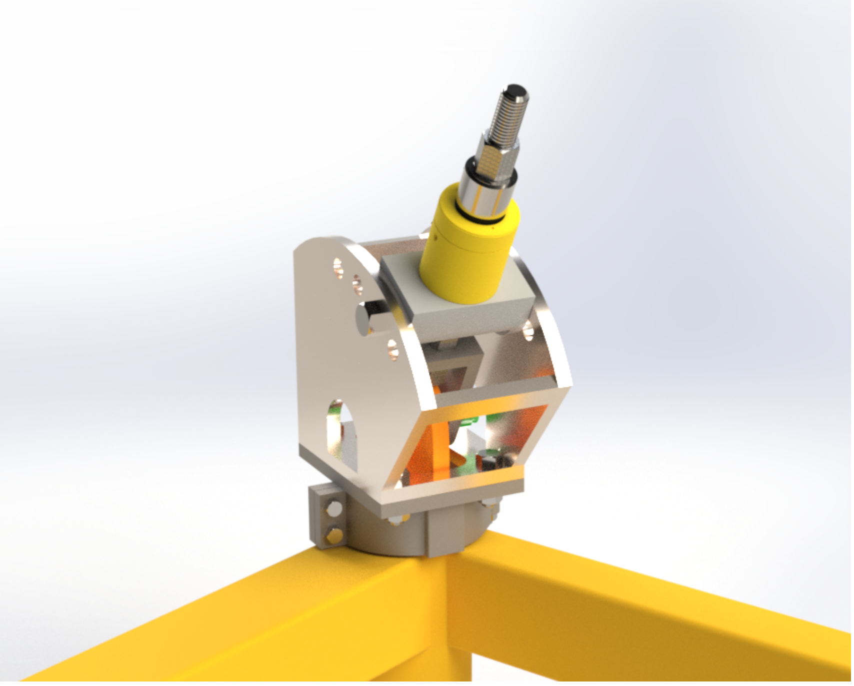
Bachelor's thesis

NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Manufacturing and Civil Engineering

Bjarte Ullebust Almklow, Sigmund Linn, Åse Helgeland Pedersen

Jacking Device for Load Testing of Lifting Points

Bachelor's thesis in MASG2900
Supervisor: Stergios Goutianos
May 2024



NTNU
Norwegian University of
Science and Technology

Bjarte Ullebust Almklow, Sigmund Linn, Åse Helgeland Pedersen

Jacking Device for Load Testing of Lifting Points

Bachelor's thesis in MASG2900
Supervisor: Stergios Goutianos
May 2024

Norwegian University of Science and Technology
Faculty of Engineering
Department of Manufacturing and Civil Engineering

NTNU
Norwegian University of
Science and Technology

Preface

The primary goal of this project was to provide OneSubsea with an alternative to full-size system integration tests for load testing of subsea structures. These full-scale system integration tests are sometimes economically or practically infeasible, especially for larger structures. Historically, the industry has relied on these tests and finite element analysis (FEA) to evaluate the structural integrity of pad eyes. However, for OneSubsea's largest structures, where full-scale tests are sometimes impractical, reliance has shifted to FEA alone. This project introduces an additional testing method to deliver reliable results and enhance safety while reducing reliance on costly, extensive full-scale tests.

We extend our heartfelt gratitude to our thesis advisor, Stergios Goutianos, whose invaluable advice and profound expertise in finite element analysis (FEA) have immensely enriched this project. His guidance has enhanced our learning experience and provided us with a robust introduction to practical mechanical engineering. We are also immensely grateful to our industry partner, OneSubsea, and especially to Ole-Petter Saxrud for allowing us to engage with a real-world and significant engineering challenge.

Upon completing this project, we have each experienced significant growth in our capabilities as budding design engineers. The opportunity to collaborate closely with OneSubsea has been very educational, allowing us to apply theoretical knowledge in practical scenarios. Our hands-on experience with SolidWorks for design, Finite Element Analysis in SolidWorks and Abaqus, and identifying and selecting suitable off-the-shelf parts has provided us with a robust introduction to real-world mechanical engineering. This project has enhanced our technical skills and deepened our understanding of the practical challenges and complexities faced by engineers in the field.

In agreement with OneSubsea, this paper is open to the public. There are no confidentiality restrictions, and the contents of this document can be freely shared and disclosed to the public and third parties.

Abstract

The objective of this project is to design and develop a system capable of load testing pad eyes on subsea structures for OneSubsea. Named the "Local PadEye Angular Load Tester" (LPALT), this system is engineered to apply loads of up to 80 metric tons on pad eyes at varying angles from vertical to 45 degrees. The design process, conducted using SolidWorks, involved conceptual and detailed phases, while finite element analysis (FEA) was performed in both SolidWorks and Abaqus to validate the structural integrity.

