The present work was submitted to the Faculty of Engineering

Creating and Improving a Product Model with Simulation and Topology Optimization

Bachelor Thesis

by

B. Ariunsanaa (ID: 15667999618364)

Supervisor 1 / Examiner 1	**Prof. Sungchil Lee**
Supervisor 2 / Examiner 2	**Prof. N. Battulga**

Ulaanbaatar/Nalaikh, May 10, 2023

Acknowledgement

"I would like to express my gratitude to Professor Sungchil Lee, my thesis advisor, for his guidance and support throughout my research project. His unwavering commitment to my academic and personal growth has been instrumental in the completion of this thesis.

I would also like to extend my thanks to the faculty and staff of German Mongolian Institute for Resources and Technology, whose expertise and feedback have contributed to the development of my research skills and academic abilities.

Additionally, I would like to acknowledge the support and encouragement of my family and friends, who have been a constant source of motivation throughout my academic journey.

STATUTORY DECLARATION

I, B. Ariunsanaa, declare that this thesis, titled “Creating and Improving a Product Model with Simulation and Topology Optimization”, is my original work and that it has not been submitted for any degree or examination at any other university.

I acknowledge that the research work reported in this thesis has been carried out by me under the supervision of Professor Sungchil Lee, in the Mechanical Engineering Department, German Mongolian Institute of Resources and Technology.

I declare that all sources of information and assistance used in the research work have been duly acknowledged and referenced in the thesis. Any contributions made by others to the research work have also been duly acknowledged.

I declare that this thesis is in compliance with the guidelines and regulations of the German Mongolian Institute of Resources and Technology regarding academic honesty and integrity.

Signed: Ариунсанаа

Full Name: B. Ariunsanaa

Date: May 9, 2023

Table of Contents

Table of figures

Glossary

1. **Deflection:** Deflection is the bending or deformation of a structural element under a load, crucial for ensuring structural integrity and safety.
2. **Load Swing:** Load swing refers to the abrupt movement or change in direction of a lifted load, which can pose safety risks and requires careful control.
3. **Boundary conditions:** Boundary conditions are the restrictions or limitations imposed on a system or structure by its surrounding environment, influencing its behavior and performance.
4. **Design space:** Design space encompasses the range of possible design solutions available for a project, allowing exploration and selection of the most suitable option.
5. **Iteration:** Iteration involves repeating a set of steps or processes to refine and enhance a design or solution, often leading to iterative improvements.
6. **Hoist:** A hoist is a mechanical apparatus used for lifting and moving heavy objects, employing mechanisms like pulleys or chains to raise or lower loads.
7. **Girder:** A girder is a horizontal or sloping structural component that supports a load across an expanse, commonly found in bridge construction and large structures.
8. **Flange:** A flange is an outward rim or edge on a structural element, often used for strength, stability, or connecting different components together.
9. **Web:** The web is the central vertical section of a structural element, connecting the flanges and providing additional strength and support.
10. **Steel grade:** Steel grade refers to the classification of steel based on its chemical composition and properties, determining its suitability for specific applications.
11. **Young's Modulus:** is a measure of the stiffness of a material. It describes the relationship between stress (force per unit area) and strain (deformation per unit length) in a material when it is subjected to tension or compression.

Introduction

Literature Review

- **What is CAD Simulation?**

Computer-Aided Design (CAD) simulation and topology optimization are widely used in mechanical product design and are considered to be important tools in the development process. These technologies have advanced significantly in recent years and are now capable of simulating complex scenarios, optimizing designs, and providing valuable insights into product performance.

CAD simulation involves using computer software to simulate the behavior of a product under different conditions. This can include testing the product's structural integrity, analyzing its thermal and fluid dynamics, and predicting its performance in real-world scenarios. By simulating these scenarios, designers can identify potential issues and optimize their designs before building a physical prototype, leading to faster and more cost-effective product development.

- **What is Topology Optimization?**

To explain the concept first I need to explain what topology itself is.

Topology is a branch of mathematics that studies the properties of geometric objects that are preserved under continuous transformations, such as stretching, bending, or twisting. In simpler terms, it is the study of the properties of space that are unaffected by deformations. Topology is concerned with concepts such as continuity, compactness, connectedness, and convergence, and it is used in a wide range of fields, including physics, engineering, computer science, and biology.

Topology optimization is a design optimization technique that maximizes the performance of a structure or component by determining the optimal layout of material within a given design space. The method is used to generate lightweight, efficient designs that meet specific design constraints. We can create lighter, more efficient designs that still meet all of the necessary performance requirements

Topology optimization is another important tool that is commonly used in mechanical product design. This approach involves using algorithms to optimize the shape and material distribution of a product, based on predefined performance goals and constraints. By taking into account factors such as stress distribution, weight, and manufacturing constraints, topology optimization can help designers create products that are stronger, lighter, and more efficient.

- **For what they are used?**

Topology optimization is typically used in the design of structures and components where weight, strength, and stiffness are critical factors. The method is commonly used in industries such as aerospace, automotive, and manufacturing, where lightweight and efficient designs are important for improving performance and reducing costs. Topology optimization is also used in the design of medical implants and prosthetics, as well as in architectural and civil engineering applications. Overall, topology optimization is used in any application where a designer needs to determine the optimal layout of material within a given design space to achieve specific performance goals while satisfying design constraints.

- **For what manufacturing technologies can be applied?**

Topology optimization can be applied to a wide range of manufacturing technologies, including traditional manufacturing methods such as casting, forging, and machining, as well as advanced manufacturing methods such as additive manufacturing (3D printing), laser cutting, and electrochemical machining. The method is particularly well-suited for additive manufacturing, as it allows designers to create complex, lightweight structures that would be difficult or impossible to produce using traditional manufacturing methods. Additionally, topology optimization can be applied to materials such as metals, polymers, and composites, making it a versatile design optimization tool for a variety of manufacturing applications.

- **What aspect will I optimize?**

Topology optimization can optimize a wide range of aspects of a structure or component, including:

Weight: Topology optimization can help designers reduce the weight of a structure or component while maintaining or improving its performance. This is particularly important in industries such as aerospace and automotive, where weight reduction can lead to significant improvements in fuel efficiency and performance.

Stiffness: Topology optimization can optimize the stiffness of a structure or component to ensure that it can withstand the loads and stresses it will encounter during use.

Strength: Topology optimization can optimize the strength of a structure or component to ensure that it can withstand the loads and stresses it will encounter during use.

Thermal performance: Topology optimization can optimize the thermal performance of a structure or component, such as minimizing thermal distortion or maximizing heat dissipation.

Fluid dynamics: Topology optimization can optimize the flow of fluids through a structure or component, such as optimizing the shape of a heat exchanger or turbine blade.

For this thesis I will be focusing on Weight, Strength and Stiffness of the components.

- **What problem will I tackle with this method?**

Although topology optimization is mainly used for additive manufacturing, my main interests in the future are subtractive methods, CNC machining, milling and turning to be specific.
So I want to study the way to use topology optimization for my interests.
The way I intend to do so is using constraints where the cnc will not be able to machine, while also obtaining the benefits of using the optimization method.

Problem Statement:

Suppose that I have been assigned the challenging task of designing a mechanical crane that must fulfill a specific set of requirements. The crane must be not only lightweight and stylish, but also capable of safely lifting up to 5 tons of weight. Known as Electric Overhead Traveling (EOT) Crane, it is expected to adhere to the industry's safety standards. To ensure that the crane blends seamlessly with its surroundings, the design language must be aesthetically pleasing and complement the building's style.

The exhibition center hosts year-round conferences that serve as a vital link between creators and consumers. These events showcase new ideas, creations, and inventions, bringing producers and companies together to exchange insights and explore the latest innovations across a broad range of fields.

“Heavy exhibitions are transported into the hall using forklifts and boom trucks. But recently, it has been observed that the floor tiles are being damaged and are showing signs of wear and tear caused by these transporting machines. Using carpets for the travel paths were suggested and being used, but the setup and removal of such increased working time.
There are also risk and difficulty concerns of operating boom trucks as to not damage the transporting item or the already settled exhibitions.

Presented with these problems, the owners of the facility opted to install an overhead crane. The market selections are incompatible with the structure of the building, so they reach to us requesting to manufacture a customized crane to fulfill their requirements.”

Delimitations:

While CAD Simulation and Topology Optimization are powerful tools that can greatly benefit the engineering design process, they also have their limitations and disadvantages.

One major limitation of CAD Simulation is that it relies on mathematical models to predict the behavior of complex systems. These models are based on simplifying assumptions and may not accurately capture all of the real-world factors that can influence a system's performance. This can lead to errors and inaccuracies in the simulation results, particularly when dealing with highly complex systems or when simulating extreme conditions.

Another limitation of CAD Simulation is that it can be computationally expensive and time-consuming. Simulating large and complex systems can require significant computational resources and may take hours or even days to complete. This can be a major barrier to using CAD Simulation effectively, particularly for small companies or organizations with limited resources.

Similarly, Topology Optimization also has its limitations and disadvantages. One major limitation is that it can be difficult to incorporate manufacturing constraints into the design process. Topology Optimization is typically used to generate highly efficient designs that may not be practical or feasible to manufacture using traditional methods. As a result, engineers may need to spend additional time and resources to modify the design for manufacturability, which can negate some of the efficiency gains from Topology Optimization.

Another limitation of Topology Optimization is that it can be sensitive to the input parameters used to generate the design. Small changes in the input parameters can lead to significant changes in the resulting design, which can make it difficult to achieve consistent and reliable results. This can be particularly challenging for engineers who are not familiar with the software or who do not have the necessary expertise to interpret the results.

In addition to these limitations, both CAD Simulation and Topology Optimization can also be subject to user error. Engineers may unintentionally introduce errors into the simulation models or input incorrect data, which can lead to inaccurate or misleading results. Similarly, engineers may be biased towards certain design approaches or solutions, which can lead to less optimal designs.

Despite these limitations and disadvantages, CAD Simulation and Topology Optimization remain valuable tools for engineering design. By being aware of their limitations and addressing them appropriately, engineers can take advantage of the strengths of these tools to design better products and systems that meet the needs of their users.

State of the art

In terms of the state of the art in mechanical product design, there are several emerging trends and technologies that are driving innovation in this field. One of these is the use of artificial intelligence (AI) and machine learning to enhance the design process. By analyzing large datasets and generating predictive models, AI can help designers identify design patterns and optimize their designs for improved performance and efficiency.

Another emerging trend is the use of additive manufacturing, such as 3D printing, to create complex geometries and reduce manufacturing costs. This technology allows designers to create parts with intricate shapes and internal structures that would be difficult or impossible to produce using traditional manufacturing methods.

Finally, there is a growing focus on sustainability and environmentally conscious design in mechanical product development. This includes designing products that are more energy-efficient, have a smaller environmental footprint, and can be recycled or repurposed at the end of their life cycle.

Overall, CAD simulation, topology optimization, and other advanced technologies are widely used in mechanical product design and are helping to drive innovation in this field. By incorporating emerging trends and best practices into their design processes, companies can stay ahead of the curve and create products that are both innovative and sustainable.

Material and Methods

Methodology

How does the software work?
The software uses mathematical algorithms to solve problems presented, the most widely used, and the one to be used for this thesis, is the Finite Element Method.

The finite element method (FEM) is a numerical technique used to solve complex engineering problems. It involves dividing a complex geometry into small, simpler parts called finite elements, which can be represented mathematically. Each element is then connected to its neighboring elements through nodes, which form a mesh. The behavior of each element is described by a set of equations, which are then combined to solve the overall system. By applying boundary conditions, such as loads and constraints, the behavior of the entire system can be analyzed and simulated. The FEM is widely used in various fields of engineering, such as structural analysis, heat transfer, fluid dynamics, and electromagnetism. Its flexibility and accuracy make it a powerful tool for predicting the behavior of complex systems under various conditions, but its results are only as good as the accuracy of the model and the assumptions made.

How does it "optimize" the model?
We understand that in Simulation, the model is dissected into mesh elements and every node behavior is represented, but Optimization is the creation of an altered model, so how does the software recreate an alternate model using FEA?
After defining the design space and boundary conditions and generating a finite element mesh within the design space, the computer repeats the following steps:

1. Assign material properties and constraints to each element
2. Solve the finite element equations to determine the structural behavior of the system
3. Update the material distribution within the design space based on the optimization algorithm

Each step is repeated until the objective function is minimized within the given constraints.

The number of iterations can vary depending on the complexity of the model and attainability of the goal.
To ensure the reliability of my analysis, I will focus on accurately determining load forces, to correctly set up study cases and use my background knowledge of engineering to assess and interpret the results that the computer delivers.
I will guide myself for the design process using Simulation and improve my own design using Optimization.

Additional guidelines:

- Classification of the crane

The importance of classification and determination of the type of crane lies with design factors like the crane runway support, beam design and bridge type.
Cranes are classified by

1. Class of utilization: required number of operation cycles for the crane
2. State of loading: conditions of loading to be subjected

Class of utilization:
Determined from the assumed total number of operating cycles during its lifespan.
One operating cycle: from picking a load to when the next load is ready for pick up.
Max. number of operating cycles = 6 times a year * 100 cycles * 40 yrs lifespan = 2.4 x 10^4. This falls under the class U1 (See Figure 1.1).

Table 1. Class of utilization

NOTE. The number of cycles used in selecting the class of utilization is a figure used only for classification purposes and as a design parameter. It does not imply a guaranteed life.

Class of utilization	Max. number of operating cycles	Remarks
U1	3.2 $\times 10^4$	Infrequent use
U2	6.3 $\times 10^4$	
U3	1.25 $\times 10^5$	
U4	2.5 $\times 10^5$	Fairly frequent use
U5	5 $\times 10^5$	Frequent use
U6	1 $\times 10^6$	Very frequent use
U7	2 $\times 10^6$	Continuous or near-continuous use
U8	4 $\times 10^6$	
U9	Greater than 4 $\times 10^6$	

Figure 1.1 /Class of utilization of Cranes/

- State of loading and nominal load spectrum factor.

