Hypermesh and afterwards optimised with Optistruct. The set-up in Hypermesh consists of the following steps:

- -1. Create basic geometry.
- -2. Define material properties.
- -3. Create design variables for all bar elements.
- -4. Define properties for all bar elements.
- -5. Create generic section properties for all bar elements.
- -6. Define properties for all bar elements.
- -7. Create 1-D line mesh for the truss.
- -8. Create load collectors for the loads and support conditions.
- -9. Create load step to prepare for the analysis.
- -10. Define responses, constrains and objective.

In Hypermesh the basic geometry is defined by generating 6 nodes with an internal grid spacing of 9144 mm. The truss structure is defined by connecting these nodes with line elements. Afterwards the nodes can be deleted. A material is defined for the rod elements with the corresponding material properties. For this benchmark problem, aluminium is used as construction material. The next step is to define the section properties for the line elements. In Hyperbeam, a generic section is selected for the standard rod section type. 10 sections are generated for each rod element. Rod elements only allow tension and compression forces and do not allow bending. The next step is to define a property for each bar element in which the elements are related to the material and beam section. It is important to select PROD as card image. Now the 1-D line elements can be meshed and assigned to the corresponding property. Then the meshed elements can be updated to CROD elements. 10 different design variables are generated in order to allow the variation of the cross section area of the members. In these design variables the minimum, maximum and initial cross sectional area of the members is specified. After this step, it is possible to define the generic relationship for the 10 bar elements. This generic relationship relates the element properties to the design variables.

The loads and support conditions can be applied to the model. Therefore two load collectors are generated for the loads and support conditions. When the current load collector is selected, forces and boundary condition can be applied to the desired nodes. Design variables for the sizing optimization are defined for each rod element. In these design variables the type of optimization, subjected elements, upper and lower bounds are defined. Two support conditions are applied at the nodes 5 and 6 which only allows rotation around the Z-axis and constrains all the other translations or rotations. Therefore these supports can be considered as hinged supports. The next step is to define the loads on the structure, therefore two point loads are applied to node number 2 and 4 with a magnitude of 445,82 kN in the negative Y-direction. The objective of the optimization is to minimize the total mass of the structure. Therefore responses of the structure are required in order to apply constraints on the structure. Responses must be generated for the member stresses, vertical displacement of nodes 1 till 4 and the total mass of the structure. Constraints can be applied to the member stresses for each member and displacement constraints can be applied for nodes 1 till 4. Finally the objective can be set to minimize the mass of the structure. Now the model is ready for solving and the set-up of the model can be found in Figure 99.

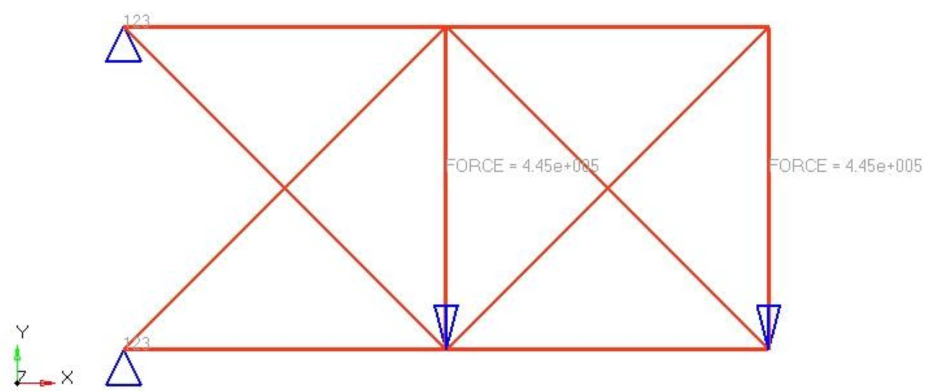


Figure 99: Model set-up in Hyperworks

Comparison of results

For the analytical solution is referred to the work of Haftka r.t, Gurdal Z. [60] in which the analytical solution of the ten bar truss structure is applied. In Table 65 the member sizes are given from the analytical solution. In the second column the values from the size optimization from Optistruct are shown.

Table 65: Comparison benchmark solution

Member	Analytical solution [mm]	Optistruct solution [mm]	Discrepancy [%]
1	19691.44	19810	0.6
2	64.52	70.25	8.2
3	14967.65	15510	3.5
4	9821.21	9529	-3.1
5	64.52	64.52	0
6	355.74	64.52	-451
7	4811.09	5495	12.4
8	13571.84	13520	-0.4
9	13889.26	13380	-3.8
10	64.52	64.52	0
Total mass	2299.65 kg	2303.9 kg	0.18

A minimum mass of 2303.9 kg was obtained by using the size optimization algorithm from Optistruct. The vertical displacement constraint of -50.8 mm is active at nodes 3 and 4. Member 5, 6 and 10 reaches the minimum cross sectional area and member 2 approaches this limit. The largest stress can be found in member 5 which was equal to 140 MPa. The discrepancy between the objective of the analytical solution and the Optistruct solution is 0.18%. This is quite low and the difference can be referred to the fact that the optimizer is approaching the optimum solution. The optimization parameters and applied algorithm will influence the final value of the objective function and the values of the design variables.

The iteration history is shown in Figure 100. After 27 iterations the objective function was converged to a final value of 2.3039 tonnes while the constrain violation was less than 0.1%. Therefore the optimization process was terminated because all the constraints were satisfied. The processing time was 8 seconds for which the CPU time was only 1 second.

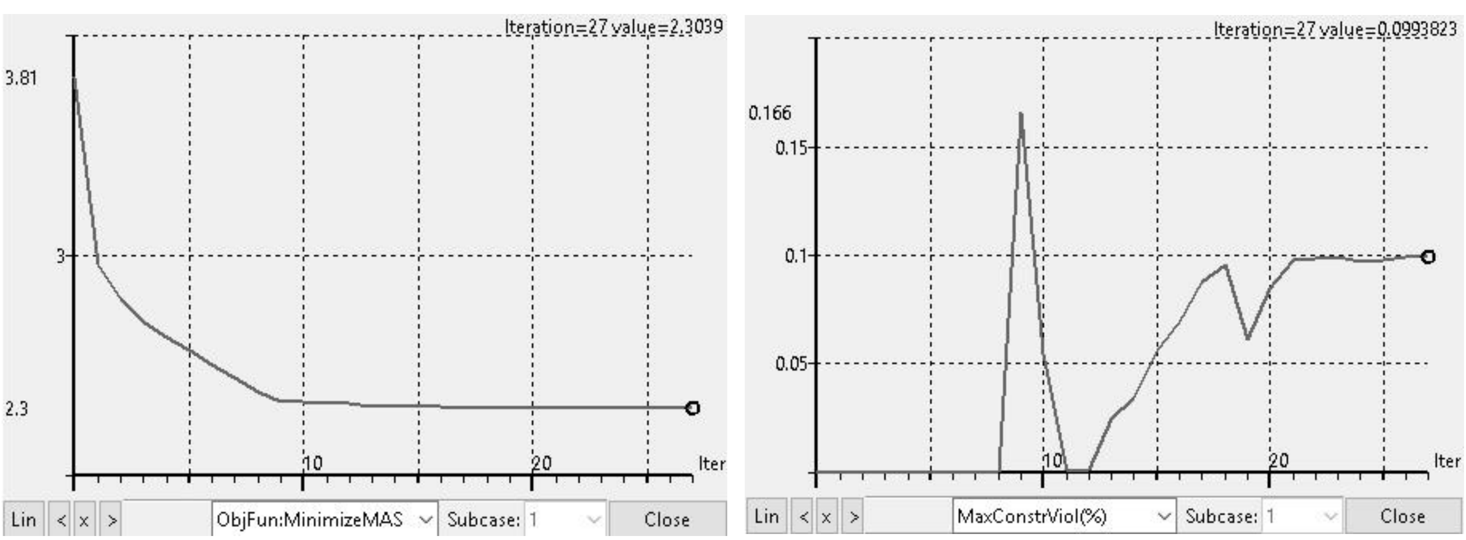


Figure 100: Iteration history for the objective function & constrain violations

A-10. Cross section classifications

For thin-walled hollow sections, it is very difficult to determine the local buckling behaviour. Especially for the interaction between global and local buckling. This is due to the local instability behaviour of thin walled cylindrical shells, which have a high susceptibility to imperfections. This causes a sudden reduction of the load bearing capacity. Local buckling has to be considered for CHS when the d/t limits for the cross sections in Table 67 are exceeded. The same holds for RHS when the b/t and h/t ratios are exceeded. These cross-section profiles are visualized in Figure 101. The problem of axial loaded beams is that the increase of the column slenderness decreases the allowable compression force which can be applied to the column. The optimization of the buckling behaviour of hollow sections, leads for a constant value of the cross sectional area, to profiles of large dimensions and small thicknesses in order to have a large moment of inertia which increases the torsion resistance and bending stiffness of the profile. These small thicknesses relative to the outer dimensions can cause failure by local buckling. Imperfections in the profiles causes a decrease for the resistance to global and local buckling.

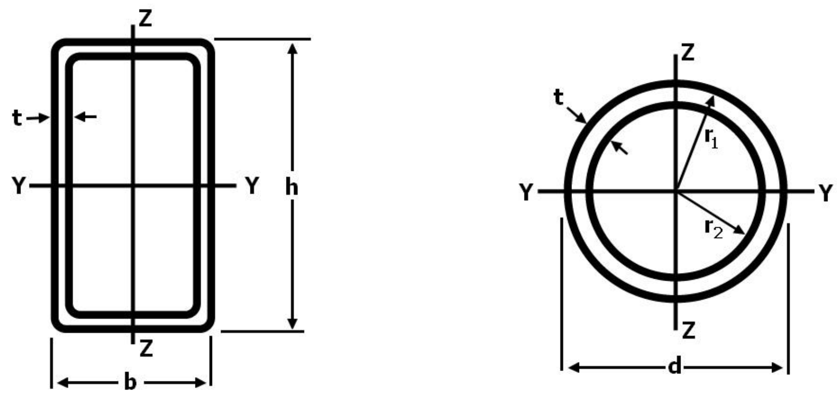


Figure 101: Hollow cross section dimensions

By keeping the profile dimensions within the specified ratios from Table 67, it is not required to check the structure for local buckling. The phenomena of local buckling, becomes more critical when materials with a higher yield strength are applied, therefore smaller ratios must be applied. These values are obtained from the Eurocode 3 which takes account of local buckling by determining the load bearing capacity by making use of effective cross section dimensions which are smaller than the real ones. In practice, circular hollow sections do not possess d/t ratios which exceeding the values given in Table 67. In general a value of C=50 is applied, which corresponds to a class 1 cross section classification. For these profiles, full plasticity is developed in the cross-section. Four different design methods, can be applied for determining the ultimate limit state design of the members. These design methods are given in Table 66.

Four classes of cross-sections are defined:

Class 1: Cross-sections are those which can form a plastic hinge with the rotation capacity required from plastic analysis with the reduction of resistance. Full plasticity is developed in the cross-section.

Class 2: Cross-sections are those which can develop their plastic moment resistance, But have limited rotation capacity because of local buckling. The Ultimate limit state is achieve by the formation of the first plastic hinge.

Class 3: Cross-sections are those in which the stress in the extreme compression fibre of the member assuming an elastic distribution of stresses reach the yield strength, but local buckling is liable to prevent development of plastic moment resistance.

Class 4: Cross-sections are those in which local buckling will occur before the attainment of yield stress in one or more parts of the cross-section.

Table 66: Cross section classification

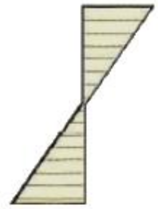
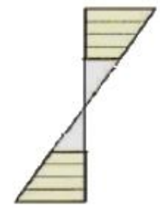
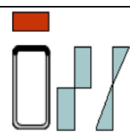
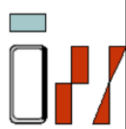
Cross section classes	Class 1	Class 2	Class 3	Class 4
Load resistance capacity	Full plasticity in the cross section full rotation capacity	Fully plasticity in the cross section restricted rotation capacity	Elastic cross section yield stress in the extreme fibre	Elastic cross section local buckling to be taken into account
Stress distribution and rotational capacity				
Procedure of the determination of stress resultants	**Plastic**	**Elastic**	**Elastic**	**Elastic**
Procedure of the determination of the ultimate resistance capacity of a section	**Plastic**	**Plastic**	**Elastic**	**Elastic**

Table 67 gives the slenderness limits d/t, b/t and h/t for the different cross-section classes based on the Eurocode 3. Other design codes are showing slightly different values.

Table 67: Limiting ratios for hollow sections

			Class	1				2				3			
				f_y (N/mm²)				f_y (N/mm²)				f_y (N/mm²)			
Cross section	Load Type	Considered element		235	275	355	460	235	275	355	460	235	275	355	460
RHS b/t	Compression	Top face		$\frac{b}{t} \leq c \cdot \sqrt{\frac{235}{f_y}} + 3$											
				C=33				C=38				C=42			
				36.0	33.5	29.8	26.6	41.0	38.1	33.9	30.2	45.0	41.8	37.2	33.0
RHS h/t	Bending	Side wall		$\frac{h}{t} \leq c \cdot \sqrt{\frac{235}{f_y}} + 3$											
				C=72				C=83				C=124			
				75.0	69.6	61.6	51.8	86.0	79.7	70.5	62.3	127.0	117.6	103.9	91.6
CHS d/t	Compression and/ or bending			$\frac{d}{t} \leq c \cdot \frac{235}{f_y}$											
				C=50				C=70				C=90			
				50.0	42.7	33.1	25.5	70.0	59.8	46.3	35.8	90.0	76.9	59.6	46.0

The next step is to calculate the required wall thickness for a certain tube dimension d. It is chosen to define a cross-section according to class 1. This means that full plasticity is developed in the cross section before failure occurs and therefore this class allows for large deformations before failure. When cross-section class 1 is applied, then it is not required to check the structure for local buckling. The structure should still be checked against local buckling. The cross-sections subsumed in class 1 allow the complete utilization of the plastic reserve of the structure and the plastic reserve of the cross-section. The optimizer is allowed to determine the outer diameter of the CHS within the defined bounds. The wall thickness is determines as a function related to the outer dimension or radius of the CHS.

From Table 67, it can be seen that a CHS class 1 which is in compression and/or bending has a d/t value which can be calculated with formula A-10.1:

$$\frac{d}{t} \leq c \cdot \frac{235}{f_y} \tag{A-10.1}$$

For a circular hollow section in class 1, the value of C is equal to 50 and the yield strength of S355 structural steel is equal to 355 N/mm2. These values can be substituted in equation A-10.1 which results in:

$$\frac{d}{t} \leq 50 \cdot \frac{235}{355} = 33.1 \tag{A-10.2}$$

The optimizer uses the radius instead of the diameter. The diameter is equal to twice the outer radius and therefore we can write:

$$\frac{2r_1}{t} \leq 33.1 \tag{A-10.3}$$

This equation can be rewritten in order to obtain the value of r_1 as a function of t:

$$\frac{r_1}{t} \leq 16.55 \tag{A-10.4}$$

$$r_1 \leq 16.55t \tag{A-10.5}$$

Or we can write this in terms of t:

$$t \geq \frac{20}{331} r_1 \tag{A-10.6}$$

If we want to define the inner diameter r_2 as a function of the outer diameter r_1, we can write:

$$r_2 = r_1 - t \tag{A-10.7}$$

The ≥ is replaced by a = sign due to the fact that the wall thickness should be at least equal to this value. Because the goal of the size optimization is to obtain the solution with the lowest weight, it is allowed to choose the minimum wall thickness which is required to prevent local buckling.

Now we substitute the value of t into this equation:

$$r_2 = r_1 - \frac{20}{331} r_1 = \frac{311}{331} r_1 \tag{A-10.8}$$

Now it is possible to define a design equation for the inner radius as:

$$Inner\ radius\ r_2 = \frac{311}{331} r_1 \tag{A-10.9}$$

This design equation will be used in the sizing optimization. This ratio will be included in the design equation which automatically relate the inner radius to the outer radius. By applying the ratio this will enable a design which is not susceptible for local buckling.

The four main horizontal beams consists of a rectangular hollow section. For this section it is also possible to define limiting ratios for the wall thickness to the height and width of the profile. The height of the profile is denoted with h and the width of the profile is denoted with b as can be seen in Figure 101. First we relate the width b of the profile to the wall thickness t.

From Table 67 it can be seen that for a RHS class 1 which is in compression has a b/t value which can be calculated with formula A-10.10:

$$\frac{b}{t} \leq c \cdot \sqrt{\frac{235}{f_y}} + 3 \tag{A-10.10}$$

For a RHS in class 1, the value of C is equal to 33 and the yield strength of S355 structural steel is equal to 355 N/mm2. These values can be substituted in equation A-10.10 which results in:

$$\frac{b}{t} \leq 33 \cdot \sqrt{\frac{235}{355}} + 3 = 29.8 \tag{A-10.11}$$

This equation can be rewritten in terms of t.

$$t \geq \frac{5}{149} b \tag{A-10.12}$$

Now the internal width of the RHS can be written as:

$$b_1 = b - 2t \tag{A-10.13}$$

When the value of t is substituted into this equation this results in:

$$b_1 = b - 2 \cdot \frac{5}{149} b = \frac{139}{149} b \tag{A-10.14}$$

Now the same procedure can be performed to determine the relation of the wall thickness to the profile height h. For a RHS in class 1 which is subjected to bending, the value of C is equal to 72 and the yield strength of S355 structural steel is equal to 355 N/mm2. These values can be substituted in equation A-10.10 which results in:

$$\frac{h}{t} \leq 72 \cdot \sqrt{\frac{235}{355}} + 3 = 61.6 \tag{A-10.15}$$

This equation can be rewritten in terms of t.

$$t \geq \frac{5}{308} h \tag{A-10.16}$$

Now the internal height of the RHS can be written as:

$$h_1 = h - 2t \tag{A-10.17}$$

When the value of t is substituted into this equation this results in:

$$h_1 = h - 2 \cdot \frac{5}{308} h = \frac{149}{154} h \tag{A-10.14}$$

If we want to have an equal wall thickness along the profile cross-section, then we must relate the wall thickness obtained from the height of the RHS to the wall thickness at the width of this section. Therefore we can write:

$$\frac{5}{308}h = \frac{5}{149}b \tag{A-10.15}$$

The ≤ sign disappears due the fact that the value of t must be equal along the cross section and the goal is to minimize the mass of the structure, therefore the minimum wall thickness which is required to avoid local buckling is allowed. Now we can relate the height of the RHS to the width of the section.

$$h = \frac{5}{149} \cdot \frac{308}{5}\ b = \frac{308}{149}b \approx 2 \tag{A-10.15}$$

Or this can be rewritten in term of the width b:

$$b = \frac{149}{308}\ h \approx 0.5 \tag{A-10.16}$$

This means that the height of the cross-section is approximately twice the width of the structure. This is a common ratio for rectangular cross-sections. The design equations for the optimization are defined in Table 68.

Table 68: Design equations size optimization

Cross section	**Design parameter**	**Design relation(s)**	**Formula**
CHS	Outer radius, r_1	Inner radius, r_2	$Inner\ radius, r_2 = \frac{311}{331}r_1$
RHS	Profile height, *h*	Profile width, *b*	$External\ width, b = \frac{149}{308}h$
		Wall thickness, *t*	$Wall\ thickness, t \geq \frac{5}{308}h$

Now all the design equations are generated which incorporate the minimum wall thicknesses required in order to avoid local buckling. The global buckling behaviour should be examined with a hand calculation and/or FEA.

A-11. Mesh convergence check

Mesh size is an important aspect in FEA. The element size determines the quality of the output but it is also related to the required solving time. Usually smaller mesh elements means more accurate results, but the solving gets significant as well. Therefore a balance should be found between the mesh size and the required computational effort. When a finite element analysis is performed, it is important to perform a mesh convergence check. A mesh convergence check relates the element size to the output result. A mesh convergence check is used to determine the required mesh size. It relates how small the mesh needs to be to ensure the results of the FEA are not affected by changing the size of the mesh. A mesh can be considered as converged when two subsequent mesh refinements do not change the result substantially.

In some cases, it is possible to check your solution with the analytical solution. The analytical solution is a solution to the problem which can be derived exactly and therefore the solution is known. Unfortunately the analytical solution is not always available or known. Therefore other approaches are available to check the mesh convergence. A method which can be applied is to check the node count. A chart is generated in which the outcome of the analyses is written on the vertical axis and the amount of nodes in the model is written on the horizontal axis. Reduction of the element size leads to more elements which also leads to more nodes in the model. If different values are plotted in this chart, then this chart will show the dependence of the outcome on the amount of nodes in the model. The line will asymptotically approaching the correct answer. Instead of node count, element size can be used for determining the relation between the outcome of the solution. Node count is the most used method because it is the easiest one to do.

A mesh convergence check is performed for the size optimization. An analysis is performed for several different mesh sizes. The mesh size is reduced stepwise from 150 mm to 3.9 mm. Every step the mesh size is reduced by a factor 1.5. Which is a common rule for mesh convergence studies. The maximum stresses in the elements are related to the amount of nodes in the model. The mesh size is related to the amount of nodes in the model. The steps for decreasing the mesh size are given in Table 69. In which the maximum von misses stress is given for each run.

Table 69: Mesh size reduction steps

Step nr.	Node count [-]	Mesh element size [mm]	Maximum stress [MPa]
1	715	150	193.2
2	1085	100	193.2
3	1625	66.67	194.5
4	2459	44.44	187.1
5	3704	29.63	184.4
6	5570	19.75	181.1
7	8363	13.17	179.9
8	12552	8.78	179.2
9	18844	5.85	179.1
10	28281	3.9	178.7

Now it is possible to plot these values in a graph. In Figure 102 the node count is plotted to the amount of nodes in the model. This shows the relation between the mesh size and the output of the FEA, which is in this case the maximum stress in the members.

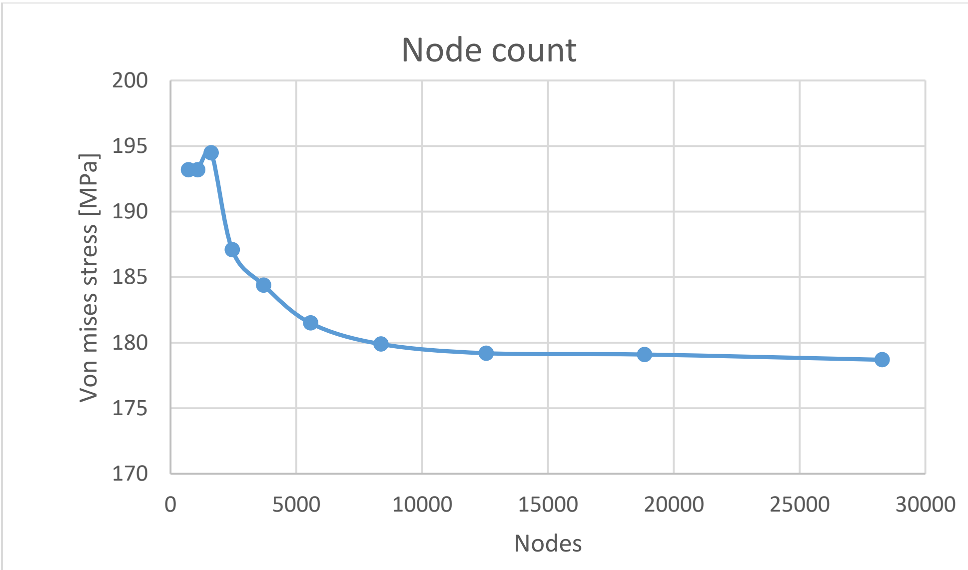


Figure 102: Node count versus von misses stress

The maximum stress for a mesh size of 100 mm, which corresponds to 1085 nodes, is lower than the successive value. The same value is obtained for a mesh size of 150mm. Therefore it is considered that the mesh size is not refined sufficiently and therefore these two mesh sizes are neglected in the further analysis. The exact asymptotic value from the node count may be difficult to obtain. It is somewhere below the 180 MPa. Instead of the node count, the values on the horizontal axis are replaced by 1/node count. Then the correct answer is at the location where the horizontal axis reaches zero. This value can be easily calculated by doing a linear approximation of this line and to determine the value for X=0. The linear approximation of this line is equal to equation A-11.1

$$Y = 26938x + 177{,}13 \tag{A-11.1}$$

For X=0, the von mises stress is equal to 177,13 MPa. Now the approximated answer is known, it is possible to estimate the magnitude of the error compared to the computational time or mesh size. This is shown in Figure 104.

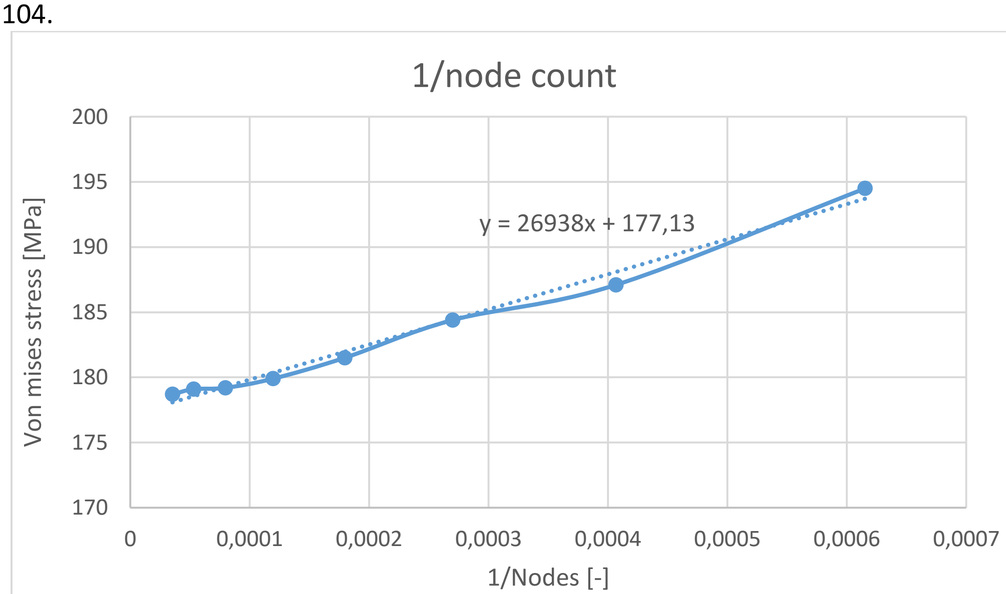


Figure 103: 1/node count versus von mises stresses

From the graph in Figure 104 it is easy to notice that after a certain mesh size the error will be reduced significantly. This graph gives the errors related to the mesh size. Now it is possible to determine which error is acceptable and what mesh size is required for the analysis. A convergence criterion can be defined which gives the allowance of the error. With this criterion, the required mesh size can be determined which is required to satisfy this criterion. So now we know the accuracy of our results for a given mesh. It is also possibility to generate a plot in which the solving time is plotted against the error. In this case the required solving time can be determined which is required to satisfy the convergence criterion. For this situation this is not implemented due to the fact that the 1D problems are solved within a very short period of time.

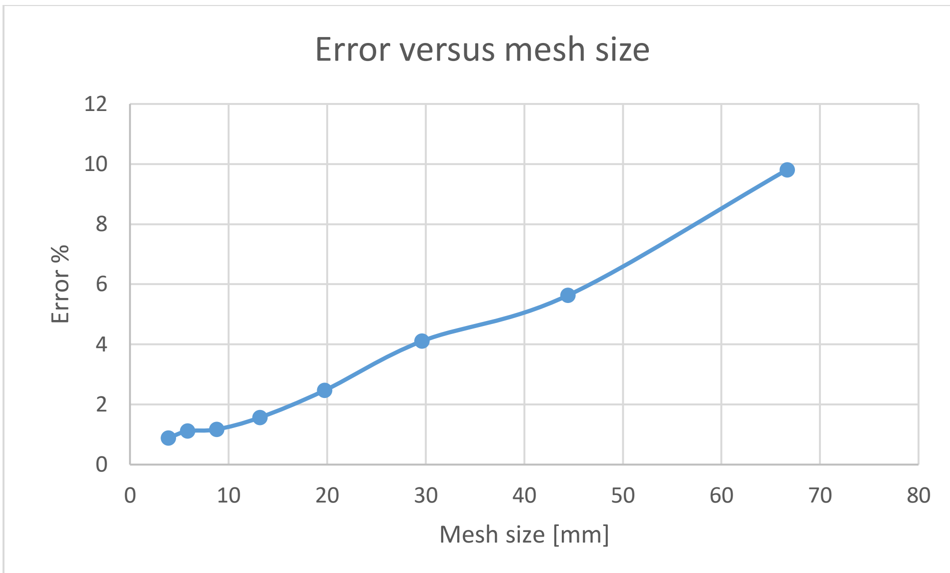


Figure 104: Error versus mesh size

A-12. Offshore access methods

90% of all the maintenance activities at an offshore wind farm consists of small operations [15]. These are for example inspections or small repairs. Therefore an increased accessibility for crew and small equipment would result in a large increase in availability. The accessibility of a wind turbine depends on the type of transport towards the turbine location and the method of transferring crew and equipment to the turbine. At this moment there are several methods of transferring crew and cargo from and to a wind turbine. The most common types of transport that are used to reach offshore structures are vessels and helicopters.

Helicopter access

For fast transport of personnel and light equipment helicopters are used. They have cruise speeds up to 290 km/h and they are not limited by the wave conditions. If a helicopter landing deck is available, the helicopter can land and the passengers can board and exit the helicopter safely. But for offshore wind turbines this is unpractical and expensive. Therefore a hoisting platform is located on the nacelle and the passengers are transported from the hovering helicopter by a rope towards the hoisting platform. This method is fast but expensive. A hoisting platform is required on each wind turbine and a helicopter is required. Also there is a safety risk, in case of a crash the probability of casualties is high. The accessibility of wind turbines by helicopters is limited by the wind speeds and visibility. Another limitation is that helicopter flights are only allowed during daylight.

Figure 105: Access of a wind turbine by a helicopter (fiberline.com)

Vessel-based access

Access from a vessel to an offshore structure is enabled by three different transfer methods. The crew can be lifted from the vessel to the offshore structure by a crane that lifts a personnel basket. But this option is not practical for an offshore wind turbine. Because in heavy weather the ship would make large excitations and the personal basket would swing dangerously. Another option is to use a swing rope to swing from a vessel to a landing platform at the same level. This option is less preferred because it is for safety reasons only restricted to very calm weather conditions. Offshore structures can also be accessed by using a modified vessel which is equipped with bumpers. This type of vessel is specially designed for the transfer of personal and equipment and is called a Crew Transfer Vessel (CTV) which is shown in Figure 106. The connection to the transition piece is accomplished by pushing a rubber piece at the ship bow against the offshore structure. The thrusters push the boat against the boat landing and the personal is able to step from the bow of the ship on the ladder at boat landing. The aim of this method is to have no vertical vessel motions at the point of contact. This is the most commonly used method for accessing offshore wind turbines. The disadvantage of this method is that it can only be used in mild wave conditions up to a significant wave height of 1.5m [16]. In certain areas this results in a very low accessibility of the wind turbines.

Figure 106: Vessel-based access (*offshorewind.biz*)

Limited conditions for vessel-based access

The vessel-based access methods are always limited by the allowable wave and wind conditions. When the wave conditions get rougher, the ship motions increases. The ship may lose its contact point with the offshore structure and the ship starts moving relative to the offshore structure. This is unacceptable for the safety of the crew and therefore there are limited conditions for vessel-based access. These limiting conditions are specified by the significant wave height. If the significant wave height is exceeding this limiting criteria, the operation is too dangerous and may not be performed.

Significant wave height

Wave conditions can be characterized by wave parameters such as the significant wave height Hs and the zero crossing wave period Tz. Wave conditions are assumed to be stationary for a period of three hours. The significant wave height is defined as the average height of the largest 1/3 of all waves in the series. The mean zero-crossing wave period Tz is defined as the mean upward or downward zero-crossing period within that series. The significant wave height correlates well with the visually estimated wave height. The significant wave height is defined as the average of the 1/3 largest waves. Therefore the maximum expected wave will be larger than this significant wave height. A rule of thumb for calculating the maximum wave height is: Hmax=1.68·Hs [4]. Currently most standard boat transfers may not be performed in sea states where the significant wave height is greater than 1.5m [16][17] and the wind velocities are exceeding 12 m/s.

Scatter diagram

To predict the percentage of time for which offshore access can performed safely, scatter diagrams are used. These scatter diagrams provides information on the long-term distribution of sea states for a certain location. A scatter diagram presents the probability of occurrence for a combination of the significant wave height Hs and the mean zero-crossing period Tz. Not every combination of Hs and Tz occur in nature due to the stability of the waves. Scatter diagrams are based on measurements or hind-cast data. From this scatter diagram, the combinations of the significant wave height and the mean zero-crossing period can be selected for which the access method can be performed. This is equal to the sum of the probabilities of sea states up to the limiting significant wave height. This is the percentage of time that the access method can be performed. In Table 70, the scatter diagram is given from the Campos Basin which is located near the coast of Brazil. From this scatter diagram it can be seen that the accessibility of the offshore structure can be increased from approximately 25% up to 98% by using a motion compensated gangway [21] which is able to operate up to a significant wave height of 3.5m. But this example also shows that a motion compensated gangway is only a cost-effective solution for offshore areas where the significant wave height is often above the 1.5m.

Table 70: Scatter diagram and accessibility at different limiting sea states [21]

H_s \ T_z		**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**			
		4	5	6	7	8	9	10	11	12	13	14	**Total**		
5.5	6							0.01					0.01		
5	5.5						0.01	0.01					0.03		
4.5	5					0.02	0.05	0.02	0.01		0.01		0.10		
4	4.5					0.11	0.12	0.06	0.03				0.32		
3.5	4				0.09	0.43	0.28	0.17	0.04	0.01			1.02		Motion compensated
3	3.5			0.07	1.17	1.07	0.74	0.25	0.06				3.35	98%	Gangway access
2.5	3			2.14	4.28	2.75	1.17	0.31	0.06				10.72		
2	2.5		0.04	10.77	8.00	4.14	1.03	0.18	0.05				24.20		
1.5	2		4.28	17.63	9.88	3.07	0.63	0.12	0.01				35.61		Ship-based
1	1.5		6.64	10.71	4.44	1.01	0.07						22.87	25%	
0.5	1		0.71	0.84	0.17	0.02							1.74		
												Sum	100.0		

A-13. Operational procedure gangway

The operational procedure of a gangway is described in this section. The actual operational procedure can differ per manufacturer or type of gangway.

System modes

For the system there are four system modes. These modes describe the state of the system.

-The first mode is *the sea-fastened mode*. In this mode the boom is fully retracted, lowered and secured in a sea-fastening frame. All the support equipment is shut down. This mode is used when the ship is in transit mode and the gangway is not required.

-*The neutral mode*: In this mode the boom is fully retracted and horizontal. It points horizontal perpendicular outwards of the vessel. No motion compensation is applied. This mode is used as preparation for operation of the gangway.

-*The compensation mode*: This mode is activated when the gangway is in neutral mode. The active motion compensation function is switched on and all the vessel motions are compensated at the tip of the gangway. Now the system can be connected to the offshore structure.

-*Operational mode*: The gangway is connected with the target platform and locked into position. The system is in passive motion compensation mode. The transfer of people and cargo is possible.

Procedure gangway

The first step is to remove the sea-fastening. The operator is in his control cabin and starts the HPU. The gangway is lifted to horizontal position and slewed outward with its tip pointing outwards with respect to the vessel. Now it is in neutral mode. The compensation mode is switched on and all the vessel motions are compensated in the tip of the gangway. Now the operator can start to steer the gangway towards the offshore structure in order to make a connection. After contact with the landing platform, the position is locked and the system switches to passive motion compensation. Now the gangway is in operational mode and the transfer of people is allowed. The traffic light is green and the operator remains in his cabin.

For the recovery procedure, the traffic lights are switched to red. The system is switched to active compensation mode. The position is unlocked and the gangway is disconnected from the target platform. The telescopic boom is retracted and the system switches to the neutral mode. The boom is rotated 180 degrees towards the sea-fastening frame. The gangway is lowered onto the sea-fastening frame and the sea-fastening is applied. Now the system can be shut down.

A-14. Set-up topology optimization

In this section, the set-up of the topology optimization is explained. The goal of this optimization is to maximize the stiffness of the gangway structure with a restriction on the available volume. The optimization process needs to be constrained in order to find a balance between the stiffness and the amount of material which is used. The software will define a new topology for the structure, by removing unnecessary material from the design domain. The set-up of the optimization process will be defined in this section. The following steps must be followed in order to perform the optimization process:

1. Define geometry / design domain.
2. Import geometry.
3. Split model in design and non-design domain.
4. Assign components to the design and non-design domain.
5. Generate mesh.
6. Define support conditions.
7. Define loads.
8. Define load steps.
9. Define material.
10. Assign properties.

-Optional: Perform analysis

11. Define responses.
12. Define constraints.
13. Define objective.
14. Create design variable.
15. Run optimization.
16. Visualize results.

An important aspect of Hyperworks is that is has no default units. The units in the software are defined throughout the model units. Therefore it is up to the user which unit system is applied for the set-up of the optimization process. The units in the software must be consistent with the base units. For example: If the length is defined in millimetres, mass in tonnes, and time in seconds then pressure will be MPa, because pressure is a force per area. A unit consistency table can be used to avoid these problems. An example of an unit consistency table is given below. The MEGA unit system is used during the optimization process.

Table 71: Unit consistency table

Modelling unit	**Unit system**			
	SI	***MEGA (MPA)****	*CGS*	*BG*
Length	[m]	***[mm]***	[cm]	[ft]
Mass	[kg]	***[T]***	[g]	[slug]
density	[kg/m^3]	***[T/mm3]***	[g/cm^3]	[slug/ft^3]
Time	[S]	***[S]***	[S]	[S]
Pressure	[Pa]	***[MPa]***	0.1 [Pa]	[lbf/ft^2]
Force	[N]	***[N]***	[N]	[lbf·ft]
Velocity	[m/s]	***[mm/s]***	[cm/s]	[ft/s]
Acceleration	[m/s^2]	***[mm/s2]***	[cm/s^2]	[ft/s^2]

*applied during the optimization process

Step 1: Define geometry / design domain.
The first step in the optimization process is to define the design domain. The design domain is the volume in which material may be placed and attached. This design domain can be generated in a computer-aided design software package. In this research, the geometry of the design domain is generated by using the Ansys Spaceclaim: 3D modelling software, but other CAD programs can be used as well. The design domain consists of a rectangular tube with the inner and outer dimensions of the current gangway design. The outer dimensions of the different design domains can be found in Table 72. For the reaction forces of the roller bearings, a flat surface area of 100 by 200 mm is generated. These locations for the reaction forces are located at the left side of the gangway with an internal spacing of 4 meter. This can be seen in Figure 107 in which the supporting areas are denoted in red. They are located at the top and the bottom of the telescopic section.