In designing the LPALT, particular emphasis was placed on using off-the-shelf components to minimize custom fabrication, thereby reducing costs and simplifying assembly. The LPALT's compact design allows it to be transported on a standard euro pallet, enhancing portability and operational flexibility across different sites. This feature positions the LPALT as a cost-effective and efficient onsite pad eye testing solution. Moreover, the LPALT increases confidence in the structural assessments made during design validations by providing physical testing, which complements the FEA simulations.

In conclusion, the LPALT represents an advancement in subsea testing equipment, providing OneSubsea with a robust tool for enhancing the safety and reliability of their subsea installations.

Sammendrag

Problemstillingen til dette prosjektet er å designe og utvikle et system som er i stand til å belastningsteste løfteører på undervannsstrukturer for OneSubsea. Systemet, kalt "Local PadEye Angular Load Tester" (LPALT), er konstruert for å påføre belastninger på opptil 80 metriske tonn på løfteører i varierende vinkler fra vertikalt til 45 grader. Designprosessen, utført ved bruk av SolidWorks, inkluderte konseptuelle og detaljerte faser, mens Finite Element Analyser (FEA) ble utført i både SolidWorks og Abaqus for å validere den strukturelle integriteten.

Ved utforming av LPALT ble det lagt spesiell vekt på å bruke hyllevarer for å minimere spesialfabrikasjon, og dermed redusere kostnader og forenkle montering. LPALT sitt kompakte design gjør det mulig å transportere det på en standard europall, noe som øker portabiliteten og operasjonell fleksibilitet på forskjellige steder. Denne funksjonen posisjonerer LPALT som en kostnadseffektiv og effektiv løsning for testing av løfteører og sveiser på stedet. Videre øker LPALT tilliten til de strukturelle vurderingene som gjøres under designvalideringer ved å gi en fysisk test som komplementerer til FEA-simuleringerene.

Avslutningsvis representerer LPALT en fremgang innen undervannstestutstyr, og gir OneSubsea et robust verktøy for å forbedre sikkerheten og påliteligheten til deres undervannsinstallasjoner.

Acronyms

- **ANI** - Angle Interface
- **BOM** - Bill of materials
- **CAD** - Computer Assisted Design
- **CH** - Cylinder Holder
- **EF** - Enhancement Factor
- **FEA** - Finite Element Analysis
- **FEM** - Finite Element Method
- **FMECA** - Failure Mode, Effects, and Criticality Analysis
- **MGW** - Maximum Gross Weight
- **LCA** - Life Cycle Assessment
- **LPALT** - Local Padeye Angular Load Tester
- **PEP** - Pad Eye Puller
- **SDG** - Sustainable Development Goals

Symbols

Symbol	**Description**
α	Angle in degrees
σ_f	Yield stress
σ_b	Bearing stress
σ_t	Allowable stress
σ_{tear}	Tear out stress
τ_f	Shear stress
τ_{tear}	Shear tear out stress
τ_t	Shear stress at yield
F	Force
F_v	Vertical load
F_p	Load per pad eye
F_0	Preload
D	Diameter
D_h	Pad Eye Puller bolt hole diameter
r	Radius
R_{pad}	Radius of the pad eye
t	Thickness of the plate
A	Cross-sectional area
A_s	Tension area of a bolt
f_s	Safety area factor for structural calculations
k	Friction coefficient
Γ	Torque needed to tighten a bolt

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

1.1 Background and motivation

The design, production, and installation of large-scale subsea structures is a complex process that involves numerous specialized vessels, equipment, and engineering expertise. Subsea structures are typically installed using cranes or winches. The structure is lifted off the installation vessel before it's lowered through the splash zone. During lifting, the structure and the vessel will encounter wave forces and other hydrodynamic influences. To ensure the safety of the operators and equipment, the installation must be thoroughly prepared. This involves a detailed design report with calculations and a mandatory load test with conservative safety factors. Because of the importance of conservative calculations, performing a load test may require a lot of ballast and different lifting arrangements. Depending on the structure's weight, the test can be demanding regarding safety, crane capacity, and rigging, and therefore often expensive and time-consuming to perform.

(a) *(b)*

Figure 1.1: Private pictures of a load test. Used with permission from OneSubsea

As a result, OneSubsea expressed interest in developing a load test that uses fewer resources. Their objective was to design a new tool or jig capable of validating pad eyes without the need for a full-scale physical lifting test involving ballast.

1.2 Objective

The objective of this report is to craft a design proposal for a jig, aiming to create a viable concept that the commissioner, OneSubsea, may choose to pursue further in the future. As requested by the commissioner, the design will be developed in SolidWorks and verified with calculations and computer simulations that prove the jig to have sufficient capacity.