Load spectrum factor:

$$K_p' = \sqrt[3]{\left\{ \frac{C_1}{C_t}\left(\frac{P_1}{P_{max}}\right)^3 + \frac{C_2}{C_t}\left(\frac{P_2}{P_{max}}\right)^3 + \frac{C_3}{C_t}\left(\frac{P_3}{P_{max}}\right)^3 + \ldots + \frac{C_n}{C_t}\left(\frac{P_n}{P_{max}}\right)^3 \right\}}$$

Figure 1.2 /Load spectrum factor formula/

The load spectrum factor **K_p'** is a coefficient that determines the state of loading. In other words, it answers the question "How much of the capacity of the crane will be used?". For example, if a crane with a rated capacity of 10 tons will be lifting weights of only 5 tons its load spectrum factor will be closer to 0.5; otherwise, if the loads are always between 8 to 10 tons the load spectrum factor is much closer to 1.0.

In our case, this factor cannot be calculated because the loads are not consistent. Unlike in a factory, we don't have a predetermined weight. However, the client reported the usual maximum weight to be 3 tons. So I'll assume the load spectrum factor is close to 0.5 and the state of loading to be Q1 (Light).

Table 2. State of loading

State of loading	Nominal load spectrum factor K_p	Descriptive definition
Q1 Light	0.5	Cranes which hoist the safe working load very rarely and, normally, light loads.
Q2 Moderate	0.63	Cranes which hoist the safe working load fairly frequently and, normally, moderate loads.
Q3 Heavy	0.8	Cranes which hoist the safe working load frequently and, normally, heavy loads.
Q4 Very heavy	1.0	Cranes which are normally loaded close to safe working load.

Figure 1.3 /State of loading classification/

The classification for our crane is Q1 x U1 = A1.

A1 classification are light duty cranes, normally single girder cranes not needing support beams.

- Deflection

  $\delta = (5 * w * L^4) / (384 * E * I)$

where:

δ = deflection (millimeters)
w = load (Newtons per millimeter)
L = length of the beam (millimeters)
E = modulus of elasticity (Pascals)
I = moment of inertia (millimeters^4)

Steps:

1. Determine the modulus of elasticity (E) of the material the beam is made of.
2. Calculate the moment of inertia (I) of the I-beam. This is a measure of the beam's ability to resist bending and is dependent on the cross-sectional shape of the beam. I can use standard formulas for the moment of inertia of common shapes, or consult reference materials or software tools for more complex shapes.
3. Substitute the values for w, L, E, and I into the equation above, and solve for δ. This will give me the deflection of the beam at the point where the load is applied.

This equation assumes that the beam is simply supported, meaning that it is supported at both ends and the load is applied at the center of the beam. If the beam is supported in a different way, or if the load is applied at a different location, additional equations or modifications may be necessary to calculate the deflection accurately.

- Load Swing

To calculate the load swing on an I-beam, I need to know the moment of the applied load and the moment of inertia of the beam.
Here is the equation for calculating the maximum load swing on an I-beam:

Max Load Swing = (M * L) / (4 * I)

where:
Max Load Swing is the maximum deflection of the beam due to the applied load (in inches or millimeters, depending on the units used)
M is the moment of the applied load (in pound-feet or Newton-meters, depending on the units used)
L is the length of the beam (in inches or millimeters, depending on the units used)
I is the moment of inertia of the beam (in inches^4 or millimeters^4, depending on the units used)

- Sway and Skew

I-beams are generally designed to resist bending and shear loads along their principal axes, and they are typically not designed to resist sway or skew. In order to calculate the sway or skew of an I-beam under a specific load, here are some steps to follow:

1. Determine the forces acting on the beam, including any vertical loads, horizontal loads, and moments.
2. Calculate the reactions at the supports of the beam. This will require solving the equations of static equilibrium.
3. Calculate the shear force and bending moment along the length of the beam using the equations of equilibrium and the properties of the beam, including the cross-sectional area, moment of inertia, and modulus of elasticity.
4. Using the deflection values, I can determine the sway and skew of the beam by calculating the horizontal and vertical displacement of each end of the beam. Sway is the horizontal displacement, while skew is the vertical displacement.

List of demands

The requirements for a crane to be installed in a building with an area of approximately 8000 m^2.

- Maximum Weight and Lift Height:

The crane must be capable of lifting a maximum weight of 5 tons and have a lift height of minimum 6 meters. These requirements are essential for the efficient movement of materials and equipment within the building. The crane's lifting capacity must be able to accommodate the heaviest load that is required to be lifted.

- Aesthetics and Lightweight:

The crane must aesthetically match the building's design language, and it must be lightweight. These requirements are necessary for the crane to blend in with the building's overall design and not detract from the ambient. The crane's lightweight design will ensure that it does not add unnecessary weight to the building's structure, as it was originally not designed to accommodate a crane.

- ISO Standards:

The crane must comply with ISO standards, specifically ISO 4301 - Cranes - Classification. Compliance with these standards will ensure that the crane meets international standards for safety, reliability, and efficiency.

- Safety Margin:

The crane must have a safety margin of 2. This means that the crane must be able to lift twice the maximum load capacity required without failure. The safety margin is necessary to prevent any accidents or damage to equipment due to overloading.

- Travel Capability:

The crane's design must incorporate features that allow it to reach all parts of the building.

- Estimated Operating Frequency and Lifespan:

The crane is estimated to operate up to six times a year, with a maximum of 100 cycles per time. The crane must be designed to withstand the estimated operating frequency and cycles. Additionally, the crane must have a lifespan of at least 40 years to ensure a long-term solution for the building's lifting needs.

The crane installed in the building must meet several requirements, including maximum weight and lift height, aesthetics, lightweight, central and outer wall height difference, compliance with ISO standards, safety margin, travel capability, estimated operating frequency and lifespan. Meeting these requirements is crucial for the crane's safe and efficient operation, ensuring the building's lifting needs are met for years to come.

Modeling the design in CAD and testing (Simulation)

Modeling a product from scratch is a very time consuming task. It all starts with notebook sketches, ideas, brainstorming and a lot of planning. Any concept ideas and sketches will be omitted to avoid filling with content irrelevant to the scope of my thesis. My work is focused on the frame structure of the overhead crane and mechanics of the trolley and end trucks.

Disclaimer: All the design process was entirely made from scratch and every part was modeled by the author on Autodesk Inventor. The model does not consider any electrical components, safety systems, power source or control system. Nor does it contain the hoisting mechanism, as it is a different set of design parameters and calculations. The electric motors on the model are assumed sizes depending on the power demand and are not real product models. Gear ratios are not calculated as it is dependent on the motor outputs.

About the model

The two main design options found upon research are single girder and twin girder bridges.
Single girder EOT cranes are lighter and have a smaller footprint. The crab system mounts on the lower flange of the girder and additional rails along the bridge are not necessary. By design standards, maximum load capacity is usually 25 tons or lower; this is mainly to avoid buckling on the single beam due to torsional forces.

Double girder or Twin girder EOT cranes are designed for heavy applications, longer bridges and have more contact points on the rails that distribute the load on a larger area. They can be safely designed to lift more than 500 tons and have a wide range of heavy duty applications and environments. The larger design format allows us to have maintenance platforms and better accessibility for repairs and servicing.

The design I opted for is a single girder bridge. It matches our needs for lightweight installation and lighter load capacity.

1. **The Bridge**: The bridge is the main horizontal beam of the crane that spans the width of the building and supports the hoist and trolley.

Main loads are vertical. I beams are best known for its strength and it's ideal for supporting heavy loads and spanning long distances without sagging or bending. I beams are commonly used in construction for the framework of buildings, bridges, and other large structures, as well as in the manufacturing of heavy equipment, vehicles, and machinery.

The following dimensions are extracted from a table representing European standard IPE beams with parallel flanges. Manufactured according to standards :

DIN 1025
Euronorm 19-57 (Dimension)
EN 10034: 1993 (Tolerances)
STN 42 5550, 0135

Beam height (h)
Flange width (b)
Web thickness (t1)
Flange thickness (t2)
Beam cross section area (A)

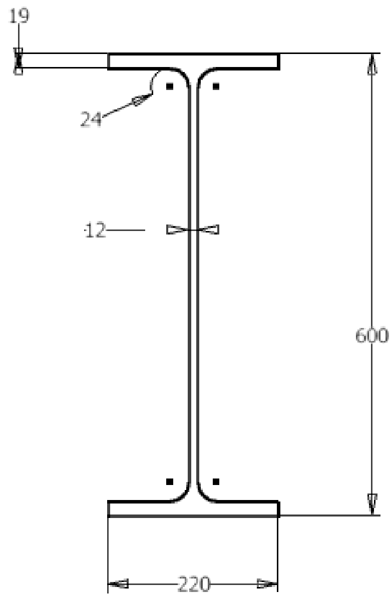

Figure 2.1.1 /I-Beam Dimensioning/

Identification	Nominal weight 1m	Nominal dimensions					Cross-section	Dimensions for detailing					Surface	
	kg/m			mm			A	h1	d	Ø	pmin	pmax	AL	AG
		b	h	t1	t2	R1	cm2	mm	mm		mm	mm	m2/m	m2/m
IPE 600	122,0	220	600	12,0	19,0	24,0	156,00	562,0	514,0	M27	116	118	2,015	16,45

Figure 2.1.2 /DIN 1025 Standard Beam dimensions, IPE 600/

Beam material: Steel ASTM A572

ASTM A572 is a specification for high-strength low-alloy (HSLA) structural steel plate. This specification covers five grades of high-strength low-alloy shapes intended for riveted, bolted, or welded construction of bridges, or for other construction applications.

The five grades of ASTM A572 are: (followed by its minimum yield strength)

- Grade 42 (290 MPa)
- Grade 50 (345 MPa)
- Grade 55 (380 MPa)
- Grade 60 (415 MPa)
- Grade 65 (450 MPa)

The atmospheric corrosion resistance of this steel is better than that of carbon structural steels with or without copper addition. When properly exposed to the atmosphere, this steel can be used unpainted.

The composition of ASTM A572 steel includes carbon, manganese, phosphorus, sulfur, silicon, columbium (niobium), vanadium, and sometimes nitrogen. The combination of these elements produces improved properties such as strength, toughness, and weldability.

ASTM A572 steel can be produced in the form of plates, structural shapes (e.g. H-beams, I-beams, angles), and bars. The steel can also be heat-treated to enhance its strength. The maximum thickness for plates is 4 inches (101.6 mm), while the maximum thickness for shapes and bars is 6 inches (152.4 mm).

I insist on researching the properties because it is one of the main factors for predicting the behavior of the material software wise.

For the purpose of our design, I will be using ASTM A572 Grade 50. It’s properties are:

- Density: 7.85 g/cm^3 (0.284 lb/in^3)
- Elastic modulus: 190-210 GPa
- Poisson's ratio: 0.3
- Yield strength: 345 MPa (50 ksi)

- Tensile strength: 450 MPa (65 ksi)
- Elongation at break: 18%
- Charpy impact strength: 20 J (15 ft-lb) at 0°C

Steel ASTM A572 Grade 50 is already included in the library of our software, I'll make sure the properties of the material are correct:

**Behavior*: Isotropic and orthotropic are terms used to describe the mechanical behavior of materials. An isotropic material has the same mechanical properties in all directions, while an orthotropic material exhibits different mechanical properties in different directions.

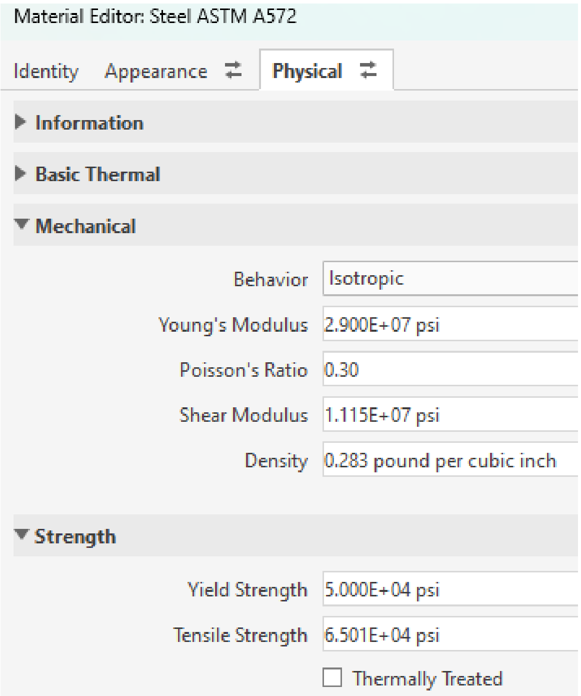

Figure 2.1.3 /Material properties editor menu in Autodesk Inventor/

In the case of steel, the material is typically considered to be isotropic, meaning that its mechanical properties, such as stiffness, strength, and toughness, are the same in all directions.

**Heat treatment:* I-beams are typically heat treated when they need to be hardened or to improve their strength and durability. Heat treatment can also be used to relieve stress and improve the machinability of the material. The decision to heat treat I-beams depends on the specific application and the properties required for the part to perform its intended function. For example, if an I-beam is going to be used in a high-stress application, it may be heat treated to increase its strength and durability.

Our beam will carry the entirety of the weight of the hoist and the load on the lower flange. In these circumstances I will consider heat treating it.

According to the perimeters of the hall to be installed, the beam needs to be at least 39 meters long.

2. **Trolley**: The trolley is the device used to move loads horizontally along the bridge. It is typically mounted on wheels or rollers that travel along the rails of the bridge. The trolley is connected to the load by a hoist. By moving the trolley along the bridge, the load can be positioned precisely over the desired location for placement or further processing.