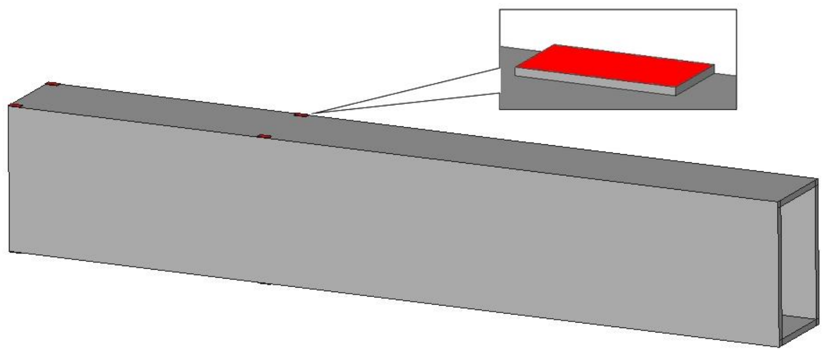


Figure 107: The design domain of the telescopic section

Table 72: Outer dimensions design domain

	Fixed section				Telescopic section			
Outer Dimensions	Length [mm]	Width [mm]	height [mm]	Thickness [mm]	Length [mm]	Width [mm]	height [mm]	Thickness [mm]
magnitude	24000	1600	2500	100	13000	1400	2300	100

The design domain is generated as a solid single part. This part will be cut and divided into different parts which will be assigned to the design and non-design domain in Optistruct. The reason for this procedure is that this method reduces the probability of mesh failures. The model can be exported from the CAD program to a .step or .iges file extension in order to import this file into Optistruct.

Step 2: Import geometry.
When the design domain is defined, it is possible to import this file into Optistruct. At the start-up of Optistruct, the program will open the users profiles dialog. Select the Optistruct user profile to perform an optimization task. The basic design geometry can be imported by the following steps:

File → *Import* → *Geometry* → *Select geometry file* → *Import*

This sequence of steps is visualized in Figure 108.

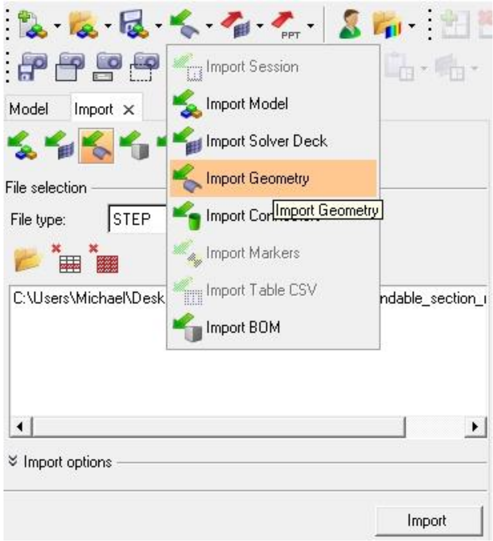


Figure 108: Import geometry file

Step 3: Split model in design and non-design domain.
Now the geometry is loaded and can be split into different parts. The model is split into sections by using sweep lines. This can be achieved by using the following steps:

Geometry ⟶ *Solid edit* ⟶ *Trim with lines* ⟶ *Trim with sweep lines* ⟶ *Trim*

During this procedure it is important to select the solids which will be cut and the lines which will be used as guideline for the cutting process. The direction of the operation must be defined to. The results of the cutting process is indicated with different colours. Green means that it is a free edge or plane, yellow means that is a shared plane or edge. This can be seen in Figure 109. Red means that the lines or surfaces are intersecting or disconnected.

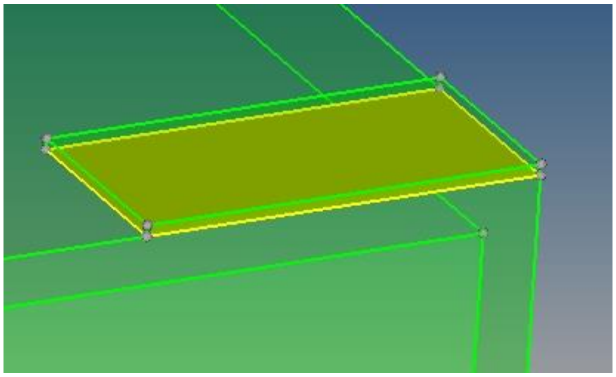


Figure 109: Different parts with a shared plane

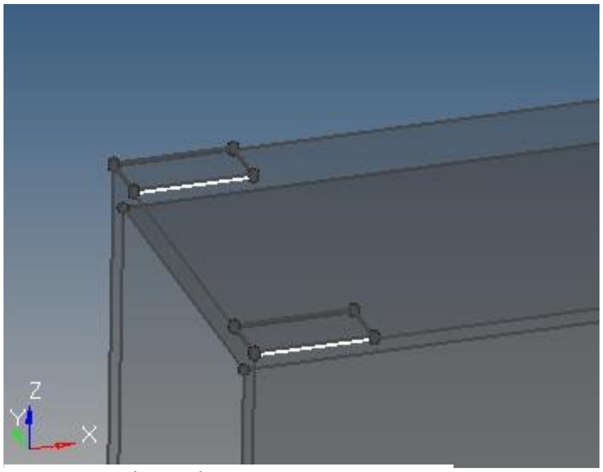


Figure 110: The cut lines

Step 4: Assign components to design and non-design domain.
The next step is to create a new component for the non-design space and to assign the non-design components. Click with the right mouse button in the model browser which is located on the left side of your screen to create a new component and name it: Non-design space. Use the following commands:

Model browser → *Create* → *Component*

It is important to split the model in a design and a non-design domain. This allows the optimizer to distinguish the parts which are subjected to the optimization and allows the user to specify different properties for each specific domain. Different colours can be chosen for the different collectors in order visualize which parts belongs to which collector. This shows which parts are subjected to the optimization. The parts can be assigned to desired collector by the following commands:

Tools → *Organize* → ▾ *Solids* → *Select non-design solids* → *Move solids*

Step 5: Generate mesh.
Now that all the parts are assigned to the design and non-design space, it is possible to mesh the parts. Unnecessary lines can be switched off during the meshing process in order to avoid mesh failures. The following steps are used for generating the mesh:

3D → *Tetramesh* → *Volume tetra* → *Enclosed volume: All* → *Specify element size* → ▾ *Elements to surf/solid comp* → *Mesh*

It is important to select a proper and well defined mesh size. The mesh size will influence the final result of the optimization even as the amount of the required computational effort. The mesh sizes according to Table 73 are applied. These are the minimum mesh sizes which can be applied without exceeding the limitations of the educational edition of Optistruct.

Table 73: Applied mesh sizes

	Mesh size [mm]	Nodes [-]	Elements [-]
Fixed section	110	79902	314956
Telescopic section	55	90660	344370

Step 6: Define the support conditions.
After discretizing the model into small mesh elements, the loads and boundary conditions can be applied. Therefore at least two load collectors needs to be defined. One for the support conditions and one for the loads. A combination of a load collector and collector for the support conditions defines a load step. Follow these steps for each load collector:

Model browser → *Create* → *load collector*

Right click inside the model browser window, activate the menu over create and click load collector. Define the name for the load collector. Leave the card image set to none and click on create. Now we have to define the support conditions which are acting on the design domain. Use the model browser and right click on the load collector and click on make current. Now the load collector is active and all the loads or constraints which are defined are stored in this load collector. Then from the analysis page at the bottom of the program, enter the constraints panel. This panel allows the user to select the nodes, lines or surface area which are constrained in single or multiple DOFS. The DOFS 123456 refer to the translations and rotations with respect to the X,Y and Z-axis.

Analysis → *constraints* → *Select Nodes/ lines/Surfaces* → *Define constrained Dof* → *Create*

Step 7: Create loads.
In this step the loads acting on the structure are defined. Use the model browser and click on the right mouse button on the force collector and make it current.

Analysis ⟶ *Forces* ⟶ *Select Nodes /Surfaces* ⟶ *Define magnitude* ⟶ *Create*

Use the analysis panel to define forces on nodes, lines or surfaces. With the dropdown arrow constant components can be defined for the force. The uniform size indicates the size of the arrow, this has no influence on the magnitude of the force. The gravitational acceleration can be included by selecting the GRAV card. The operational accelerations are defined as a ratio of the gravitational acceleration.

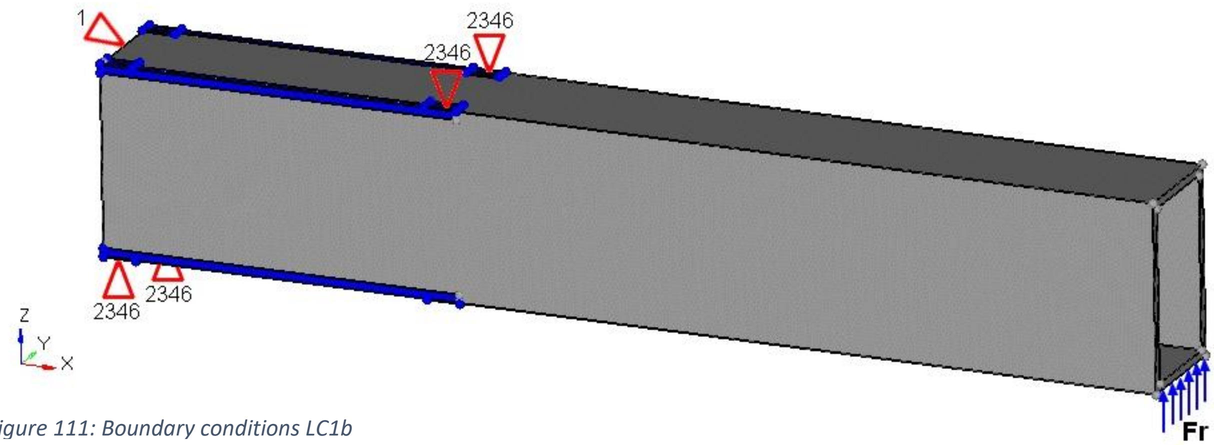


Figure 111: Boundary conditions LC1b

Step 8: Define load cases.
The final step in establishing the boundary conditions is to define the load steps. From the analysis page, enter the load step panel. Define a name for the load step and set the type to linear static. Select the support conditions (SPC) and the loads collector from the list of load collectors. Finally click on create to define the load step.

Analysis ⟶ *Load steps* ⟶ *Select SPC Select Loads* ⟶ *Type: linear static* ⟶ *Create*

Each combination of SPC and Loads define a load case. So this means for multiple load cases, multiple load steps must be defined.

Step 9: Define materials.
Components needs to be referenced to a material. Therefore a material collector must be defined. This can be achieved by right click inside the model browser and select create material. A name and the specific properties of the material must be defined. MAT1 is used as card image. So with the following steps can be used to define a new material:

Model browser ⟶ *Create material* ⟶ *Define: -Young's modulus -Poisson's ratio -Density*

For the optimization process, only the Young's modulus, Poisson's ratio and the density of the material must be defined. It is important to check your input parameter with the unit consistency table. The building material of the gangway is S355 structural steel which is defined in Table 22. It is important to apply the correct units.

Step 10: Define and assign properties.
Properties must be defined for the components. Properties must be defined for the design space and for the non-design space. Right click in the model browser to create a new property. Specify the name of the property and use Psolid as card image and select the material. Finally, use the right mouse button to click on the components in the model browser to assign the properties to these components.

Model browser → *Create property* → *Card image: Psolid* → *Assign material* → *Assign to component*

Optional: Run analysis.
This step is optional but it will allow the user to check if the response of structure is as expected and if the boundary conditions are well defined. It is a linear static analysis of the design domain prior to the optimization process. This analysis identifies the responses of the structure before the optimization to ensure that the constraints defined for the optimization are reasonable.

The analysis can be performed by selecting the Optistruct button on the analysis panel. Then by selecting the dropdown button for the run options, analysis can be selected. Then the Optistruct button can in order to start the analysis. When the analysis is finished, select the results button in order to start HyperView. In HyperView, select the slider to move to final iteration. Click on the contour button and select the preferred results type. Click on apply in order to visualize the results from the analysis.

Analysis → Optistruct → Run options: analysis → Optistruct → Results → Select final iteration → Contour → Select output → apply

The discretized model, consisting of mesh elements, material properties, loads and boundary conditions has been defined. Now a topology optimization will be performed with the goal of minimizing the compliance of the structure for a specified volume. Typically, a reduction in the material of the structure will reduce the stiffness of the structure and makes it more prone to deformation. Therefore, we need to track the displacements, which represent the stiffness of the structure, to obtain the maximum stiffness of the structure for the available volume.

Step 11: Define the responses of the structure.
Before an objective or a constraint can be defined, it is important to define the responses. A response is a numerical measure of a design variable due to the input on the model. These responses can be used as an objective function or as a constraint. In this situation, two responses are required for the optimizer: the total volume of the design and the weighted compliance of the design. The volume of the design domain is required in order to check if the solution satisfies the design constraint. The weighted compliance of the design is the objective function which must be minimized.

Use the analysis page and select the optimization tab, Click on responses and define the name of the response. With the dropdown arrow the response type can be selected. For the response of the weighted compliance, the corresponding load steps must be selected. This results in the following sequence of actions:

Analysis → Optimization → Responses → Response type: *weighted comp* → Load steps: *all*

For the response of the volume of the design domain, the property of the design domain must be selected. This results in the following steps:

Analysis → Optimization → Responses → Response type: *Volfrac* → Property: *Design domain*

Step 12: Define the constraints.
In this step the constraints for the optimizer are defined. The upper and lower bound constraint criteria must be defined for the analysis. An upper bound constraint will be defined for the allowable volume of the design domain. Select the optimization tab in the analysis panel and select dconstraints. Define the name for the constraint and select the response for which the constraint will hold. Upper and lower bounds can be defined with their corresponding values. For the optimization process, it is determined to set an upper limit of 25% of the volume of the design domain as a constraint for the optimizer. Use the following steps to define this constraint:

Analysis → Optimization → dconstraints → Select response → Define bound(s) → Create

Multiple constraints can be defined for the optimization process.

Step 13: Define the objective function.
The objective function must be defined for the optimizer. The objective function is the goal of the optimizer. For this optimization, the objective is to maximize the stiffness of the structure. This is equivalent to minimize the weighted compliance of the structure. The following steps must be performed in order to define the objective of the optimization:

Analysis → Optimization → objective → ▾ Min/max → Select response → Create

Only one single objective function can be defined for the optimization process.

Step 14: Create design variable
The final step in the set-up of the optimization process is to create a topology design variable. This is required in order to specify which parts of the model are subjected to the optimization process. The design variable can be generated by using the following steps:

Analysis → Optimization → topology → ▾ Type: *Psolid* → Properties: *design domain* → Create

Additional constraints such as: geometric, stress or member size constraints can be defined. For the gangway structure it is chosen to define a mirror plane at the Z-X. This will generate a symmetric structure at the defined plane and reduces the amount of computational effort which is required. To achieve this, use the pattern grouping option and select 1-plane symmetry. For this, an anchor node and first node must be specified. The anchor node defines the location of the mirror plane and the first node determines the orientation of the plane. A normal vector will point perpendicular from this mirror plane to the location of the first node. For the gangway structure, these nodes are defined by using interpolated nodes as can be seen in Figure 112.

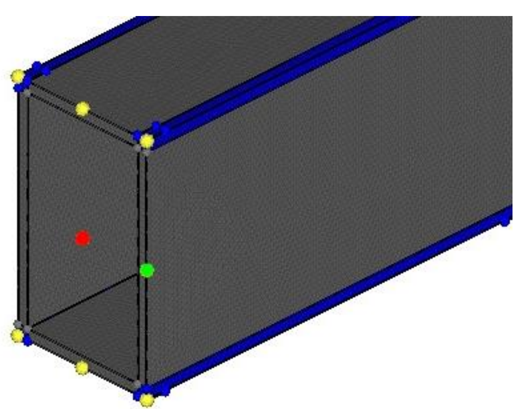


Figure 112: Design domain with mirror nodes

The anchor node is shown in red and the first node is green. The red node is located in exactly the middle of the gangway. These nodes can be created by the following steps:

Geometry → Nodes → On geometry → Select corners → Create

When the four nodes on the corners of the design domain are generated, it is possible to define new nodes and use the interpolate nodes in order to obtain mid-nodes. Now the anchor node and the first node can be selected for the plane symmetry:

Analysis → Optimization → Topology → Pattern grouping → Pattern type: 1-pln sym → Select anchor node → Select first node → Update

Additionally, control cards can be selected. Control cards are used to create solver control cards. For example: The output control card determines the required output files from the optimization. This control card can be selected to obtain the shape optimization file (.Sh file) for a particular iteration step. In order to get the .sh files for all the iteration you can use the following steps:

Analysis → Control cards → Output → Keyword: SHRES → FREQ: all

When this option is applied, the optimizer will write a .sh file for each particular iteration step. Also the .grid file can be selected which is related to the mesh of the particular iteration step.

Step 15: Run optimization
Now all the required steps are performed in order to start the optimization. Go to the analysis page and select Optistruct. Make sure the run option is set to optimization and click on Optistruct.

Analysis → Optistruct → Run options: *Optimization* → Optistruct

An additional screen will open which shows the progress of the optimization process. The following message appears in this window at the completion of the job:

OPTIMIZATION HAS CONVERGED FEASIBLE DESIGN (ALL CONSTRAINTS SATISFIED).

Optistruct will also report any error messages if they exist. Select the results button to visualize the optimization results in HyperView.

Step 16: Visualize results
HyperView can provide an ISO value plot of the element densities. This plot provides information about the element density in the model. ISO value retains all of the elements at and above a certain density threshold. The user can pick the density threshold which provides the structure which suits the needs. In HyperView, select the final iteration step of the optimization. This can be done in the HyperView menu or with the slider next to the play buttons. Select the ISO button and click on apply. Now the results from the topology optimization are shown and the slider on the right side of the panel can be used to vary the ISO values. A reduction of this threshold means a reduction in the element density threshold and therefore more mesh elements are shown.

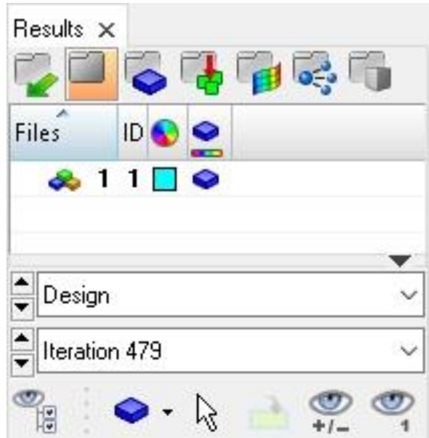


Figure 113: HyperView menu

Use the slide to change the density threshold. The ISO value in the graphics window update interactively with the change of the ISO value. This will help the user to understand the material lay-out and the load paths in the design.

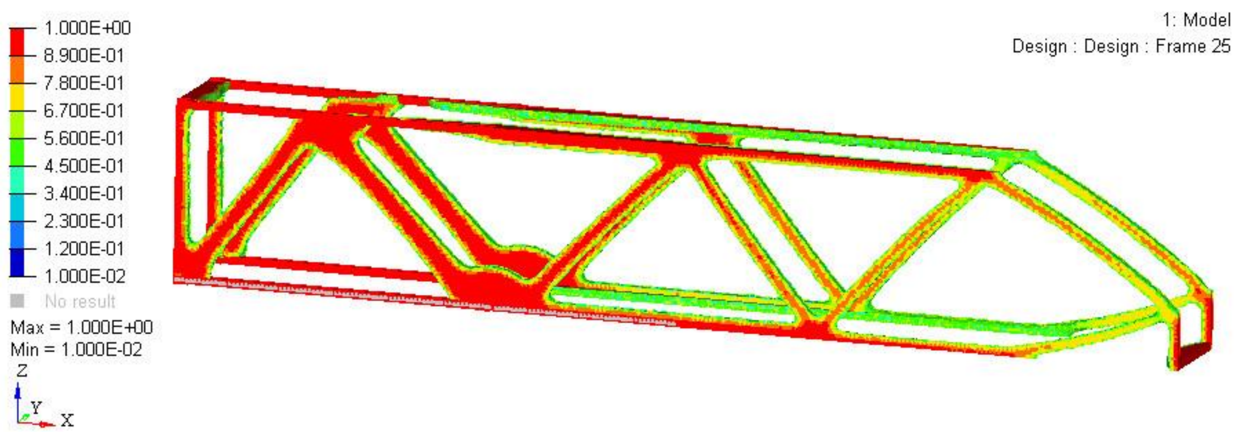


Figure 114: Density ISO Value

A-15. Set-up size optimization

This appendix describes the set-up of the sizing optimization process. The results from the topology optimization process are converted to a feasible design.

The following steps needs to be performed in order to set-up the optimization:

-Step 1: Open the topology optimization results and define temporary nodes.
-Step 2: Define linear lines between the temporary nodes.
-Step 3: Define the beam element cross-sections.
-Step 4: Define the material properties.
-Step 5: Define element properties.
-Step 6: Mesh the model.
-Step 7: Assign element properties to mesh elements.
-Step 8: Create load collector and define constraints and loads.
-Step 9: Define the load cases.
-Step 10: Discrete size optimization
-Step 11: Define the design variables.
-Step 12: Define the generic relationship between the design variables.
-Step 13: Define the required responses.
-Step 14: Define the design equations.
-Step 15: Define the design constraints.
-Step 16: Define the objective
-Step 17: Run the optimization.

-Step 1: Open the topology optimization results and define temporary nodes

The first step is to open a new Hypermesh session and to load the results from the optimization process. To load the optimization results, Select the optimization toolbar on top of the screen and select OSSmooth. This will open an panel at the bottom of the screen. In here, select the .fem model file and the corresponding .sh file. This .sh file can be selected for the preferred iteration step. The geometry of the topology optimization can be selected by using the following steps:

Optimization → OSSmooth → Select model file → Select result file → ▾ Autobead: none → OSSmooth

Now the model must be remeshed in order to define the nodes this structure. There a mesh size must be defined and click on remesh. OSSmooth will now discretize the loads from the topology optimization.

Temporary nodes can be defined on the geometry at the intersection points of the topology optimization. In between these nodes a line model can be defined. Therefore temporary nodes can be defined by selecting geometry and select create nodes on the geometry. Nodes must be defined on the intersection points and at the endpoints of the topology result.

Geometry → Create → Nodes → On geometry

The coordinates of the bounding nodes are shown in Figure 195.

-Step 2: Define linear lines between the temporary nodes.
Now it is possible to define lines between these nodes. Therefore a component collector must be defined. This is a storage container for the lines or geometry. Click with the right mouse button in the model browser and create a new component:

Model browser → Create → Component

Start to define lines between the previously defined temporary nodes. The geometry lines are built by using linear nodes. To achieve this, use the following commands:

Geometry → Lines → Linear nodes → Select nodes → Create

Finally all the temporary nodes can be deleted as they can be confusing. Therefore select the geometry button on the top of the screen and click on delete nodes. Select all to delete al the nodes.

-Step 3: Define the beam element cross-sections.
The frame will be modelled with beam elements. Beam elements allow for axial loads, shear forces, bending and torsion moments. For the 1-D elements, a cross-section must be defined. The beam cross-section are defined by using Hyperbeam. Use the bar on the top of the screen and click on properties and select Hyperbeam. For the standard section library, selection Optistruct and for the type of standard section select Optistruct. Click on create to open Hyperbeam. In here, different types and sizes of tubes can be generated for the different parts. The preliminary dimensions of the cross-section can be defined. The beam element cross-sections will be referenced later. Exit Hyperbeam by clicking the model view icon the top left corner.

Properties → Hyperbeam → *Section library: Optistruct*
Section type: Tube → Define beam sections

-Step 4: Define the material properties.
The material properties must be defined. Click with the right mouse button in the model browser and select create material. Define the name of the material and select the card image MAT1. The young's modulus, density and poisons ratio must be defined. Multiple materials may be defined.

Model browser → Create → Component

-Step 5: Define element properties.
The frame will be partitioned into different domains with different radiuses. Therefore each group of members a property must be defined. For the gangway, it is chosen to define each member separately and to put each two symmetrical adjacent members in a group. This means that the right and left side of the gangway will have the same radiuses. To define a property, use the right mouse button and click on the model browser to create a new property. As card image select PBEAML and select the corresponding material. The beam section of interest must be selected.

Model browser → Create → Property → *Card image: PBEAML*
Select beam section
Select material

-Step 6: Generate mesh

The line model must be meshed with CBEAM elements. Two different approaches can be used: Mesh all the lines and subsequently assign the mesh elements to the corresponding property. Or all the domains are meshed individually. The frame is meshed with a mesh size of 4mm which is determined by performing a mesh convergence check. The following steps are performed in order to discretize the model:

1D ⟶ Line mesh ⟶ All lines ⟶ *Specify element size*
Element config: Bar2
Select property
Define orientation vector ⟶ Mesh

The orientation and size of the elements can be visualized by selecting the 1D element representation.

Now it is possible to check the orientation of the mesh elements. The orientation of the mesh elements can be achieved by defining a normal vector which is perpendicular to the element orientation. This orientation vector is defined with respect to the global coordinates.

Before proceeding with the definition of loads and constraints, make sure that the mesh is compatible. This means that the orientation of the elements is correctly defined and all the elements are connected to each other. The mesh compatibility can be defined by using the following steps:

Tools ⟶ Edges ⟶ Preview equiv.

The highlighted nodes indicates that the mesh is not merged. When this is the case check your model or increase the tolerance value.

-Step 7: Assign element properties to mesh elements.

With this step the element properties for the different beam elements are assigned to the mesh elements. This can be done by using the right mouse button and click on assign. Now select all the mesh elements which must be assigned to this property. This procedure must be performed for each property until all the mesh elements are assigned the right corresponding property.

Property ⟶ *Assign* ⟶ *Select mesh elements*

When this procedure is performed, all the mesh elements are assigned to the right property.

-Step 8: Create load collector and define constraints and loads.

The model is discretized and all the mesh elements are assigned to the corresponding property. The next step is to defined the loads and support conditions which are applied to the model. The analysis set-up starts with defining load collectors for loads and boundary conditions respectively. A load collector can be defined by a right click in the model browser and to click on create load collector:

Model browser ⟶ *Create* ⟶ *Collector*

Once the load collector exists, the constraints can be applied. Go to analysis and select the constraints button. Now the location, area and constrained DOF can be selected and applied to the model. Make sure that the right collector is set to current. This can be done by use the right mouse button and make current.

Analysis ⟶ *constraints* ⟶ *Select nodes/ lines/surfaces* ⟶ *Define constrained Dof* ⟶ *Create*

Once the constraints are defined, the loads can be applied. A load collector must be defined for the loads. Set this load collector to current. The force subpanel can be entered by the following steps:

Analysis ⟶ *Forces* ⟶ *Select nodes/ lines/surfaces* ⟶ *Define magnitude* ⟶ *Create*

-Step 9: Define load steps.
In the load steps the combination of loads and boundary conditions are related. It defines the different load cases for the optimizer. These loads and constraints must be taken into account during the analysis. The load steps can be defined by using the following steps:

Analysis ⟶ *Load steps* ⟶ *Select SPC Select Loads* ⟶ *Type: linear static* ⟶ *Create*

Now the model-set is completed and a design analysis may be performed. An analysis is optionally but highly recommended because it allows the user to inspect the behaviour and set-up of its model. The analysis can be performed by selecting the Optistruct button on the analysis page. Set the run options to analysis and click on run the analysis.

Analysis ⟶ *Optistruct* ⟶ ▾ *Run options: Analysis* ⟶ *Optistruct*

The results of the analysis will be shown in HyperView. In here, the elements stresses and displacements of the model can be shown.

-Step 10: Discrete size optimization.
In this step discrete design variables are generated for the size optimization. Discrete size optimization will allow/enforce discrete changes of the design variables in user defined steps. So for instance, the inner and outer radii will vary with steps of 0.1 or 0.5 mm instead of a continuous variation. The optimization set-up will therefore be extended with the definition of discrete design variables which will allow for more realistic profile dimensions. Two discrete design values are defined:

Optimization ⟶ *Create* ⟶ *Discrete design variables*

The parameters for these discrete design variables are defined in Table 74 in which the value ranges for the discrete values are given and the incrementation steps.

Table 74: Discrete value ranges

Name	**Value range [mm]**		
	From	*To*	*Increment*
DR1	25	100	0.5
DH	50	200	0.5

-Step 11: Define the design variables.
The following steps are related to the size optimization of the structure. In this size optimization study the design variables are the inner and outer dimensions of the profiles. The design parameters are the outer diameter of the hollow tube and the height of the square tube. All the other dimensions are related to these two design variables as can be seen in A-10. Cross section classifications. The outer diameter of the hollow tube and the height of the rectangular hollow tube are varied within their user defined upper and lower bounds. In the following steps the outer diameter of the tube is abbreviated as R1 and the height of the rectangular hollow section is abbreviated as h. Design variables needs to be defined for each property which is defined for the members. This means that for all the different elements or members these design variables must be defined. Use the following steps to define these design variables:

Analysis → *Optimization* → *Size* → *Desvar* → *Define Bounds* / *Select ddval* → *Create*

The initial value is the starting value for the optimizer, the upper and lower bounds define the limiting values for the optimizer. The upper and lower bounds define the range in which the values may occur. Ddval are the discrete design variables which defines the incrementation steps for the design variables.

-Step 12: Define the generic relationship between the variables.
In this step the design variables are related to the model. The generic relationships can be generated by using the optimization tab and to create a new desvar relationships and click on generic. Use the following steps:

Optimization → *Create* → *Desvar relationships* → *Generic*

Assign a name for the generic relationship. Use the property button to select the corresponding property collector. This property collector contains the mesh elements or beams which are related to these design variables. The next step is to relate the design variables to the dimensions of the profile. Define which relationship is for the outer “1” or inner “2” radius of the CHS. This can be done by selecting the dropdown button and to select the dimension 1 or 2. For a RHS the external height and width of the profile and the internal height and width of the profile must be defined. This results in 4 dimension relationships as can be seen in Figure 115.

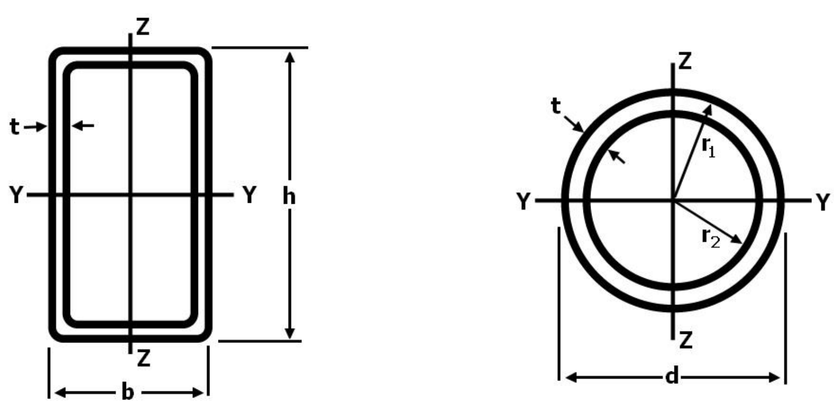


Figure 115: Relationships design variables

These steps needs to be repeated for all the design variables. The generic relationship can be checked by activating the review option and to selection the relation of interest

Generic relationship → *Define name* → *Select property* → Dim 1: *Select designvars* / Dim 2: *Select designvars* → *Create*

-Step 13: Define the required responses.
In this part the required responses from the model are defined. For the gangway, the maximum stresses in the members and the maximum deflection of the structure is required. Because stress and deflection limits are defined by the guidelines. Also the mass or total volume of the structure are of importance due to the fact that the structure is optimized for minimum weight. Consequently, the responses of this size optimization are mass, stress and displacements. These responses can be defined by using the following steps:

Optimization → *Create* → *Responses* → *Create*

The response displacement is defined for the tip of the gangway. The tip of the gangway may not exceed a certain value for the vertical deflection and therefore the displacement of this point is required. The response type is set to static displacement and the corresponding nodes are selected. The displacement in the Z-direction is required, so therefore select ◉ dof3 and click on create.

The total mass of the structure is required for the optimization process and therefore a response of the mass of the structure is required. The response name can be defined as mass with the response type mass. The total mass of the model is required and therefore select total.

The stress responses must be defined for each member separately. The response type is static stress and the corresponding property must be selected. This response must be defined for each member in order to apply the stress constraints for each member separately. An overview of the responses is shown in Table 75.

Table 75: Optimization responses

Response name	***Response type***	***Applied to***
Tip displacement	Static displacement	Nodes at gangway tip
Total mass	Mass	Total design domain
Member stress	Static stress	Each property collector

-Step 14: Define the design equations.
Design equation can be defined which relates the other dimensions of the cross-sections to the design variable. These functional relationship are used to simplify the optimization process and to include additional constraints like local buckling. The functional relationships are defined in Table 76 which are derived in section A-10. Cross section classifications.

Table 76: Functional relationship

Cross section	**Design parameter**	**Design relation(s)**	**Formula**
CHS	Outer radius, r_1	Inner radius, r_2	$Inner\ radius, r_2(r_1) = \frac{311}{331} r_1$
RHS	Profile height, h	Profile width, b	$External\ width, b(h) = \frac{149}{308} h$
		Wall thickness, t	$Wall\ thickness, t(h) \geq \frac{5}{308} h$

The design equations can be defined by using the following steps:

Optimization → *Create* → *Design equations* → *Create*

The name of the design equation can be defined in this section. Furthermore, the design equation itself can be defined. Three design equation are defined as can be seen in column 4 of Table 76. It is important to define three different design equations in order to relate the design equations to the existing design variables. The design variable links can be generated by:

Optimization ⟶ *Create* ⟶ *Desvar links*

Specify or assign a name and select the dependent design variable and the equation which defines how the design variables are linked together. Before creating this design variable, use the Edit button to check the relation between the independent and dependent design variable.

Step 15: Define the design constraints.
Now that all the responses are defined, it is possible to define the design constraints. The goal of this optimization is to investigate if the weight of the gangway structure can be reduced while keeping the tip displacement of the structure within its limits. Therefore a displacement constraint must be defined for the maximum displacement at the tip of the gangway. Use the following steps to define this constraint:

Optimization ⟶ *Create* ⟶ *Constraints* ⟶ *Select response* ⟶ *Define bounds* ⟶ *Create*

In the design constraint panel the define response "tip displacement" is selected and the allowable lower bound is selected is set to -195mm. The minus sign is due to the fact that the displacement is in the negative Z-direction. The magnitude of 195mm is defined by the guidelines with a safety factor of 1.5 due to the connection of the roller bearings. This constraint defines the maximum allowable vertical deflection of the nodes located at the tip of the gangway.

Step 16: Define the optimization objective.
The objective of the optimization is to minimize the mass of the structure while maintaining its stiffness and stress requirements. The objective is set to minimize the mass of the structure:

Optimization ⟶ *Create* ⟶ *Objective* ⟶ ▾ *min* ⟶ *Response = Mass*

Step 17: Run the optimization.
The optimization parameters are defined and the optimization can be performed. The optimization run is started with the following steps:

Analysis ⟶ *Optistruct* ⟶ ▾ *Run options: Optimization* ⟶ *Optistruct*

If the optimization has converged and the constraints are satisfies, it is possible to investigate the results. The maximum displacement of the tip is reported. The different values for the design variables are reported which are respectively the dimensions of the cross-sections. Another important results is the response value of the total mass of the structure. The deflection of the structure can be shown by using the following steps:

Optistruct ⟶ *Results* ⟶ *Select final iteration* ⟶ *Contour* ⟶ *Select output* ⟶ *apply*

A-16. Finite element analysis

The static structural strength and deflection of the gangway is evaluated in this section. The following cases have been performed in FEA for total evaluation of the steel structures within the model and are considered worst cases for these structures:

- Operational:
  - Stresses
  - Deflections

- Emergency Disconnection:
  - Stresses
  - Deflections

The gangway consists of two different sections. In this report, both sections will be analysed even as the composition of the two parts. These parts are visualized in Figure 116.

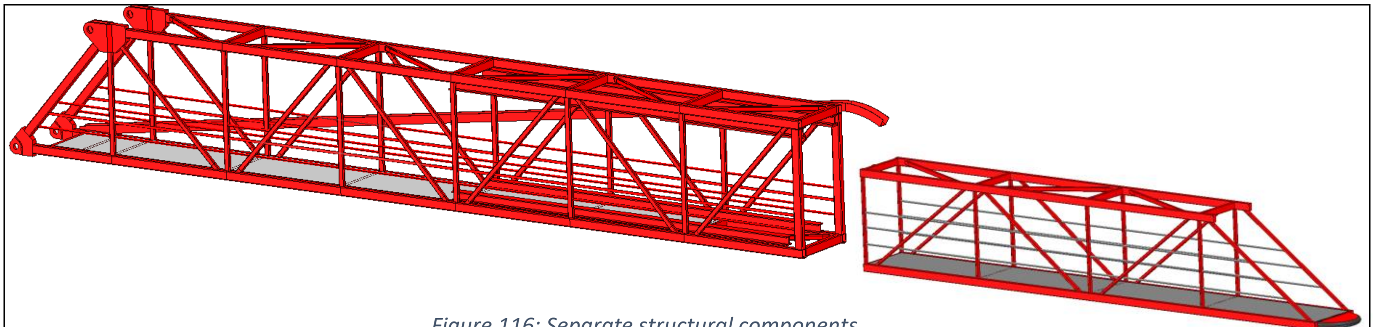


Figure 116: Separate structural components

Acceptance criteria

For this construction the following criteria are used according to the DNV [25]:

- Operational load cases $\sigma_{allow} = \frac{\sigma_y}{1,5}$
- Emergency disconnection $\sigma_{allow} = \frac{\sigma_y}{1,10}$

In which σ_y is the yield stress of the construction material.

The maximum allowable deformation of the gangway is given by the DNV-guidelines [25]:

- Operational load cases $\delta_{allow} = \frac{L}{200}$
- Emergency disconnection $\delta_{allow} = \frac{L}{100}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway.

16.1 Gangway current design – Normal working conditions LC 1c

In the normal working condition the motion compensated gangway is in operational mode. This means that gangway structure is able to transfer people and cargo from and to the offshore structure. For the normal working condition the gangway shall be designed for the following scenario:

- The live load (LL) on the gangway shall be the maximum number of persons, including hand tools and luggage allowed on the gangway at the same time. The load will be applied at the most limiting location.

The design load of the gangway shall be equal to two times the live load. The live load represents the allowable number of persons on the gangway at one moment. Only one person is allowed on the gangway during operation.

Mesh

The model is mainly built with solid elements and not with solid shell elements. The reason for this is that a solid shell mess causes an error in the numerical solution. The global mesh size is 50mm. The total number of elements is equal to 83586.