To reduce unbalanced loads generating moment forces on the beam I opted to install the hoist lengthwise of the bridge. It will be anchored on the two ends by one non-drive trolley and the other one with the drive motor:

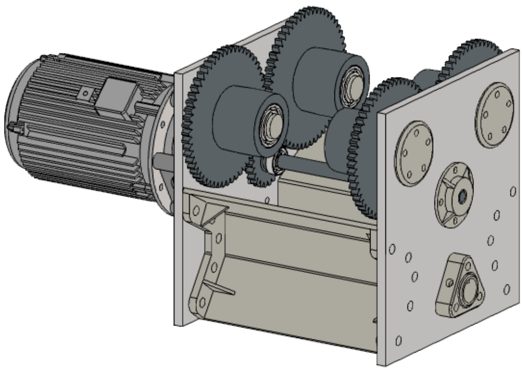

Figure 2.2.1 /Trolley Assembly, Drive side/

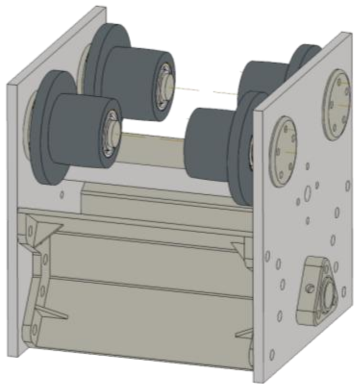

Figure 2.2.2 /Trolley Assembly, Non-drive side/

The components of the drive trolley are:

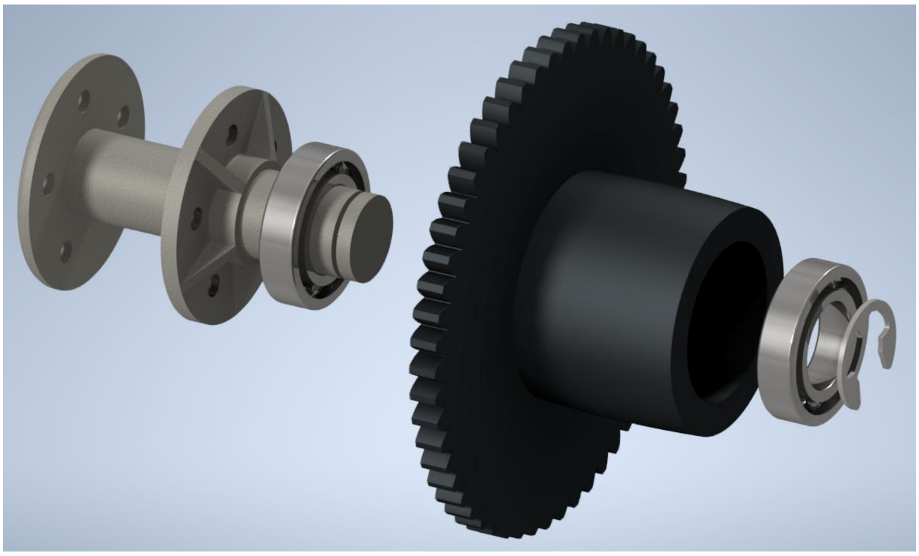

Figure 2.2.3 /Trolley wheel assembly/

- Four geared wheels that are simultaneously driven by the geared shaft. The inner side walls help with the alignment along the travel axis. Grooves on each side of the wheel are designed to accommodate the two bearings. The support shaft is clamped down by five M8 bolts with an inner disk that also offers protection against shearing and bending. The whole is locked with a c-clip.

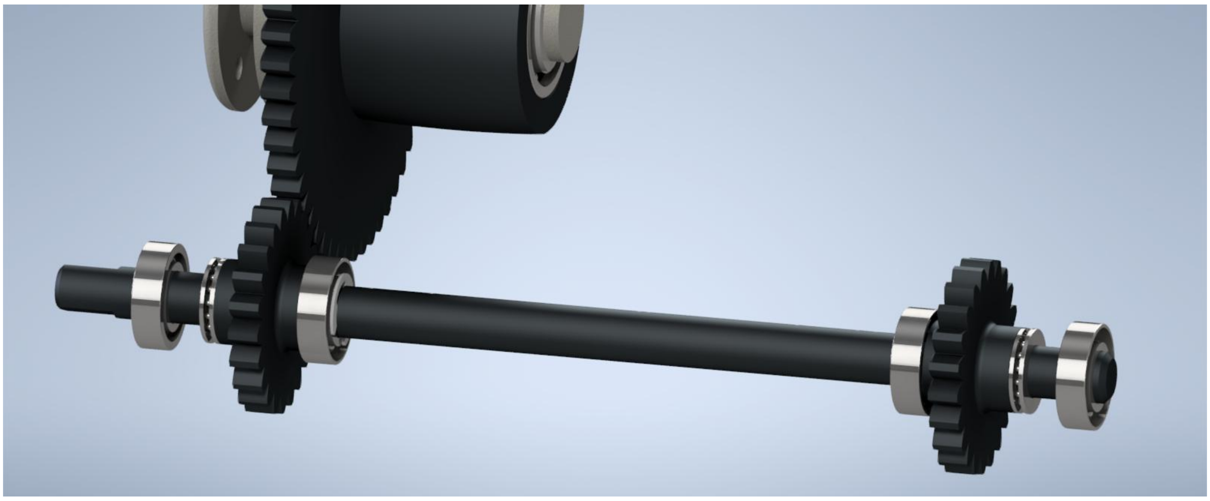

Figure 2.2.4 /Trolley Geared shaft assembly/

- The geared shaft contains three sets of bearings. The outer ones are to assist the rotation of the shaft, meaning the shaft is mounted on those bearings. The inner bearings roll against the bottom part of the beam; while not necessary, it restricts the degree of freedom in directions we don't want the assembly to travel, they are held in place by c-clips. And the thrust bearings are placed as a measure to reduce friction against the side plates.

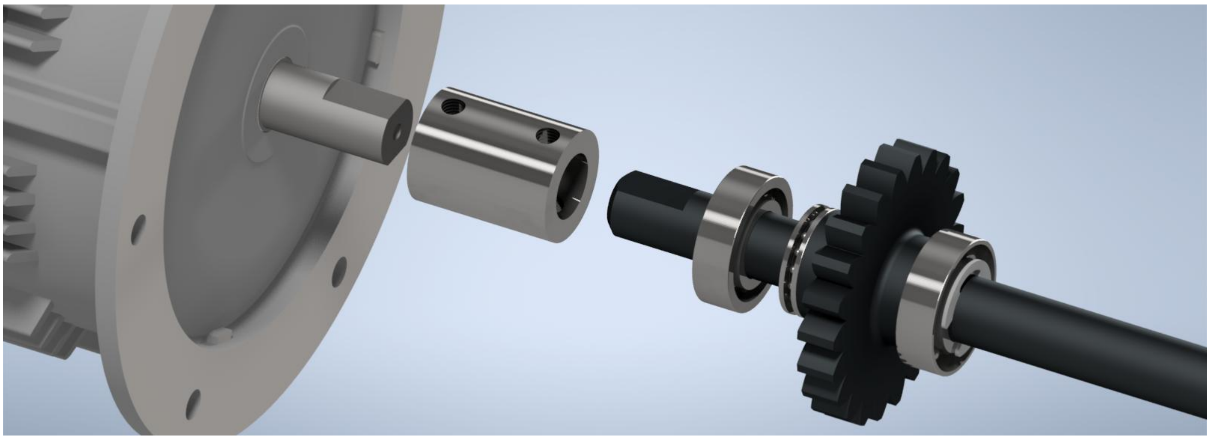

Figure 2.2.5 /Shaft and motor coupling/

- To transfer torque and rotational power from the motor to the shaft, a coupling is used. It can be bolted from both sides to relieve stresses on a single bolt thread.

Figure 2.2.6 /Trolley motor mount and bearing housing/

The shaft bearing is inserted into a disk that is bolted to the side plate by four M8 bolts. Because of the configuration and small footprint of the assembly, the motor is mounted offset with raised mounts with six M8 Bolts per side.

At last, a strong connection between the two plates is essential. This shape is chosen as it will stand vertical and horizontal forces, as well as moments.

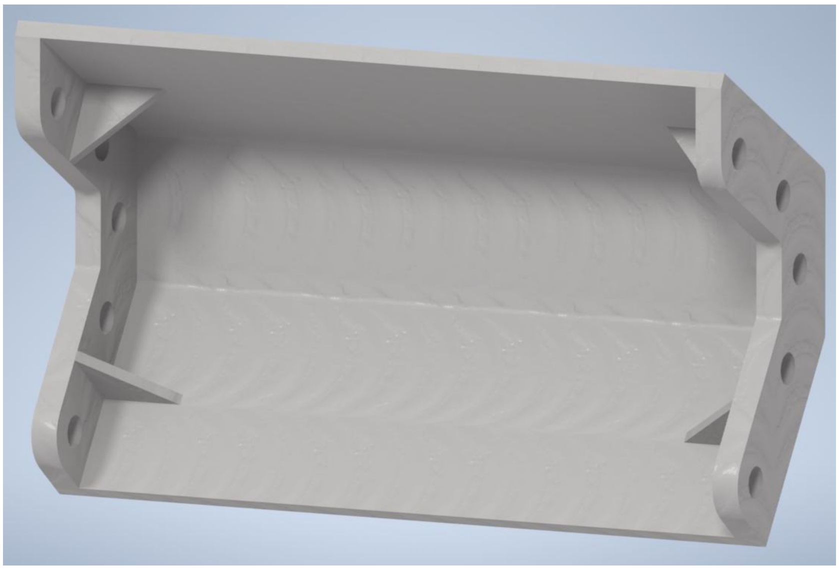

Figure 2.2.7 /Trolley reinforcement bracket/

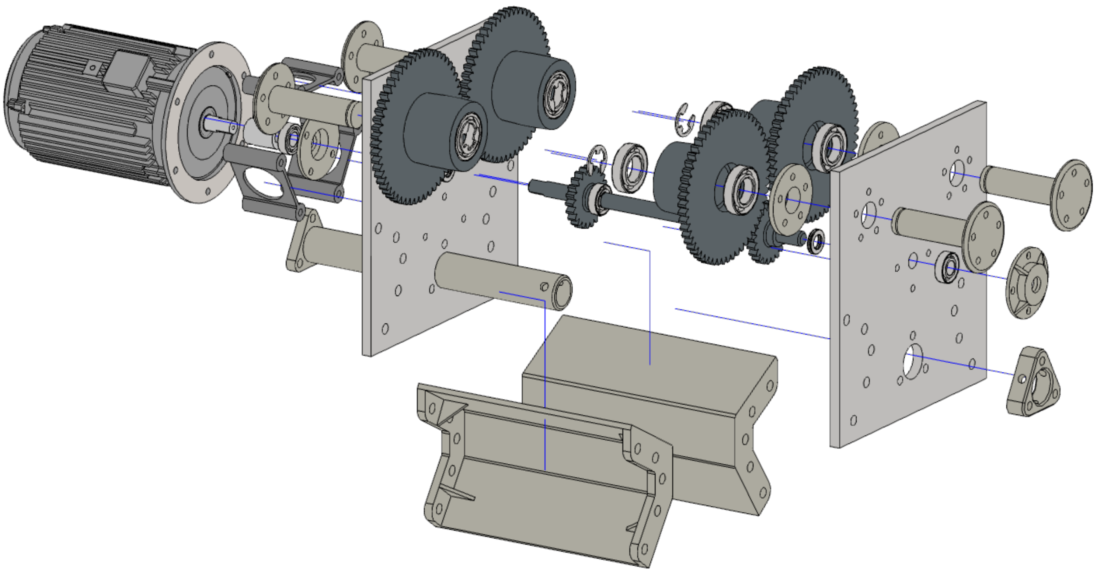

Figure 2.2.8 /Exploded view of drive-trolley/

3. **Hoist**: The hoist is the part of the crane that lifts and lowers the load. It is typically a motorized unit that is attached to the trolley and can move up and down on a set of ropes or chains.

As mentioned before, my model will only include the housing for the hoisting mechanism. The design will be discussed in the results section.

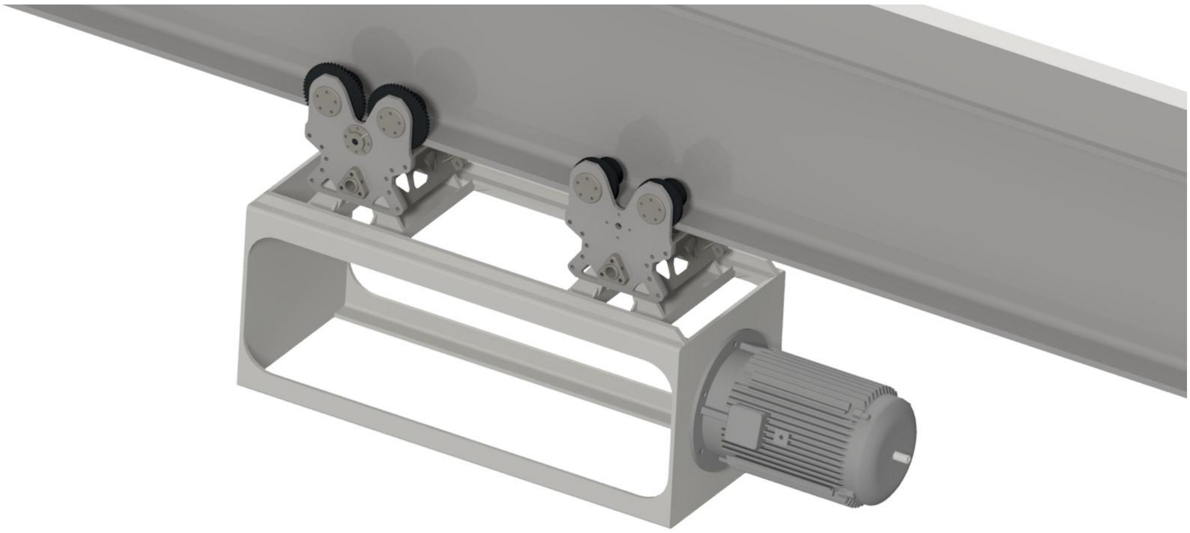

Figure 3.1 /Hoist Assembly/

4. **End trucks**: The end trucks are the structures that support the bridge and allow it to move side to side along the length of the building. In our case, because the building is circular, we have to design tracks that follow the radial path of the walls.

Figure 2.4.1 represents the outer truck. The side wheels have been aligned to the rails and the middle one is centered. Regardless of which direction it moves, the outer wheels will guide the path.