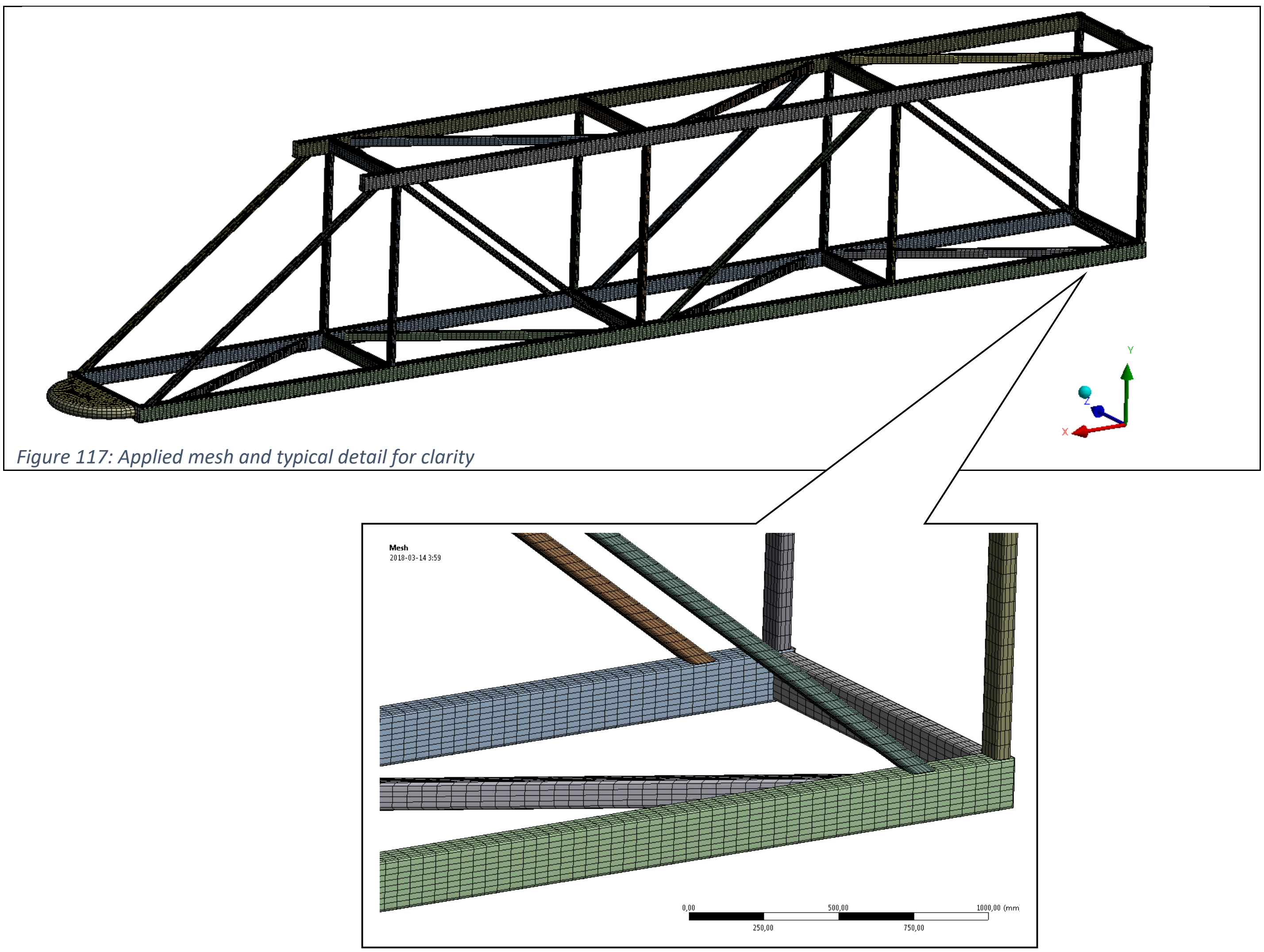


Figure 117: Applied mesh and typical detail for clarity

Connections and boundary conditions

All connections in the model are bonded. A tolerance setting of 1mm is allowed for detection of the contact regions. There are no special connections in this model.

The structure is constrained by means of remote displacement supports and frictionless supports. At the hinge point for connecting the cable for the linear motion, a remote displacement is added which constrains the translation in the X-direction. The landing mechanism is located at the tip of the gangway. This system constrains the displacements in the X, Y and Z-direction and only allows rotation around the Y and Z-axis. The sliding pads consists of frictionless contact regions, Frictionless sliding may occur and these regions can only transfer compression and separates if put under tension. This means that the reaction force can only work in the direction of the object and not in the opposite direction.

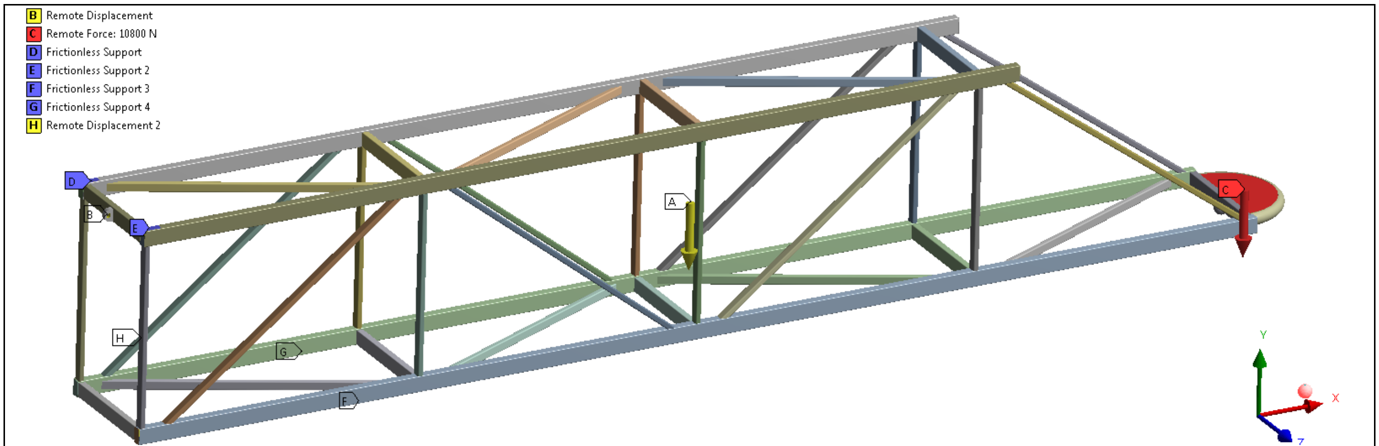


Figure 118: Remote displacements cylinders and hinges

The DOF of the remote displacements are given in Table 77.

Table 77: Constraints normal working condition

DOF	Hinge point b	Roller bearings	Landing mechanism
X Component	0 mm	Free	0 mm
Y Component	Free	0 mm (Only compression)	0 mm
Z Component	Free	Free	0 mm
Rotation X	Free	Free	0°
Rotation Y	Free	Free	Free
Rotation Z	Free	Free	Free

Material properties

The assumed primary structure material is S355 or similar, with the following applicable properties according to Table 78.

Table 78: Properties S355 steel

Property	**Value**
Yield strength	355 MPa
Modulus of elasticity	200 GPa
Poisson's ratio	0.3
Density	7850 kg/m^3

The sliding pads are made of the D-glide FC material which has the properties according to Table 79.

Table 79: Properties D-glide FC

Property	**Value**
Modulus of elasticity	1600 MPa
Poisson's ratio	0.34
Density	1350 kg/m^3

Loads and accelerations

In this chapter all the applied forces and accelerations are defined. A standard Earth gravity of 9806.6 mm/s^2 is applied to the geometry. Additionally, the accelerations are given in Table 80.

Table 80: Operational accelerations

Direction	Operational
	[mm/s^2]
X	1540
Y	836
Z	1420

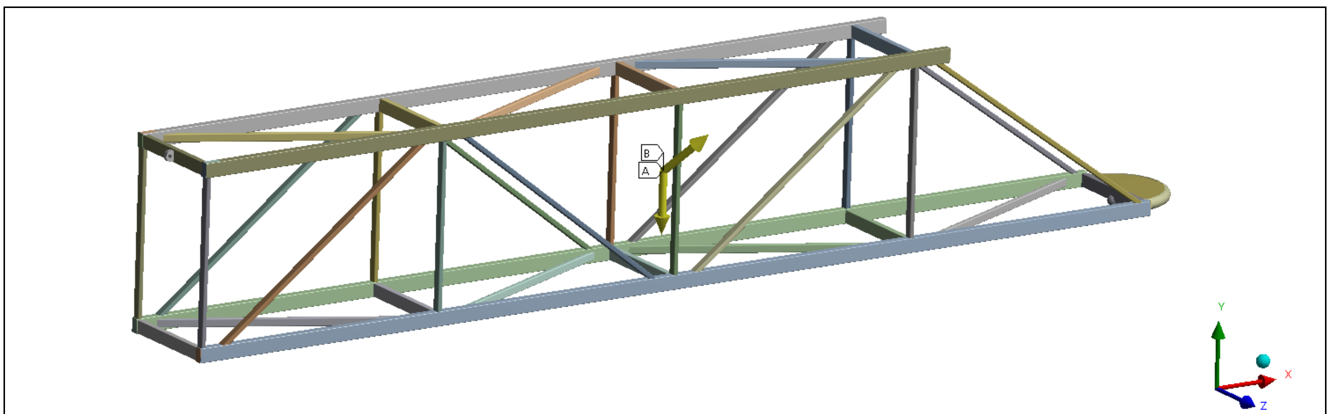


Figure 119: Typical acceleration application

Note that Ansys creates a reaction force opposite to the direction of the applied acceleration, therefore the accelerations are placed in opposite direction of the actual accelerations. Given values are multiplied with -1.

Forces

In addition to the self-mass of the gangway structure, the gangway boom will be loaded by additional components such as: the rollers, flooring, hand rails, utility lines and the landing tool. For the structural evaluation of the gangway structure these additional masses are represented by point masses. The additional loads acting gangway boom are estimated in A-5. Design calculations gangway. These loads are applied as point masses to the model. The locations of these point masses are shown in Figure 120.

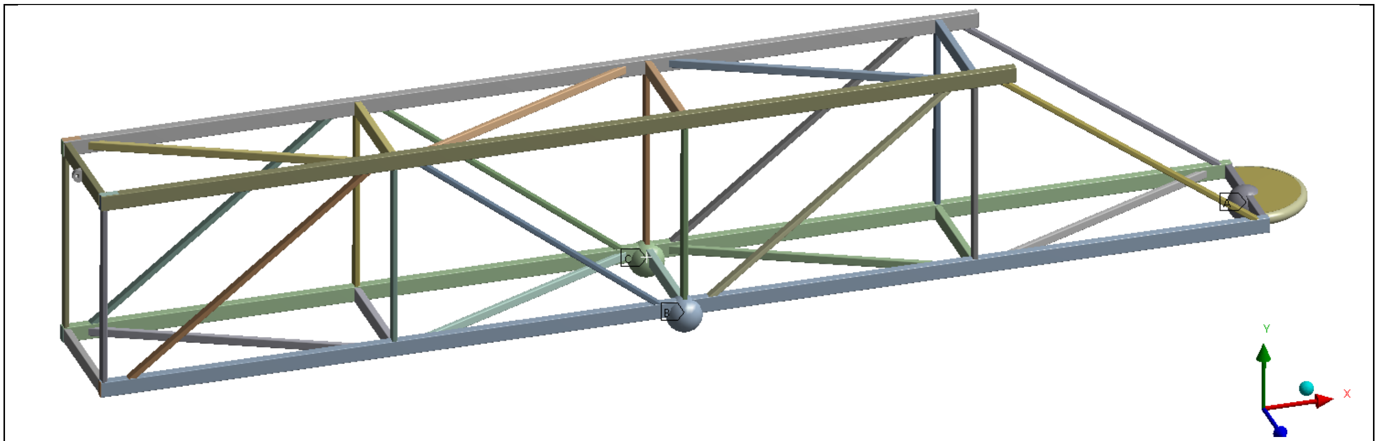


Figure 120: Point masses in model

The live load

The live load is applied as a remote force. This load is applied in the middle of the gangway structure. The design load of the gangway shall be equal to two times the live load. The live load represents the allowable number of persons on the gangway at one moment. It is defined as the sum of the cargo capacity, trolley mass and the operator mass multiplied by the number of persons allowed on the gangway. The magnitude of the live load is equal to 10800 N as is determined in appendix 5.

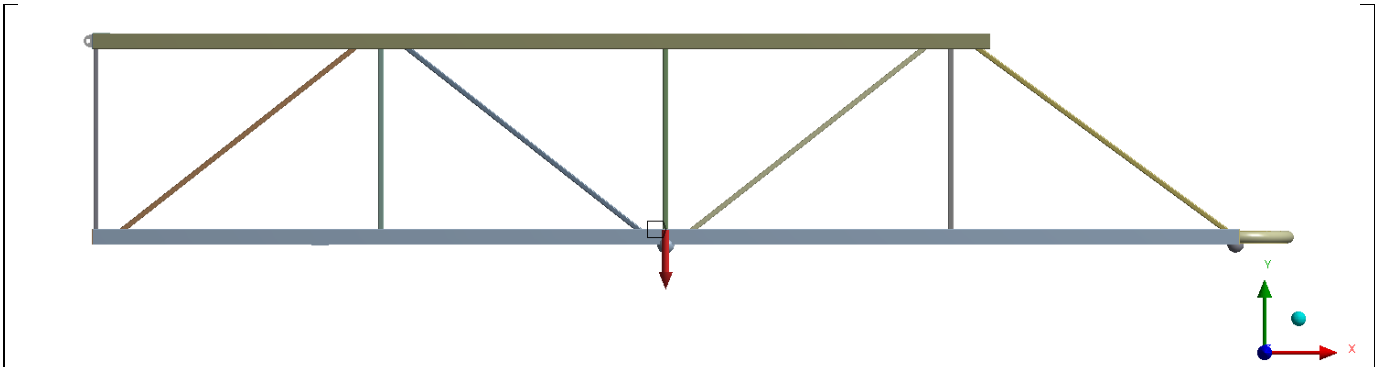


Figure 121: Live load acting on the gangway

Two times this live load means that the gangway design must be subjected to a total load of 21600 N which is acting exactly in the middle of the gangway in operational conditions.

The wind load

The gangway structure is subjected to wind loads. The wind load normal to a flat surface is calculated according to DNV. The value of the air velocity pressure is calculated and applied to the lateral surface area of the gangway structure. This pressure is only working in the normal direction of the face on which it is acting. The values in Table 81 are calculated according to the DNV. [25]

Table 81: Wind pressures

Load condition	**Wind velocity**	**Wind pressure**
	[m/s]	[Pa]
Operational wind speed	20	245
Deployment/retrieval wind speed	36	794
Transit/survival/parked wind speed	44	1186

The wind pressure is applied to the red surface area shown in Figure 122.

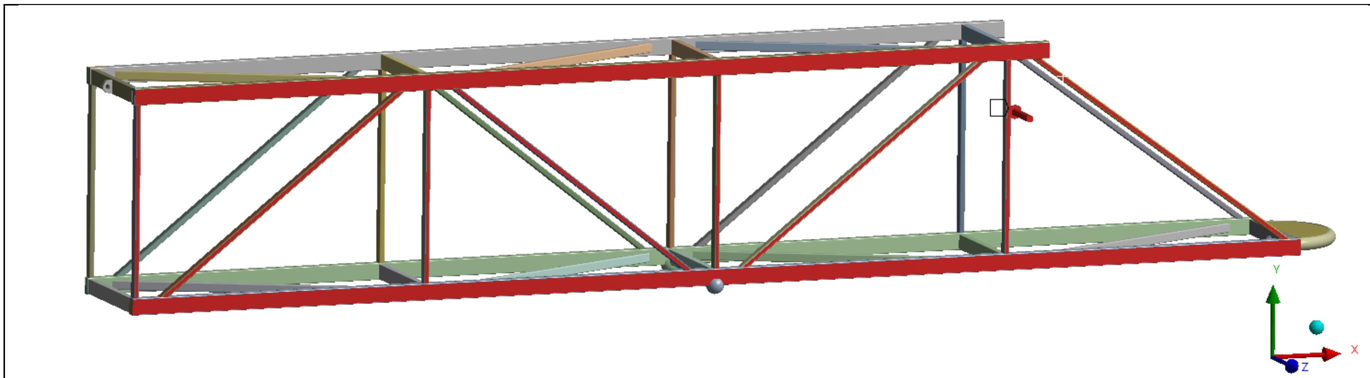


Figure 122: Lateral surface area wind pressure

Stress plots

The stress plots for the normal working conditions are reported in this section. The stresses over 50, 100, 150, 237 (Acceptance criteria) and 355 MPa (Yield limit) are plotted. Also the overall von-mises stresses are plotted.

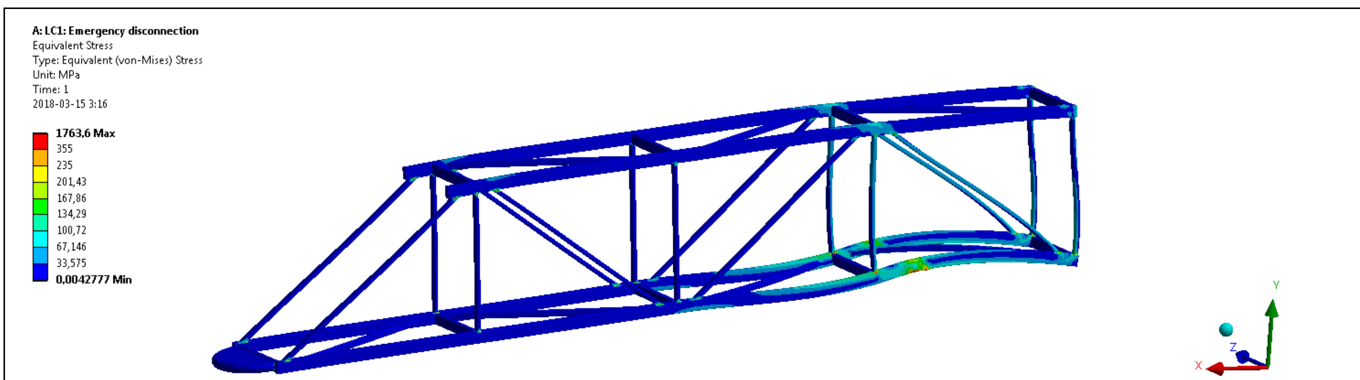


Figure 123: Overall Von Mises stress

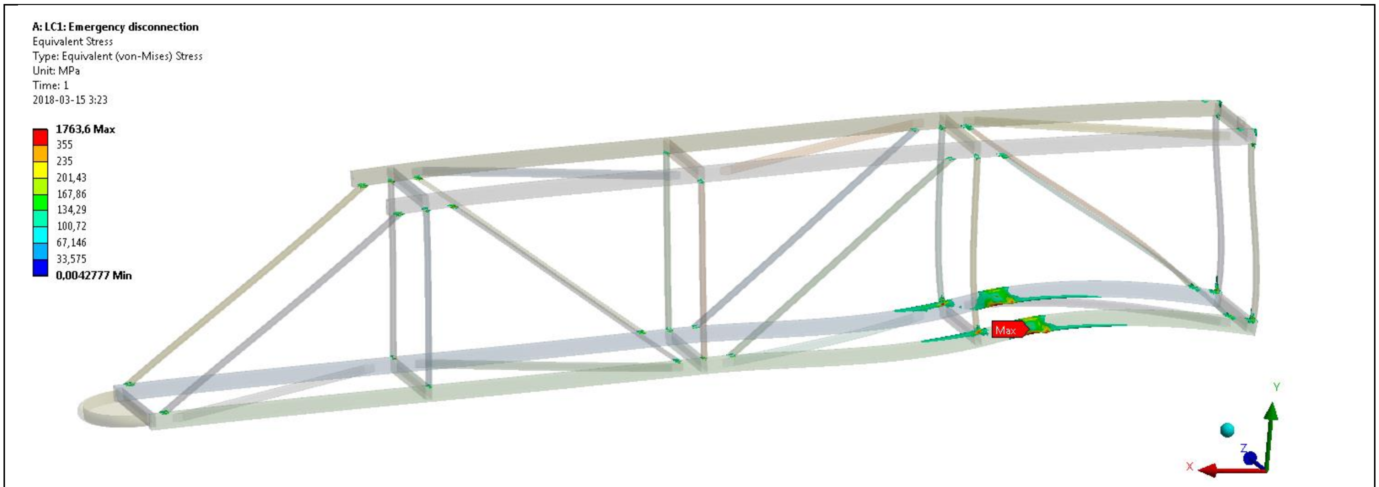


Figure 124: Stresses over 100 MPa

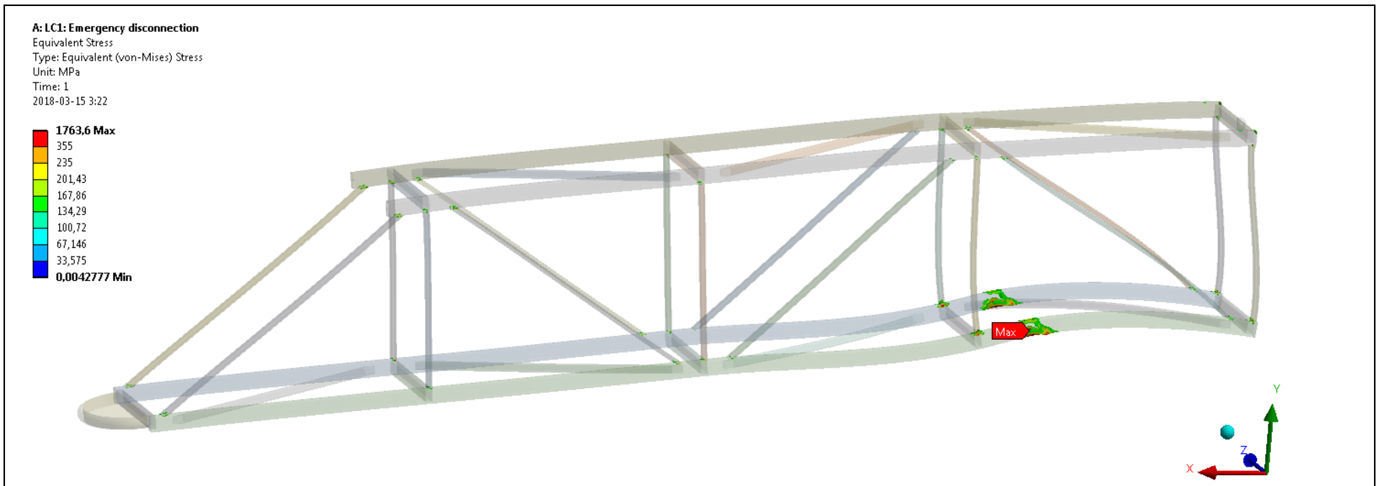


Figure 125: Stresses over 150 MPa

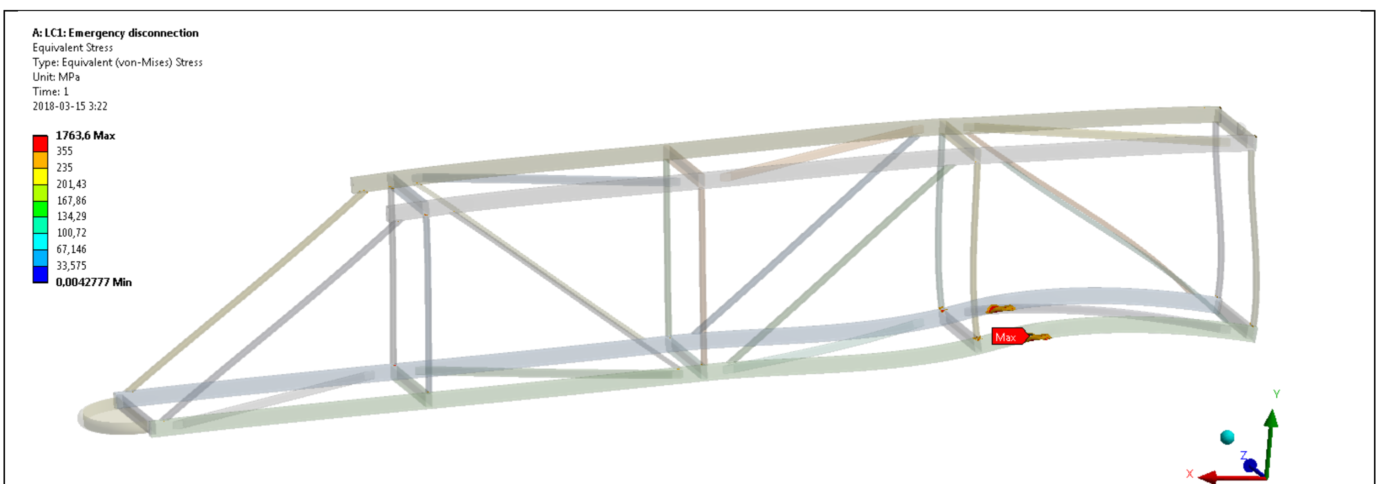


Figure 126: Stresses over 237 MPa

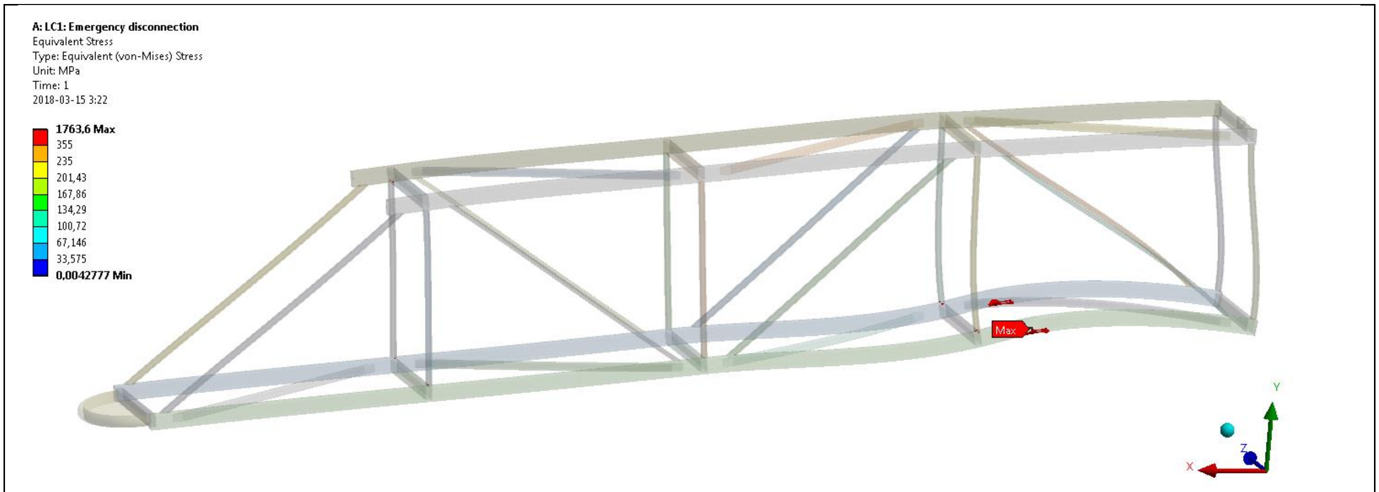


Figure 127: Stresses over 355 MPa

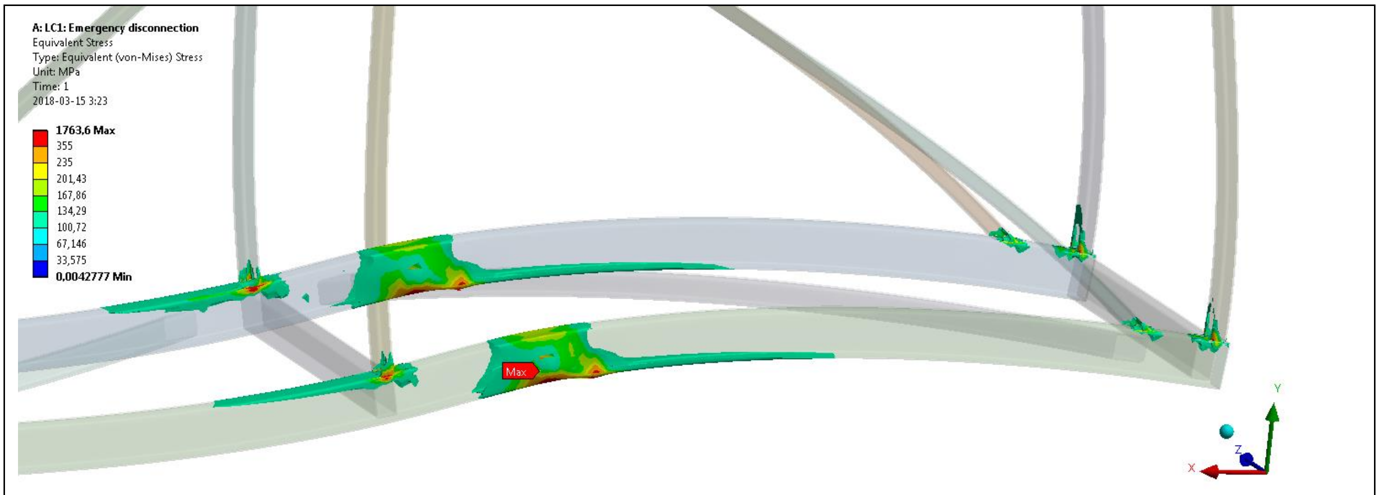


Figure 128: Detail view of highest stress locations

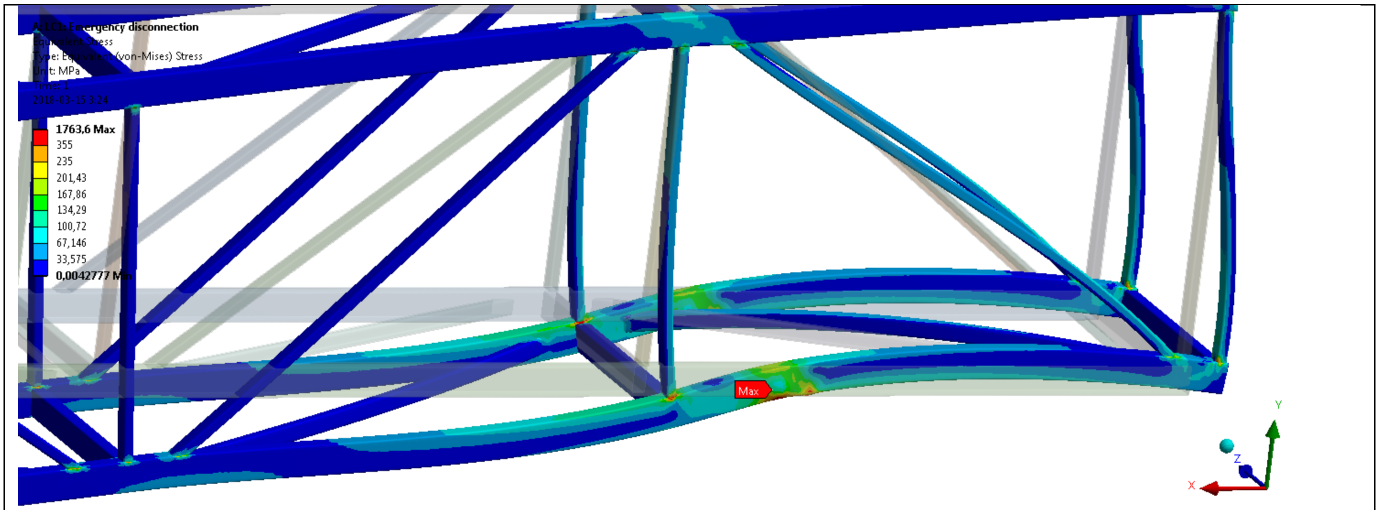


Figure 129: Detail view of the highest stress locations

The general stress level is above 355 MPa / 1.5 = 237 MPa and is therefore considered unacceptable. The plots with detail views of the peak stress areas show the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and needs to be reinforced.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines [25]. For the operational load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{200}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33 meter. This means that the maximum deflection limit is 165 mm.

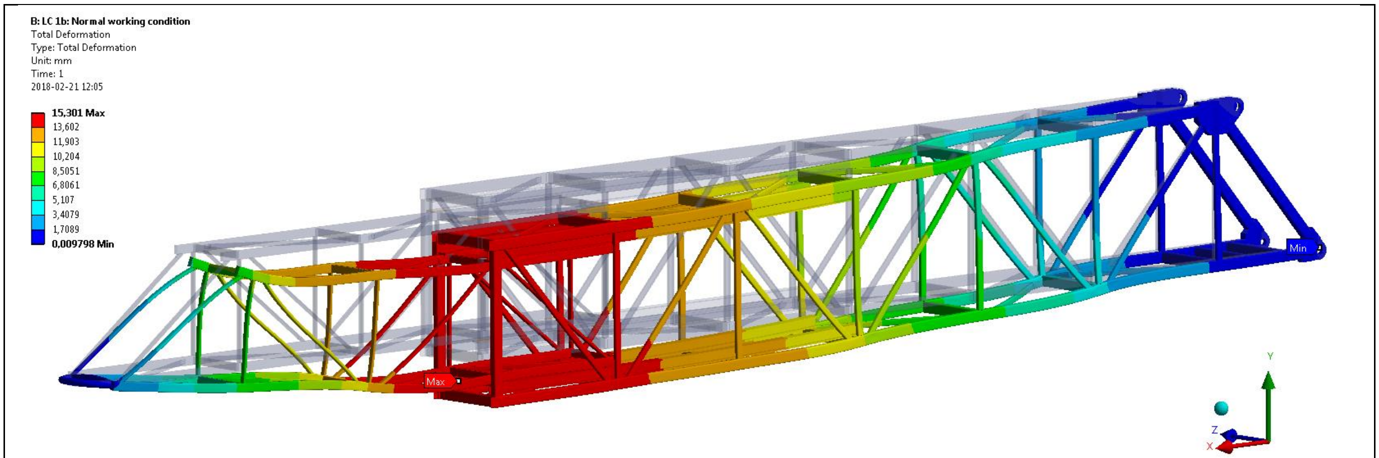


Figure 130: Total deformation, scale = 60

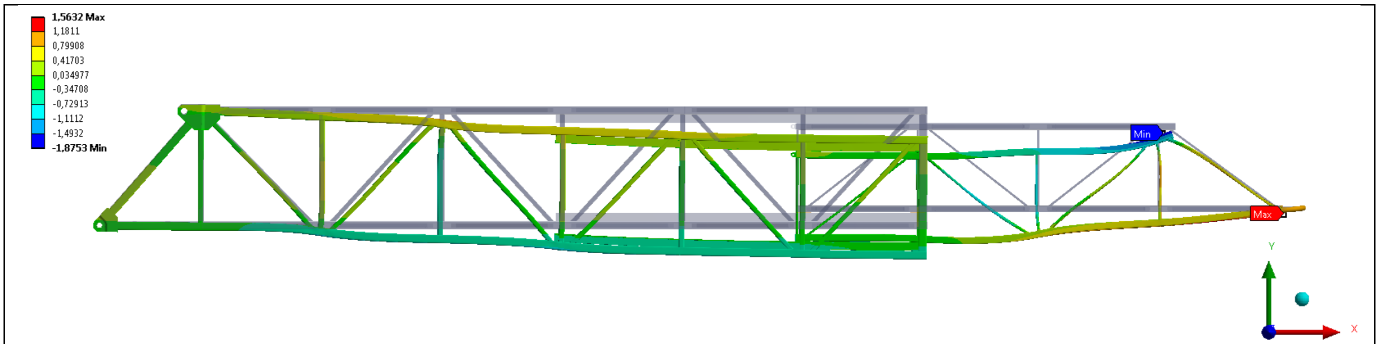


Figure 131: Directional deformation, X-direction, scale = 60

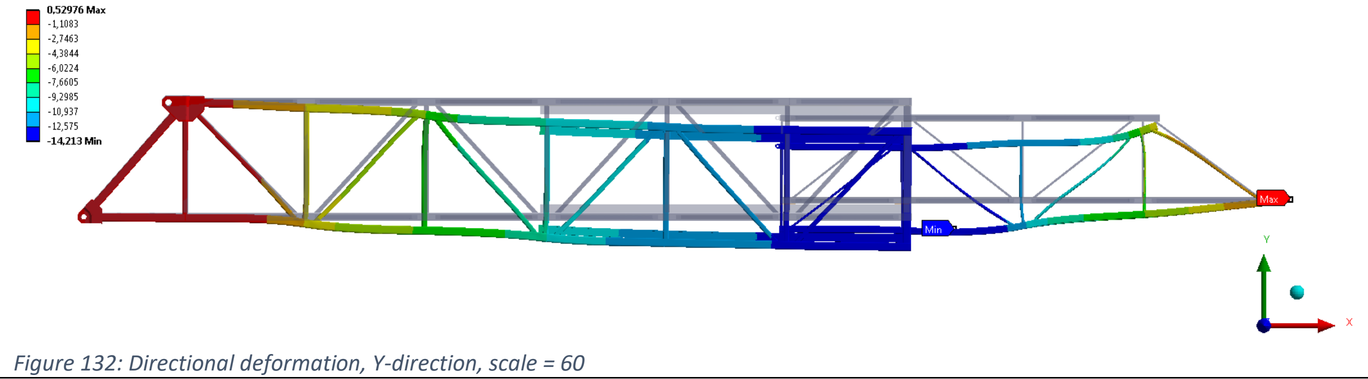


Figure 132: Directional deformation, Y-direction, scale = 60

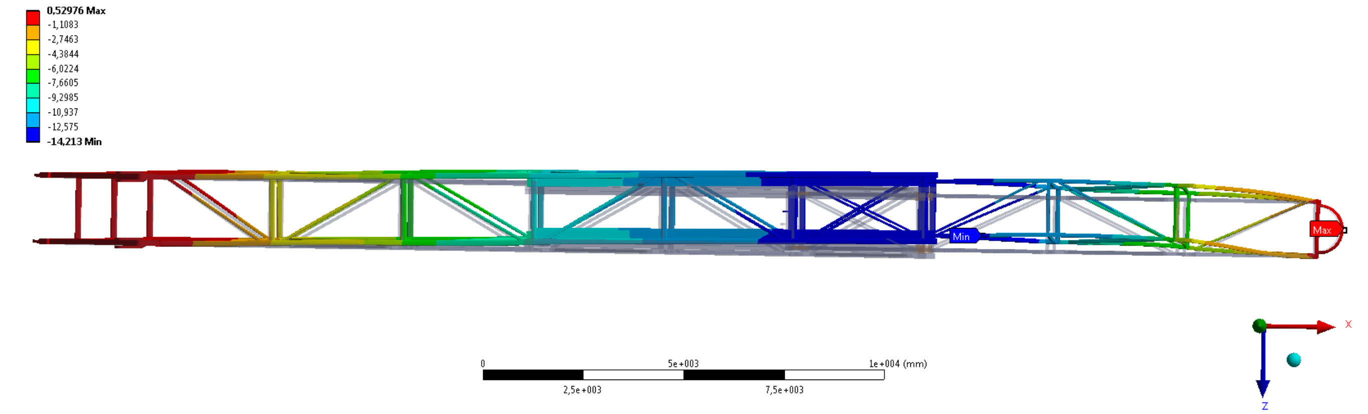


Figure 133: Directional deformation, Z-direction, scale = 60

The total deformation of the gangway structure is equal to 14,21 mm in the vertical direction. The total combined deformation is equal to 15,30 mm. These values are well within the stated limit and therefore considered acceptable.

Reaction force check

A reaction force check is performed in order to check the finite element model. The reaction forces from the finite element analysis are validated with hand calculated reaction forces. This is to make sure all applied forces and constraints are right. An overview of the results can be found in table 71: reaction force check below.

Table 82: Reaction force check

Reaction force check				
LC 1b: Normal working condition				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	21,706	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	-836	-1540	-1420
Geometry	[N]	-18146	-246289	-30823
Live load	[N]	0	-21600	0
Wind load	[N]	0	0	-5495,60
Additional mass	[N]	0	0	0
Summation	[N]	**-18146**	**-267889**	**-36318**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	226120	115490	5265,6
Force - hinge 2	[N]	312910	107730	5619,1
Force - cylinder 1	[N]	-325190	-15347	2444
Force - cylinder 2	[N]	-195730	-11061	14258
Force - Landing point	[N]	0	70386	8640,4
Summation	[N]	18110	267198	36227,1
Difference	[N]	-36	-691	-91
Deviation	[%]	**-0,2**	**-0,3**	**-0,3**

Wind pressures	**Value**	**Dimension**
Operational	0,000245	[Mpa]
Emergency discon.	0,000794	[Mpa]
Survival	0,001186	[Mpa]
Lateral surface area	2,24E+07	[mm2]

From the results we can conclude that there is a relative small deviation in the reaction forces. These discrepancies are due to numerical errors. These differences are small and are within the acceptable limits.