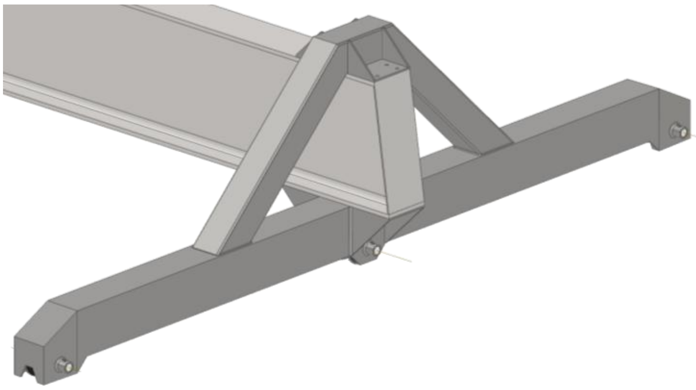

Figure 2.4.1 /External end trucks/

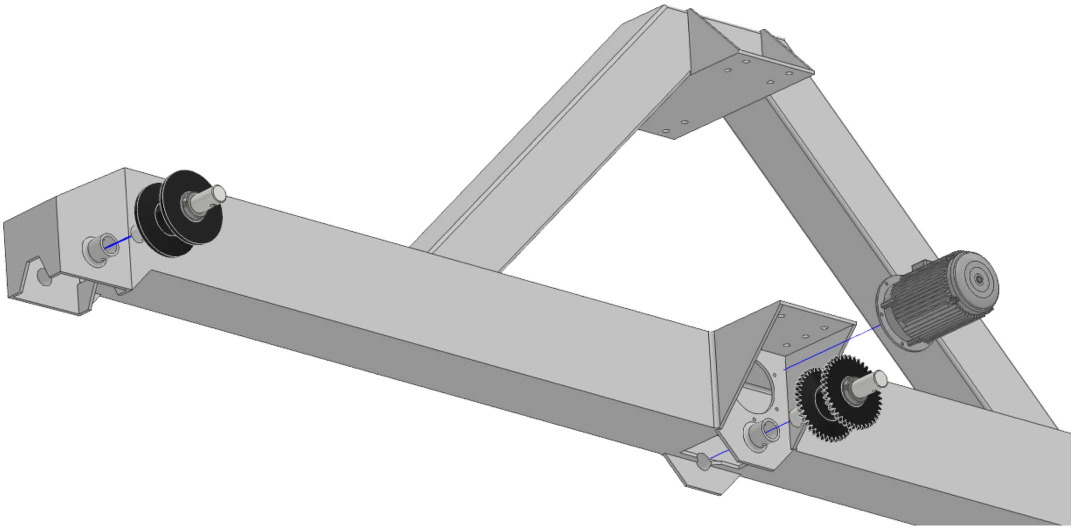

Figure 2.4.2 /External end truck components/

To distribute swing moments along the whole truck, a triangular support bar is implemented. This will ensure to share the stresses on the side wheels when lateral forces are present, otherwise tension forces will be too high only on one side of the bolt row and it could lead to failure.

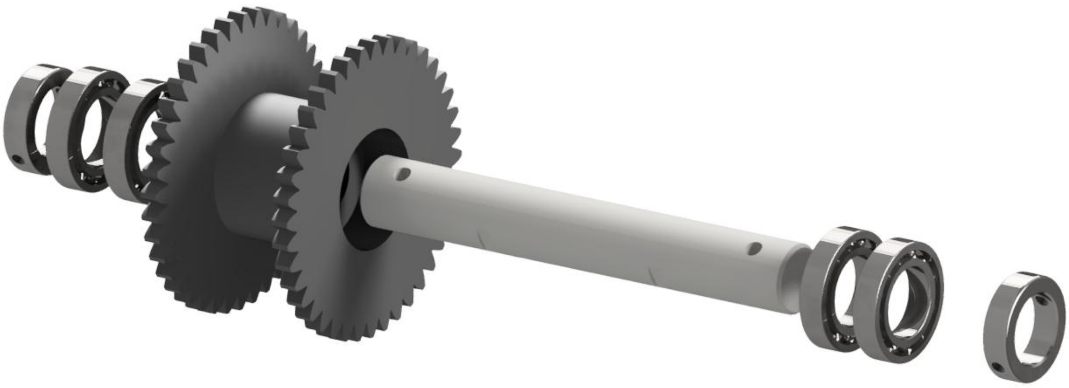

Figure 2.4.3 /End truck wheel assembly exploded view/

The wheel assembly is fairly simple. On both sides are installed one pair of bearings, which then is locked with shaft collars. I opted for collars because adding grooves for a c-clip will compromise the strength of the shaft, also horizontal shear forces can be higher in this part of the crane. There are flat edges for the collar bolts to sit on, the edges will prevent the collar from slipping along the axis.

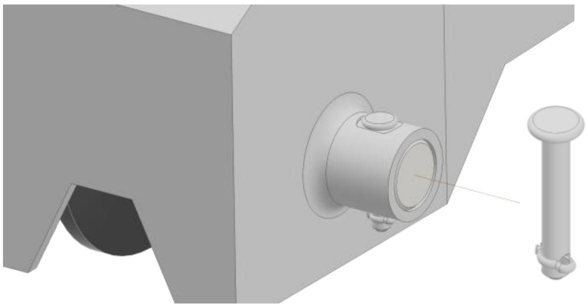

Figure 2.4.4 /End truck wheel shaft locking mechanism/

The shaft is secured to the frame with a locking pin with a diameter of 10mm.

Figure 2.4.5 represents the inner truck. Because the rails are going to be mounted on the inner walls of the building, installing the track below the bridge will restrict the reach of the hoist. Considering this fact, this side of the crane will be suspended, thus allowing the hoist to reach the wall.

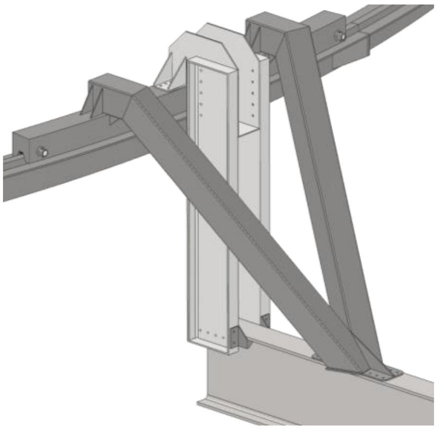

Figure 2.4.5 /Internal end truck/

The reason I used two different bars is the same as in the outer tack. Side to side forces will put too much stress on a single attaching point, so this ensures proper distribution.

We can see in Figure 2.4.6 that the wheel ensemble has the same configuration of double paired bearings and collar lock.

Lengthwise it is shorter due to the fact that the travel path is shorter too.

The two center members are standard C-Beams.

Figure 2.4.6 /Internal end truck exploded view/

5. **Runway**: The runway is the path along which the bridge of the crane travels. It is typically mounted to the ceiling or walls of the building.

In this design specifically, the outer runways are mounted on a flat surface above the wall. The inner ones are to be bolted on the wall itself.

- Outer runway

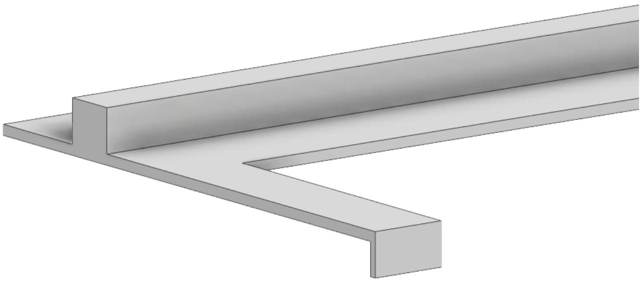

Figure 2.5.1 /Outer Runway/

Rail profile:	60 mm wide, 50 mm tall
Total length:	317.4 meters
Diameter:	101 meters
Flange length:	346 mm
Flange width:	86 mm

The function of the flange is for alignment purposes at installation. A total of 46 pieces is necessary for complete installation in the hall. They are secured by bolts.

- Inner runway

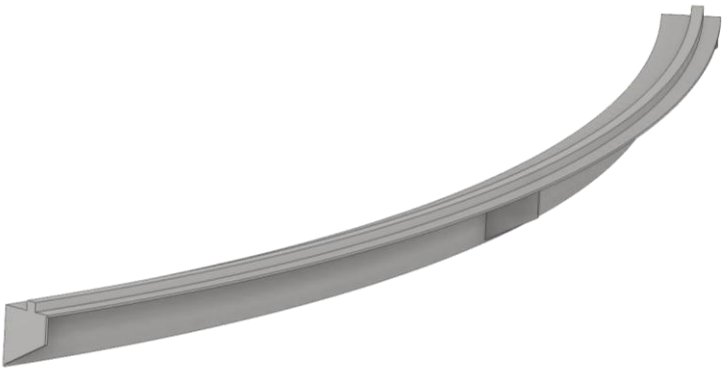

Figure 2.5.2 /Inner runway/

Rail profile:	60 mm wide, 50 mm tall
Total length:	89.6 meters
Diameter:	22.4 meters

These rails need support as the only attachment plane is parallel to the wall. A total of 8 pieces will complete the installation. Each one will connect with two M12 bolts for alignment.

Outline

Most of the components are designed with the assistance of Simulation. Parts are drawn, tested and modified repeatedly until it is considered to match requirements. No optimization has been done in this section.

Plate thicknesses follow the standard dimensions of 6mm, 10mm and 12mm. The choice of dimensions are literature and additional research based at first. Then changes are made when optimizing.

Improving model (Optimization)

This method is a powerful tool that can be used to guide the process of removing material from a model. It can help us to identify areas of a model that are not essential to its strength or functionality. However, in order to use it effectively, it is important to have a good understanding of what the results are telling us. This is because topology optimization relies heavily on our setup parameters, including the forces and constraint points used in the simulation. If these parameters are not set up correctly to accurately simulate the real case scenario, the results may not be effective or safe.

This includes accurately modeling the loads and boundary conditions that the part will experience in the real world.

Setup Parameters

Here in Figure 3.0.1 we can see the configuration list for our Optimization case.

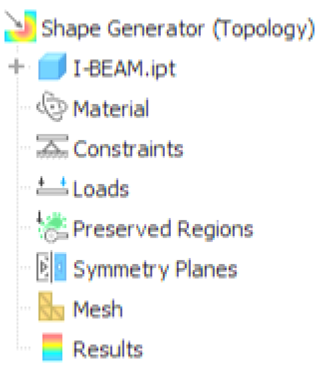

Figure 3.0.1 /Shape generator setup window/

- Material. The reason for checking our material properties before is to ensure that calculations at each node represent and simulate realistically the action and reaction forces exerted by and in the model.

- Constraints. These are restrictions to the degree of freedom of faces, edges or points. They are considered as fixed regions in space. The three constraints are Fixed, Pin and Frictionless.

- Loads. These are external forces on the model. The types are Force, Pressure, Bearing, Moment and Gravity.

- Preserved regions. This feature allows us to indicate areas where we don’t want material to be removed. This helps us to preserve threads, tolerance zones, slots and other functional features.

- Symmetry planes. Because the amount of calculations are high, it is a common practice to indicate if the model is symmetric through a plane to avoid doing the same calculations again. This saves time when generating the optimized model.

- Mesh. Meshing is the process of dividing the model's surfaces and volumes into small geometric shapes called "mesh elements". This allows for simulations to be run on the model and helps us understand how it will behave under different conditions. Here we can change “Average Element Size” to a smaller value for a more refined and smaller mesh elements, in cases necessary for more complex shapes or small curves. Default value is set to 0.05.

Sample Optimization with Additive Manufacturing

It will be shown a case where I suppose the trolley is going to be a cast.

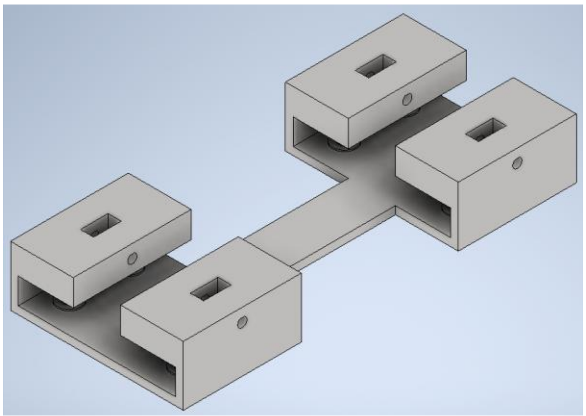
Figure 3.1.1 /Original Idea of Cast part/

My initial idea is represented in this image; the four slots on top are for the wheels and below there are 8 holes for attaching the hoist. The weight is 34.6 kg.

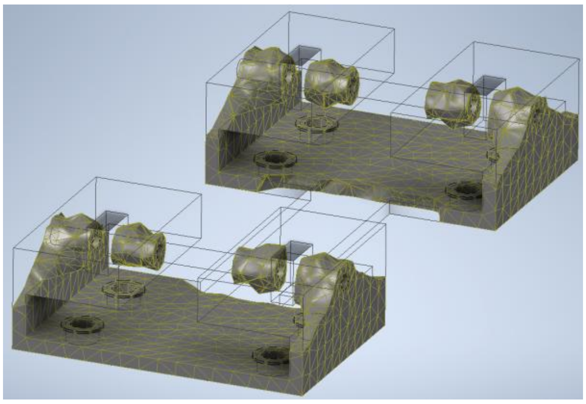
Figure 3.1.2 /First iteration shape/

In my first iteration I made the mistake of taking the whole wheel shaft slot as a Fixed geometry, that is why I got separate components along the wheel mounting area; because it assumes there is a fixed shaft. But this first iteration is not useless. It prompts that I don't need the center plate and that it can be designed as one and duplicated. It also reminds me to use symmetry planes for a faster generation.

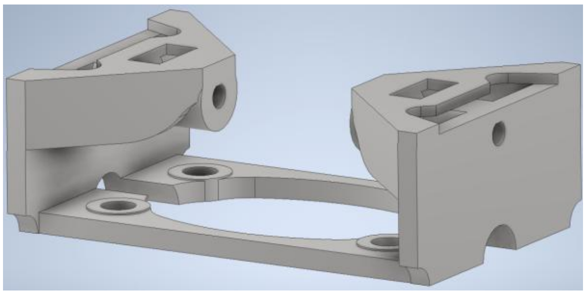
Figure 3.1.3 /Adaptation to first study/

Because this section is not an actual part of the crane, I will skip the second and third iterations and show my design after the third one.

After each adaptation on the newly generated structure, an analysis must be made to ensure our changes match our expected results. Results are shown in Figure ….