16.2 Gangway current design - Emergency disconnection case LC3

In the *emergency disconnection* case, the gangway is in uplift/cantilever position. The length of the loaded gangway is the maximum operational length without the safety length. The principle loads are applied including a live load on the gangway tip. The principal loads consists of the loads from the self-weight of the gangway structure and the loads due to the live loads. The live load shall be applied at the tip of the gangway. The inertia forces shall be taken into account by multiplying the self-weight of the gangway with the sum of the dynamic factor DF_z and the maximum vertical operational acceleration. The loads due to the motion of the vessel shall be included. The live load acting on the gangway tip shall be at least 350 kg.

Mesh

The mesh is identical to the stated mesh for the normal operational condition.

Connections and boundary conditions

The connections and boundary conditions are identical to the stated boundary conditions in the normal operational case. Except for the support at the tip of the gangway. In the emergency disconnection case the tip of the gangway is unsupported.

Accelerations

A standard Earth gravity of 9806.6 mm/s2 is applied to the geometry. In the emergency disconnection case, dynamic factors are applied. These dynamic factors are specified by the DNV [25] and given in Table 83.

Table 83: Dynamic factors

Direction	**Dynamic factor**
	[-]
X	1
Y	1,10
Z	1,10

Additionally, the accelerations are given in Table 84.

Table 84: Accelerations emergency condition

Direction	**Operational**	**Total (Incl. dyn. Factor)**	**Total (Incl. gravitation)**
	[mm/s²]	***[mm/s²]***	***[mm/s²]***
X	1420	1562	1562
Y	836	836	836
Z	1540	1694	12481,26

Point masses

The applied point masses are identical to the point masses as in normal operational situation.

The live load

The live load is applied as a remote force. This load is applied at the tip of the gangway structure as can be seen in Figure 134. The tip load is equal to the live load which is equal to 10800 N.

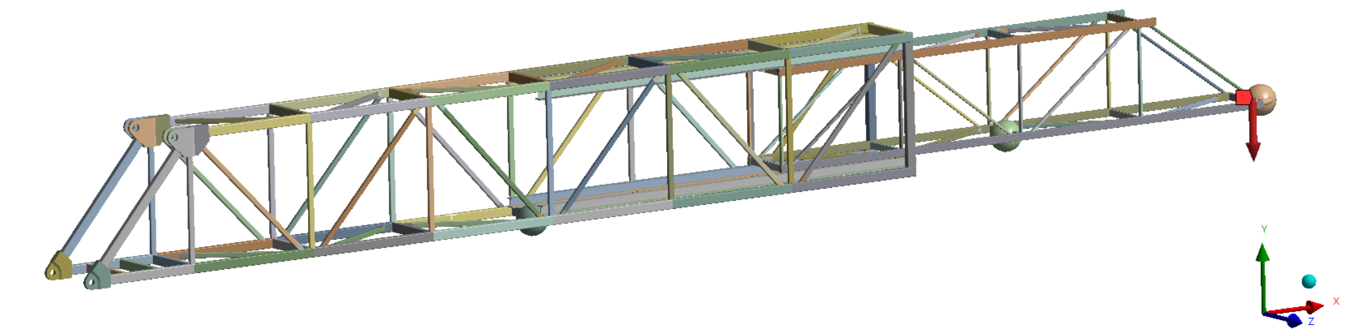


Figure 134: The applied tip load

The wind load

The deployment/retrieval wind pressure of 794 Pa is applied to the lateral surface area of the gangway structure. The lateral surface area is shown in red in Figure 135.

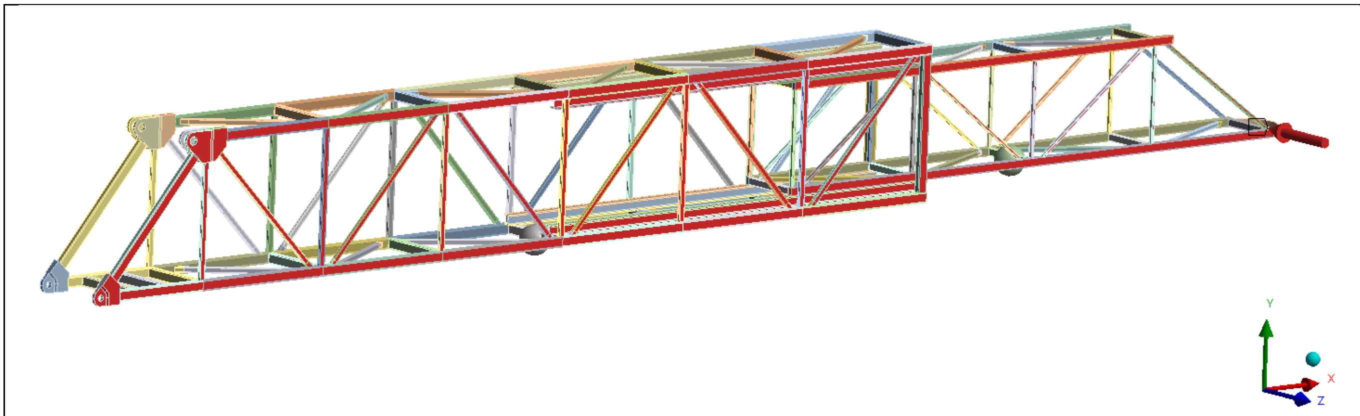


Figure 135: The applied wind pressure

Stress plots

The stress plots for the emergency disconnection case are reported in this section. The stresses over 50, 100, 150, 323 (Acceptance criteria) and 355 MPa (Yield limit) are plotted. Also the overall von-mises stresses are plotted.

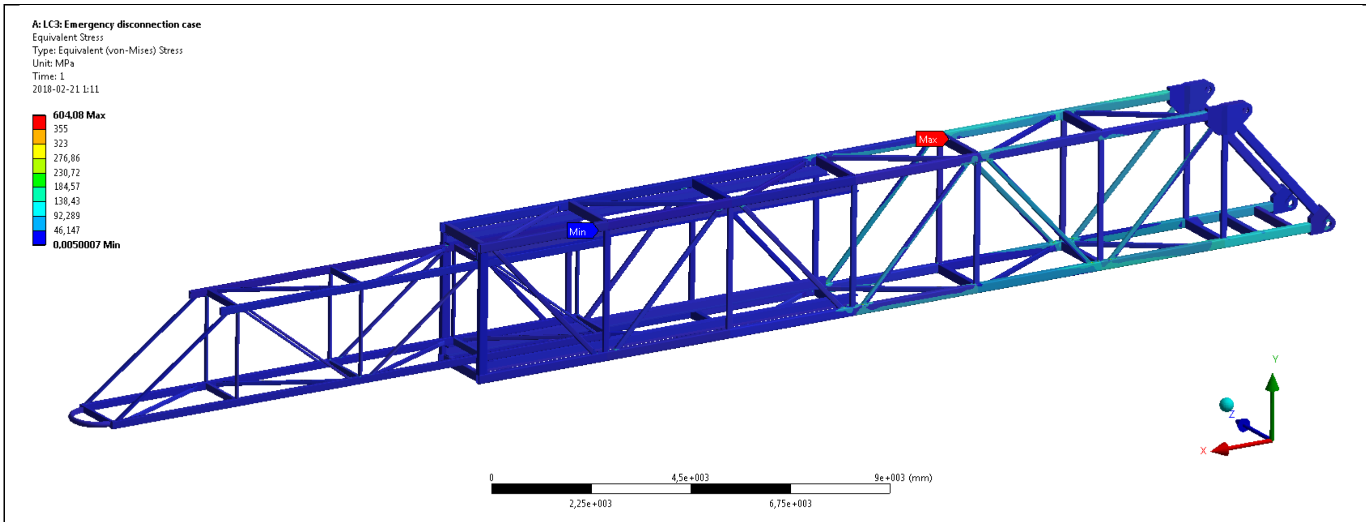


Figure 136: Overall Von Mises stresses

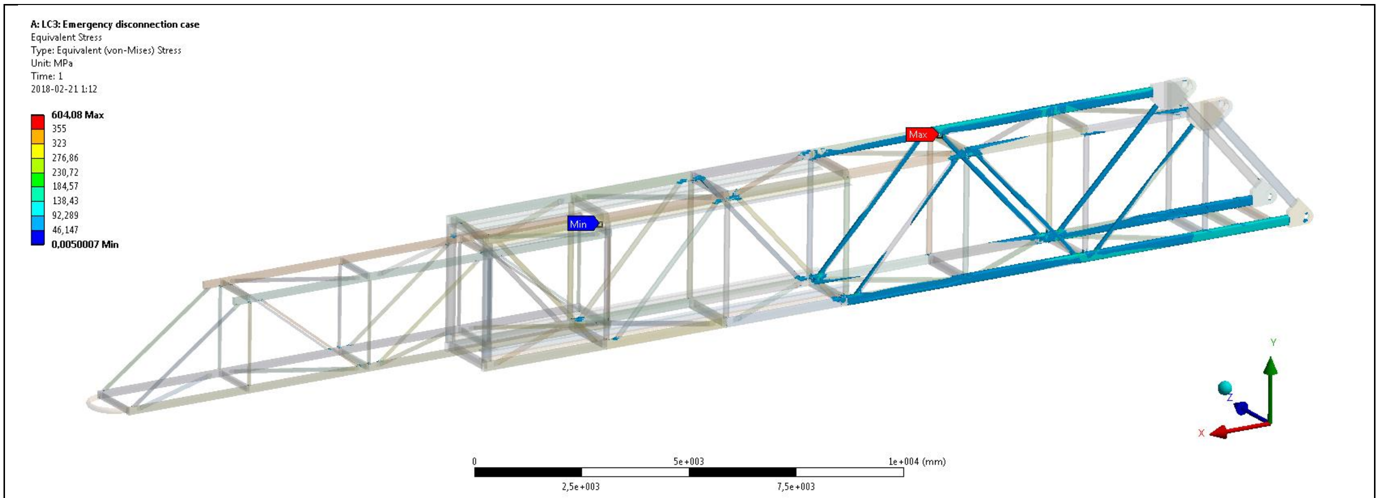


Figure 137: Stresses over 50 MPa

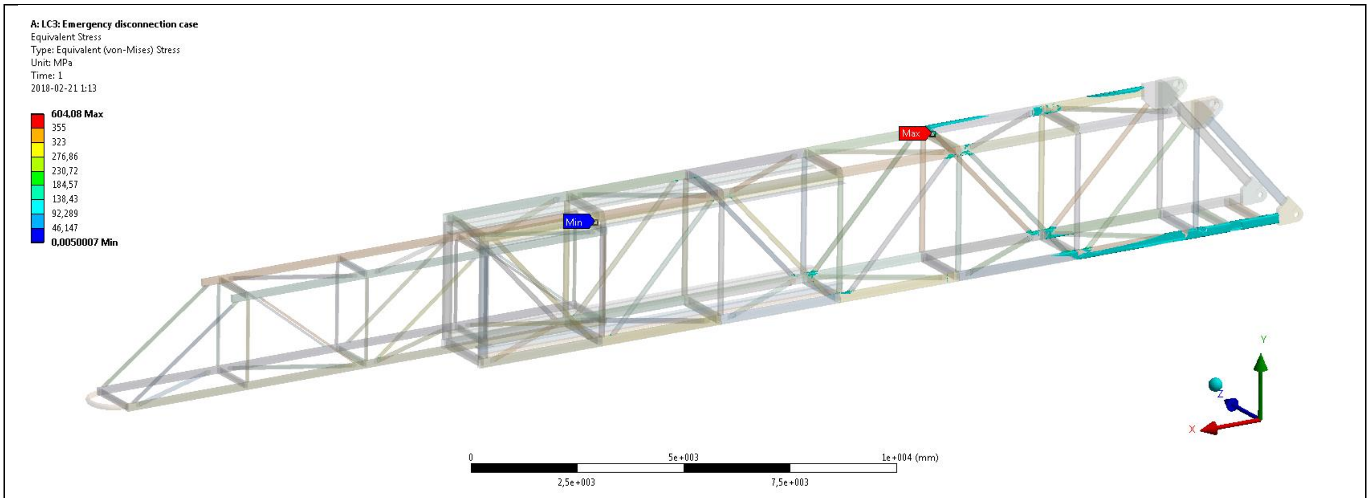


Figure 138: Stresses over 100 MPa

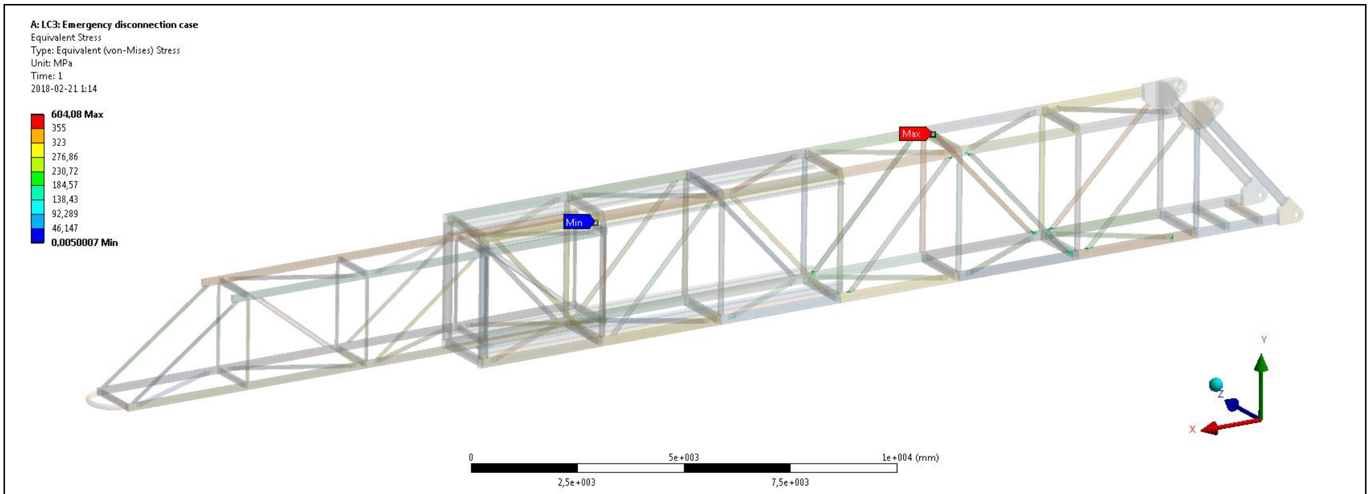


Figure 139: Stresses over 150 MPa

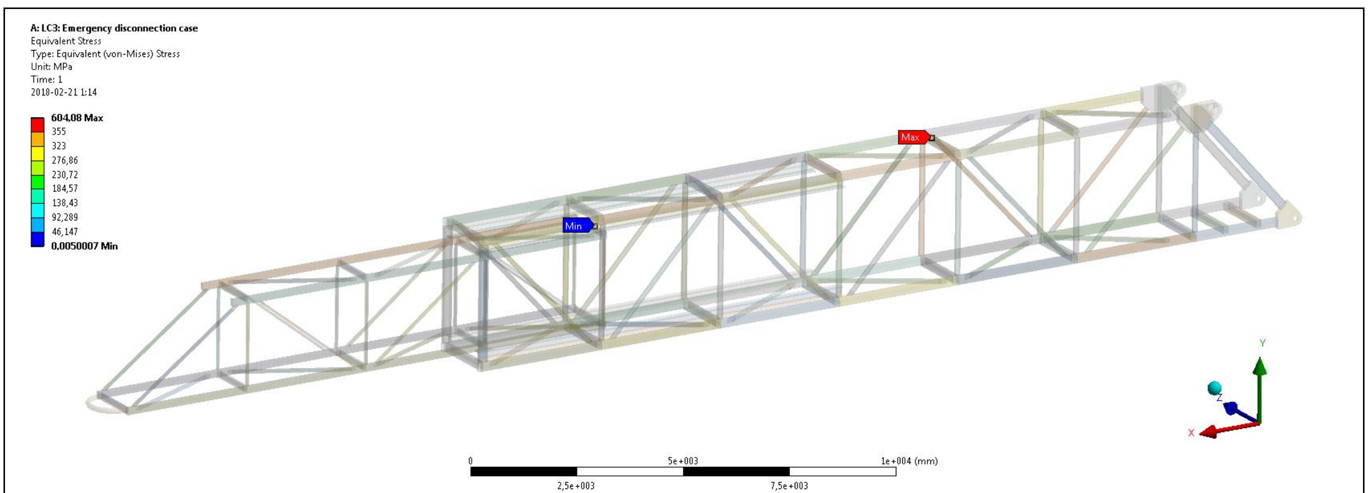


Figure 140: Stresses over 323 MPa

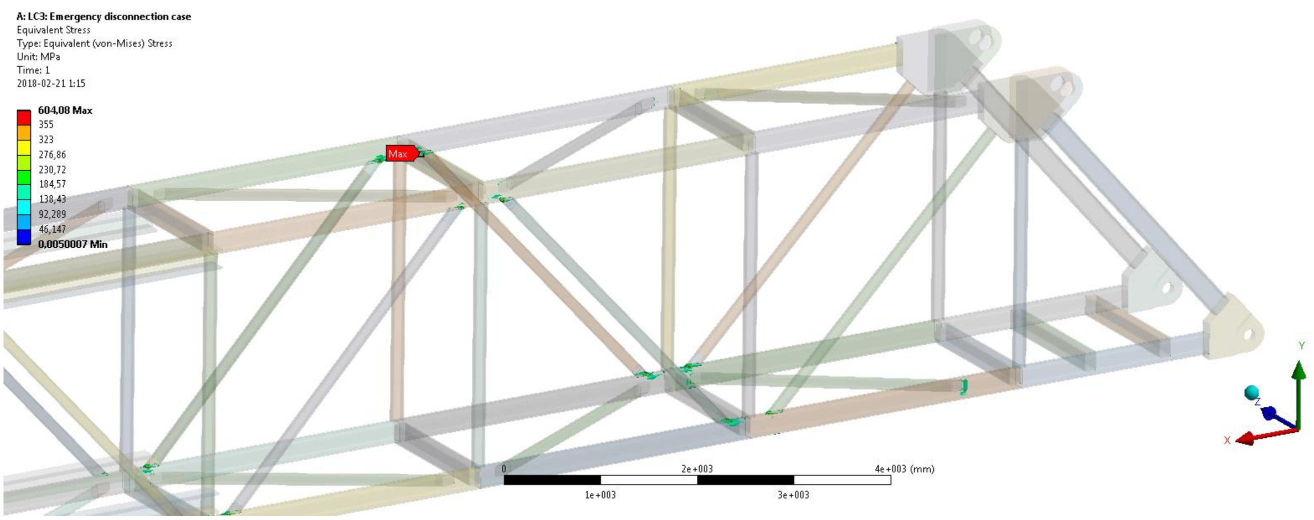


Figure 141: Detail view of high stress locations

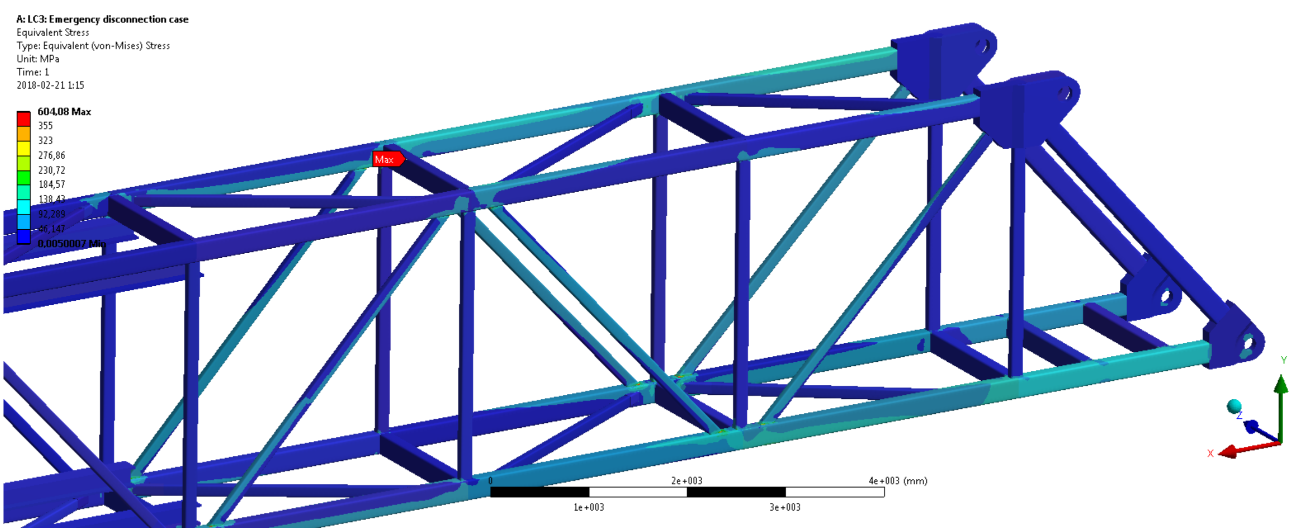


Figure 142: Detail view of the high stress locations

The general stress level is lower than 355 MPa / 1.1 = 323 MPa and is therefore considered acceptable. The plots with detail views of the peak stress areas show the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and only single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines [25]. For the emergency disconnection load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{100}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33 meter. This means that the maximum deflection limit of the tip is 330 mm.

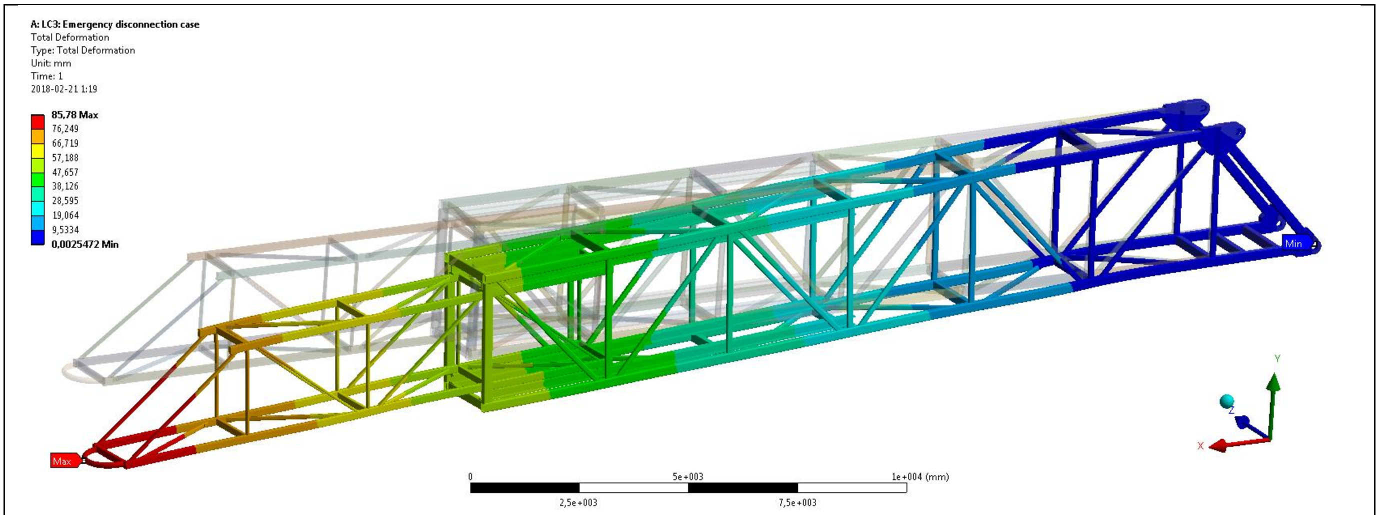


Figure 143: Total deformation, scale = 24

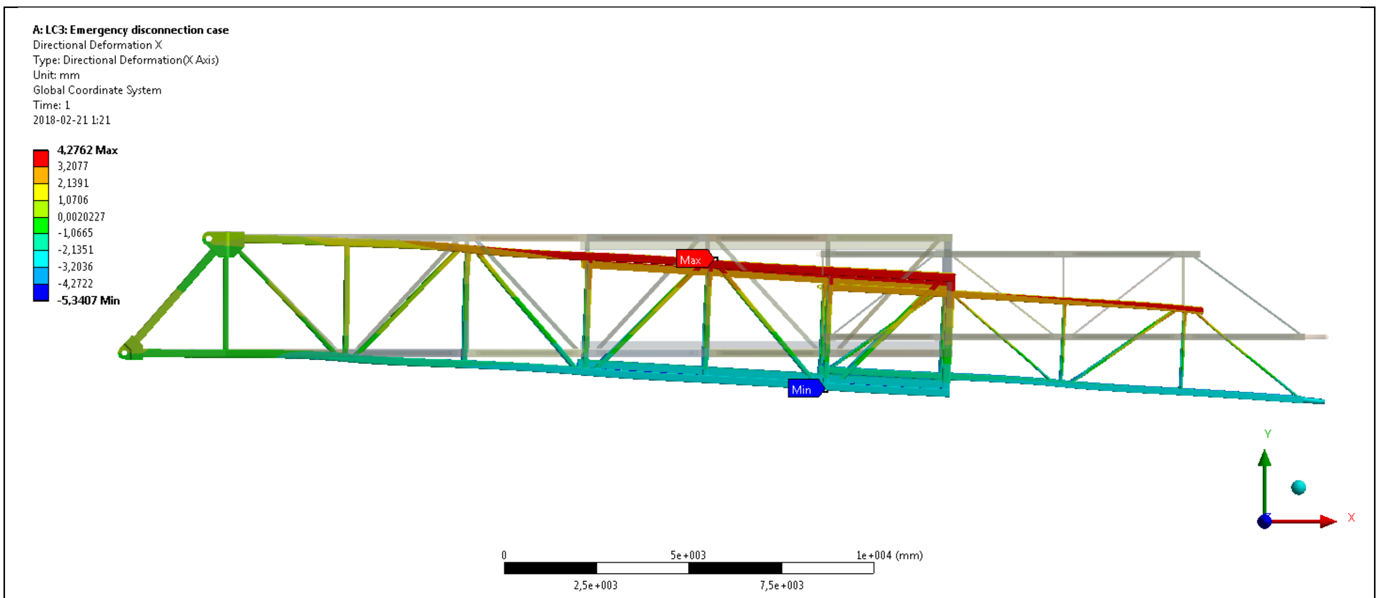


Figure 144: Directional deformation, X-direction, scale = 24

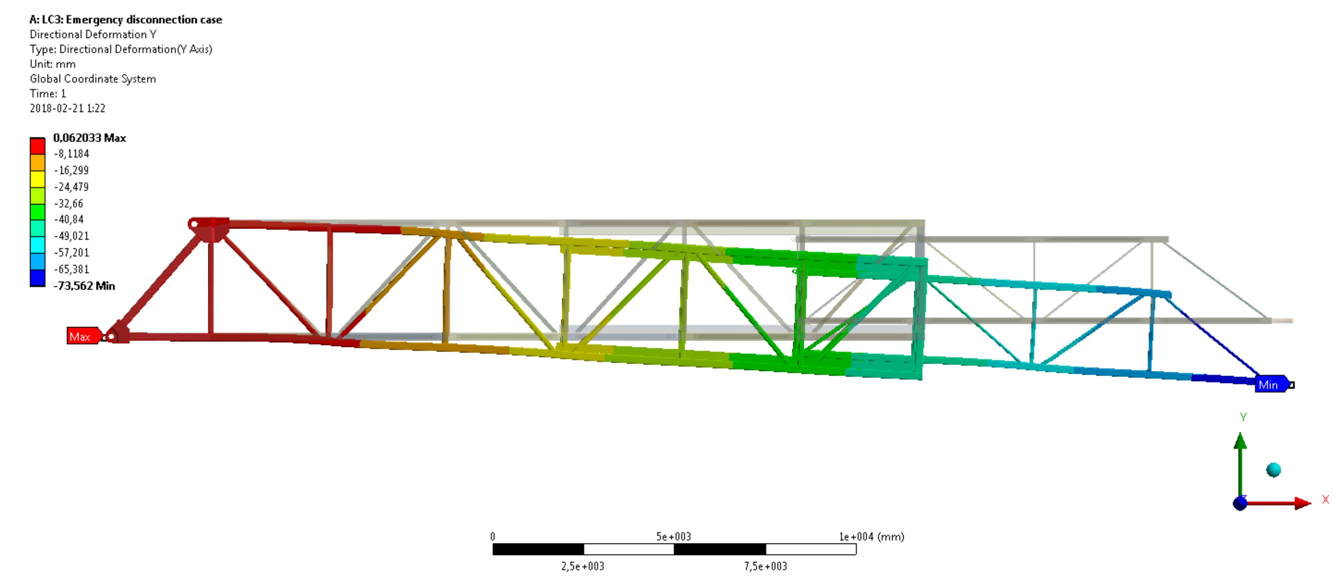


Figure 145: Directional deformation, Y-direction, scale = 24

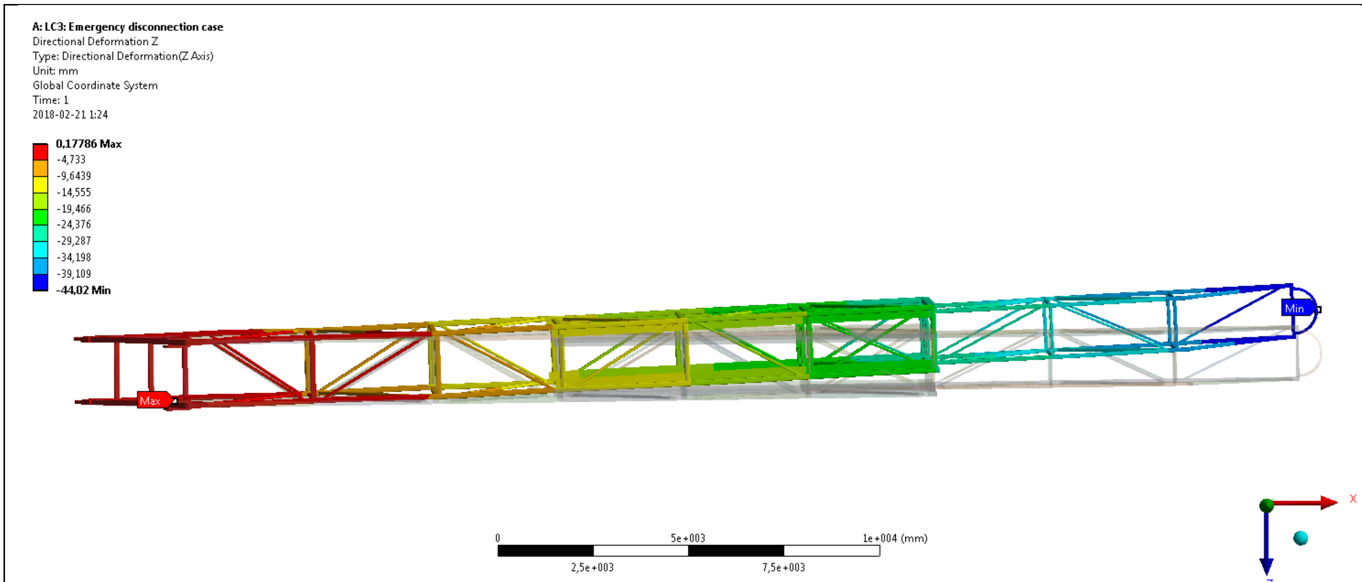


Figure 146: Directional deformation, Z-direction, scale = 24

The total deformation of the gangway structure is equal to 73,65 mm in the vertical direction. The total combined deformation is equal to 85,78 mm. These values are well within the stated limit and therefore considered acceptable.

Reaction force check

A reaction force check is performed in order to check the FEA. The results of this check can be found in the table below.

Table 85: Reaction force check

Reaction force check				
LC3: Emergency disconnection				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	21,706	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	-836	-1694	-1562
Geometry	[N]	-18146	-249632	-33905
Tip load	[N]	0	-10800	0
Wind load	[N]	0	0	-17810
Additional mass	[N]	0	0	0
Summation	[N]	**-18146**	**-260432**	**-51715**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	506810	179590	10217
Force - hinge 2	[N]	796450	142340	11258
Force - cylinder 1	[N]	-410090	-23253	24930
Force - cylinder 2	[N]	-875050	-38949	5200,5
Summation	[N]	18120	259728	51605,5
Difference	[N]	-26	-704	-109
Deviation	[%]	**-0,1**	**-0,3**	**-0,2**

Wind pressures	**Value**	**Dimension**
Operational	0,000245	[Mpa]
Emergency discon.	0,000794	[Mpa]
Survival	0,001186	[Mpa]
Lateral surface area	2,24E+07	[mm2]

There is a small deviation in all directions. This discrepancy is due to small numerical errors related to the mesh. But these results are within acceptable limits. The FEA analysis demonstrates that the gangway structure in emergency disconnection case has a general stress lower than the allowable 323 MPa and is considered acceptable. The structure has a maximum resulting deformation of 85,78 mm. This is mainly the deformation of 73,65 mm in the y-direction. A deformation and stress criteria is applied according to the DNV guidelines.

16.3 Gangway optimized design – Normal working conditions LC 1c

In the normal working condition the motion compensated gangway is in operational mode. This means that gangway structure is able to transfer people and cargo from and to the offshore structure. For the normal working condition the gangway shall be designed for the following scenario:

- The live load (LL) on the gangway shall be the maximum number of persons, including hand tools and luggage allowed on the gangway at the same time. The load will be applied at the most limiting location.

The design load of the gangway shall be equal to two times the live load. The live load represents the allowable number of persons on the gangway at one moment. Only one person is allowed on the gangway during operation.

Mesh

The model is mainly built with solid elements and not with solid shell elements. The reason for this is that a solid shell mess causes an error in the numerical solution for some of the parts. All the circular hollow members are meshed with solid shell elements. The rectangular hollow tubes are meshed with solid elements. The global mesh size is 20mm and mess refinements are applied at the hinges and the cylinder points. Also mesh refinements are applied at the connecting joints of the diagonal members. Mesh refinements on bodies with solid elements result in at least 2 elements through the thickness. The total number of elements is equal to 1212574, which is quite significant.

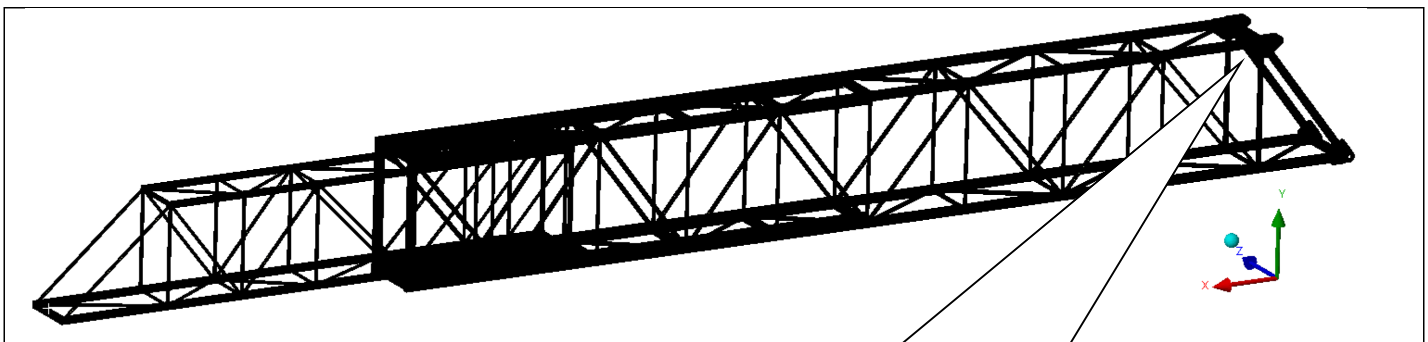


Figure 147: Applied mesh and typical detail for clarity

Connections and boundary conditions

Almost all the connections in the model are bonded. A tolerance setting of 1mm is allowed for the detection of the contact regions. The manual connections are made in the model as shown in figure 3-2. The red area represents a no separation contact, while the blue area is a bonded contact. The reason for this choice is that the study will remain linear which reduces the required solving time. The non-active supports will be suppressed during the analysis. The can be substantiated by the fact that the active support reactions can be determined by solving the free body diagram of the structure.

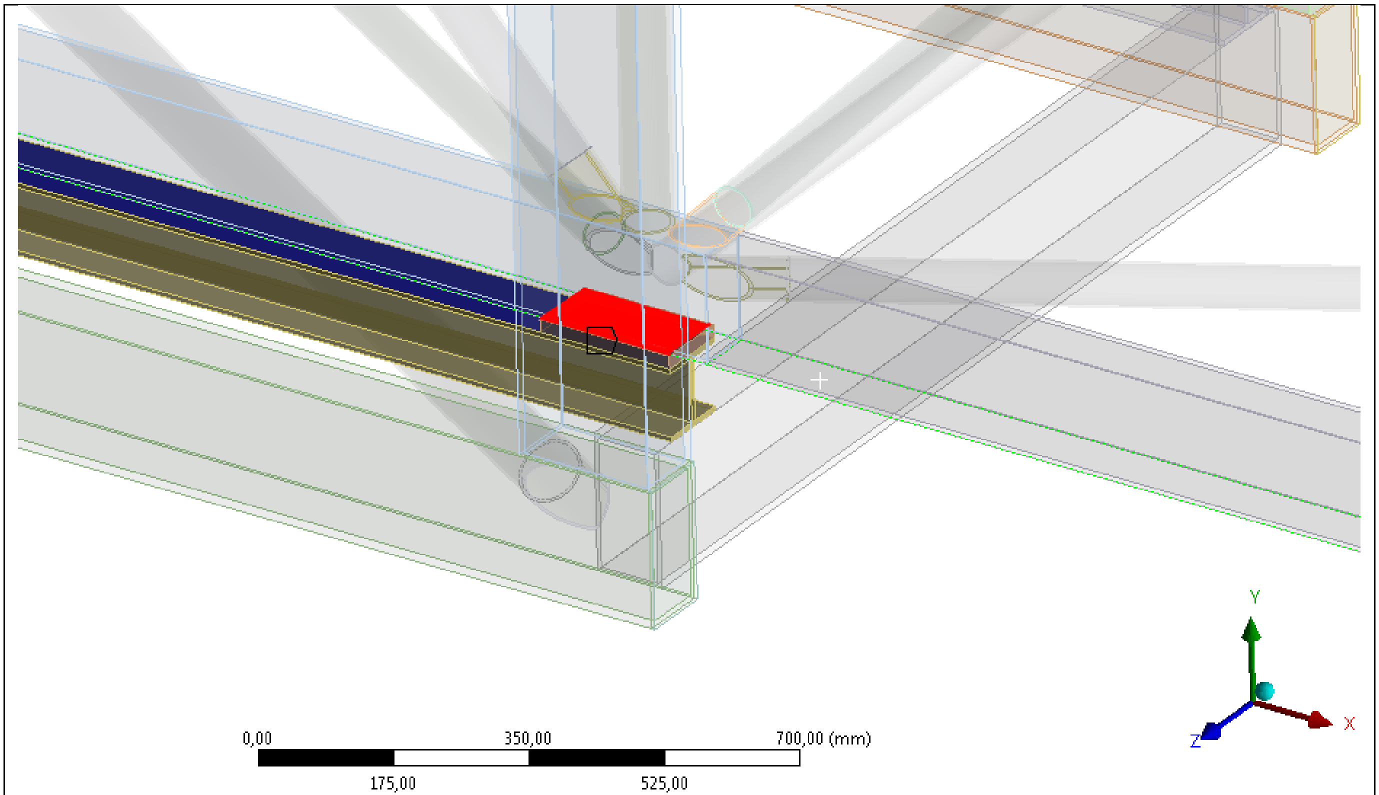


Figure 148: The no separation contacts

The telescopic part is supported by sliding pads. These sliding pads represents the supporting area and these are made of the D-glide FC material. This is a composite material which is designed for bearing and sliding capabilities. For the structural analysis of the gangway boom, these sliding pads are used to represents the guidance rollers which are used in the final design.

The linear guidance rollers are visualized in Figure 149. Two pairs of rollers are located on the fixed section of the gangway structure. Two pairs of rollers are located on the telescopic section of the gangway structure. The reason for this layout is that in this orientation, the roller bearings on the left side will move with the stroke of the bridge. The roller bearings on the right side are always located on the end of the fixed section. Therefore in this orientation, two sets of roller bearings will always be in contact with the gangway structure. The set which is not in contact with the gangway structure, will be suppressed during the analysis.

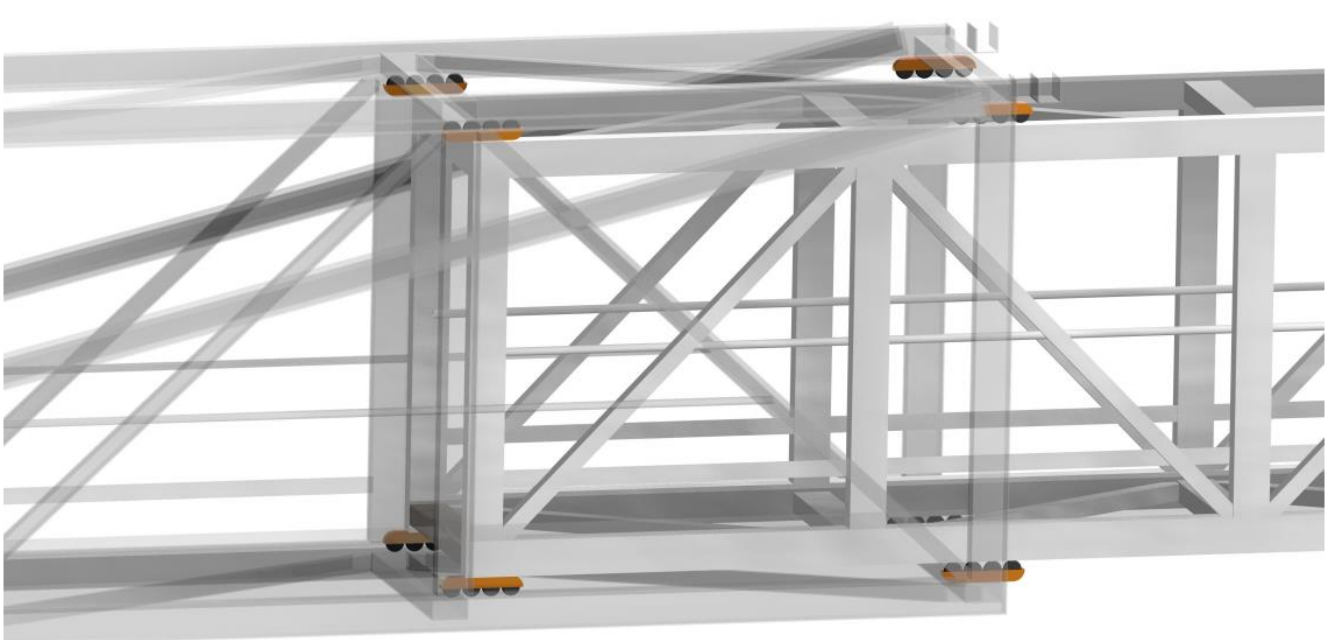


Figure 149: The guidance rollers

Joints are generated in order to restrict the movement of the telescopic section in the Z-direction. Normally, roller bearings are applied to the structure which will transfer the loads from the telescopic section to the fixed section. In this stage of the design, only the structural behaviour of the structure must be examined and therefore it is chosen to replace these roller bearings for joints in the optimization process. Four additional joints are generated which constraints the translation in the Z-direction and the rotation around the X and Y-axis.

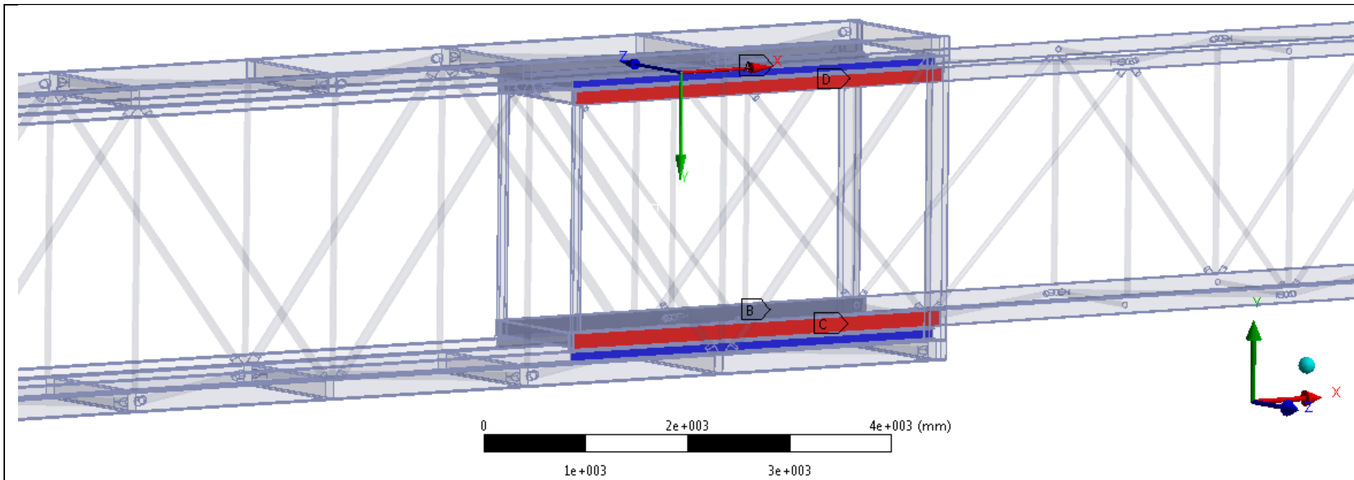


Figure 150: The applied joints

In the normal operational condition, the structure is constrained by means of remote displacement supports at the hinges and at the landing mechanism. For the hinges, every DOF except the axial rotation is constrained. The landing mechanism is located at the tip of the gangway. This system constrains the displacements in the X, Y and Z-direction and only allows rotation around the Y and Z-axis.

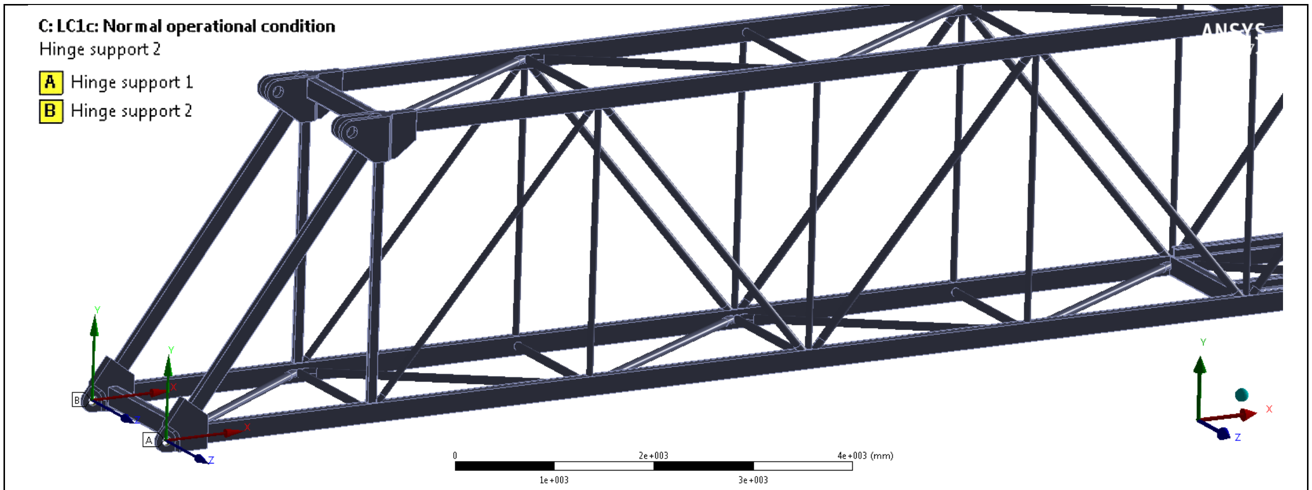


Figure 151: Remote displacements for the hinges

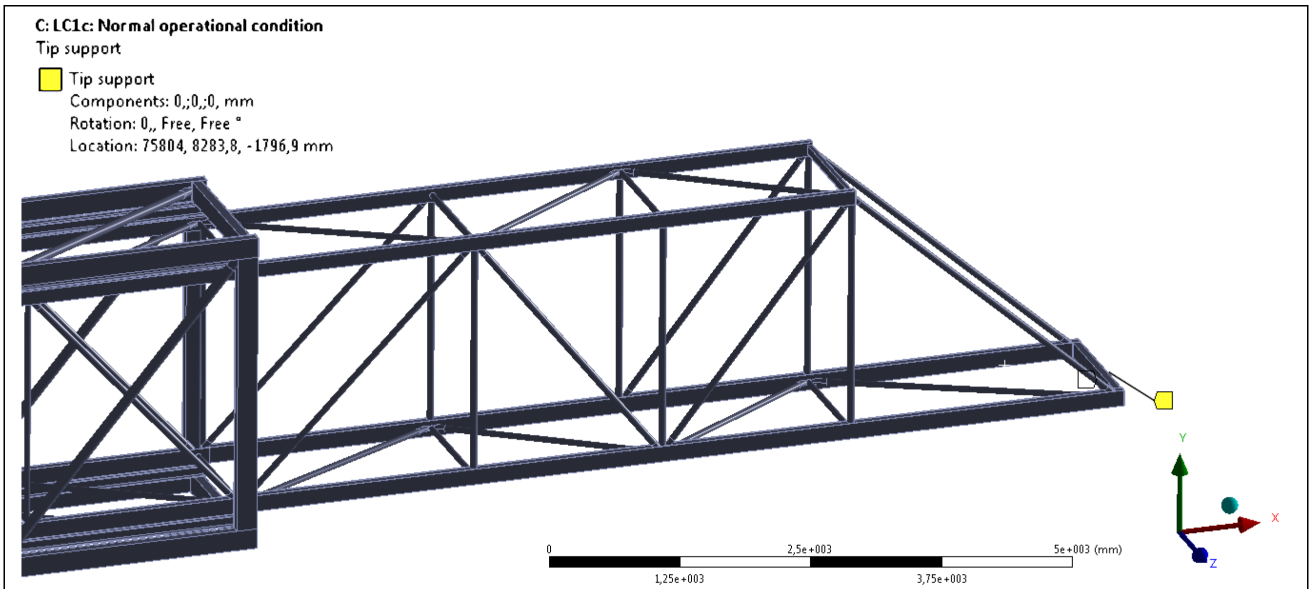


Figure 152: Remote displacement for the landing mechanism

The two sections of the gangway are connected by means of an adjusting cable. This adjusting cable is used to accommodate for the telescopic motion. A double ended hydraulic winch is used in order to pull the telescopic section in the positive or negative X-direction. The cable is connected to the hinge point which is shown in Figure 153.

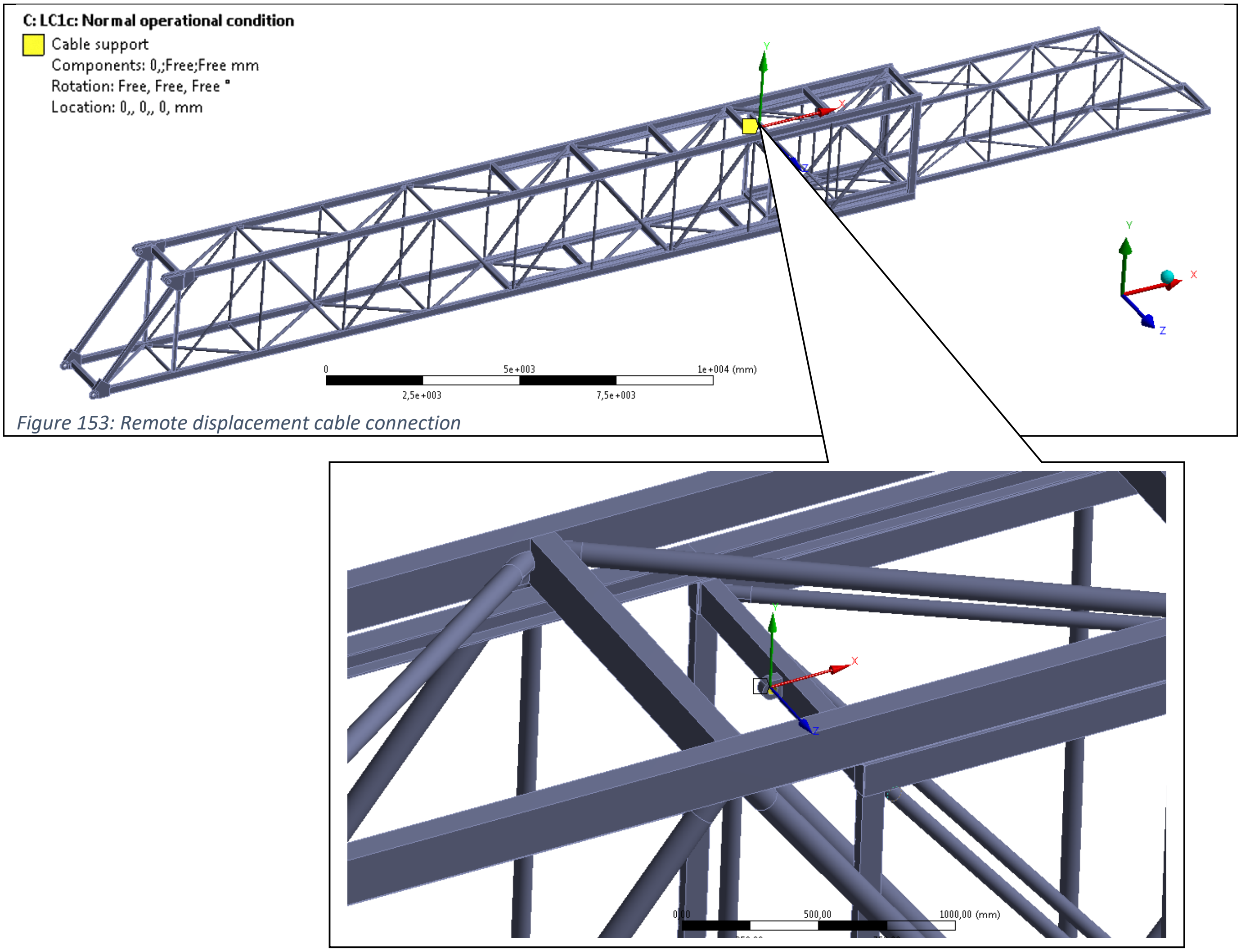


Figure 153: Remote displacement cable connection

The DOF of the remote displacements are given in Table 86.Table 84

Table 86: Constraints normal working condition

DOF	Hinges	Cable connection	Landing mechanism
X Component	0 mm	0 mm	0 mm
Y Component	0 mm	Free	0 mm
Z Component	0 mm	Free	0 mm
Rotation X	0°	Free	0°
Rotation Y	0°	Free	Free
Rotation Z	Free	Free	Free

Material properties

The assumed primary structure material is S355 or similar, with the following applicable properties according to Table 78. The sliding pads are made of the D-glide FC material which has the properties according to Table 79.

Loads and accelerations

In this chapter all the applied forces and accelerations are defined. A standard Earth gravity of 9806.6 mm/s^2 is applied to the geometry. Additionally, the accelerations are given in Table 80.

Table 87: Operational accelerations

Direction	Operational
	[mm/s²]
X	1420
Y	1540
Z	840

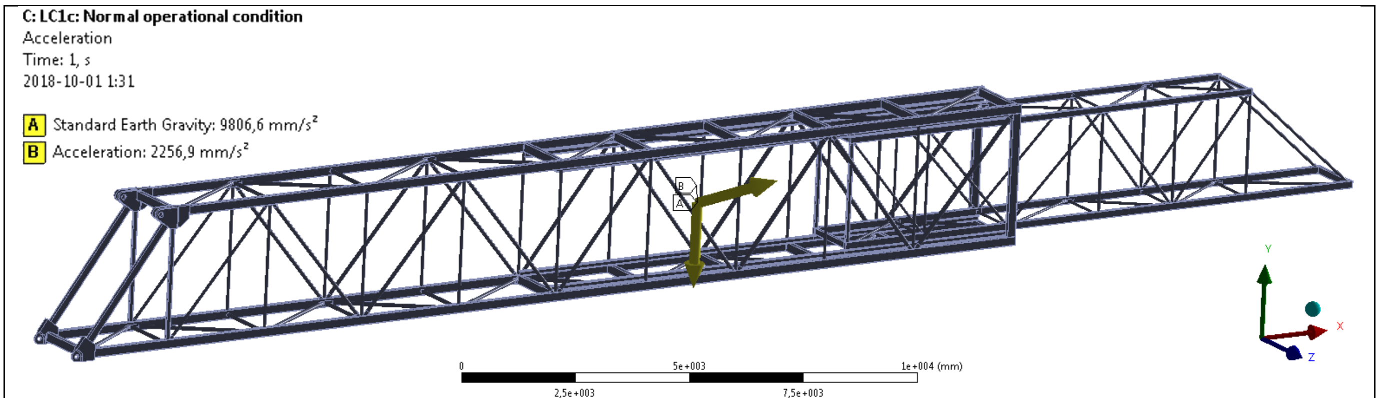


Figure 154: Typical acceleration application

Note that Ansys creates a reaction force opposite to the direction of the applied acceleration, therefore the accelerations are placed in opposite direction of the actual accelerations. Given values are multiplied with -1.

Forces

In addition to the self-mass of the gangway structure, the gangway boom will be loaded by additional components such as: the rollers, flooring, hand rails, utility lines and the landing tool. For the structural evaluation of the gangway structure these additional masses are represented by distributed masses and point masses. The locations of point masses are shown in Figure 155. The distributed masses are applied to all the components of each section.

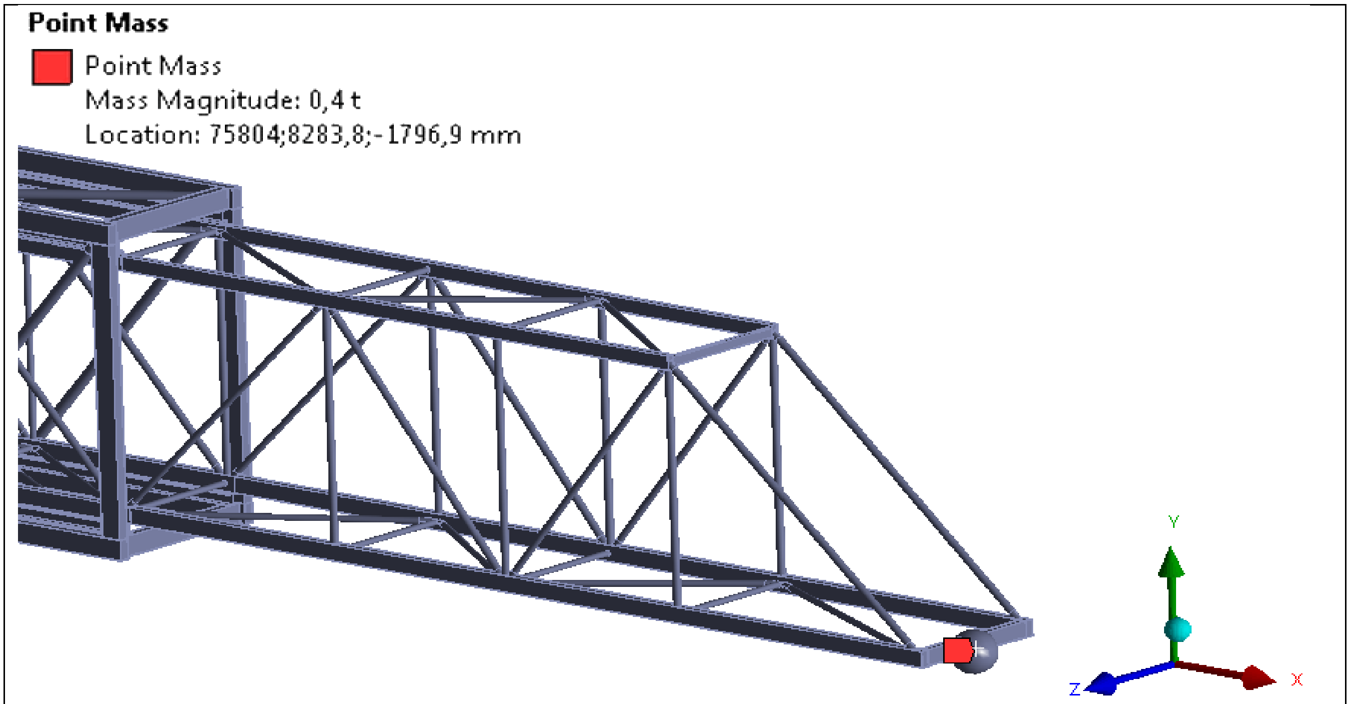


Figure 155: Point mass applied to the model

An additional mass of 3255 kg is applied for the fixed part and 845 kg for the telescopic part. The mass of the landing tool is located at the tip of the gangway structure and is equal to 400 kg.

The live load

The live load is applied as a remote force. This load is applied in the middle of the gangway structure as can be seen in Figure 156. The design load of the gangway shall be equal to two times the live load. The magnitude of the live load is equal to 10800 N as is determined in appendix 5.

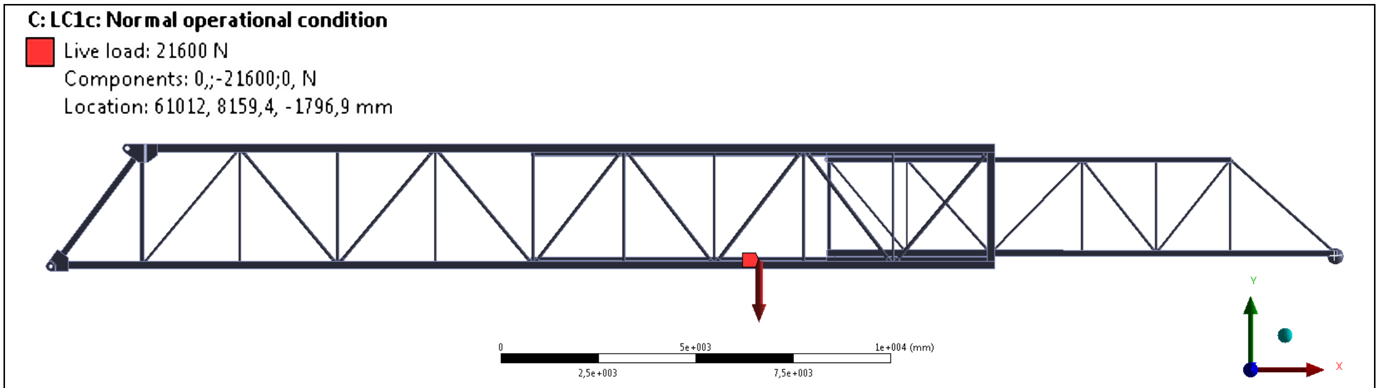


Figure 156: Live load acting on the gangway

Two times this live load means that the gangway design must be subjected to a total load of 21600 N which is acting exactly in the middle of the gangway in operational conditions.

The wind load

The gangway structure is subjected to wind loads. The wind load normal to a flat surface is calculated according to DNV. The value of the air velocity pressure is calculated and applied to the lateral surface area of the gangway structure. This pressure is only working in the normal direction of the face on which it is acting. The values in Table 81 are calculated according to the DNV. [25]

Table 88: Wind pressures

Load condition	**Wind velocity**	**Wind pressure**
	[m/s]	[Pa]
Operational wind speed	20	245
Deployment/retrieval wind speed	36	794
Transit/survival/parked wind speed	44	1186

The wind pressure is applied to the red surface area as shown in Figure 157. This wind pressure is applied on the lateral surface of the gangway structure. This pressure is only working in the normal direction of the face on which it is acting. The wind pressure is calculated for the total lateral surface area of the gangway structure and is applied to all the main members of the structure.

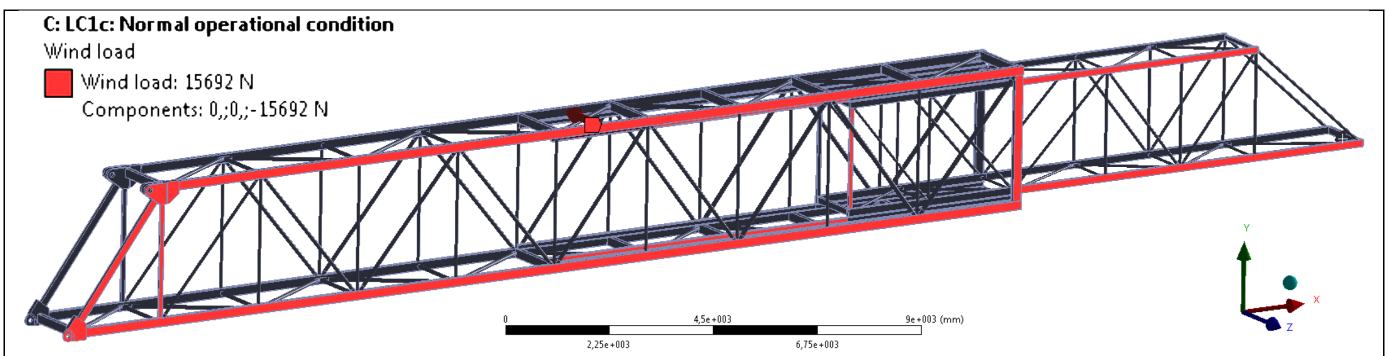


Figure 157: Lateral surface area wind pressure

Stress plots

The stress plots for the normal working conditions are reported in this section. The stresses over 50, 100, 150, 237 (Acceptance criteria) and 355 MPa (Yield limit) are plotted. Also the overall von-mises stresses are plotted.

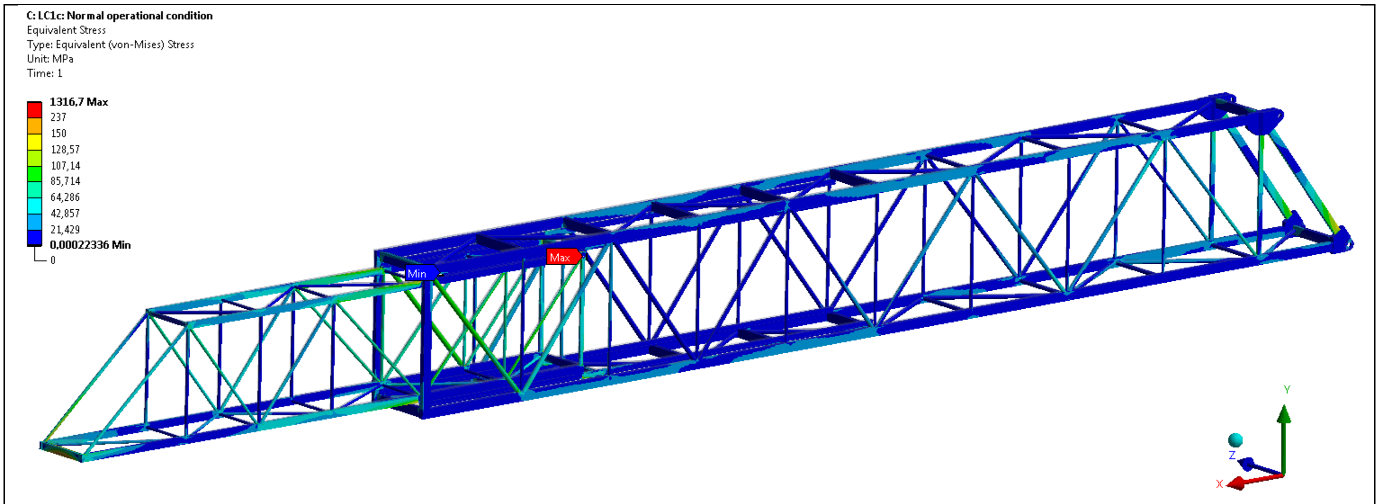


Figure 158: Overall Von Mises stress

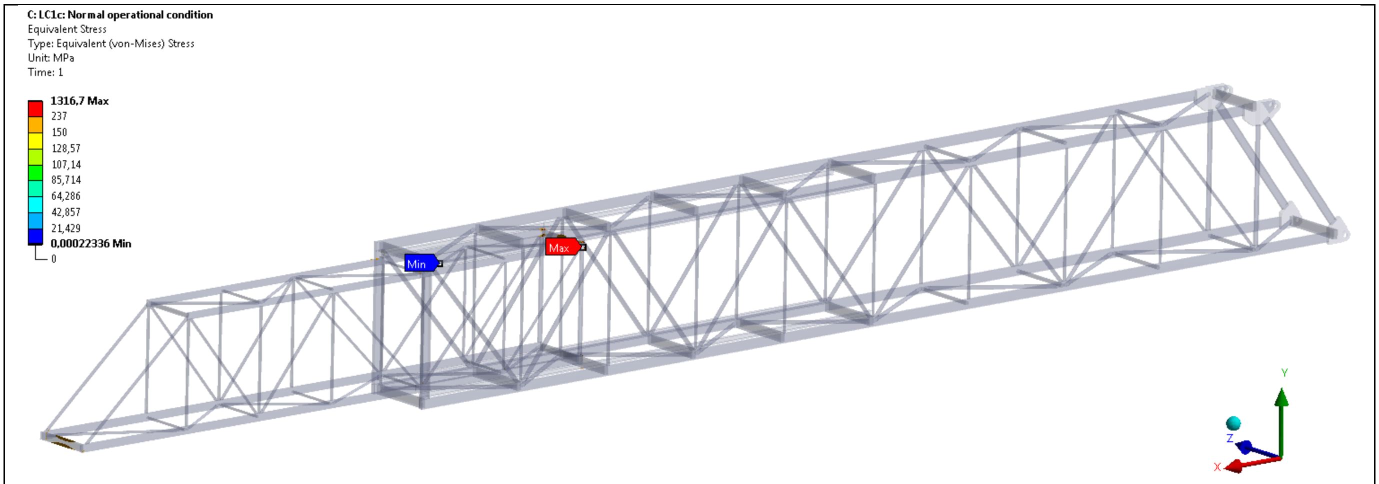


Figure 159: Stresses over 150 MPa

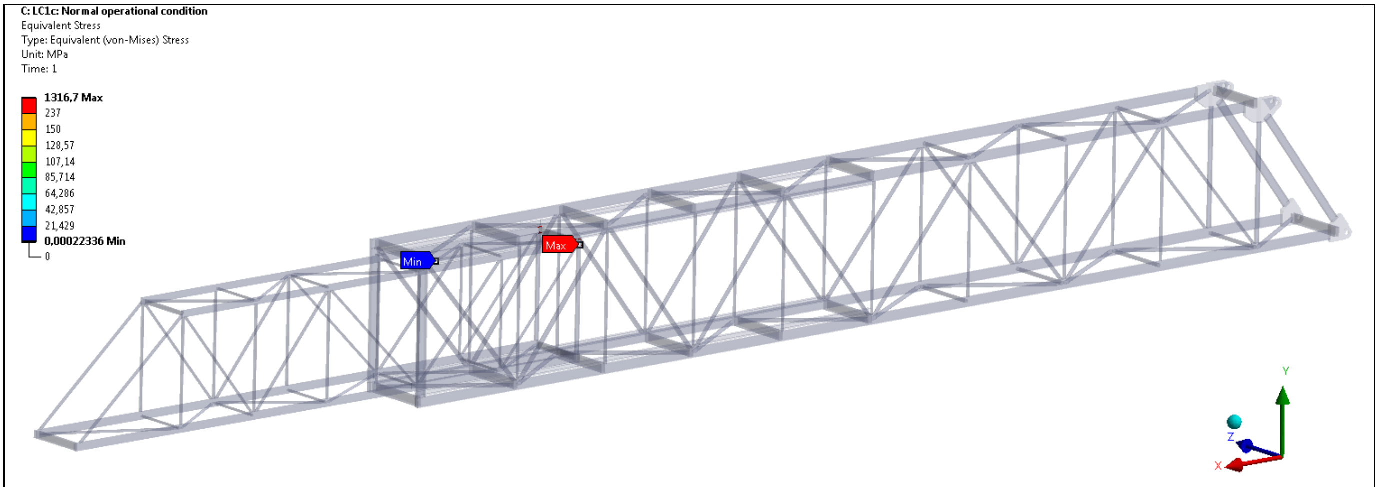


Figure 160: Stresses over 237 MPa

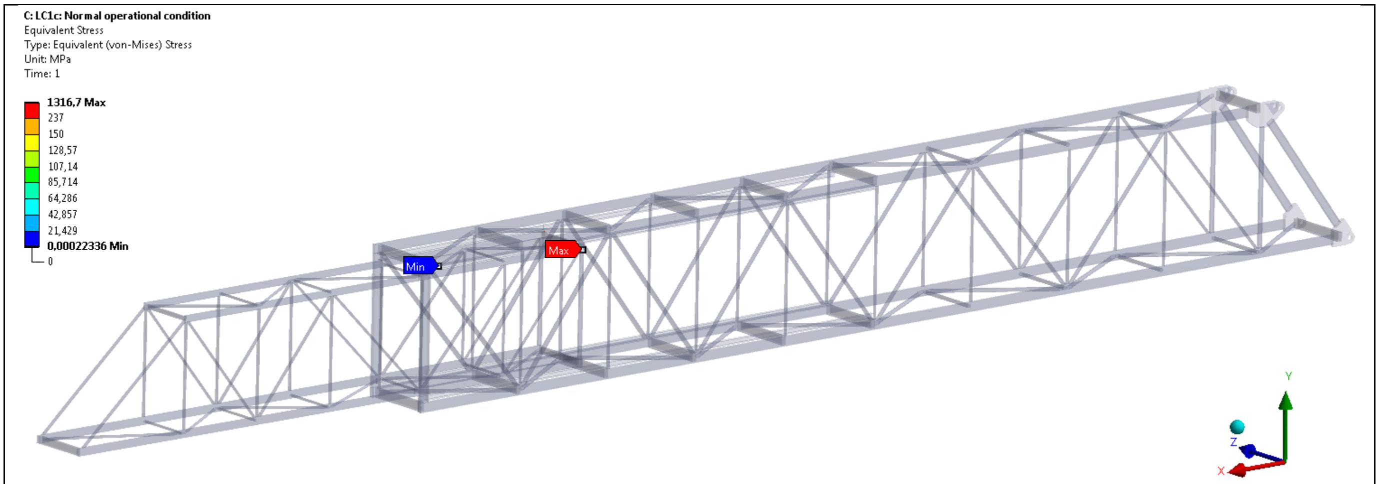


Figure 161: Stresses over 355 MPa

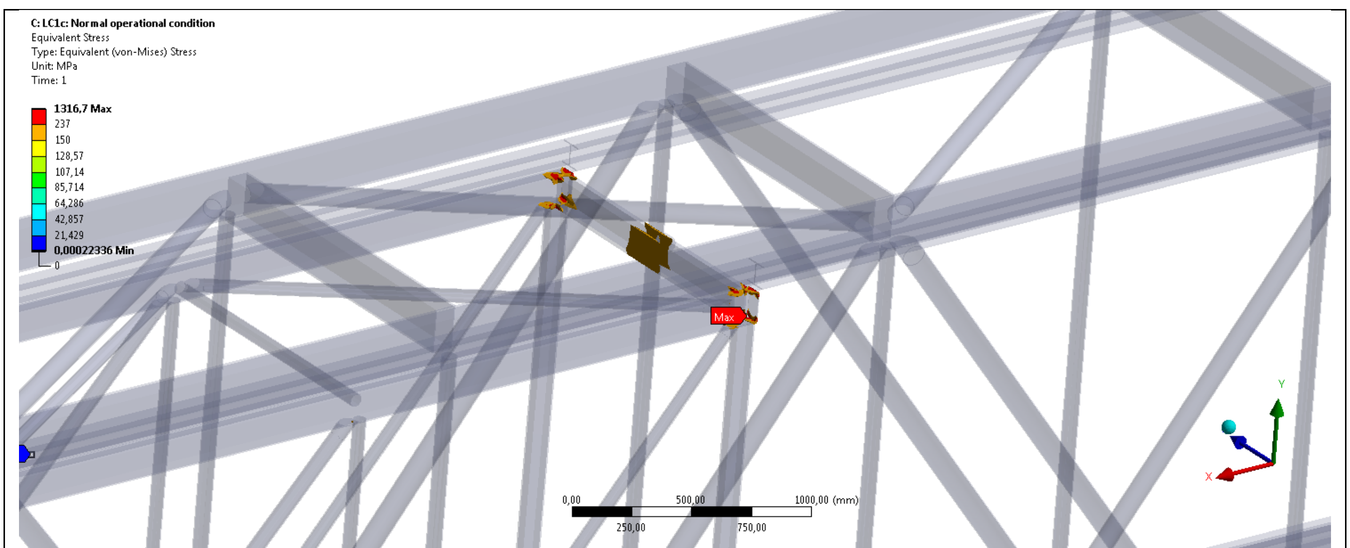


Figure 162: Detail view of highest stress locations

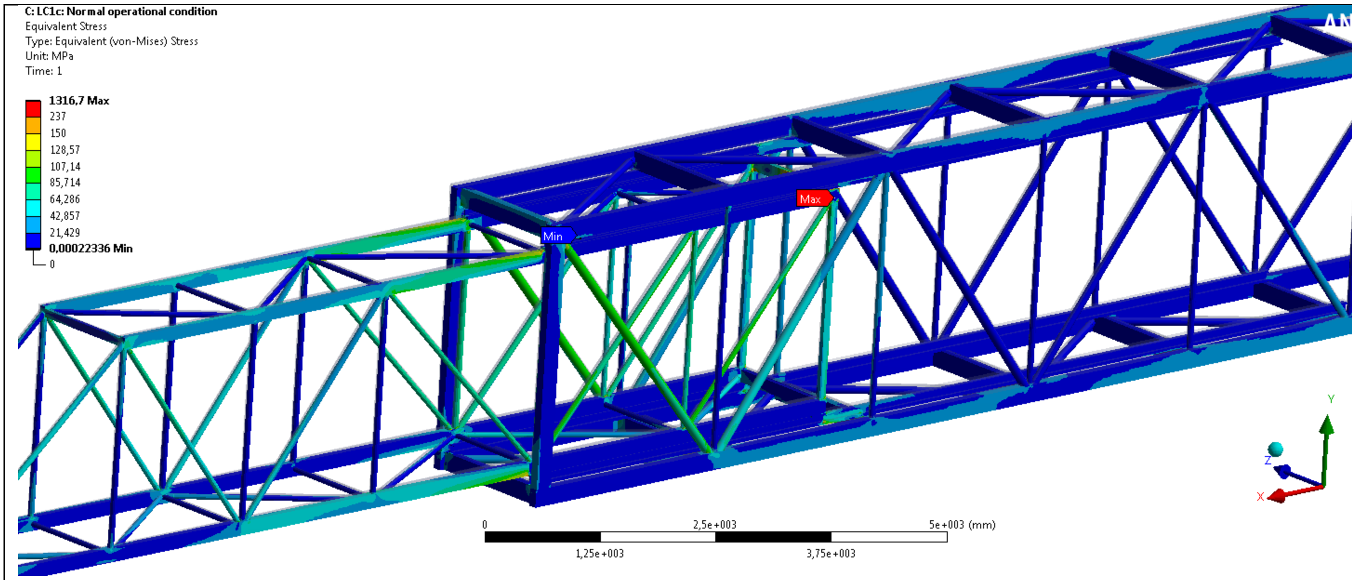


Figure 163: Detail view of the highest stress locations

The general stress level is above 355 MPa / 1.5 = 237 MPa and is therefore considered unacceptable.