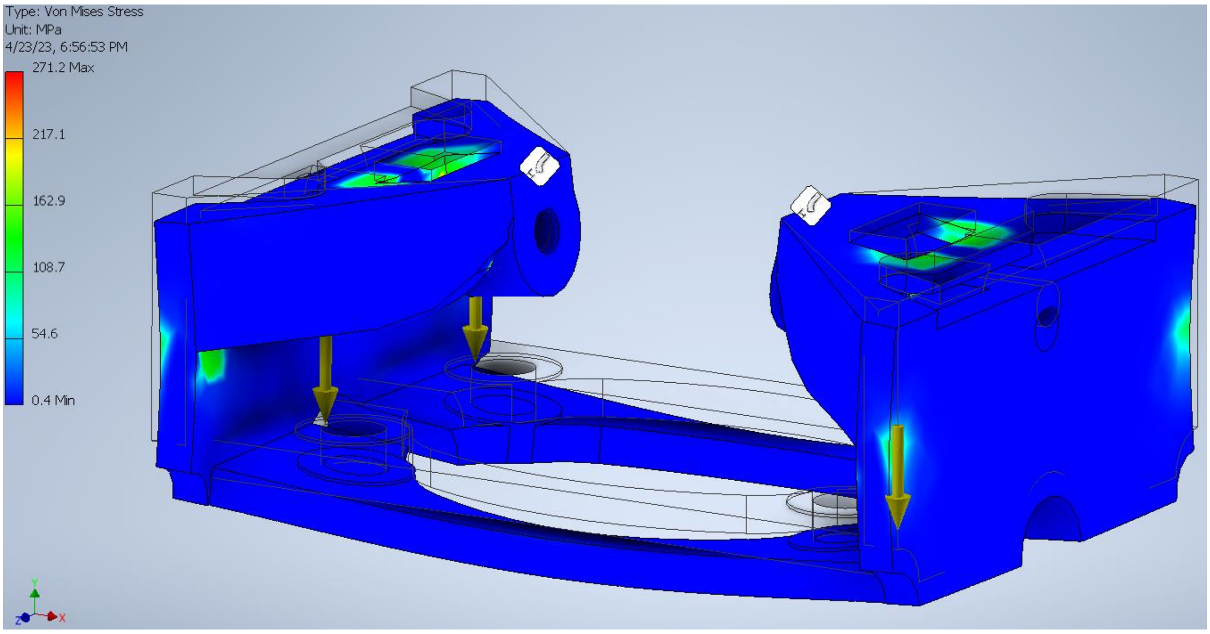

Figure 3.1.4 /Test of the first adaptation/

A load of 10 tons is applied uniformly on the four bolts and fixed constraints on the edges shown to realistically calculate deflection and forces along the whole body. The maximum stress is 271.2 MPa, which is well under our allowable stress.

Let’s skip to the 5th iteration, which is the final design.

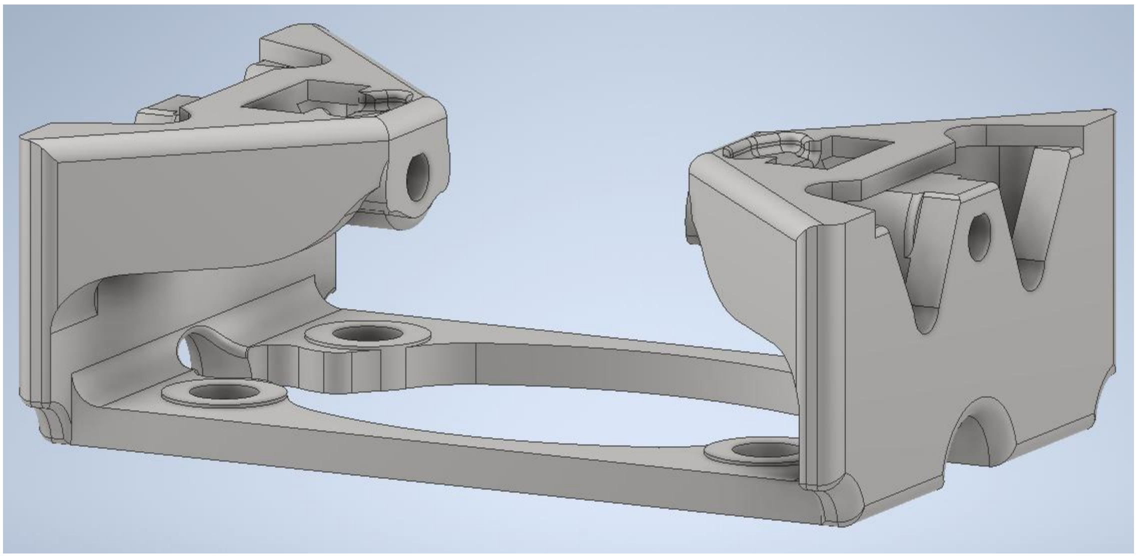

Figure 3.1.5 /Adaptation to fifth study/

Final weight = 5.6kg
Maximum deflection = 0.378 mm
Maximum stress = 269.2 MPa
Weight reduction = 65.64% (from 16.3kg to 5.6kg)
Because this is one side, the total weight is 11.2 kg

Although the model can be further improved, it can be considered as a successful optimization because of the weight reduction within its stress and strain allowances.

Results

Our final model will be discussed and the processes and steps will be explained.

Our safety factor is 2. This means that the maximum allowable stress we should aim for is 172.5 MPa. I had selected the components suitable for topology optimization and aimed to reduce its weight, thus using less material.

*Remark on loads: Because the final weight of the hoist is unknown, I will assume it to be 1 ton, and the load as 10 tons.

1. **Trolley Mount Plate**

Original weight = 8.573 Kg
The first iteration suggested the changes we see on Figure 4.1.1.

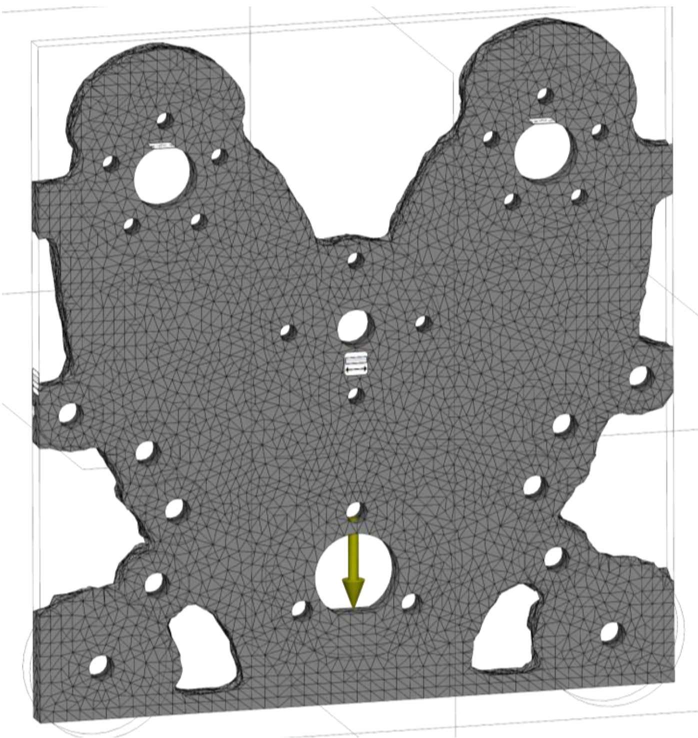

Figure 4.1.1 /Shape generated for the mount plate/

After adaptation, our maximum stress is about 60.21 MPa (see Figure 4.1.2), with a weight of 6.534 Kg.

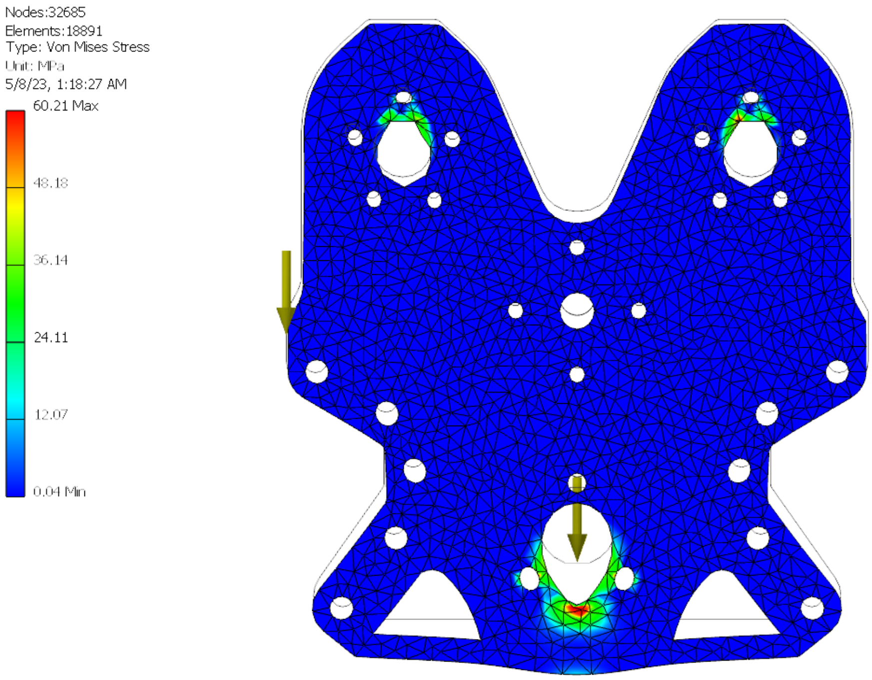

Figure 4.1.2 /Test on adapted shape/

We can still manipulate the thickness of our plate. Thinning from 12 mm to 6 mm gives a maximum stress of 151.7 MPa, bringing down the weight to just 3.267 Kg while still within our allowable stress. (See figure 4.1.3)

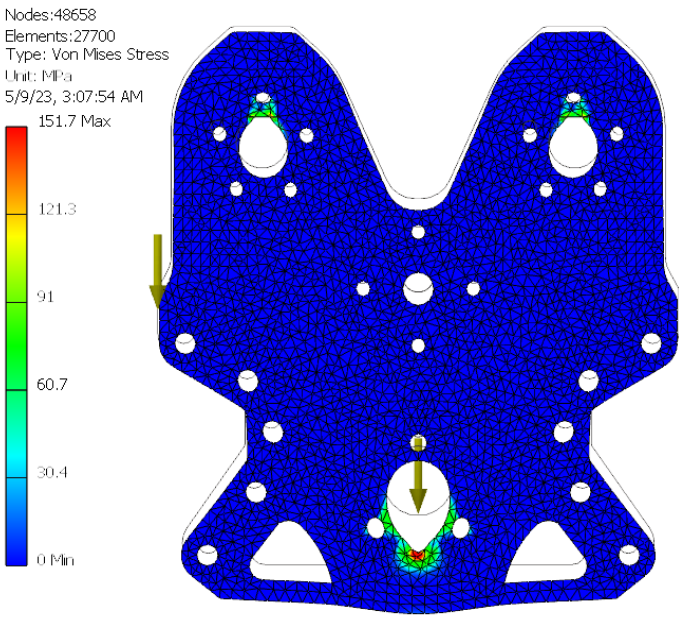

Figure 4.1.3 /Test after removing thickness/

Further reducing thickness brings our safety factor to less than 1.7; I will accept changes until the second iteration with plate thickness of 6 mm.

Total weight reduction = 5.306 Kg (62%)

Maximum stress = 151.7 MPa

Maximum strain = 0.04164 mm

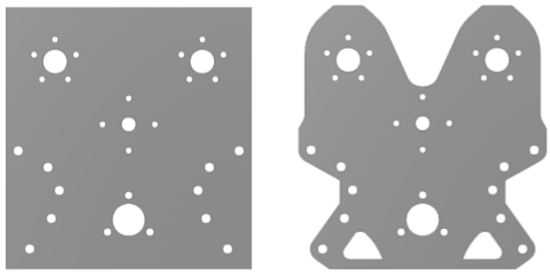

Figure 4.1.4 /Optimization 1/

2. Reinforcing Bracket

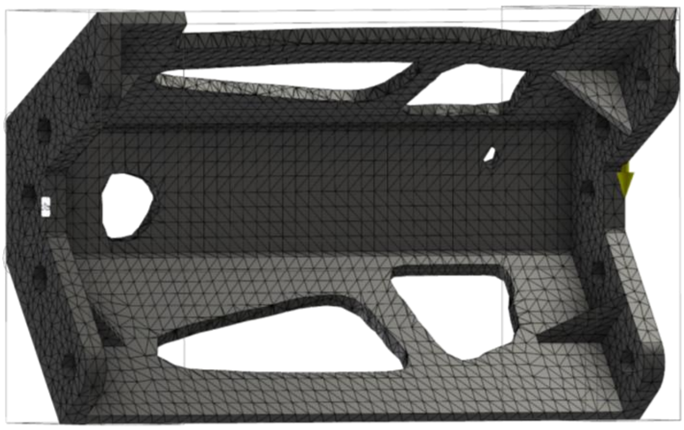

Figure 4.2.1 /Shape generated for reinforcement bracket/

This bracket experiences forces in many directions and it's not anchored to any main bodies. I will use a method where I constrain one side, and apply the forces on the other. Then I can mirror the changes from the tested side to the untested one.