The plots with detail views of the peak stress areas show the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and needs to be reinforced.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines [25]. For the operational load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{200}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33 meter. This means that the maximum deflection limit is 165 mm.

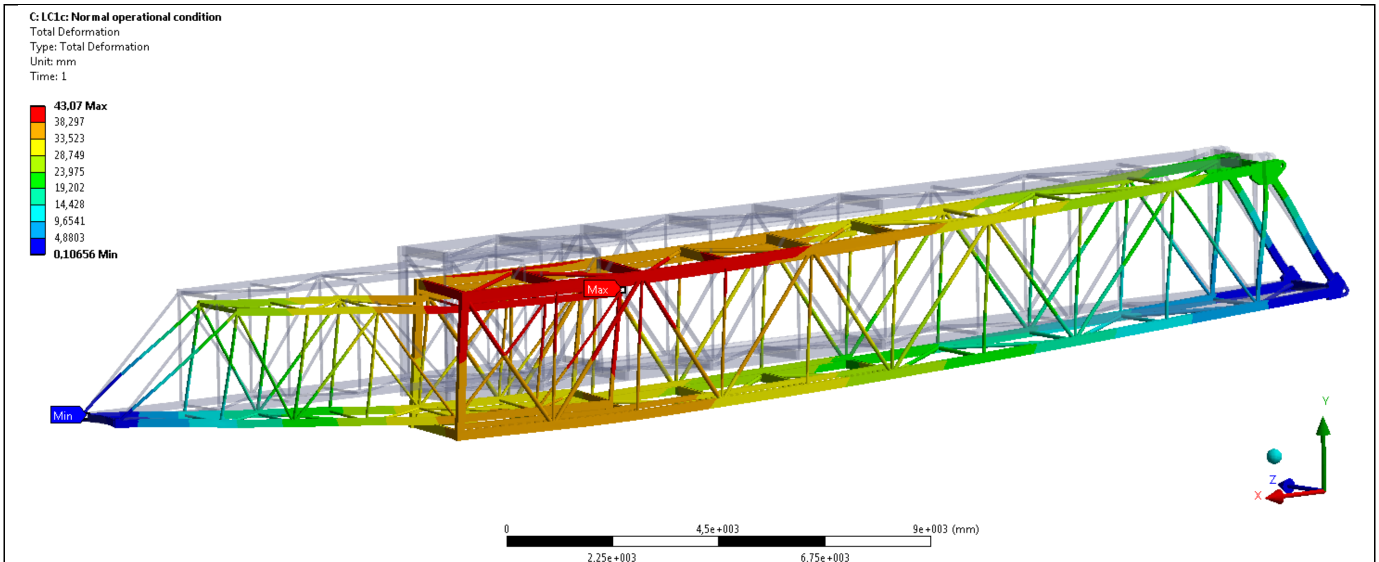


Figure 164: Total deformation, scale = 24

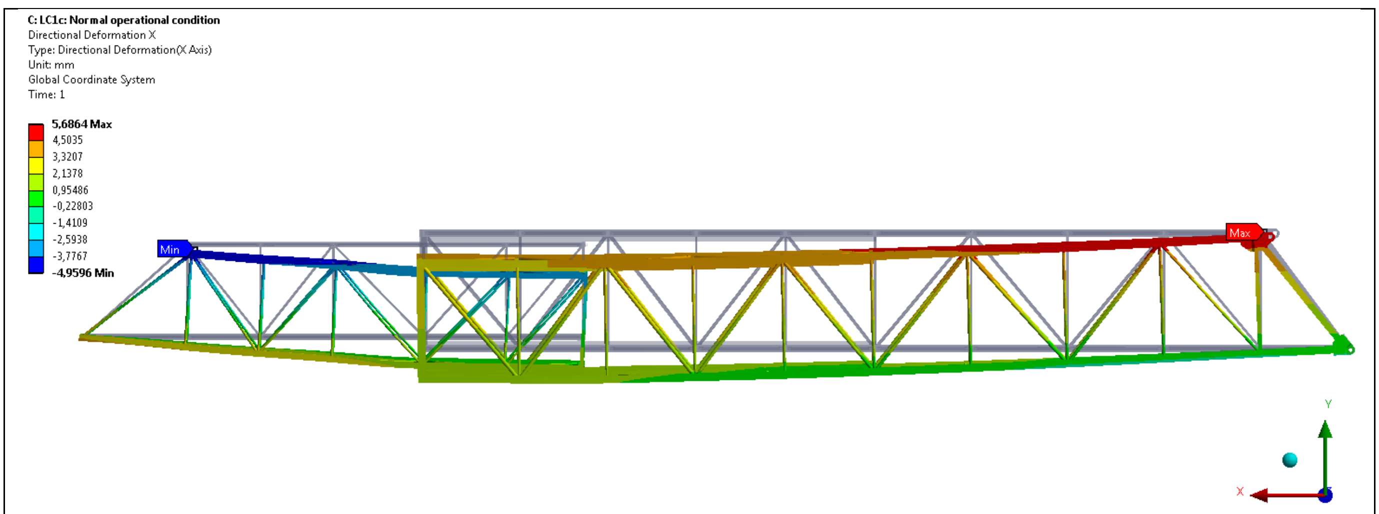


Figure 165: Directional deformation, X-direction, scale = 24

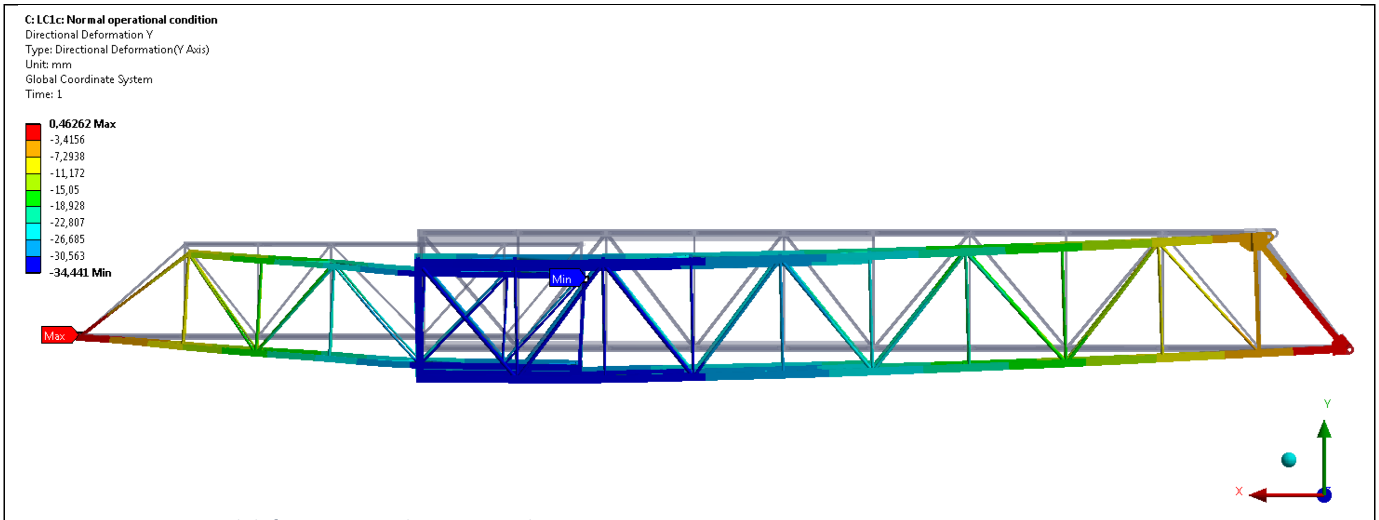


Figure 166: Directional deformation, Y-direction, scale = 24

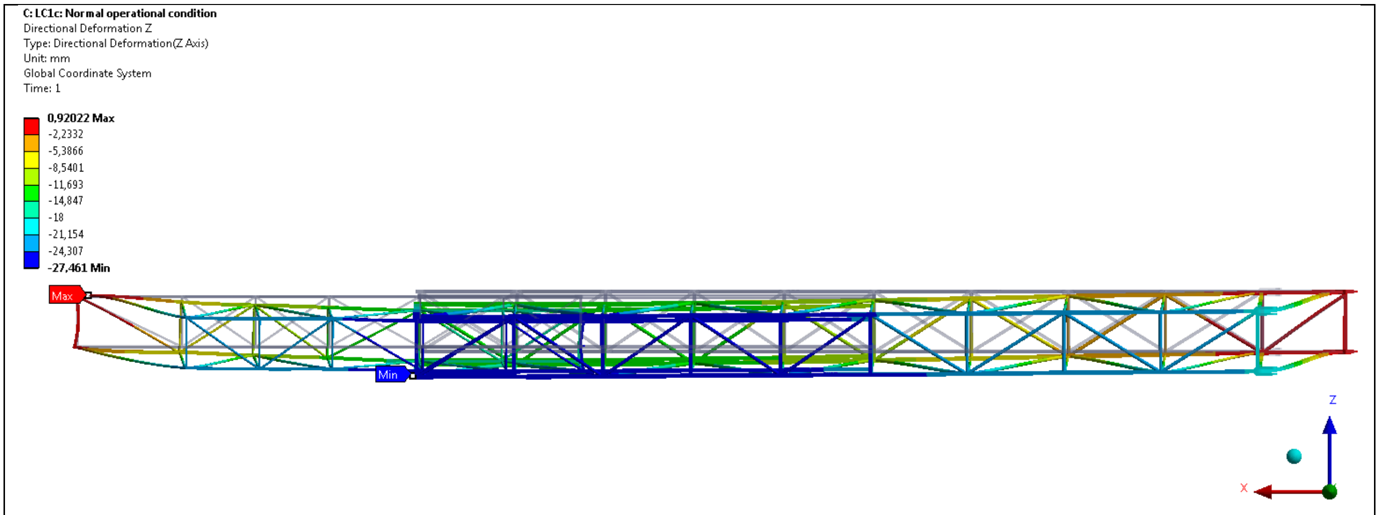


Figure 167: Directional deformation, Z-direction, scale = 24

The total deformation of the gangway structure is equal to -34,44 mm in the vertical direction. The total combined deformation is equal to 43,07 mm. These values are well within the stated limit and therefore considered acceptable.

Reaction force check

A reaction force check is performed in order to check the finite element model. The reaction forces from the finite element analysis are validated with hand calculated reaction forces. This is to make sure all applied forces and constraints are right. An overview of the results can be found in table 71: reaction force check below.

Table 89: Reaction force check

Reaction force check				
LC 1b: Normal working condition				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	12,759	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	-1420	-840	-1540
Geometry	[N]	-18118	-135840	-19649
Live load	[N]	0	-21600	0
Wind load	[N]	0	0	-15692,00
Additional mass	[N]	0	0	0
Summation	[N]	**-18118**	**-157440**	**-35341**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	127350	64096	8392,9
Force - hinge 2	[N]	-112990	12698	16404
Force - cable connection	[N]	79850	0	0
		-76224	79671	10403
Force - Landing point	[N]			
Summation	[N]	17986	156465	35199,9
Difference	[N]	-132	-975	-141
Deviation	[%]	**-0,7**	**-0,6**	**-0,4**

From the results we can conclude that there is a relative small deviation in the reaction forces. These discrepancies are due to numerical errors. These differences are small and are within the acceptable limits.

Conclusion

The FEA analysis demonstrates that the gangway structure in operational condition, incl. two times the live load has a general stress lower than the allowable 237 MPa and is considered acceptable. A few areas of stresses are higher than allowable, there are reviewed and considered acceptable.

The structure has a maximum resulting deformation of 43 mm. Note that this deformation is with the maximum operational load of 21600 N acting in the middle of the gangway. A deformation criteria is adopted from the DNV-guidelines and therefore this deformation is acceptable.

16.4 Gangway current design – Deployment / retrieval condition

In this case the gangway is in uplift condition. No live loads are acting on the gangway structure. The principle loads on the gangway consists of the self-weight and additional weight acting on the gangway structure. The wind load and the horizontal and vertical loads due to the operational motions must be included. The centrifugal force based on the maximum angular velocity and the radius of the considered mass should be included in this situation.

Mesh

The mesh is identical to the stated mesh in paragraph 16.3

Connections and boundary conditions

The connections are identical to the stated connections in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c.

The structure is constrained by means of remote displacement supports at the hinges and at the cylinder brackets. For the hinges, every DOF except the axial rotation is constrained. For the cylinders every DOF is constrained except for the rotation around the cylinder hinge and the rotation around the cylinder rod. The hinge points of the hydraulic actuators are located at a vertical displacement of 3000 mm above the lower hinge points of the gangway. In the emergency disconnection case the tip of the gangway is unsupported. At the hinge point for connecting the cable for the linear motion, a remote displacement is added which constrains the translation in the X-direction. The sliding pads consists of no separation contact, Frictionless sliding may occur and these regions.

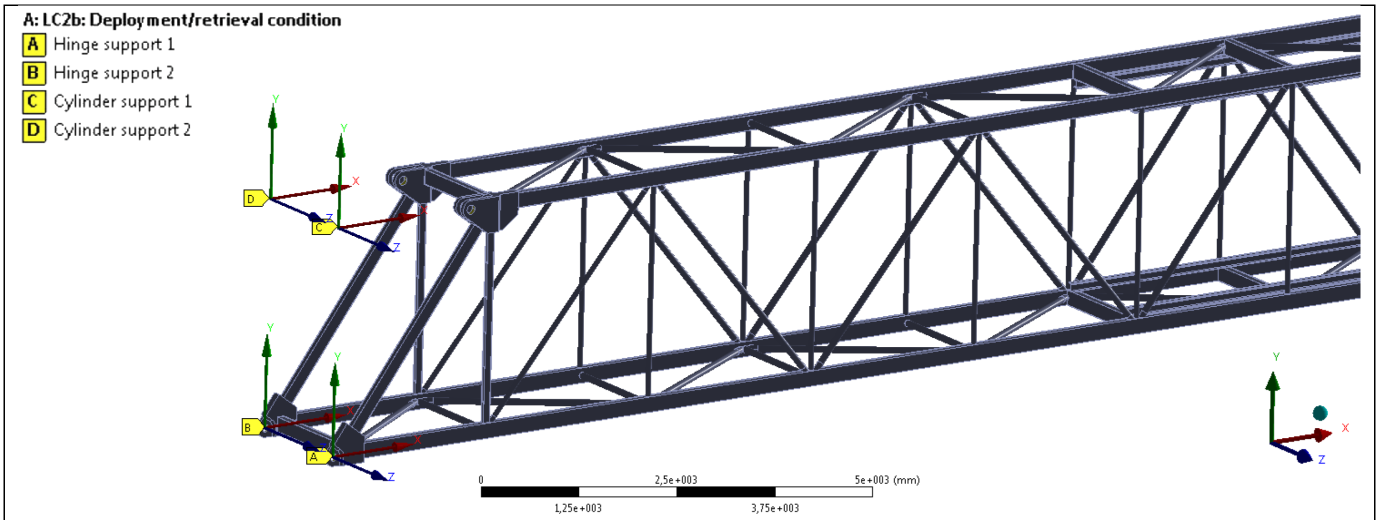


Figure 168: The remote displacements at the origin of the gangway

The DOF of the remote displacements are given in Table 95.Table 84

Table 90: Boundary conditions LC2b

DOF	Hinges	Cable connection	Landing mechanism
X Component	0 mm	0 mm	0 mm
Y Component	0 mm	Free	0 mm
Z Component	0 mm	Free	0 mm
Rotation X	0°	Free	0°
Rotation Y	0°	Free	Free
Rotation Z	Free	Free	Free

Accelerations
A standard Earth gravity of 9806.6 mm/s2 is applied to the geometry. In the deployment retrieval condition, dynamic factors are applied. These dynamic factors are specified by the DNV [25] and given in Table 83. Additionally, the accelerations are given in Table 84.

Point masses
The applied point masses are identical to the point masses as in normal operational situation.

The live load
No live load is applied in this condition

The wind load
The applied wind load is in the same way as in section 0 but now for the deployment/retrieval wind speed. This results in a larger wind pressure which is applied to the same lateral surface area. The emergency wind pressure is equal to 794 Pa which results in a wind load of 50856 N. This load is applied to the main beams in the structure. This surface area is shown in Figure 157.

Centrifugal force
The radial/centrifugal force is determined in appendix: A-5. Design calculations gangway according to the DNV. The force is applied as a remote force on the tip of the gangway structure.

Stress plots
The stress plots for the emergency disconnection case are reported in this section. The stresses over 150 and 237 MPa (Acceptance criteria) are plotted. Also the overall von-mises stresses are plotted.

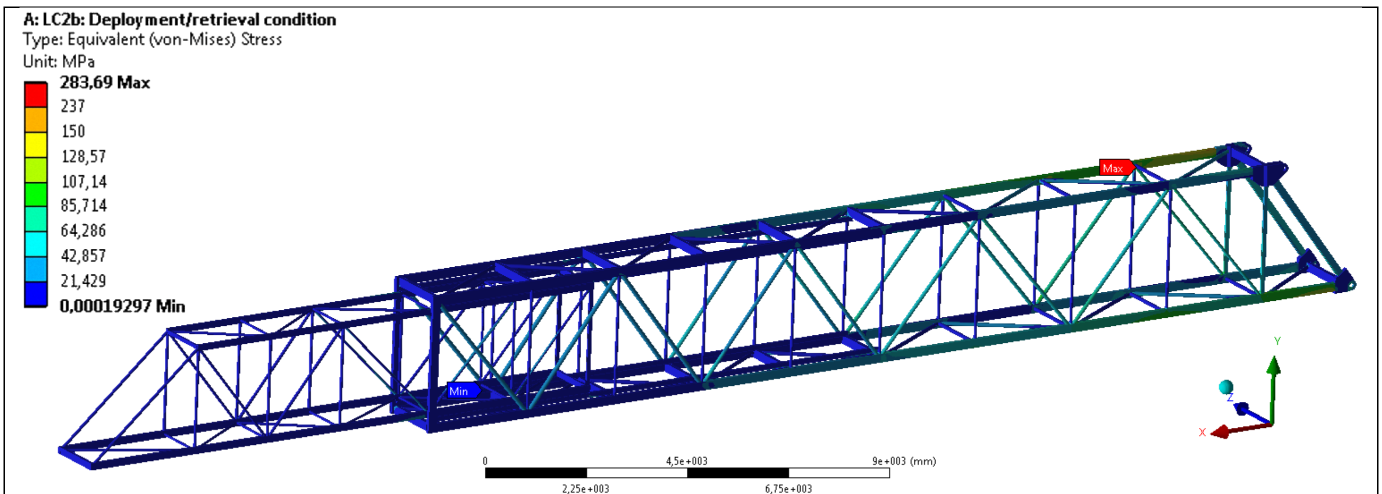


Figure 169: Overall Von Mises stresses

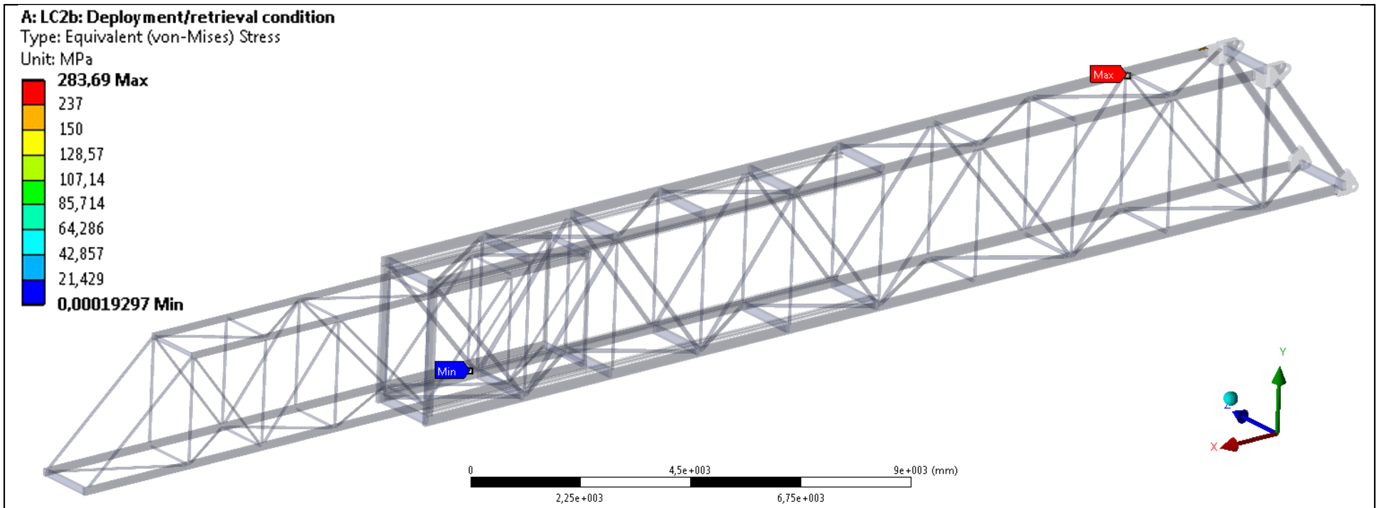


Figure 170: Stresses over 150 MPa

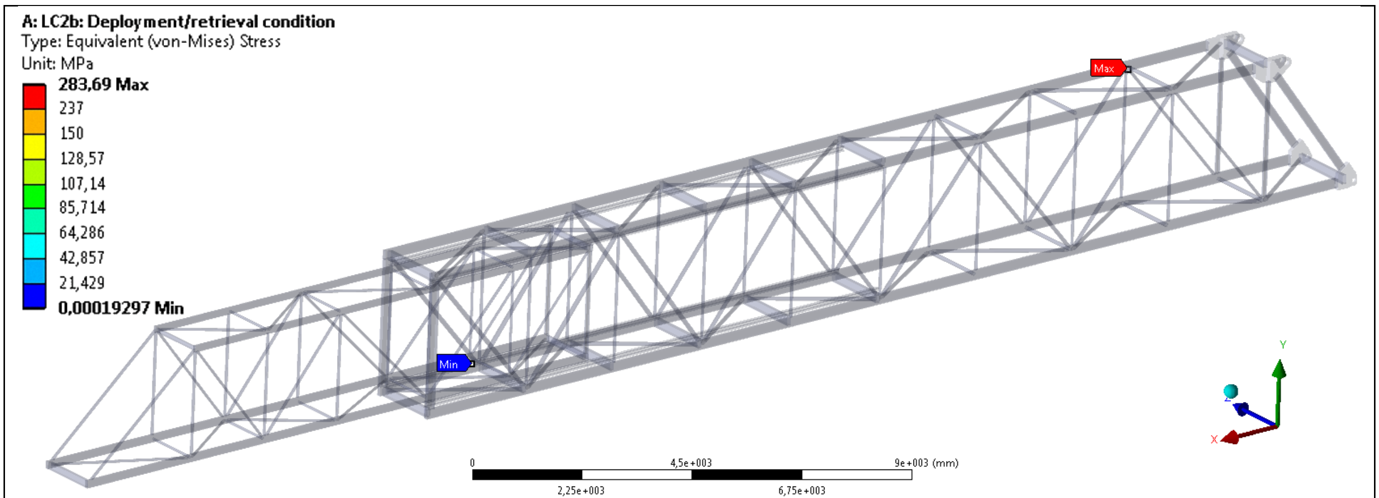


Figure 171: Stresses over 237 MPa

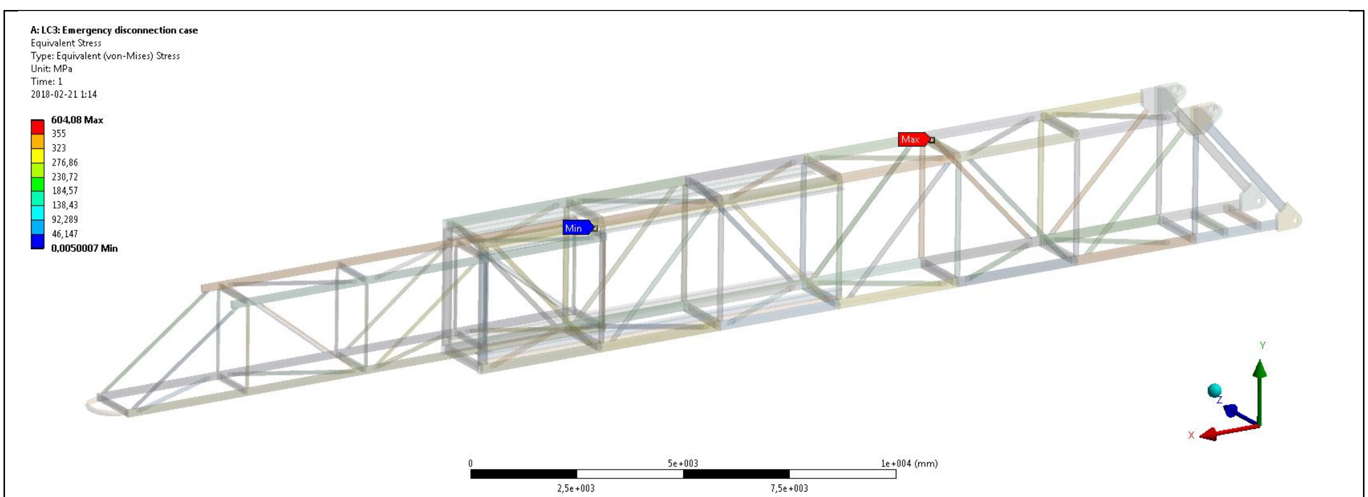


Figure 172: Stresses over 323 MPa

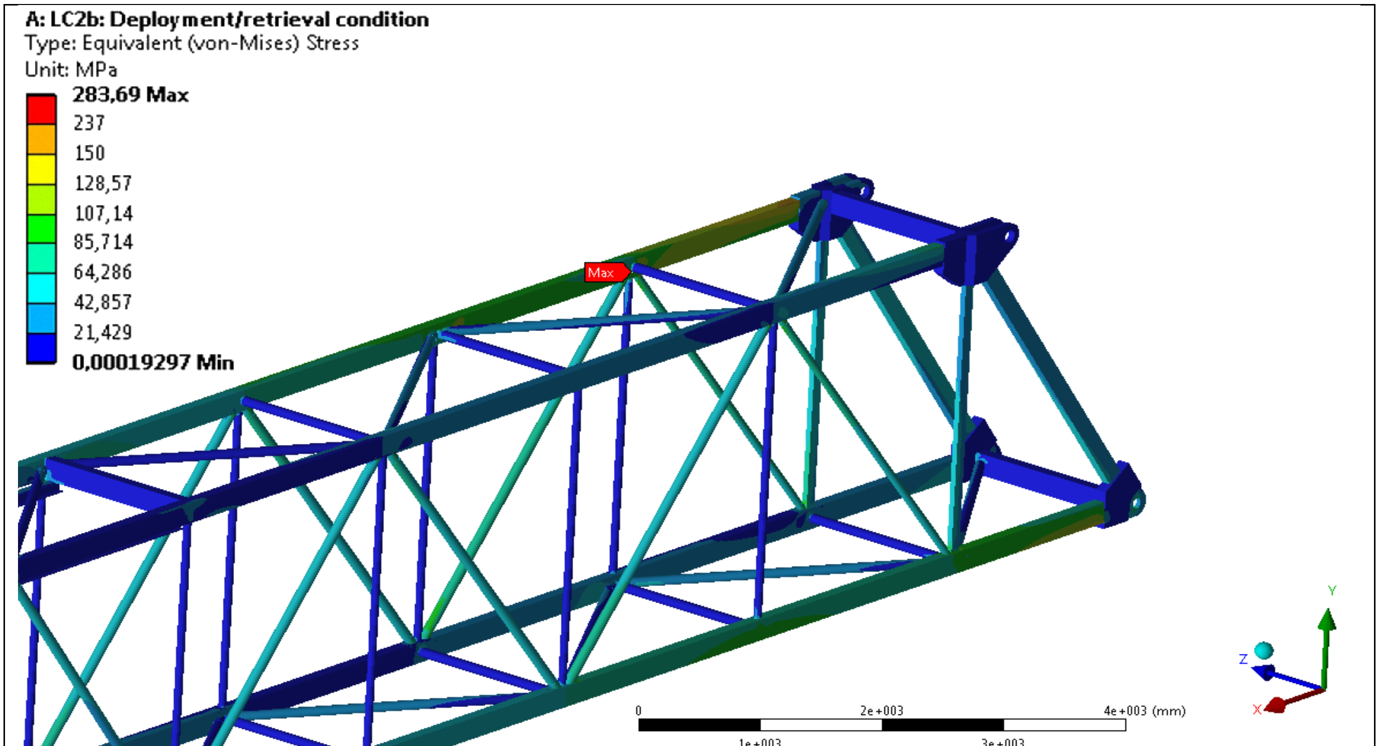


Figure 173: Detail view of the high stress locations

The general stress level is lower than 355 MPa / 1.5 = 237 MPa and is therefore considered acceptable.

The plots with detail views of the peak stress areas show the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and only single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria. Further investigation showed that these peak stresses are related to bad shaped mesh elements. Therefore these peak stresses can be neglected.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines [25]. For the emergency disconnection load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{100}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33 meter. This means that the maximum deflection limit of the tip is 330 mm.

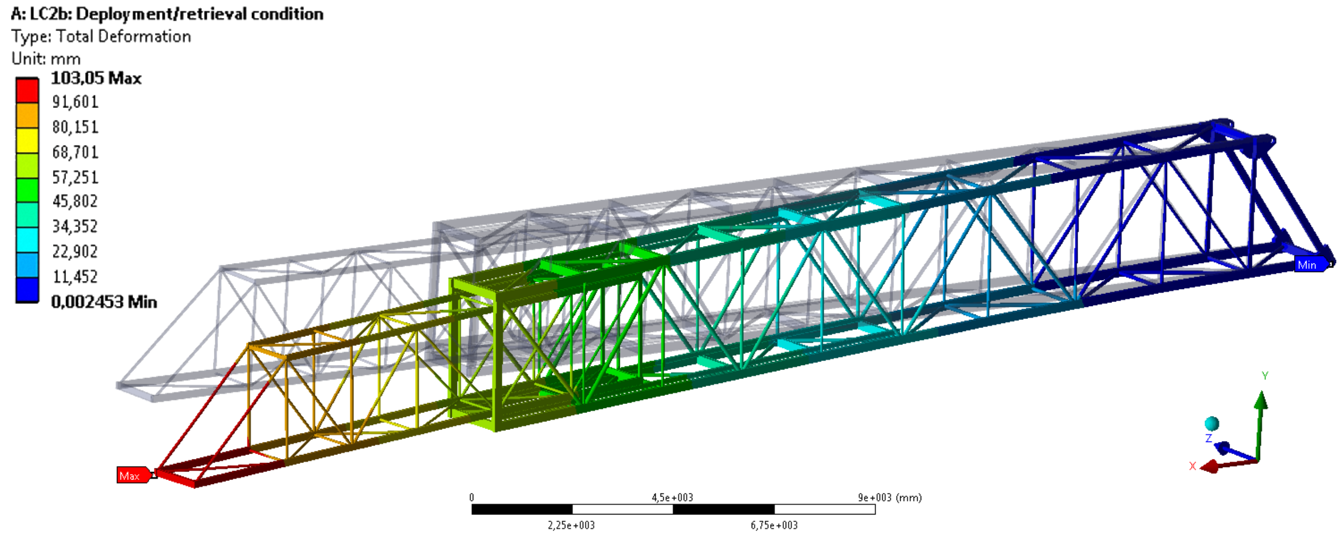


Figure 174: Total deformation, scale = 21

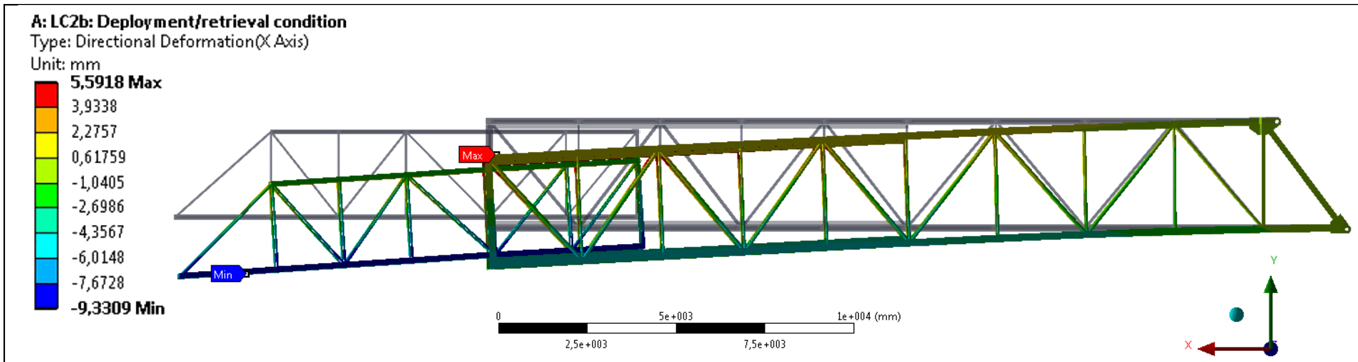


Figure 175: Directional deformation, X-direction, scale = 21

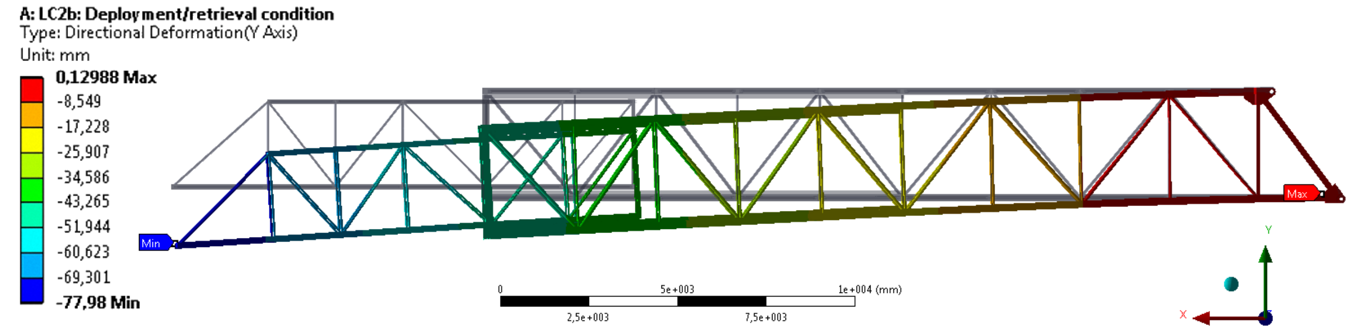


Figure 176: Directional deformation, Y-direction, scale = 21

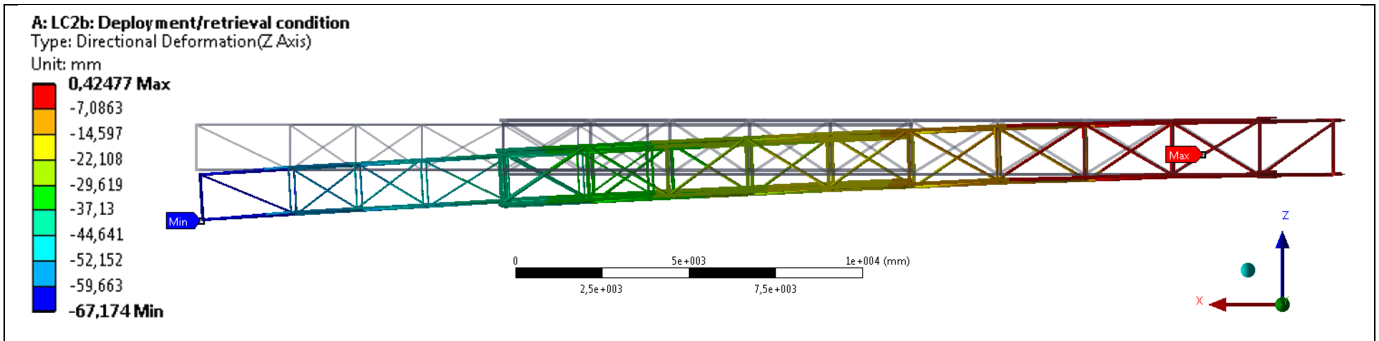


Figure 177: Directional deformation, Z-direction, scale = 21

The total deformation of the gangway structure is equal to 77,98 mm in the vertical direction. The total combined deformation is equal to 103,5 mm. These values are well within the stated limit and therefore considered acceptable.

Reaction force check
A reaction force check is performed in order to check the FEA. The results of this check can be found in Table 96.

Table 91: Reaction force check

Reaction force check				
LC2b: Deployment/retrieval condition				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	12,759	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	-1420	-840	-1540
Geometry	[N]	-18118	-135840	-19649
Tip load	[N]	1300	0	0
Wind load	[N]	0	0	-15692
Additional mass	[N]	0	0	0
Summation	[N]	**-16818**	**-135840**	**-35341**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	494200	68440	5863,3
Force - hinge 2	[N]	192620	73011	10682
Force - cylinder 1	[N]	-499140	-3282	14569
Force - cylinder 2	[N]	-173320	-3303,8	4086,4
Force - cable connection	[N]	2322,5	0	0
Summation	[N]	16682,5	134865,2	35200,7
Difference	[N]	-135	-975	-140
Deviation	[%]	**-0,8**	**-0,7**	**-0,4**

There is a small deviation in all directions. This discrepancy is due to small numerical errors related to the mesh, but these results are within acceptable limits.

Conclusion
The FEA analysis demonstrates that the gangway structure in emergency disconnection case has a general stress lower than the allowable 323 MPa and is considered acceptable.

The structure has a maximum resulting deformation of 103,05 mm. This is mainly the deformation of 77,98 mm in the y-direction. A deformation and stress criteria is applied according to the DNV guidelines.

16.5 Gangway current design – Emergency disconnection condition

In the emergency disconnection case, the gangway is in uplift/cantilever position. The length of the loaded gangway is the maximum operational length without the safety length. The principle loads are applied including a live load on the gangway tip. The principal loads consists of the loads from the self-weight of the gangway structure and the loads due to the live loads. The live load shall be applied at the tip of the gangway. The inertia forces shall be taken into account by multiplying the self-weight of the gangway with the sum of the dynamic factor DFz and the maximum vertical operational acceleration. The loads due to the motion of the vessel shall be included. The live load acting on the gangway tip shall be at least 350 Kg.

Mesh

The mesh is identical to the stated mesh in paragraph 16.3

Connections and boundary conditions

The connections and boundary conditions are identical to the connections and boundary conditions stated in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c.

Accelerations

The accelerations are identical to the stated accelerations in paragraph 16.3

Point masses

The applied point masses are identical to the point masses as in normal operational situation.

The live load

The live load is applied as a remote force. This load is applied at the tip of the gangway structure as can be seen in Figure 198. The tip load is equal to the live load without the use of a safety factor.

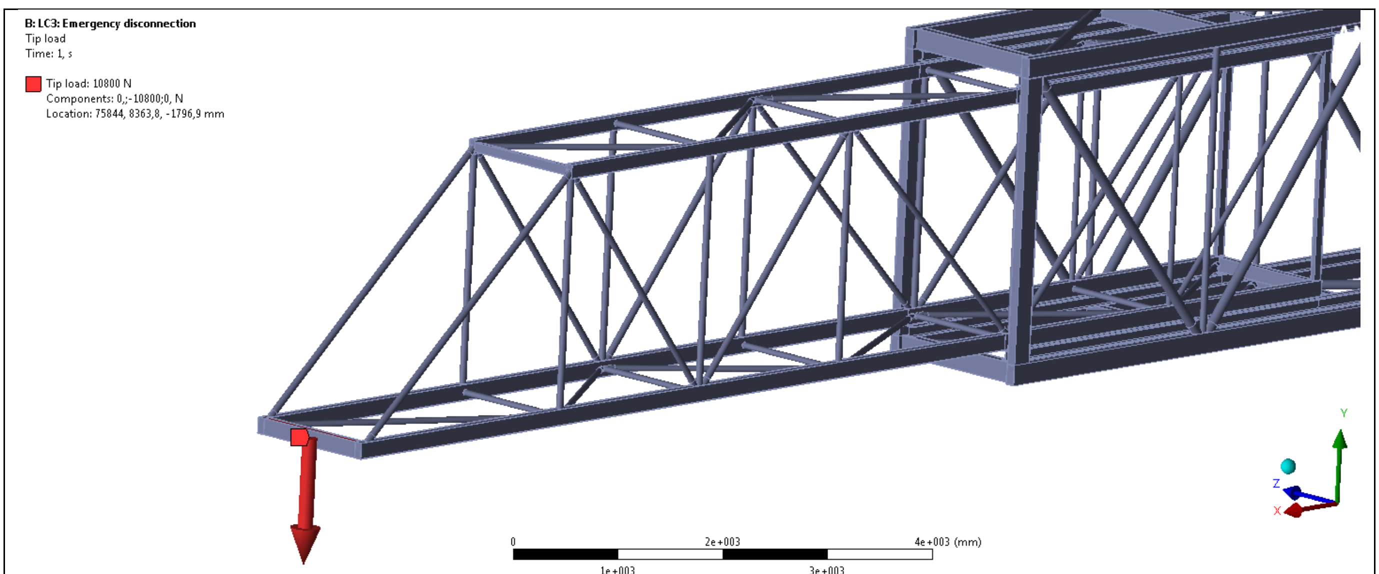


Figure 178: The applied live load

The wind load

The applied wind load is in the same way as in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c, but now for the deployment/retrieval wind speed. This results in a larger wind pressure which is applied to the same lateral surface area. The emergency wind pressure is equal to 794 Pa which results in a wind load of 50856 N. This load is applied to the main beams in the structure as can be seen in Figure 157.

Stress plots

The stress plots for the emergency disconnection case are reported in this section. The stresses over 150 and 323 MPa (Acceptance criteria) are plotted. Also the overall von-mises stresses are plotted.

Figure 179: Overall Von Mises stresses

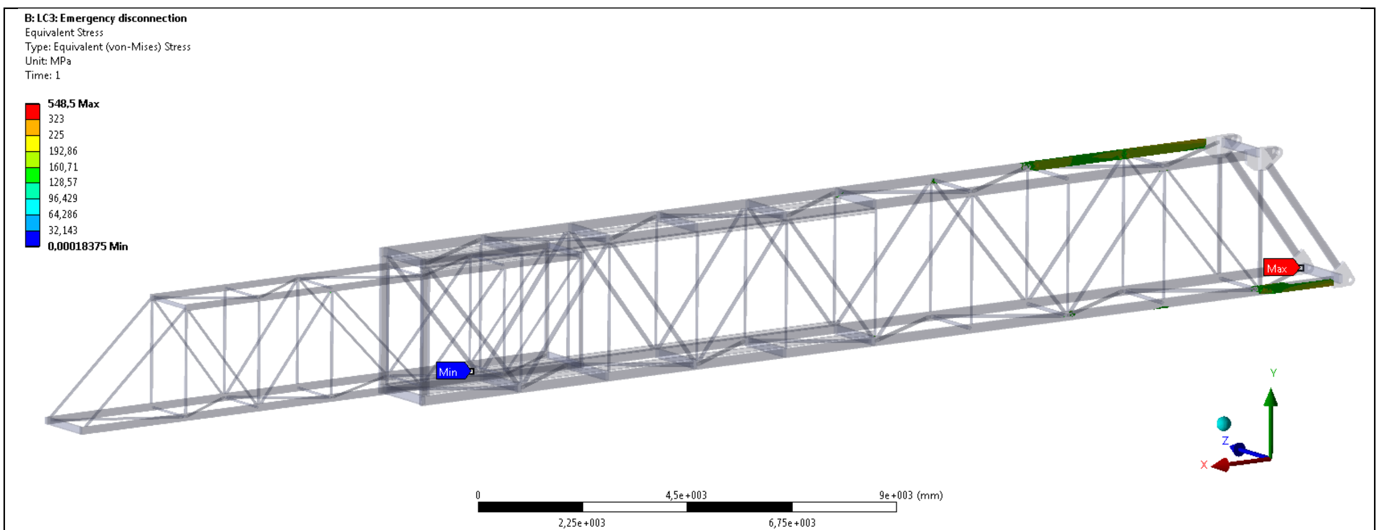


Figure 180: Stresses over 150 MPa

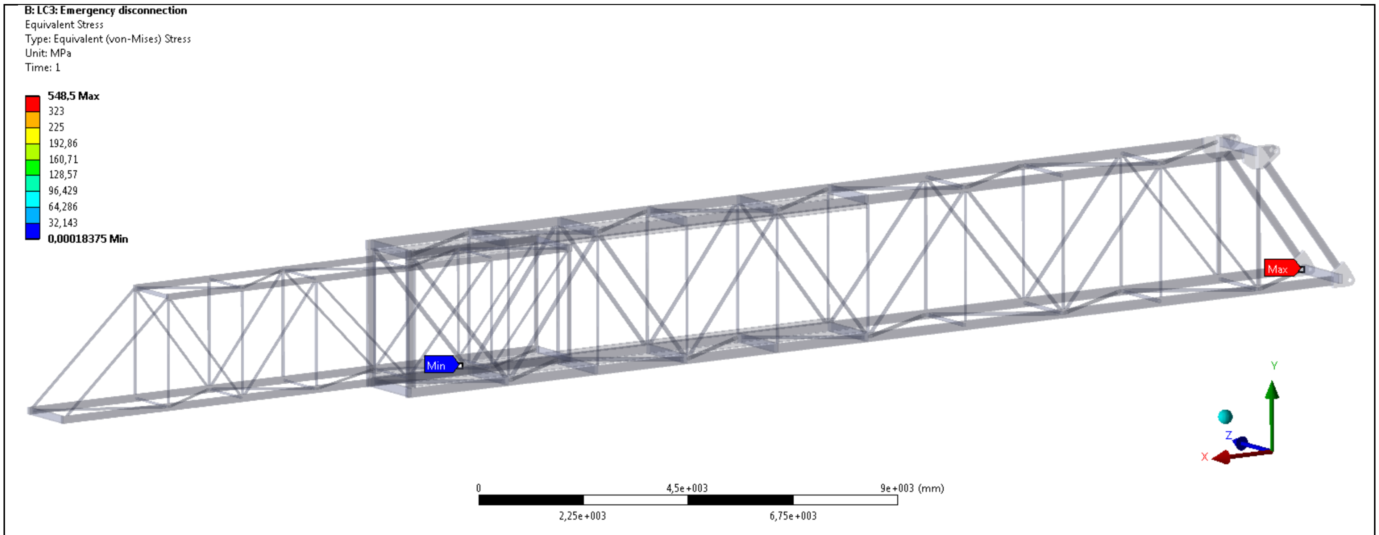


Figure 181: Stresses over 323 MPa

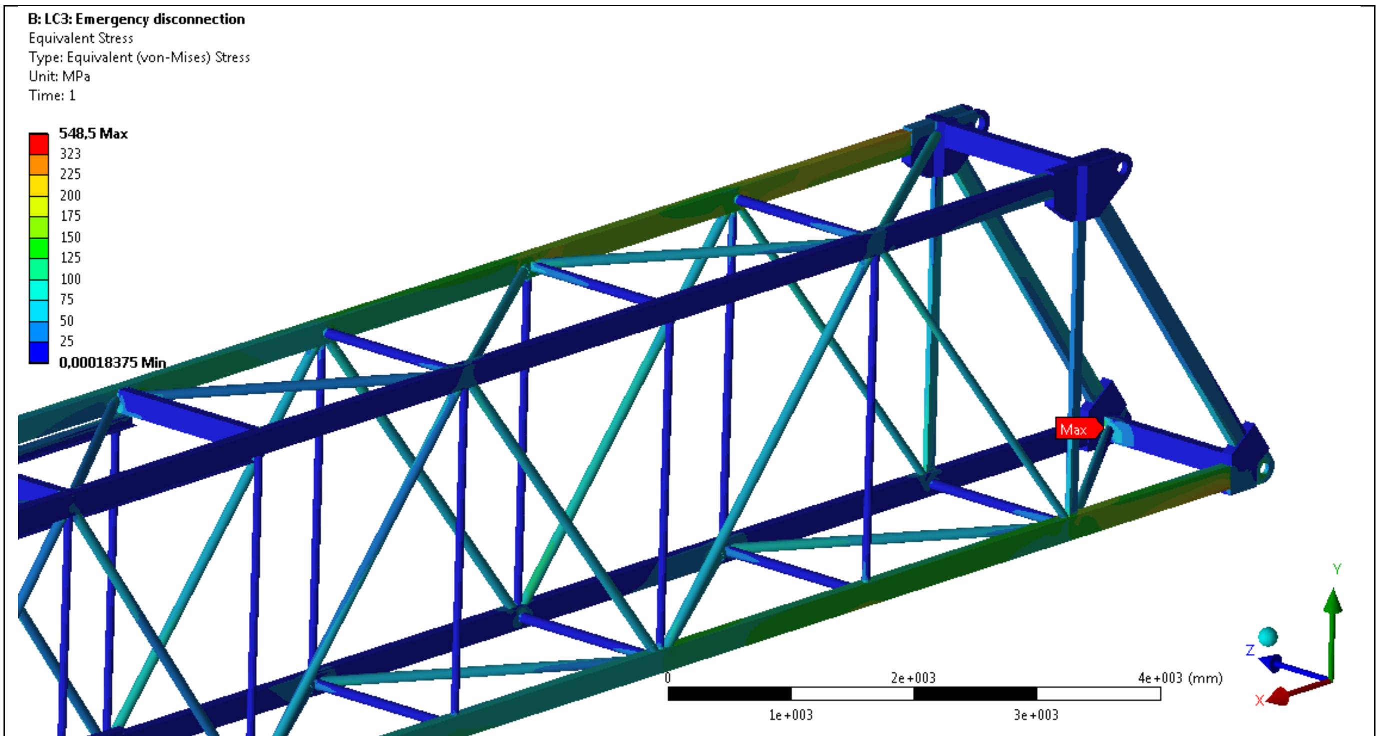


Figure 182: Detail view of the high stress locations

The general stress level is lower than 355 MPa / 1.1 = 323 MPa and is therefore considered acceptable.

The plots with detail views of the peak stress areas show the areas where the stresses exceed the allowable stress level. However, these areas are at sharp edges or sharp corner transitions. The areas are very small and only single element of size. These stresses are considered singularities and are acceptable. The overall stresses are below the stress limit and therefore the structure satisfies the stress criteria. Further investigation showed that these peak stresses are related to bad shaped mesh elements. Therefore these peak stresses can be neglected.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines. For the emergency disconnection load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{100}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33000 mm. This means that the maximum allowable deflection limit is equal to 330 mm.

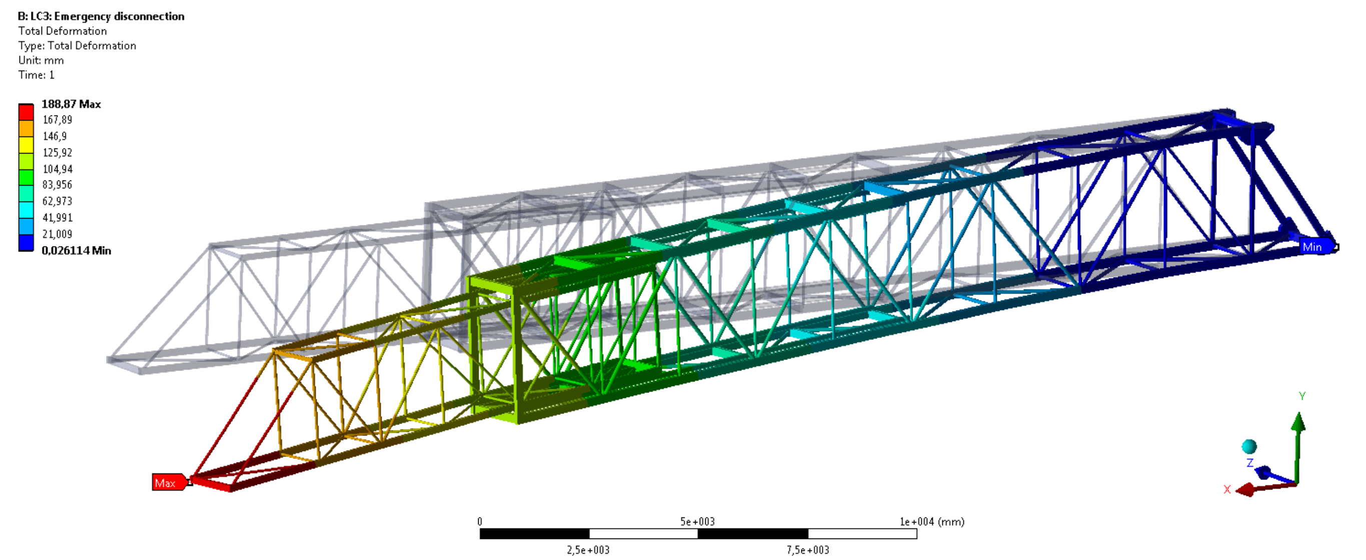


Figure 183: Total deformation, scale = 21

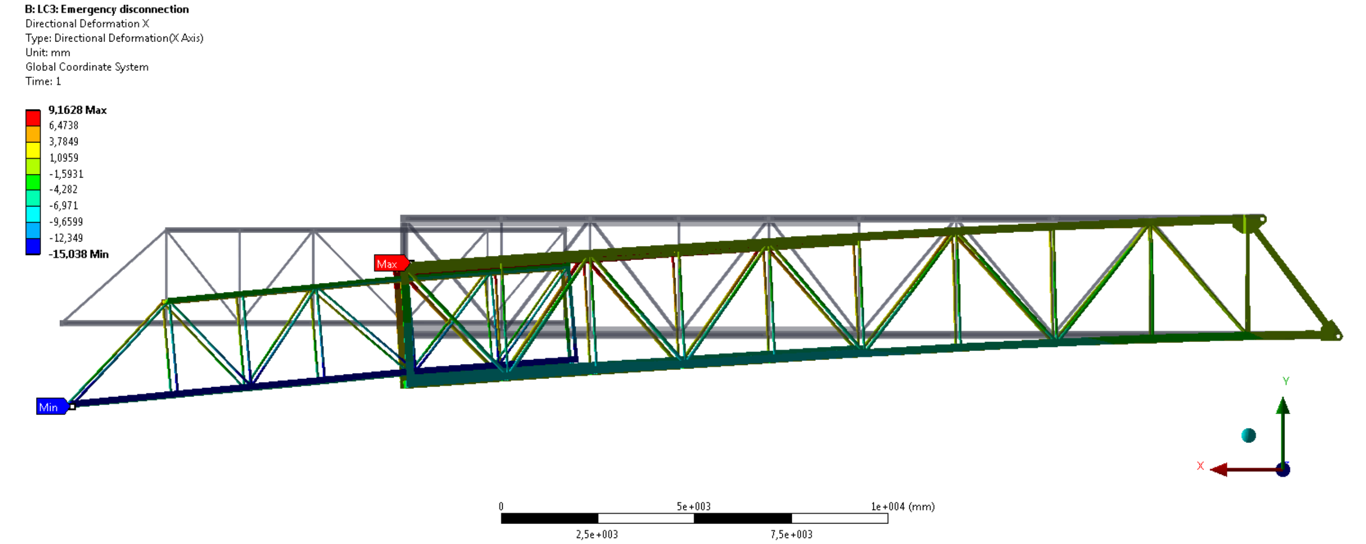


Figure 184: Directional deformation, X-direction, scale = 21

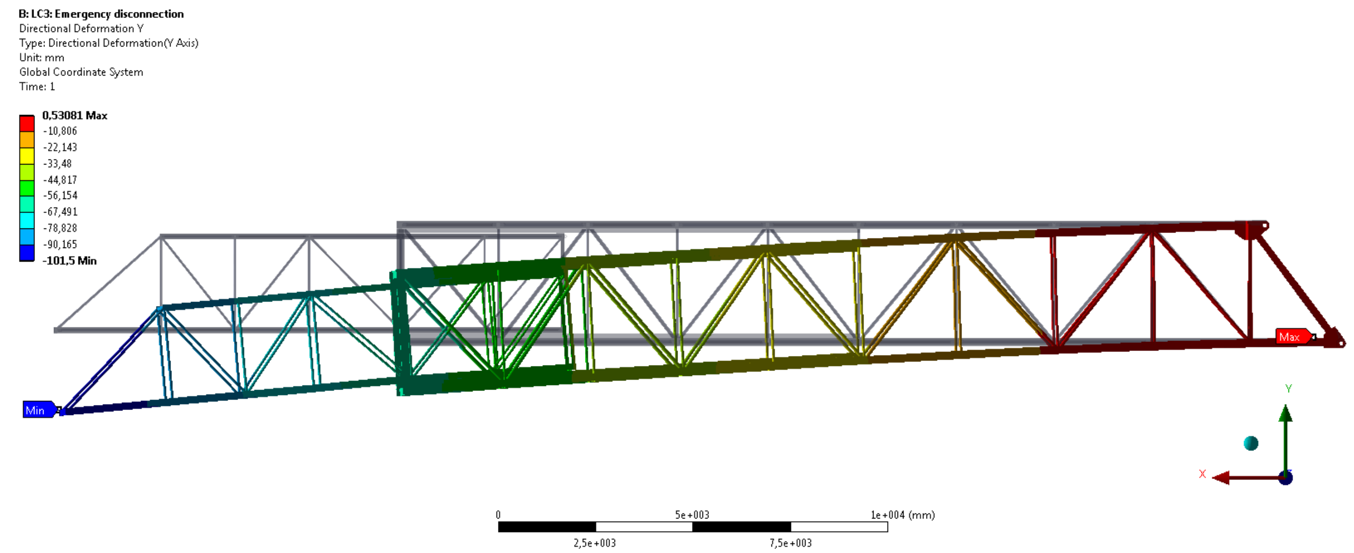


Figure 185: Directional deformation, Y-direction, scale = 21

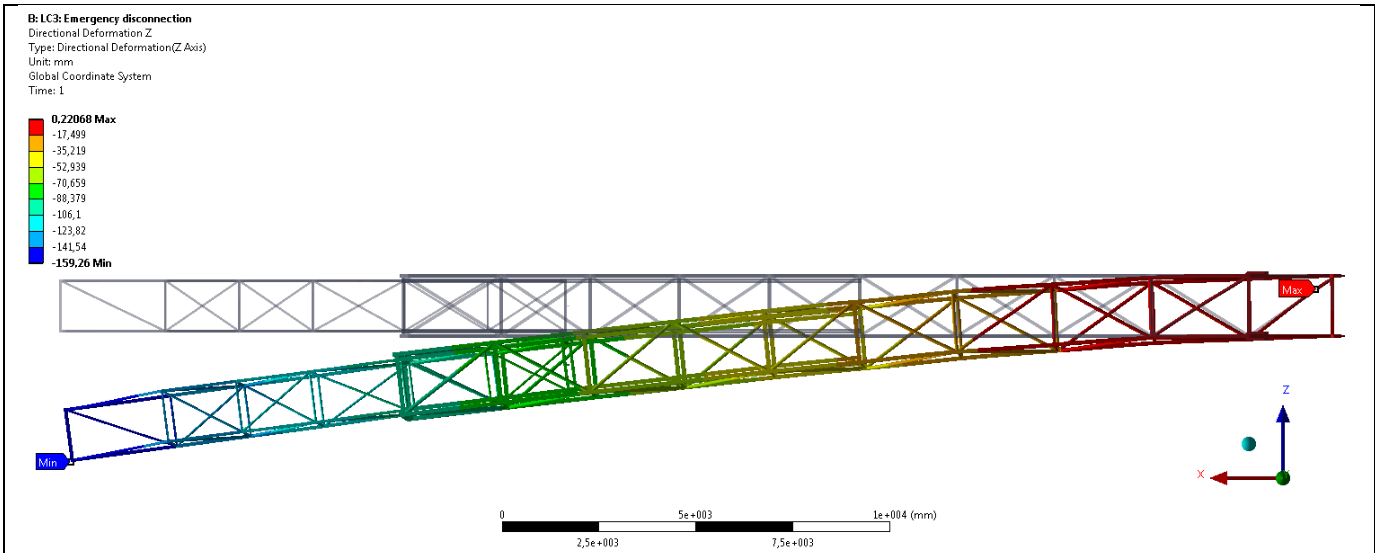


Figure 186: Directional deformation, Z-direction, scale = 21

The total deformation of the gangway structure is equal to 101,5 mm in the vertical direction. The total combined deformation is equal to 188,87 mm. These values are well within the stated limit and therefore considered acceptable.

Reaction force check
A reaction force check is performed in order to check the FEA. The results of this check can be found in Table 96.

Table 92: Reaction force check

Reaction force check				
LC3: Emergency disconnection				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	12,759	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	-2520	-840	-2640
Geometry	[N]	-32153	-135840	-33684
Tip load	[N]	0	-10800	0
Wind load	[N]	0	0	-50856
Additional mass	[N]	0	0	0
Summation	[N]	**-32153**	**-146640**	**-84540**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	771330	72138	13218
Force - hinge 2	[N]	40845	81271	27008
Force - cylinder 1	[N]	-776420	-3764,8	33221
Force - cylinder 2	[N]	-10268	-3979,9	10851
Force - cable connection	[N]	6429,1	0	0
Summation	[N]	31916,1	145664,3	84298
Difference	[N]	-237	-976	-242
Deviation	[%]	**-0,7**	**-0,7**	**-0,3**

There is a small deviation in all directions. This discrepancy is due to small numerical errors related to the mesh, but these results are within acceptable limits.

Conclusion
The FEA analysis demonstrates that the gangway structure in emergency disconnection case has a general stress lower than the allowable 323 MPa and is considered acceptable.

The structure has a maximum resulting deformation of 159,26 mm. This is mainly the deformation of 101,5 mm in the y-direction. A deformation and stress criteria is applied according to the DNV guidelines.

16.5 Gangway current design – Load test

This test case will be executed before a gangway is put into service. In this case the gangway will be extended to its maximum length and is supported in the vertical direction at both ends. A test load which is equal to 1.25 times the live load will be applied at the middle of the gangway. The acceptance criteria in this load case is the maximum deflection of the gangway structure.

Mesh

The mesh is identical to the stated mesh in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c.

Connections and boundary conditions

The connections and boundary conditions are identical to the connections and boundary conditions stated in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c.

Accelerations

The accelerations are identical to the stated accelerations in paragraph 16.3 Gangway optimized design – Normal working conditions LC 1c.

Point masses

The applied point masses are identical to the point masses as in normal operational situation.

The live load

No live load is applied in the load case

Test load

This test load is equal to 1.25 times the live load and it will be applied in the middle of the gangway. The live load is equal to 10800 N. Therefore a test load of 1.25 x 10800 = 13500 N is applied in the middle of the gangway structure. This is shown in Figure 187.

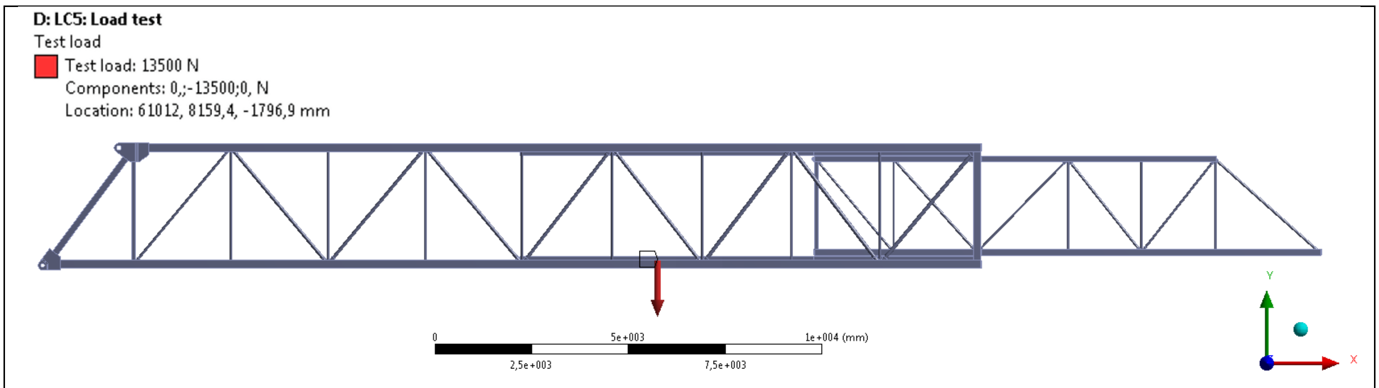


Figure 187: The applied test load

Stress plots

For the test load only the deflections of the gangway are considered as acceptance criteria. Therefore the stress is not reported in this chapter.

Deformation plots

The maximum allowable deformation of the gangway is given by the DNV-guidelines. For the emergency disconnection load case the maximum allowable deflection of the gangway structure is equal to:

- $\delta_{allow} = \frac{L}{200}$

In which δ_{allow} is the maximum allowed deflection of the structure. L is the maximum extracted length of the gangway. The maximum extracted gangway length is 33000 mm. This means that the maximum allowable deflection limit is equal to 165 mm.

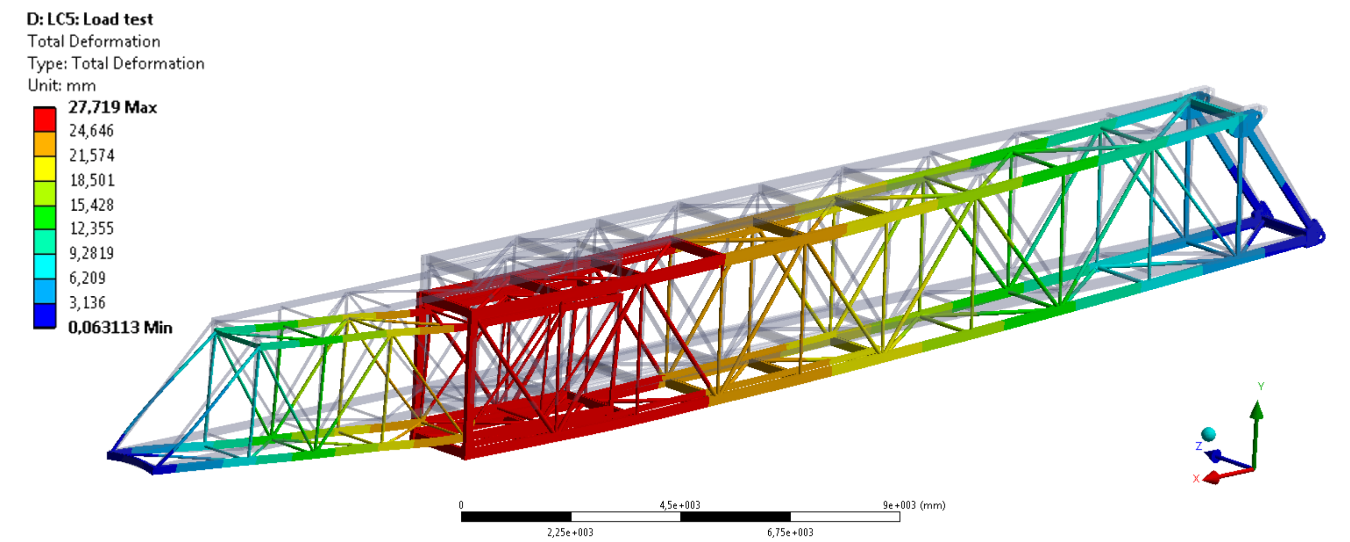


Figure 188: Total deformation, scale = 30

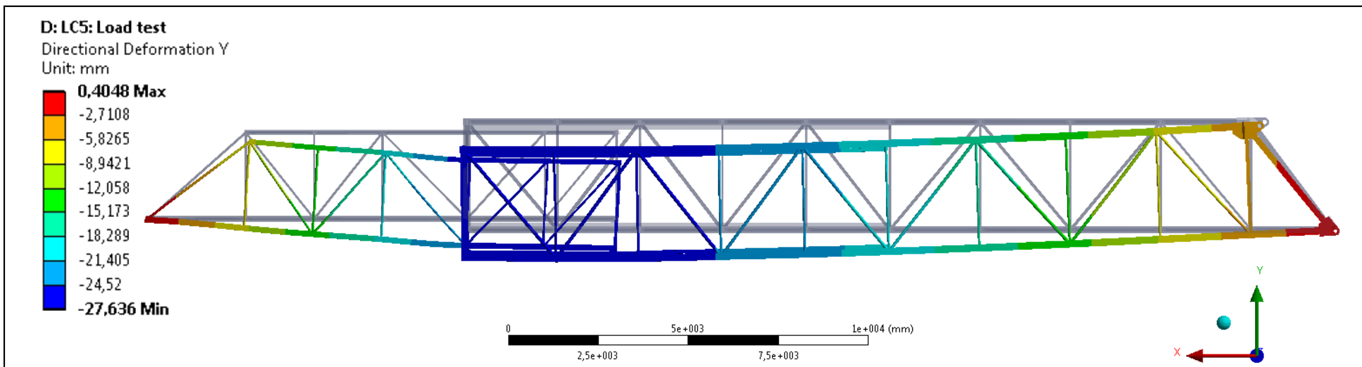


Figure 189: Directional deformation, Y-direction, scale = 30

The total deformation of the gangway structure is equal to 27,72 mm in the vertical direction. The total combined deformation is equal to 27,64 mm. These values are well within the stated limit and therefore considered acceptable

Reaction force check

A reaction force check is performed in order to check the FEA. The results of this check can be found in Table 96Table 98.

Table 93: Reaction force check

Reaction force check				
LC5: Load test				
Input	**Dimension**	**X dir.**	**Y dir.**	**Z dir.**
FEA Model mass	[t]	0	12,759	0
Earth gravity	[mm/s2]	0	-9806,6	0
Acceleration	[mm/s2]	0	0	0
Geometry	[N]	0	-125122	0
Test load	[N]	0	-13500	0
Wind load	[N]	0	0	0
Additional mass	[N]	0	0	0
Summation	[N]	**0**	**-138622**	**0**
FEA output		**X dir.**	**Y dir.**	**Z dir.**
Force - hinge 1	[N]	1871,9	33986	194,67
Force - hinge 2	[N]	-1874,6	33547	-110,31
Force - cable connecti	[N]	68607	0	0
Force - Landing point	[N]	-68605	70191	-84,361
Summation	[N]	-0,7	137724	-0,001
Difference	[N]	-1	-898	0
Deviation	[%]	-	**-0,6**	-

There is a small difference in the X-direction and a deviation of 0.6% in the Y-direction. This is a very small value and therefore the result is within the acceptable limits.

Conclusion

The FEA analysis demonstrates that the structure in the test load case and is considered acceptable.

The structure has a maximum resulting deformation of 27,72 mm. This is lower than the deflection limit which is stated by the DNV and therefore this deflection is considered acceptable.

A-17. Topology optimization: Fixed Part

In this section the boundary conditions are defined which are acting on the design domain. According to Table 7, three load cases are incorporated in the optimization process:

-LC1c: Normal operational condition.
-LC2b: Deployment/retrieval condition.
-LC3: Emergency disconnection.

Load collectors are defined which contains the support conditions or forces per load case. A combination of a load collector which contains supports conditions and a load collector which contains forces defines a load case.

The design space is the volume in which material may be placed and attached. It defines the geometric restrictions for the optimizer. The design domain for the fixed part is visualized in Figure 190. The rectangular tube has inner dimensions of 2.3 by 1.4 meter and has a total length of 24 meter. The outer dimensions of the tube are equal to 2.74 by 1.72 meter. This rectangular tube is supported by four roller bearings in each load case. These supporting wheels have a contact area of 0.1 by 0.2 meter. The grey transparent part is the design domain and the blue parts belongs to the non-design domain. At the origin of the gangway, hinge point are created for the connection of the hinges and the hydraulic actuators.

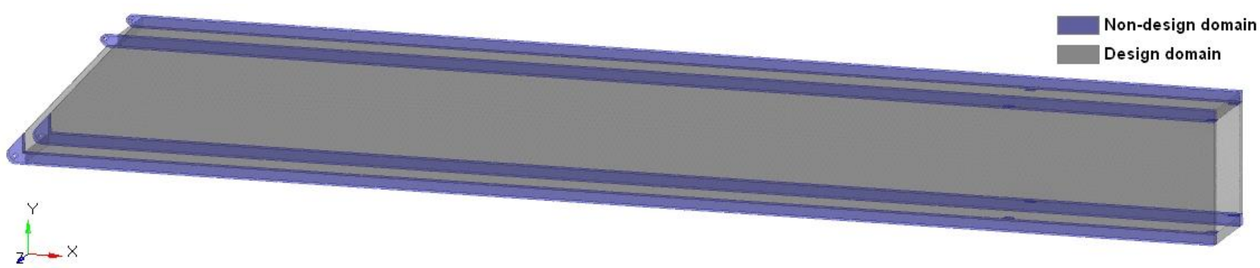


Figure 190: Design domain fixed part

17.1. Point masses

During the optimization process it is chosen to set the density of the design material equal to zero. The masses of the gangway structure are incorporated in the design by means of point masses. These point masses are equally distributed over the length of the design domain and they represent the mass of the current design and the additional masses of the structure.

The mass of fixed part of the gangway is equal to 11000 kg and the additional masses are estimated at 2355 kg. The gangway mass and the additional masses are added together and divided over the nodes of the two main beams located at the bottom of the gangway. These are denoted by the white nodes in Figure 34. These point masses are applied for all load cases.

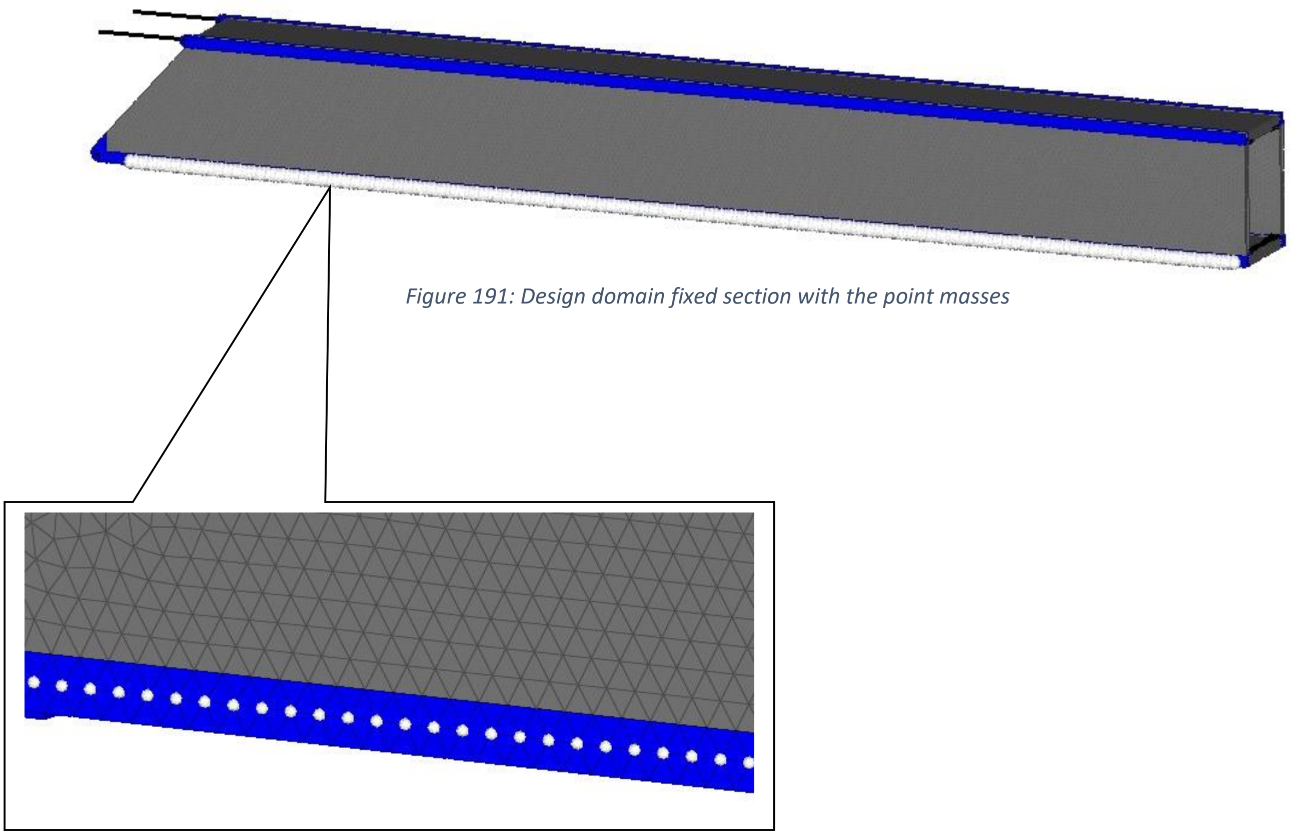


Figure 191: Design domain fixed section with the point masses

17.2 LC1c: Normal operational condition

First the *operational situation* is considered. In this situation the gangway is supported on both sides and the structure is able to transfer people and cargo from and to the offshore structure. In this condition, the structure is supported by the following displacement constraints: At the hinge point for connecting the cable for the telescopic motion, a displacement constraint is added which constrains the translation in the X-direction. This constraint is applied in the middle at the end of the fixed section at location F as can be seen in Figure 192Figure 36. This constraint is acting on 5 nodes. The gangway structure has a hinged connection at the origin of the gangway. RB2 elements are used to constrain all the nodes located at the surface area of the hole. These nodes are connected by spider nodes to the centre of the hole. The centre of the hole is constrained in all directions and only a rotation around the Z-axis is allowed. These constraints are active at location G and H as can be seen in Figure 192.

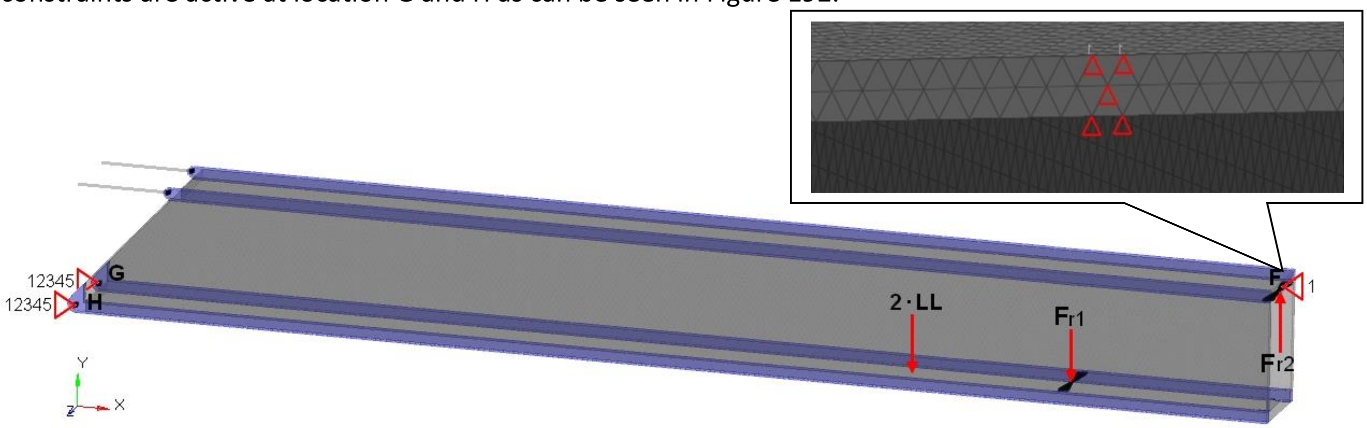


Figure 192: Boundary conditions for LC1c

The forces are applied at the active locations of the roller bearings. RB3 elements are used to distribute the loads over the support areas. The supporting areas are selected and connected by using spider nodes to one common node. The support reaction force is applied to this node which is then distributed over all the nodes which are connected to this node. The live load is acting in exactly the middle of the total length of the gangway and is applied to one node. A safety factor of 2 is applied according to the DNV. All the applied loads are shown in Table 94. Notice here that the reaction forces are a little bit lower due to the fact that the new mass of the telescopic section is included.