A first iteration result has 52.3 MPa of maximum stress. (Figure 4.2.2)

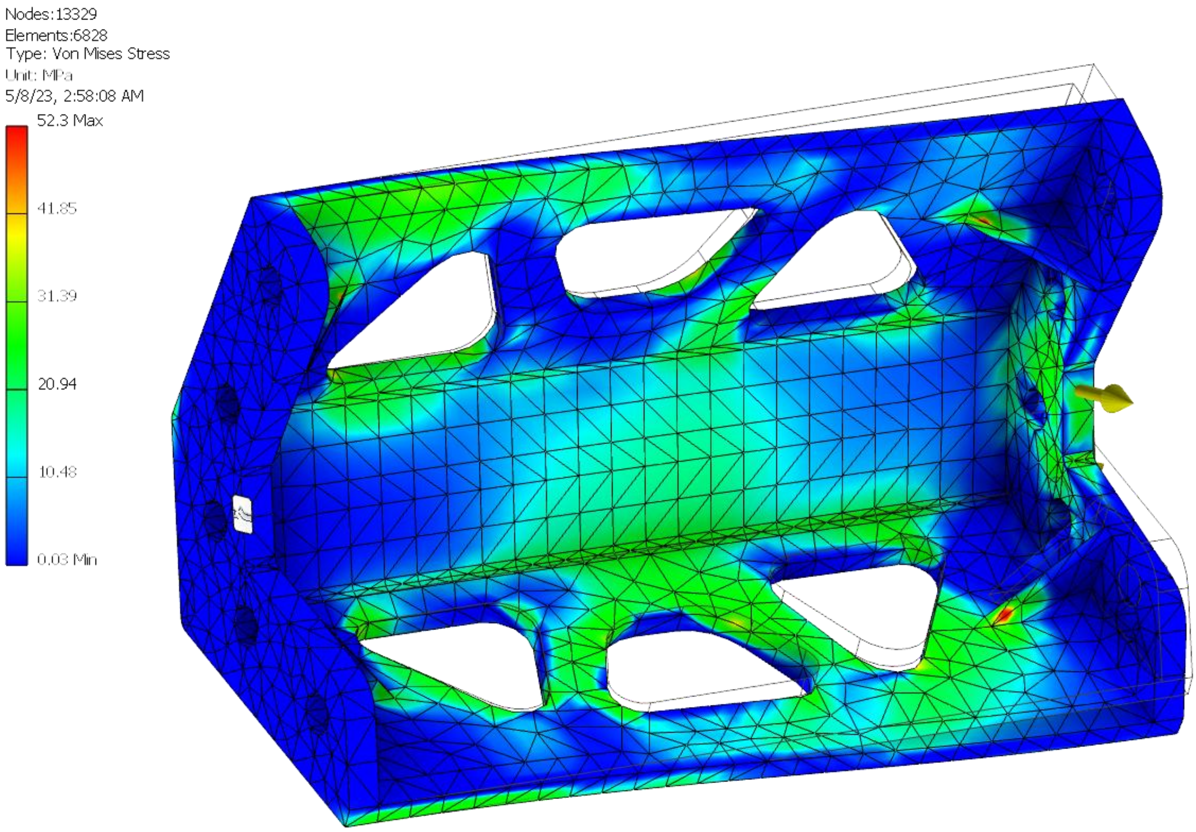

Figure 4.2.2 /Simulation of the new bracket/

From 3.949 Kg to 3.403 Kg.
Due to the forces acting on the member being random and complex I will not subtract any more material.

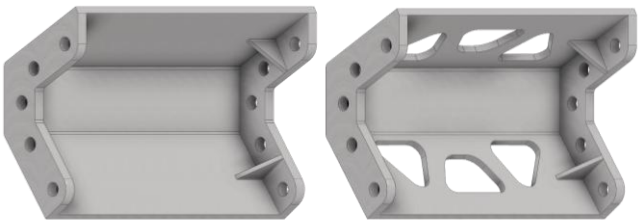

Figure 4.2.3 /Optimization 2/

Weight reduction = 0.546 Kg (14%)

3. Hoist Frame

This is a special case where the frame was preliminarily designed using multiple simulation iterations. There is material only where it is needed and the removal of any parts will cause damage to the structural integrity of the whole.

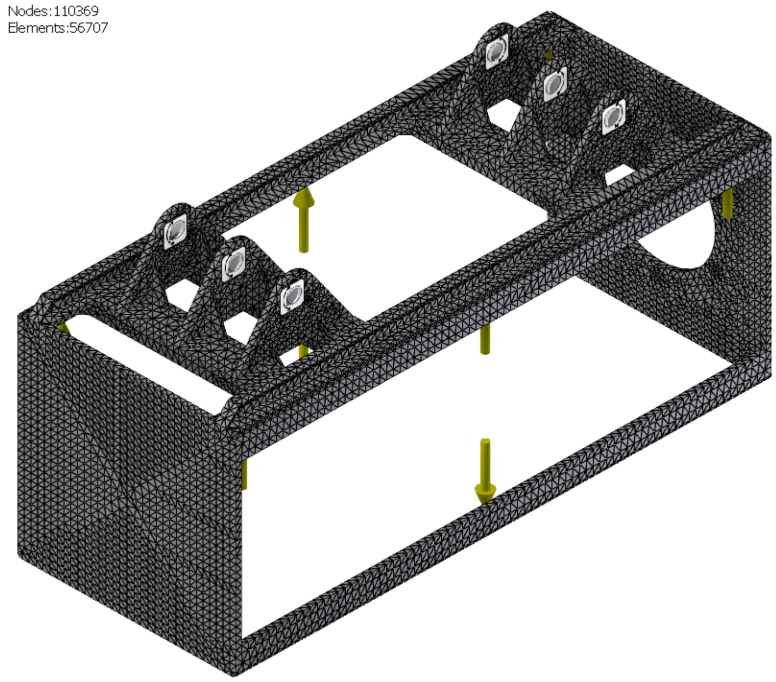

Figure 4.3.1 /Original design of the hoist frame/

There is only one part I assume it can be improved, and it is the hook plates. On each side I have included three plates, in total of 6 plates to hold the weight of the hoist assembly and the loads to lift.

Let us take a look at the simulation analysis. (Figure 4.3.2)

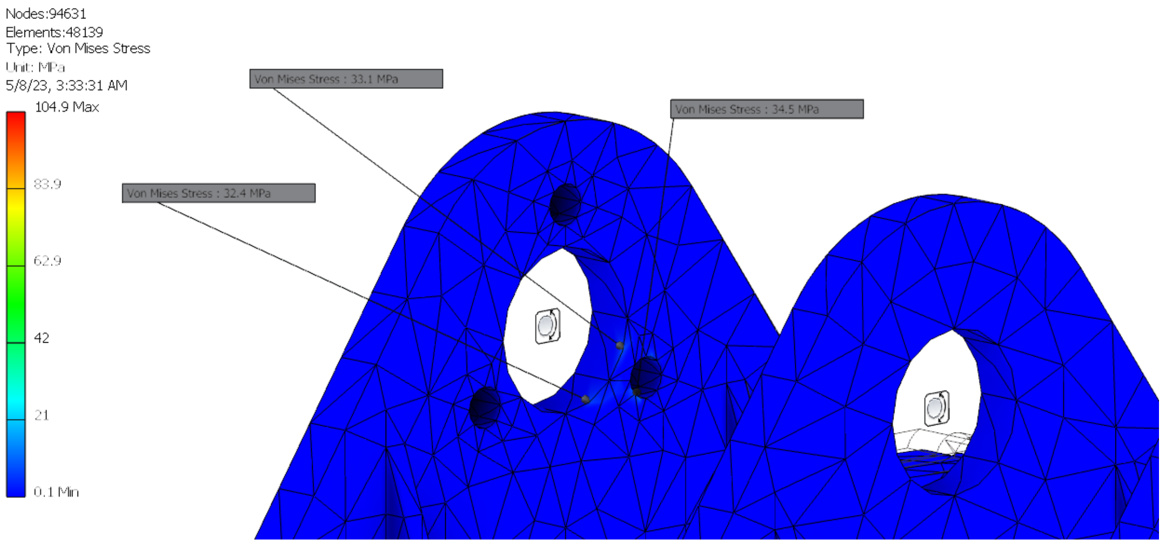

Figure 4.3.2 /Analysis on the attachments of hoist frame/

Although the maximum stresses are at 104.9 MPa, at the hook points we have a maximum of 34.5 MPa. With this information we can safely assume that not all the hook plates are necessary without the need of running a topology optimization. Let's do our changes and reevaluate them.

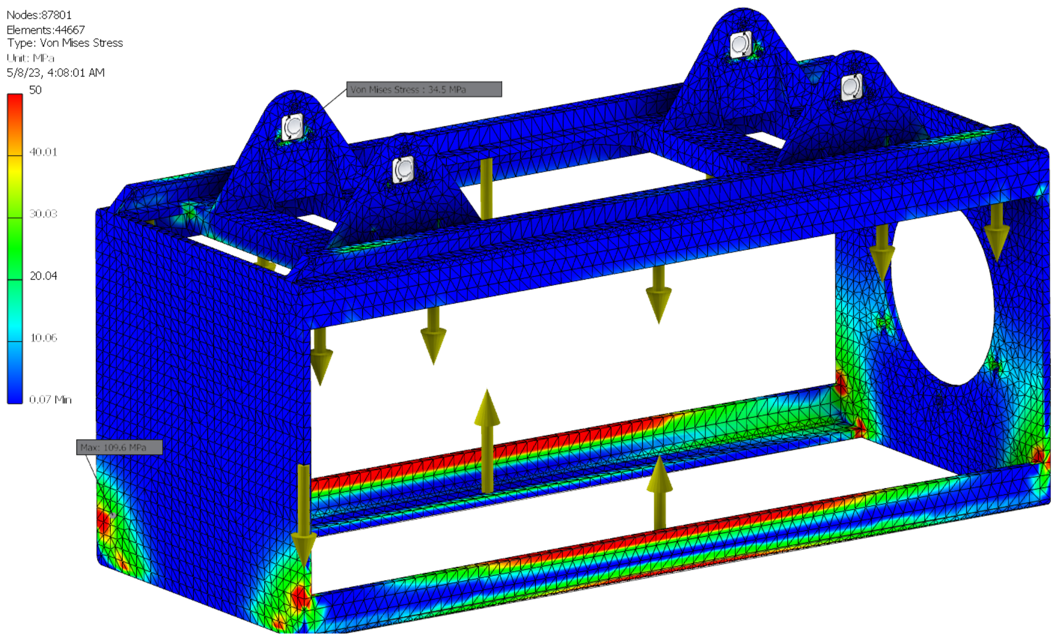

Figure 4.3.3 /Testing the second version of the hoist/

The stress at previous points remained the same. This means that the center hook plates were not carrying any load and indeed was unnecessary.
I will remove more material from underneath the hook plates where no stress is present and take care of the edges causing higher stress points.

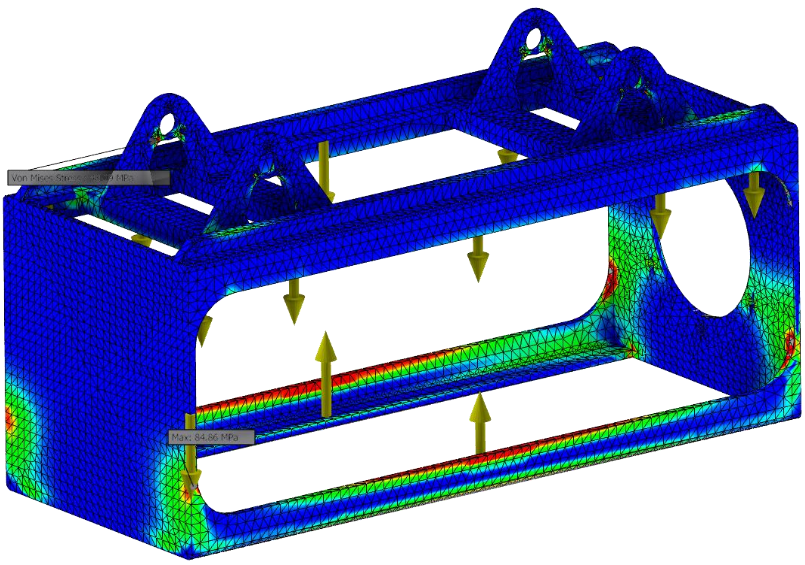

Figure 4.3.4 /Simulation of the final version of hoist/

With the last change I have reduced maximum stress from 109.6 KPa down to 84.86 KPa without adding any weight.

Total weight: 121.9 Kg

Maximum stress: 84.86 KPa

Maximum strain: 1.274 mm

4. I-Beam

Although the beam is a standard component, if the deflection generated by the loads is concerning, we will need to add additional supports.

One important load to consider is the swing angle of the load. The more the angle of swing the more horizontal forces the beam has to withstand.

To calculate this, from basic physics we know that mass is not important, rather the acceleration. For a crane this size, safe speeds vary from 5 to 10 m per minute.

For a safety margin, I will assume it travels at 15 m/min (0.25m/s).

The angle of deflection is given by $2 \times arctan\ (a/g)$, where a is acceleration and g is gravity constant. The result is 2.92 degrees.

I will do two tests:

First case is when load is at its maximum deflection.

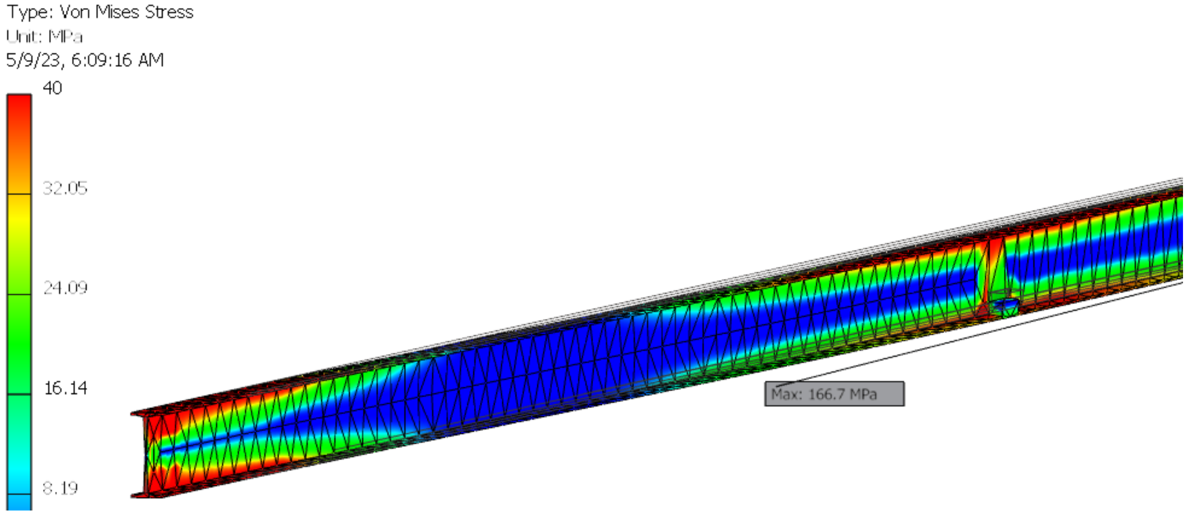

Figure 4.4.1 /Stress test of original I-beam/

The maximum stress is 166.7 MPa, within our safety margin.