Table 94: Applied loads for LC1c

L.C.	*Name*	*Magnitude [N]*
LC1c	LL	10800
	F_{r1}	161082
	F_{r2}	220360

An overview of the boundary conditions for the operational conditions are given in Table 95.

Table 95: Boundary conditions for LC1c

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC1c	Nodes	F	0	Free	Free	Free	Free	Free
	Nodes	G,H	Free	0	0	0	Free	0

A standard earth gravity is applied to the geometry. Additionally, the operational accelerations are included in the model. These accelerations are shown in Table 17.

17.3 LC2b: Deployment/retrieval condition.

In *deployment/retrieval condition* the gangway is in uplift condition. No live loads are acting on the gangway structure. The principle loads on the gangway consists of the self-weight and the additional weights which are acting on the gangway structure. The horizontal and vertical loads due to the operational motions must be included. The centrifugal force based on the maximum angular velocity and the radius of the considered mass should be included in this situation. The centrifugal force is visualized in Figure 193 and is acting at the location of the cable connection. The displacement constraint at the hinge point is defined in exactly the same way as in in the normal operational condition. In this condition, the gangway structure is supported by two hydraulic cylinders which are keeping the gangway in cantilevered condition. In order to incorporate the behaviour of these hydraulic actuators, RBE2 elements are created. These RBE2 elements create a connection between two nodes. All the nodes located on the surface area of the holes, are connected to a single node which is located in the middle of the hinge point. This point is only allowed to move in the Y-X plane and rotate around the Z-axis. This node is connected to a node which is located exactly above the hinge point at the bottom. The result of this approach is that the design domain is only allowed to rotate at the lower hinge points and at the cylinder hinge points. The resulting force in the cylinder can only work in the longitudinal direction of the cylinder.

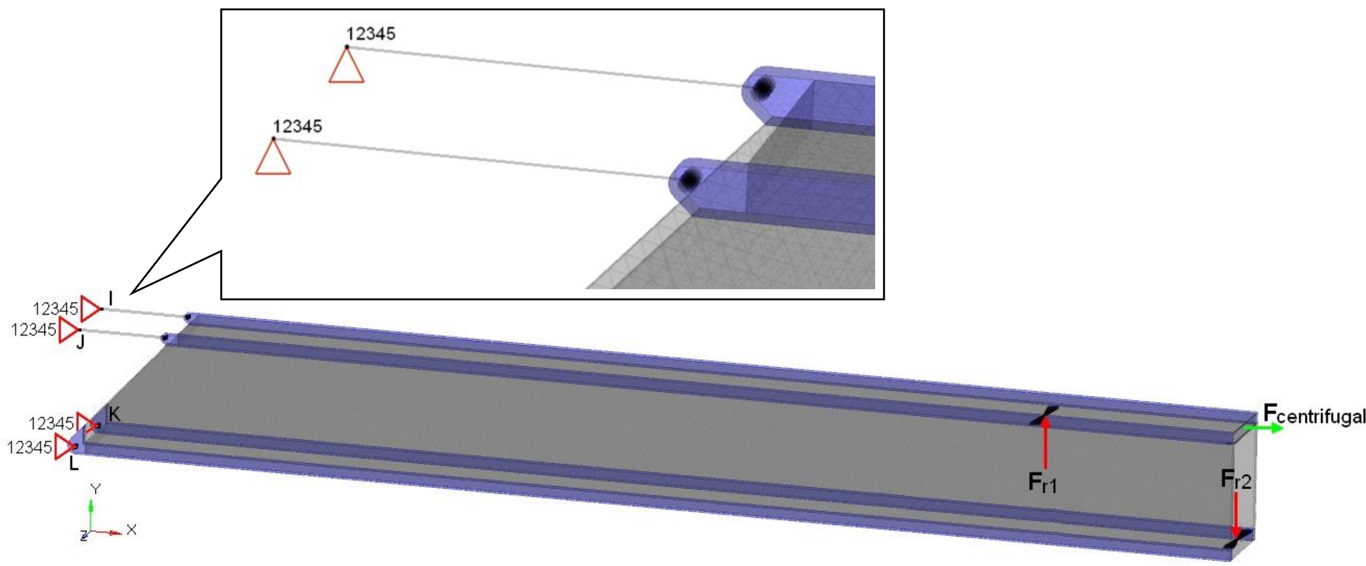


Figure 193:Boundary conditions for LC2b

An overview of the boundary conditions for the operational conditions are given in Table 96.

Table 96: Boundary conditions for LC2b

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC2b	Nodes	I,J,K,L	0	0	0	0	0	Free

A standard earth gravity is applied in this load case. Additionally, dynamic factors are applied. These dynamic are specified by the DNV and incorporated into the gravity acceleration. This results in a gravity acceleration of 10791 mm/s^2. Operational accelerations are not included in this load case.

17.4 LC3: Emergency disconnection condition.

In the *emergency disconnection* case, the gangway is in uplift position. Therefore the tip of the gangway is not supported by the landing tool. The live load is applied at the tip of the telescopic section and has a magnitude of 10800 N. The gangway is supported by means of two hydraulic cylinders and two hinges which are all located at the origin of the gangway structure. The reaction forces which originates from the telescopic part are applied to the locations of the active roller bearings. These reaction forces are denoted by F_{R1} and F_{R2}. The boundary conditions for LC3 are shown in Figure 194.

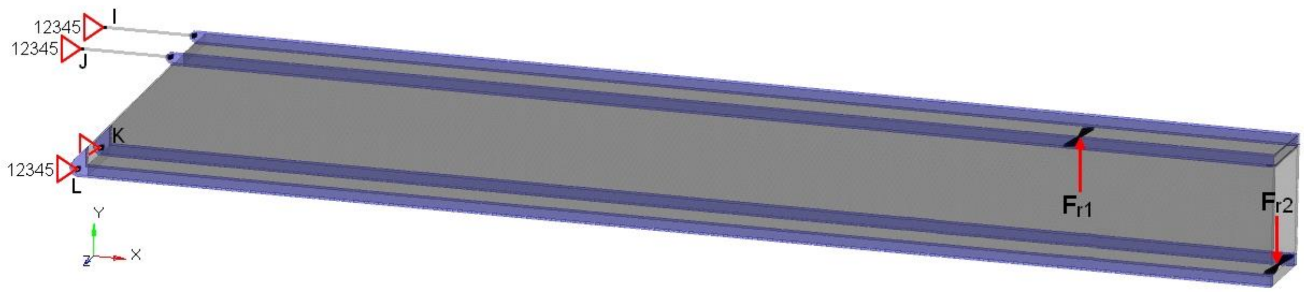


Figure 194: Boundary conditions LC3

Table 97: Boundary conditions for LC3

L.C.	*Location*		*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC3	Nodes	I,J,K,I	0	0	0	0	0	Free

A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the emergency disconnection case, dynamic factors are applied. These dynamic factors are specified by the DNV and are given in Table 27. The accelerations are defined as a factor of the gravity acceleration. In this situation orthogonal components are defined which are a factor of the vertical gravity acceleration. These dynamic factors combined with the vessel accelerations results in the acceleration combinations stated in Table 20.

Table 98: Acceleration combinations LC3

Acceleration combinations [mm/s²]				
	Additional load cases			
Load case	**LC3a**	**LC3b**	**LC3c**	**LC3d**
Gravity acceleration [mm/s²]	12450			
Longitudinal	-0.2024	0.2024	0.2024	-0.2024
Transversal	0.0675	-0.0675	0.0675	-0.0675
Vertical	-1	-1	-1	-1

A-18. Size optimization: Fixed part

This appendix is concerned with all the post-processing steps which are performed in order to obtain a feasible design. The same procedure is performed for the telescopic section. The approach, set-up and description of all the steps are given. The results from the size optimization are presented in section 8.4 Size optimization results.

The gangway structure is subjected to several load cases according to Table 7. The following load cases are incorporated in the sizing optimization process:

-LC1C: *Normal operational conditions.*
-LC2B: *Deployment / retrieval load case.*
-LC3: *Emergency disconnection.*

According to Table 6, five load cases needs to be analysed. It is chosen to only use three load cases for the optimization process. The other load cases will be analysed during the FEA of the detailed design. A description of the loads and support conditions per load case is given in the following sections.

The first step in the design realisation, is to convert the topology result to a line model. The line model will represent the orientation and location of the 1D beam elements. The result of the topology optimization is imported into Hypermesh by using OSSmooth. The model can be imported by selecting the model and the corresponding result file. Temporary nodes are defined on the intersection point of the beams. The nodal coordinates are measured manually and created on the intersection points of the beam elements in the centre of the beam. These temporary nodes are interconnected by linear lines, which represent the orientation and direction of the beams. The beams have rigid connections at the nodes.

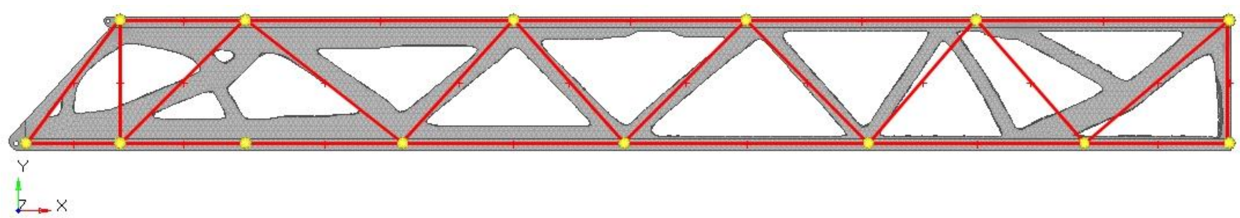


Figure 195: Nodal configuration of the topology results

The grey transparent shell corresponds with the topology optimization results. The yellow points are the temporary nodes which are located at the midpoint of the cross-section at the intersection points of the beams. The red lines define the beam axial centrelines. Different cross-sections are defined which can be related to the different line elements. A circular hollow section is chosen for the diagonal and vertical elements and for all the horizontal elements a rectangular tube section is chosen. The cross-sections are defined by using Hyperbeam, which is a build-in tool in Hyperworks to create cross-sections. The gangway structure will be made out of steel tubes and therefore a material collector is defined with the properties of S355 structural steel. The assumed primary structure material is S355 or similar, with the properties according to Table 22.

Now the gangway structure will be partitioned into different domains. Each beam elements and its symmetric adjacent element is assigned to a property. This allows an independent size optimization of all the different beams in the design domain. For each property, the corresponding cross-section and material is selected. These properties will be assigned to the mesh elements in a later stage. The properties are related to the model according to element numbering visualized in Figure 196.

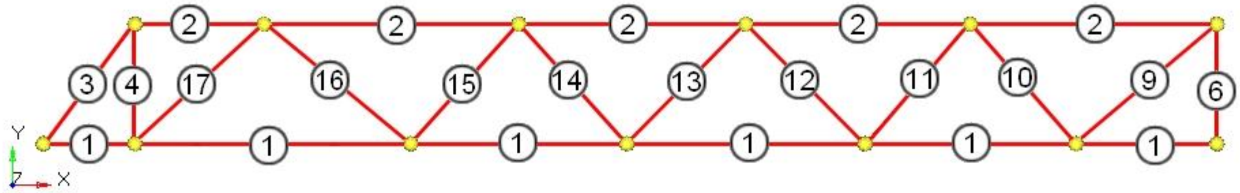


Figure 196: Property element relation

In Figure 197, the isometric view of the line model is given. All the lines which can be mirrored according to the Z-X Plane are assigned to the same property. This is done in order to obtain a complete symmetric structure around the Z-X plane.

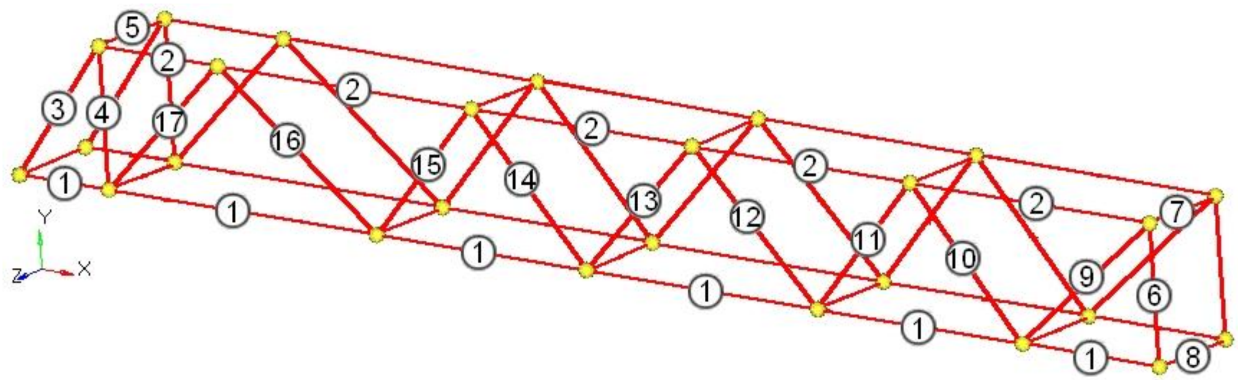


Figure 197: Property element relation isometric view

The element properties are given in Table 99, in which the materials and the beam sections are given.

Table 99: Beam element properties

Beam elements	**Material**	**Beam section**
1 - 8	S355 Steel	Hollow Rectangular
9 - 17	S355 Steel	Hollow circular

The model is meshed with 1D line elements with an element size of 4 mm. The total amount of elements is equal to 48101. The orientation of the elements needs to be specified and the different properties are assigned to the different elements. This means, that for each different beam element, a separate size optimization is performed and beam section can be assigned. It is chosen to assign the upper and lower longitudinal beams to one property in order to obtain an uniform size for these elements. This will simplify the manufacturing process.

18.1 Point masses

In addition to the self-mass of the gangway structure, the gangway boom will be loaded by additional components such as flooring, hand rails and utility lines. For the structural optimization of the beams it is important to incorporate these masses into the load collector. The calculation of these additional masses can be found in appendix: A-5. Design calculations gangway. The additional masses are split between the fixed and telescopic section according to the length ratio of the two parts. This results in a total additional mass of 3255 kg for the fixed part. These additional masses are represented by point masses. These point masses are distributed equally over the two lower longitudinal members. This can be seen in Figure 198. These point masses are included in the optimization for each load case.

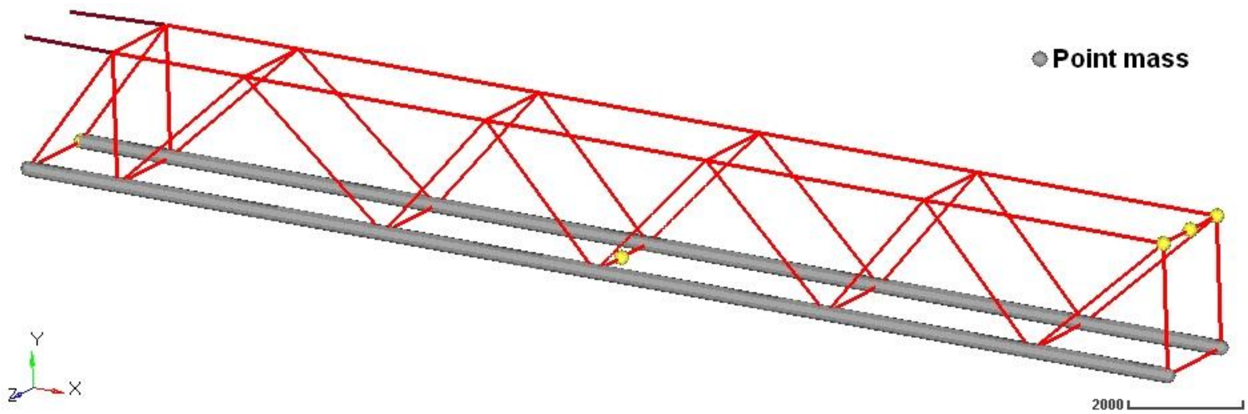


Figure 198: The applied point masses

18.2 LC1c: Normal operational condition

First the operational situation is considered. In this situation the gangway is in normal operational condition and therefore the gangway is supported on both sides. The hydraulic actuators are not activated and in free-flow mode. The structure is able to transfer people and cargo from and to the offshore structure. The structure is supported by the following displacement constraints: At the hinge point for connecting the cable for the linear motion, a displacement constraint is added which constrains the translation in the X-direction. This constraint is applied at node 256 as can be seen in Figure 199. The fixed section of the gangway is supported by two hinges located at the origin of the gangway. These hinges constrains the displacements in the X, Y and Z-direction and only allows a rotation around the Z-axis. The live load is acting in the middle of the total length of the gangway, which is the most severe condition. The reaction forces from the telescopic part are applied at the locations of the roller supports.

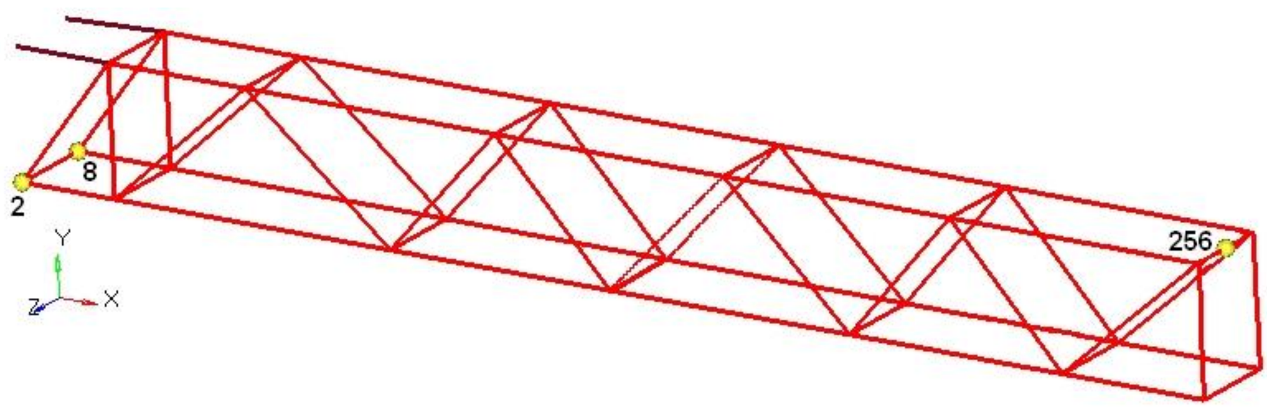


Figure 199: Constrained nodes for LC1c

An overview of the boundary conditions for the operational conditions is given in Table 100.

Table 100: Boundary conditions for $LC1_c$

L.C.	*Node*	*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
$LC1_c$	2	0	0	0	0°	0°	Free
	8	0	0	0	0°	0°	Free
	256	0	Free	Free	Free	Free	Free

In this next part, the applied forces and accelerations are defined. A standard earth gravity is applied to the geometry. Additionally, the operational accelerations are included in the model. These accelerations acts parallel or normal to the gangway deck. The downward (vertical) acceleration is considered normal to the deck. The longitudinal and transversal accelerations are considered parallel to the gangway deck and are perpendicular to each other. The vertical gravity acceleration is equal to 9810 mm/s^2. The accelerations for the different load cases are given in Table 25 which results in four additional load cases for the normal operational condition. The vertical gravity acceleration is included in these accelerations, assuming that this is always the most dominant direction due to the fact that these accelerations work in opposite direction and therefore the resultant force can be added together.

The reaction forces from the support conditions are applied as a point load on the members. All the applied loads acting on the structure are visualized in Figure 200. The live load is multiplied by a factor 2 according to the DNV. The reaction force loads are divided by a factor 2 due to the fact that the force is divided over two beams. Please notice that the new weight of the telescopic section is included in the calculation of the reaction forces.

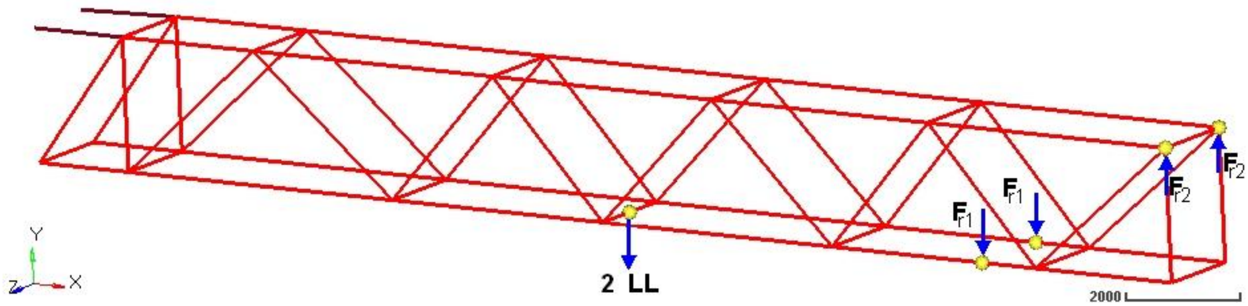


Figure 200: The loads acting on the gangway for LC1c

Table 101: Loads applied during LC1c

L.C.	*Name*	*Magnitude [N]*
$LC1_c$	LL	10800
	F_{r1}	80540.9
	F_{r2}	110179.25

The wind load is not applied in this load case. It is assumed that the wind load has a relative low impact on the structure because the wind pressure is acting on an open truss structure. The lateral surface area is low compared to the other dimensions of the structure. The wind load will be checked during the FEA of the detailed design.

During the normal operational conditions, the gangway is subjected to additional constraints. According to the DNV, a safety factor regarding the yield stress of 1.5 should be applied. The means that the member stresses may not exceed the limiting value of 237 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. In the normal operational condition, the deflection of the structure may not exceed a value of 120 mm. This constraint is applied to the nodes 52, 58, 62 and 68 may not exceed a vertical deflection of 120 mm. These nodes are visualized in Figure 201 and an overview is given in Table 102.

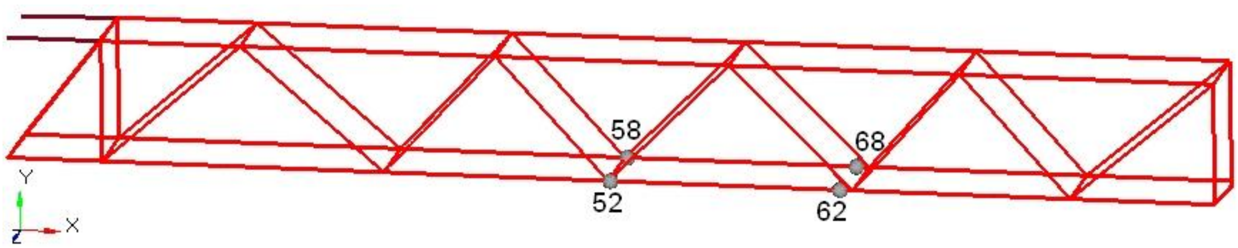


Figure 201: Deflection limits LC1c

The nodes 52 and 58 are the mid-nodes of the fixed section of the gangway. They are defined in exactly the middle of the fixed section. The nodes 62 and 68 are the mid nodes of the total gangway length. They are defined in the middle of the total gangway at its maximum length.

Table 102: Constraints applied during LC1c

L.C.	*Name*	*Applied to:*	*Limiting value*
$LC1_c$	Displacement	Nodes: 52, 58, 62, 68	120 mm
	Members stresses	All members	237 MPa

18.3 LC2b: Deployment / retrieval condition.

In this situation, the gangway is fully extracted and in uplift position. The tip of the gangway is not supported by the landing tool. Only the self-weight and the additional weights are acting on the structure and no live loads are considered. The loads due to the centrifugal forces are included in this situation.

In this situation the gangway is supported by the following boundary conditions: The fixed section of the gangway is supported by two hinges at the origin of the gangway. These hinges constrains the displacements in the X, Y and Z-direction and only allows a rotation around the Z-axis. In this condition, the gangway structure is supported by two hydraulic cylinders which are keeping the gangway in horizontal condition. In order to incorporate the behaviour of these hydraulic actuators, RBE2 elements are created. These RBE2 elements create a connection between two nodes. All the nodes located on the surface area of the holes, are connected to a single node which is located in the middle of the hinge point. This node is only allowed to move in the Y-X plane and to rotate around the Z-axis. This node is connected to a node which is located exactly above the hinge point at the bottom. These are the nodes J and I. The result of this approach is that the fixed section is only allowed to rotate at the lower hinge points and at the cylinder hinge points. The length between the connection will remain constant and it represents the cylinder length. The resulting force in the cylinder can only work in the longitudinal direction of the cylinder. This can be seen in Figure 202.

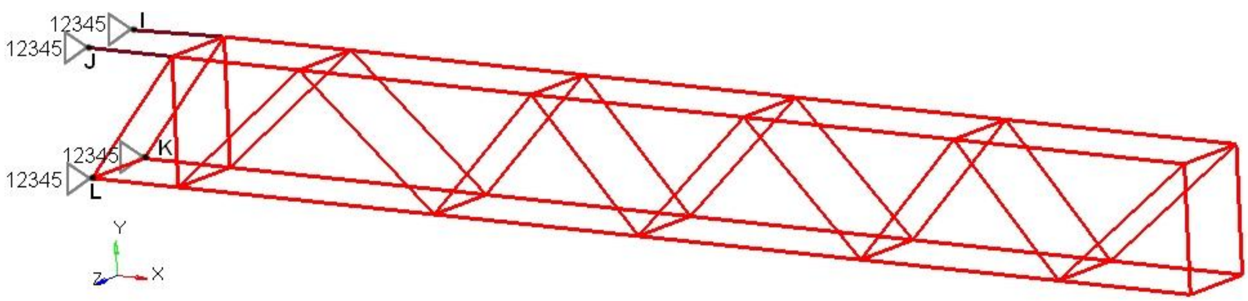


Figure 202: Boundary conditions for LC2$_b$

Table 103: Overview boundary conditions LC2$_b$

L.C.	*Nodes*	*X Component*	*Y Component*	*Z Component*	*Rotation X*	*Rotation Y*	*Rotation Z*
LC2$_b$	I, K, L, M	0	0	0	0°	0°	Free

A standard earth gravity is applied in this load case. No operational accelerations are included in this situation. In the deployment/retrieval condition, dynamic factors are applied. These dynamic factors are specified by the DNV and are given in Table 27. These dynamic factors can be multiplied with the gravity acceleration which results in a gravity acceleration of 10791 mm/s^2.

In appendix: A-5. Design calculations gangway, an approximation has been made for the centrifugal force. It is estimated that the centrifugal force is equal to 1300 N which is applied as a concentrated force located at the cable connection. This force is acting in the longitudinal direction as be seen in Figure 203. Also the self-weight and additional masses of the telescopic section are included in this load case. This means that the reaction forces from the roller bearings must be taken into account. In this situation the reaction forces from F_{r1} and F_{r2} are lower due the fact that the live load is not included in this load case.

Table 104: Magnitude reaction forces

L.C.	*Name*	*Magnitude* [N]
LC2b	F_{r1}	5734
	F_{r2}	21659
	$F_{centrifugal}$	1300

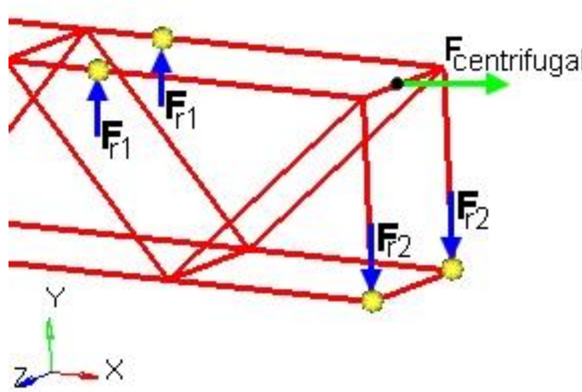


Figure 203: Centrifugal force at the cable connection

Also wind loads needs to be considered during the deployment and retrieval load case. This wind load is not included in the optimization process but will be applied and checked during the final FEA analysis.

During the deployment and retrieval condition, the gangway is subjected to following additional constraints. According to the DNV, the safety factor regarding the yield stress is equal to 1.5, which corresponds to acceptance criteria I. The means that the member stresses may not exceed the limiting value of 237 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. For the gangway in cantilevered condition, the nodes located at the tip of the gangway may not exceed a vertical deflection of 240 mm. The nodes: 280, 284 and 288 are shown in Figure 204.

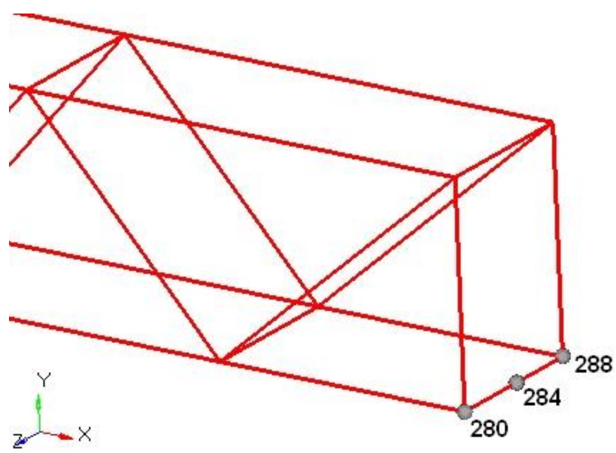


Figure 204: Deflection limit for LC2b and LC3

18.4 LC3: Emergency disconnection condition.

In the emergency disconnection case, the gangway is in uplift position. Therefore the tip of the gangway is not supported by the landing tool. A live load is located at the tip of the gangway structure. In this situation the gangway is supported by the boundary conditions which are identical to the stated boundary conditions in 18.3 LC2b: Deployment / retrieval condition. A standard earth gravity is applied in this load case. Additionally, the operational accelerations are included in the model. In the emergency disconnection case, dynamic factor are applied. These dynamic factors are specified by the DNV and are given in Table 27. These dynamic factors combined with the vessel accelerations results in the acceleration combinations in Table 30. During the emergency disconnection condition, the reaction forces from the roller bearings are applied. The location of these reaction forces can be seen in Figure 205.

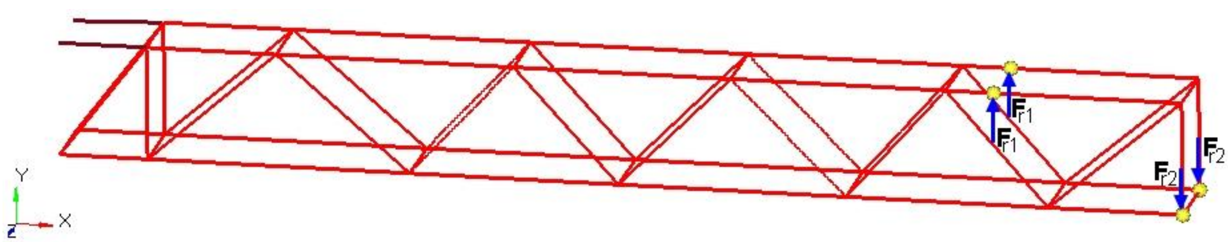


Figure 205: The applied loads for LC3

The reaction forces from the roller supports are applied as a point load on the members. The reaction force loads are divided by a factor 2 due to the fact that they are divided over two beams. Please notice that the new weight of the telescopic section is included in the calculation of the reaction forces. The magnitude of the reaction forces is given Table 105.

Table 105: Magnitude reaction forces

L.C.	*Name*	*Magnitude* [N]
LC3	F_{r1}	21978.13
	F_{r2}	37903.13

A retrieval wind load wind loads is acting on the structure during the emergency disconnection case. This wind load is not applied during the size optimization and it will be included in the FEA of the final design.

During the emergency disconnection condition, the gangway is subjected to the following additional constraints. According to the DNV, in this condition the design is subjected to acceptance criteria III, which corresponds to a safety factor of 1.1 regarding to the yield stress. The means that the member stresses may not exceed the limiting value of 323 N. Also deflection constraints are applied in this load case. These deflection constraints are calculated according to the length of the gangway structure. For the gangway in cantilevered condition, the node located at middle of the tip of the gangway may not exceed a vertical deflection of 240 mm. These nodes are shown in Figure 204.

In het emergency disconnection case, the gangway is in cantilevered condition and is slowly retracted while a live load is acting on the gangway tip. When the gangway is retracting, the support conditions located at the bottom of the gangway structure, are moving into the positive X-direction. Due to this linear motion, the horizontal distance between the support condition increases and the momentum which is generated by the live load is decreased. This is favourable but still it is important to check the situations between a full extended and a complete retracted gangway. Therefore it is chosen to add four additional load cases in which the support conditions are shifted to an intermediate and unfavourable condition. The locations for the support condition related to each load case is shown in Figure 62.

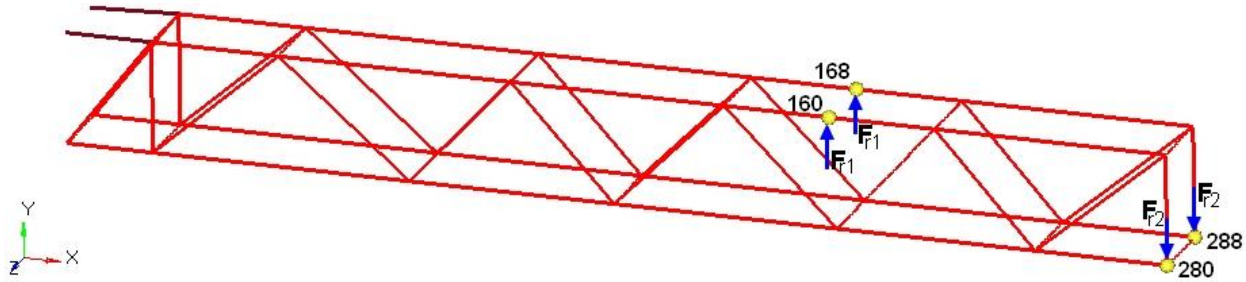


Figure 206: Intermediate load cases

An overview of the intermediate load cases is given in Table 31: Intermediate load cases. All these load cases are constrained in the X-direction for node 671. The subscript 1,2and 3 at each load case indicate the group of nodes to which the constraints are applied. This leads to four additional load cases which are subjected to all the conditions described in this section.

Table 106: Intermediate load cases

Load case	*Nodes*	*Name*	*Magnitude [N]*
$LC3_1$	160, 168	F_{r1}	5734
	280, 288	F_{r2}	21659

A-19. Optimization pitfalls

This appendix is concerned with all the optimization problems which where encountered during the optimization process. The goal of this appendix is to help users which are new to the phenomena of topology optimization, with the use of the software. During the research different problems where encountered which causes a delay of the research. Solving these problems can be frustrating and time consuming. These pitfalls will be addressed in this appendix in order to help new users to avoid these problems in their optimization process. This enables the users to perform the optimization process more easily and to obtain satisfactory results.

The first import aspect in the optimization process is to check if the boundary conditions are loads are applied in a correct way. This sound obvious but often this step is neglected or skipped due to the fact that the user is interested in the optimization results and therefore rushes through the steps which needs to be performed for the optimization process. The optimization process may take a lot of time before it has converged and results can be shown. Therefore it is suggested to perform an analysis before the optimization process in order to check the behaviour of the structure. Are the deflections for the different load cases as expected? Are the high stresses in a region where it is expected, like for example near the support conditions. These checks will ensure that the input parameters for the optimization process are well defined.

The second point which will be addressed is the influence of gravity on the design. Often the user would like to include the gravitational earth gravity in the optimization. This sound reasonable because the final structure will also be subjected to gravitational accelerations. It is important in incorporate all the loads acting on the structure but including the gravity has the following disadvantage: When a design space is defined with a certain volume from a certain material, the mass of the design domain is defined as volume multiplied by the density. Often this results in a large mass which is subjected to the gravitational or operational accelerations. This will result in large inertia forces which are often a few magnitudes larger compared to the body forces acting on the structure. This means that the gravitational forces will have a larger contribution to the stresses and deflection of the structure and therefore the structure will be designed to handle these loads. All the other forces acting on the structure will not have a significant impact on the structure due to the difference in magnitude. A solution to this problem is to apply point masses which are for example equal to the mass of the current design. Then the density of the design domain can be set to zero. This will result in a more realistic result. Another important aspect of this solution is that the mass of the structure will not change, therefore the resulting forces from the accelerations will remain constant. Therefore the result of the

Another interesting point is that the stress and deflection constraint have no real significant meaning in the optimization results. In almost every situation, post processing step are required in order to obtain a feasible design. The results from the optimization process can only be manufactured by using 3d printing or casting. Often this is not the preferred manufacturing solution and therefore post-processing steps are required in order to obtain a design which complies with all the manufacturing and service constraints. Therefore deflection or stress constraints have no significant meaning due to the fact that the optimization results only will serve as an indication for the final design. This means that the topology of the solution is optimum but that the dimensions of the structure are not. When the amount of input parameters is increased also the required time per iteration step increases even as the required time before convergence is reached. Therefore it is suggested to start with a simple solution with for example just one load case without any accelerations. Then review the result and rerun the optimization process with more load cases and accelerations. This will help to user to get insight on the behaviour of the optimizer and the influence of the different parameters and load cases.

The use of different accelerations in the model results in a model which has a low convergence rate. This is due to the fact that the total mass of the structure is changing which causes a change in the inertia forces. The use of accelerations in the optimization process will increase the required time for each iteration step and therefore it will result increased solving time. Therefore it is suggested to neglect the accelerations in the optimization process and to start with a simple defined problem. Then the user can simply run an optimization task and determine if the solver output is as expected. If this is not the case, then the model can be checked easily or an FEA for the design domain can be performed in order to check all the deflections and stresses in the model. If the solver output is plausible, the complexity of the model can be increased further in order check the influence of these parameters on the final result.

The final message that I will give about topology optimization is that this method gives only a concept of the optimal structure. Usually the large amount of the mesh elements have an intermediate density value and the final shape depends on the selected density limit value. Therefore the reliability of the structure should always be verified. The optimization algorithm removes all the unnecessary material from the design domain. Therefore the correct definition of the loads and boundary conditions are very important. If these are not defined properly, the structure may not survive the real conditions. Stress constraints will only been applied for the average stress in the structure and will not deal with stress peaks. Therefore it is not suggested to check the stress values during the optimization process. The use of displacements of compliance constraints will give more reasonable results. Mass constraints can be used during the optimization process but these will give only representative values when no post-processing steps are required for the manufacturing process. For example when the solution of the optimization process is manufactured by using 3D-printing. Different types of manufacturing constraints are available in the different topology optimization packages. Minimum dimensions, symmetry planes and manufacturing directions can be defined. All these options have effect on the solution of the optimizer.


BW 52
BW
52
BW
52
TU Delft
IHC