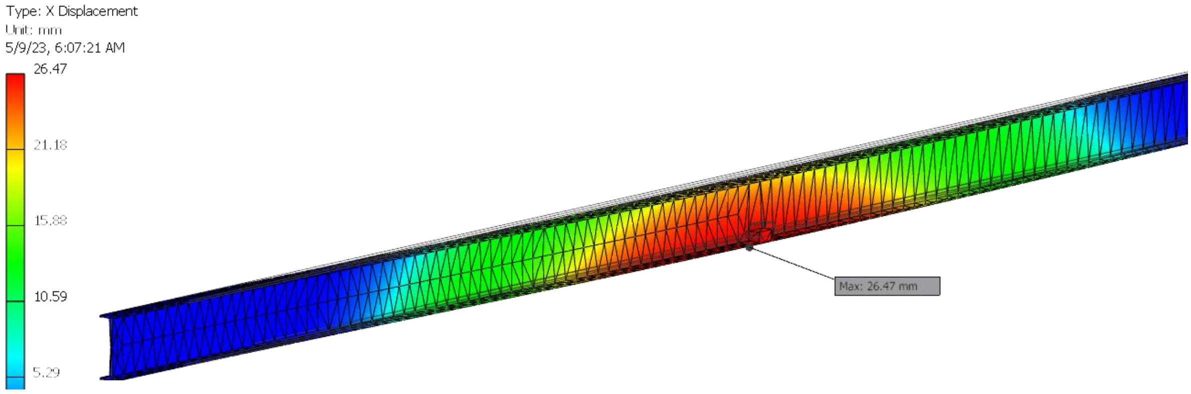

Figure 4.4.2 /Testing I-beam for lateral deflection/

The maximum lateral deflection is 26.47 mm.

The beam can safely withstand the horizontal forces.

The second test is the case when no swing occurs.

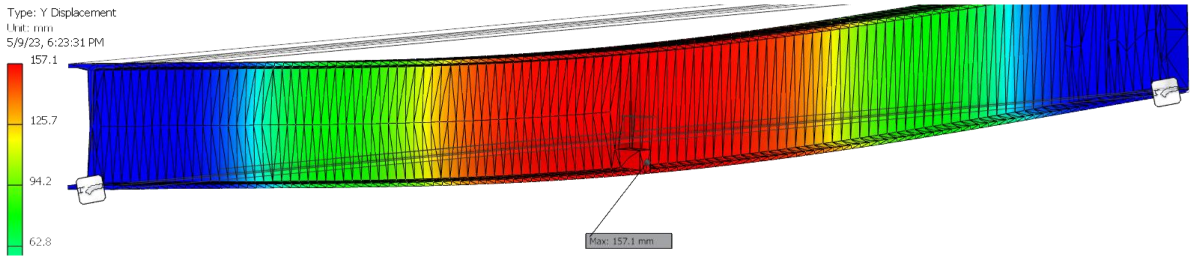

Figure 4.4.3 /Testing I-beam for vertical deflection/

The maximum vertical deflection is 157.1 mm, which is not acceptable for a load carrying system. Reinforcement of the beam is necessary. My strategy is to add a block of material additionally to the beam and run a topology optimization to locate optimal placements for addition of material.

After a few tests, this shape best suited our needs.

Figure 4.4.4 /New I-beam with support/

Analyzing the performance with the support section.

Although our weight increased from 4831.6 kg to 5870 Kg, its maximum vertical deflection reduced from 157.1mm to 117.8 mm.

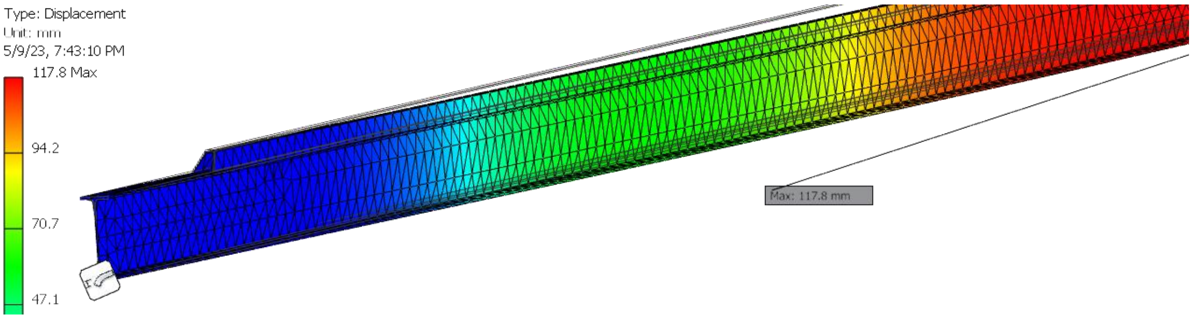

Figure 4.4.5 /Deflection test of the new I-beam/

5. Outer Carriage Frame

In this last part I used the method for optimizing parts with varying loads and force directions. I will demonstrate the steps from setting up an analysis to adapting the changes to the model.

1. Setting up the study

Apart from the parameters mentioned in the beginning of the section, it is important that the model itself does not contain errors like gaps or small shapes that can't be meshed. Gaps represent a disconnection where it shouldn't be.

Fixed constraints in the model are assumed to be immovable in space, regardless of any other parameter. This is why we generally should do separate analysis changing the constraint positions if the fixed area is expected to experience deflections.

Once the model is ready, we switch to the "Stress Analysis" Environment and "Create Study".

The first study assumes our crane is static, fully loaded and the trolley is located closest to the outer carriage.

After confirming our material selection we start setting constraints. Our frame is entirely supported by the wheels, so I will put "Pin constraints" along the shaft slots.

The loads come from the bridge and gravity (shown as yellow arrows in Figure…). If we suppose that the other end of the bridge is a pin connection, the force magnitude will be taken as the half of the beam (because the center of mass is on the center of the beam), the whole weight of the hoist and trolley assembly and the carrying load; approximately 15.4 tons.

It is not desired to remove any material from the wheel mounts and connecting section with the bridge. (Preserved regions are represented in green in Figure…)

The plane highlighted in red is a symmetry plane.

Mesh size is set to 0.030.

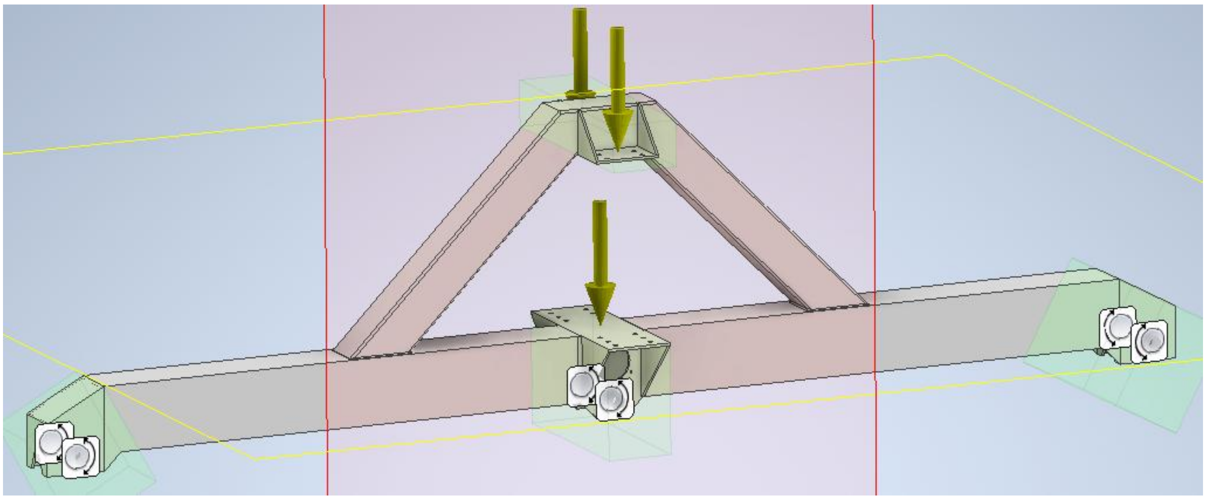

Figure 4.5.1 /Setting up for a stress analysis on carriage/

2. Processing the result

The optimization process is started and in Figure 4.5.2 is represented the result.

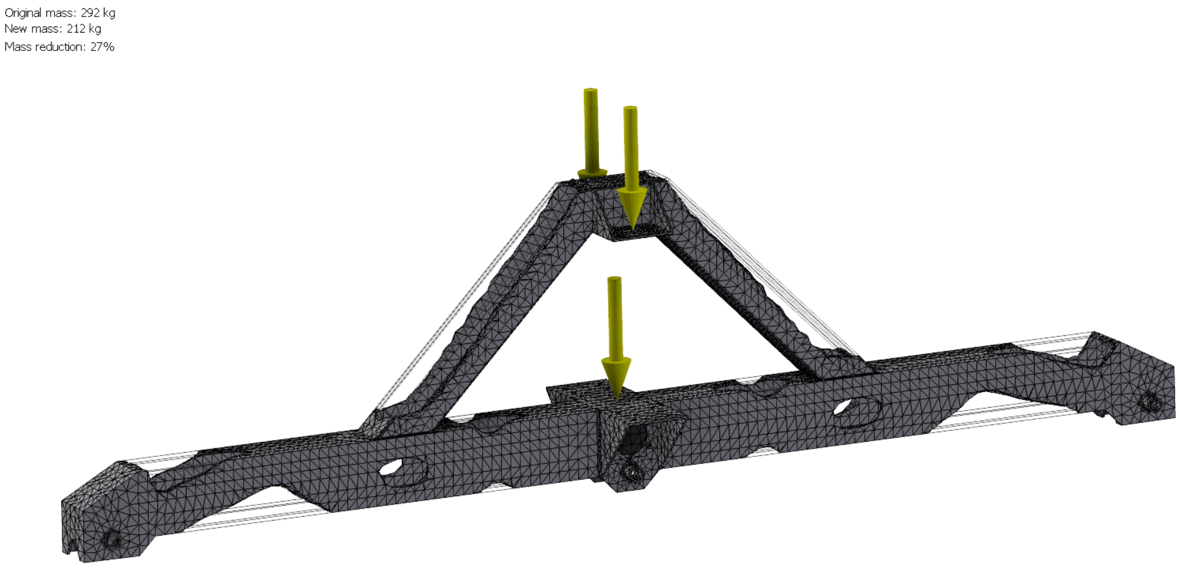

Figure 4.5.2 /Shape proposition tested for vertical loads/

My assessment is that the results are successful and accurate. The shape is promoted. Whenever a study result is promoted, a new layer over the original model is created.

3. Running the study with secondary loads

The next step is to run the study again, but this time it will be assumed that the crane is moving, other parameters remain the same.

Although the results look very similar, more material is left at the flanges of the triangular beam, which accommodates for the side loads. This shape is promoted too.

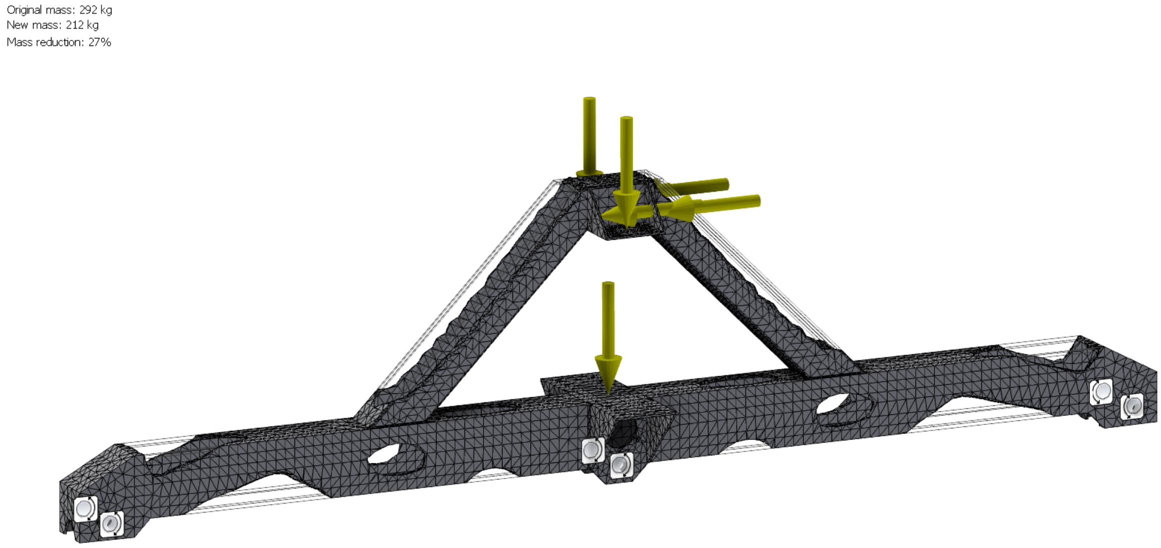

Figure 4.5.3 /Shape proposition tested for vertical and lateral loads/

4. Using promoted shapes to improve the model

Figure 4.5.4 represents the two promoted shapes layered on top of the original model, front side.

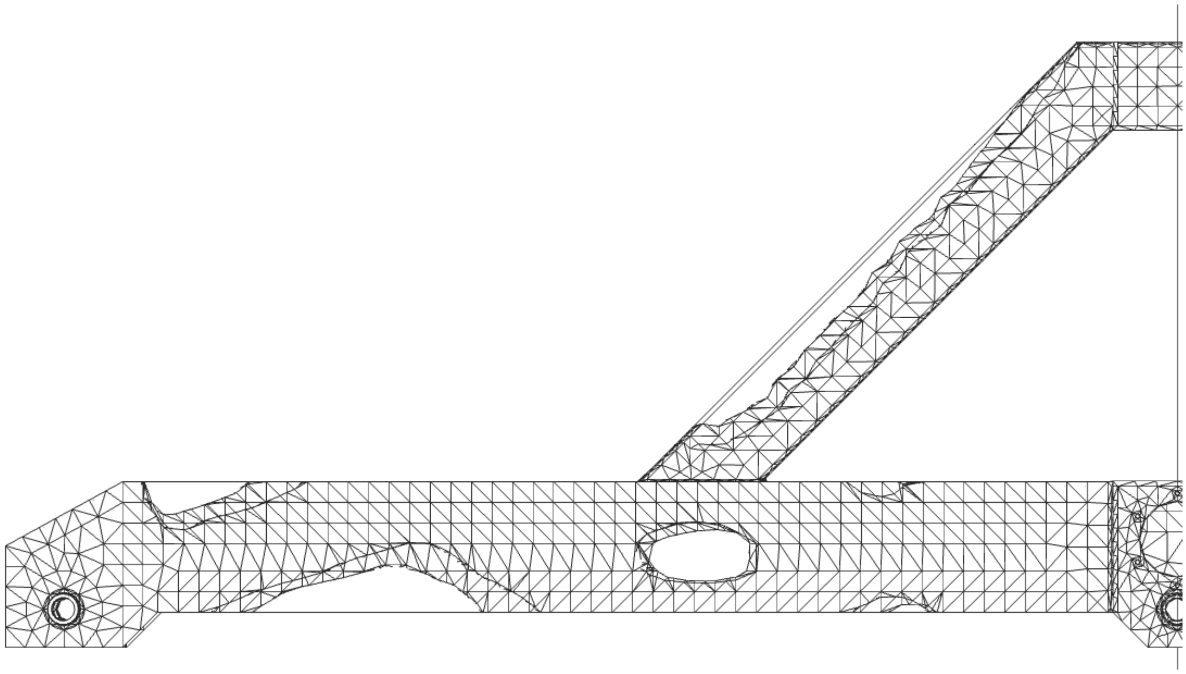

Figure 4.5.4 /Shape promotion overlapping/

These will be the guidelines for modifying the model. A new sketch is created on each view necessary and extruded to remove the material.

5. Testing and evaluation of changes.

The new model is tested and it shows a maximum stress of 34.03 MPa with a weight reduction of 20% (290.595 Kg to 230.451 Kg).

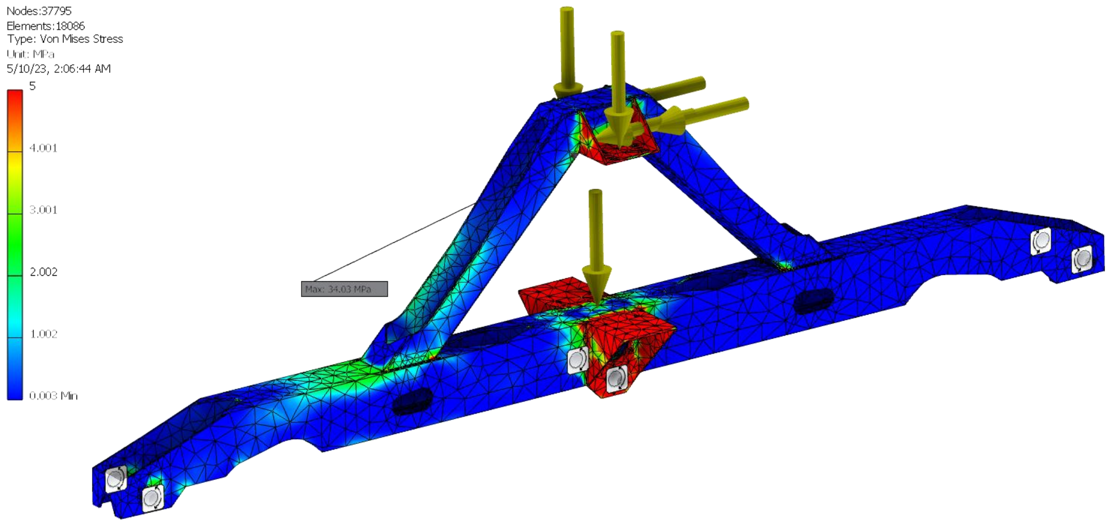

Figure 4.5.5 /Stress testing the new shape/

A Warning on Meshing Settings

One factor I have found that we must be aware of when simulating was the mesh size. The mesh should represent at its best the geometry of the model, otherwise it can have miscalculations.

Figures 4.6.1 and 4.6.2 show two case scenarios where all parameters are the same except for mesh size.

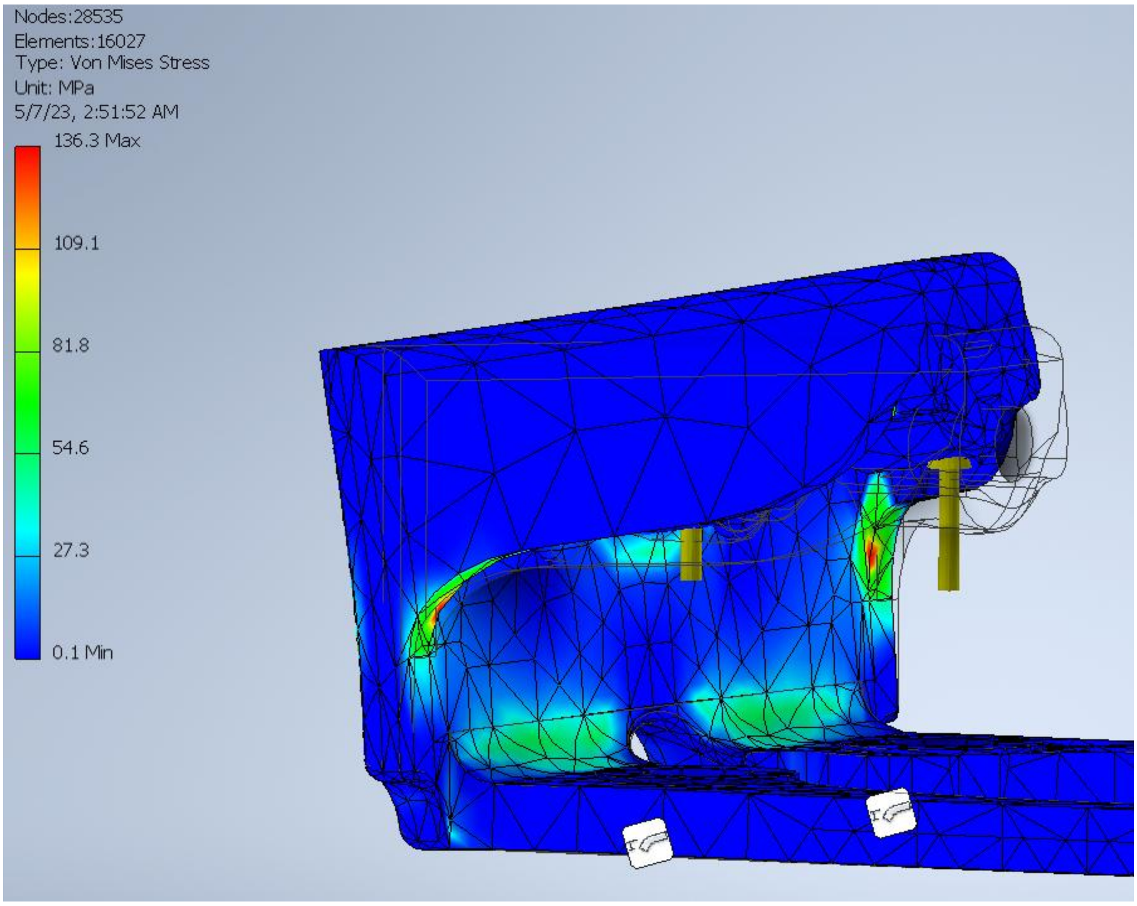

Figure 4.6.1 /Sample of bad meshing/

This Figure 4.6.1 is an example of bad meshing.
The average element size is set at 0.10
Number of Mesh Elements is 16027
Number of Nodes is 28535
The results indicate a maximum stress of 136.3 MPa. However, this is not correct because the mesh elements around corners do not accurately represent curves and other complex shapes.

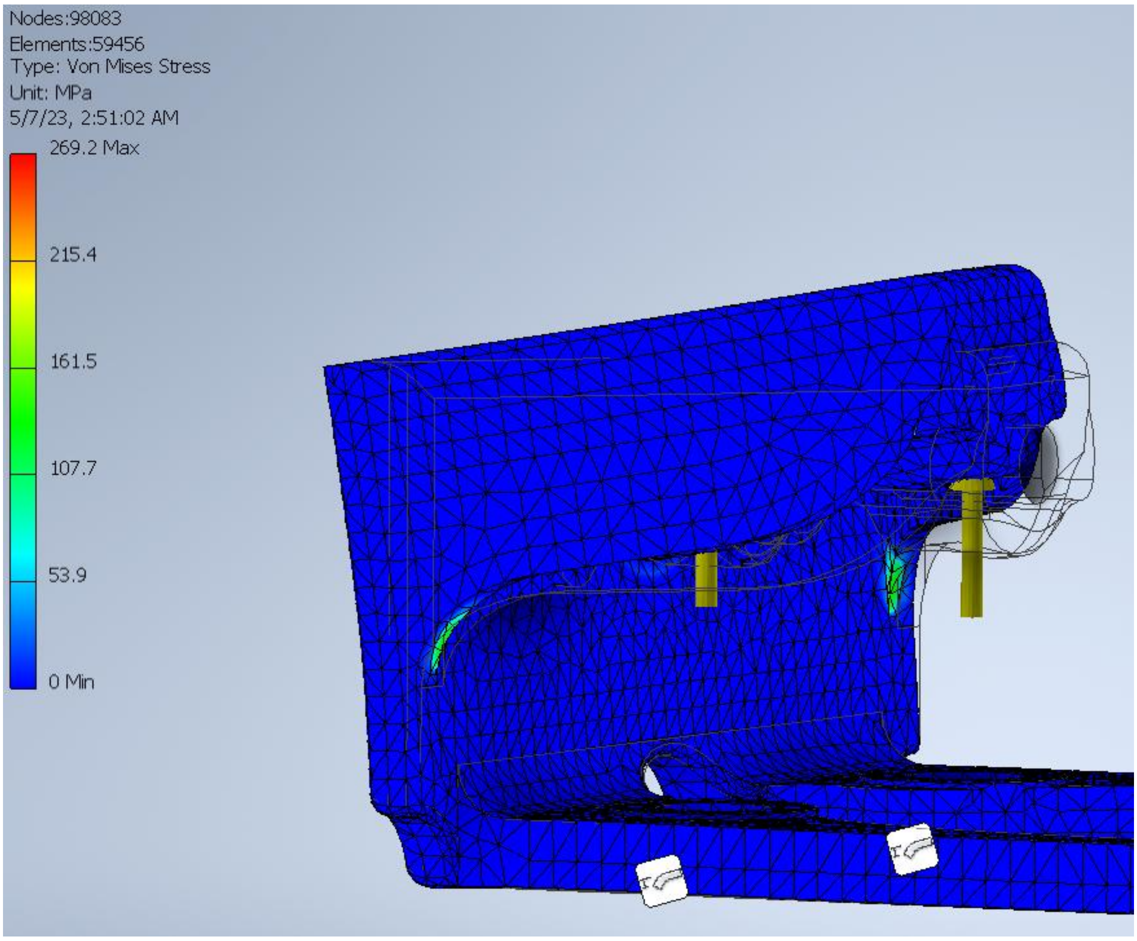

Figure 4.6.2 /Sample of acceptable meshing/

Figure 4.6.2 on the other hand, is accepted as a proper meshing.

The average element size is set at 0.020

Number of Mesh Elements is 59456

Number of Nodes is 98083

The results indicate a maximum stress of 269.2 MPa.

Conclusion

In conclusion, the design of the EOT crane's structural frame was successfully completed with the use of 3D modeling software and simulation tools, resulting in a robust and optimized design. While the process of creating lightweight components without sacrificing performance can be challenging, the application of these tools proved to be instrumental in guiding and shortening the process. Furthermore, the use of simulation tools allowed for a better understanding of how reaction forces propagate along a body, providing valuable insight for future designs.

Throughout this thesis, a set of design rules was consistently followed, combining knowledge acquired during the undergraduate years, experience with product design, prototyping, and working with mechanical devices. While the scope of this work was limited to static structural analysis, further dynamic analysis is needed to fully evaluate the crane's behavior under different loading conditions.

Overall, the successful completion of this thesis provides a solid foundation for the design of EOT cranes with the use of 3D modeling software and simulation tools. The knowledge gained from this project can be applied to a range of structural design problems, improving the efficiency and accuracy of the design process. Moving forward, further research can build upon this work to optimize the crane's performance under dynamic loading conditions and explore additional design considerations such as cost-effectiveness and environmental sustainability. Ultimately, this thesis highlights the importance of applying modern design tools and principles to create innovative and functional solutions to engineering problems.

References list

[1] Revacranes. Single girder and double girder crane. [Internet]. [cited 2022 May 7]. Available from: https://www.revacranes.com/single-girder-and-double-girder-crane/#:~:text=Single%20Girder%20Cranes%20generally%20have,and%20platform%20for%20crab%20maintenance.&text=Single%20Girder%20cranes%20does%20not%20consist%20Under%20bridge%20light.

[2] B2B Metal. I-Sections (IPE), Specification. http://www.b2bmetal.eu/i-sections-ipe-specification (accessed May 11, 2023)

[3] Tayebi-Azar A, Yazdani-Asrami M, Ghasemi Rad S, et al. A hybrid approach to design an EOT crane with topology optimization and finite element analysis. International Journal of Mechanical Sciences. 2018;137:51-62.

[4] Wei Y, Zhang Y, Ma S, et al. Design optimization of an EOT crane main girder using topology optimization and strength check. Engineering Failure Analysis. 2019;96:248-259.

[5] Wang K, Wang J, Sun G, et al. Topology optimization of an EOT crane boom with multiple loading cases. Journal of Mechanical Science and Technology. 2017;31(6):2937-2945.

[6] Chen H, Huang W, Zhou Z, et al. Multi-objective topology optimization design of an EOT crane. Structural and Multidisciplinary Optimization. 2019;60(3):1075-1093.

[7] Leng J, Wu X, Zhou J, et al. Multi-objective optimization design of EOT crane main beam based on topology optimization. Journal of Mechanical Engineering. 2017;53(14):204-215.

[8]Yang L, Li X, Wu J, et al. Topology optimization of EOT crane trolley structure. Journal of Mechanical Science and Technology. 2020;34(4):1753-1762.