1.3 Limitations

While the intention is for the jig to accommodate different frame types, the primary emphasis is on testing OneSubsea's heavier constructions, like HIPPS, as illustrated in *Figure 1.2.* This limitation was agreed upon after discussions with the project commissioners. Continuing the existing load testing procedures is deemed more efficient for subsea structures described by the DNV-ST-E273 standard. The DNV-ST-E273 standard, which governs load testing practices for 2.7-3 Portable offshore units, prohibits testing a single lifting point in isolation. For heavier structures like HIPPS, the DNV-ST-N001 standard is followed. This standard has no formal requirement to conduct a physical load test with overload by an overhead lift through all lifting points. These types of structures are mainly verified only through finite element analysis (FEA). However, customers of OneSubsea often express the desire for a physical load test anyway. OneSubsea will then refer to the API-SPEC-17D standard when conducting a load test.

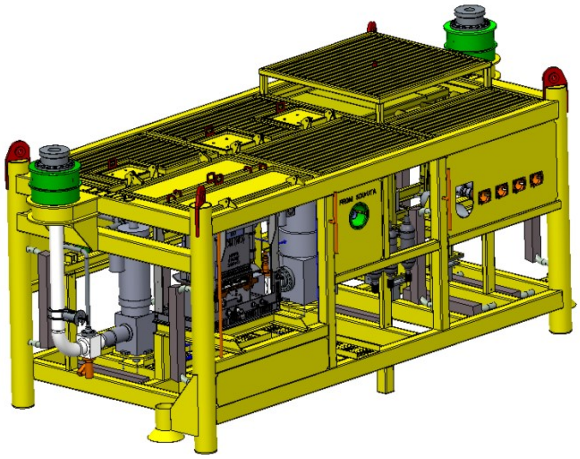


Figure 1.2: HIPPS module in SolidWorks, used with permission from OneSubsea

OneSubsea set forth specific requirements for the jig design, aiming for a versatile solution capable of conducting load tests on structures with varying sizes, shapes, and weights. The proposed solution should offer adaptability to meet these varied specifications. Their first requirement is flexibility in adjusting the angle of the force applied during the test from the vertical axis ranging from 0 to 45 degrees. The frame's side beams may range from 0 to 1000 mm in height, and the jig should accommodate both square and circular tube profiles. The jig should be able to exert a force up to 2.5 times the Maximum Gross Weight (MGW) of the subsea structure. Using the HIPPS model as a foundation for the jig design, the minimum force required is 71 tons per lifting point, considering angular loads. OneSubsea, which has numerous heavy structures, has opted to limit the project to structures that don't require a load of more than 80 tons per lifting point.

The project is limited to only developing a possible design for the jig. This means that practical aspects, such as production costs or manufacturing a physical product, will not be examined.

1.4 Challenges

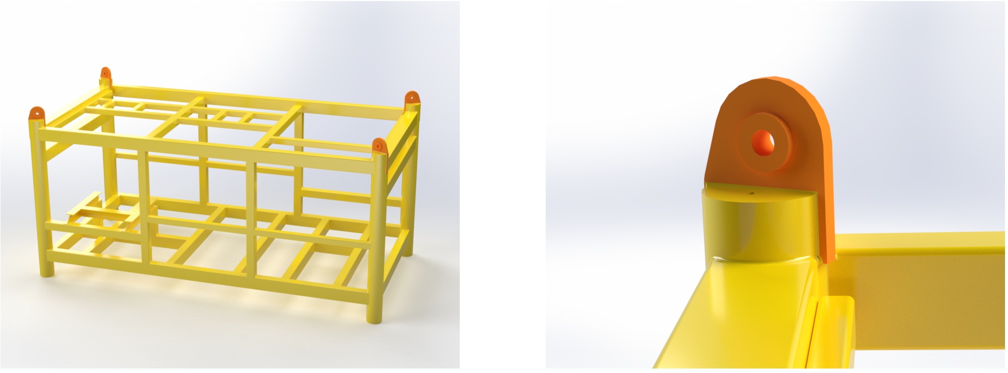
Conducting a fully loaded lift test is an important step in getting a structure approved, following most standards used for subsea structures. There are strict safety margins and requirements for the conservative nature of the calculations. The main challenge of the task is to create a design that can compete with the current solution. As mentioned earlier, it is important that the new lift test can work on several different types of subsea structures, which can vary significantly. Finding a universal solution that fits all kinds of structures may prove to be very challenging. Because of this, there will also be a risk of needing to develop multiple design variants to accommodate all the features OneSubsea requests. For the concept to be considered advantageous for OneSubsea to continue, the design must also be competitive economically, environmentally, and in terms of safety.

Chapter 2

Theory

2.1 Subsea structures

Subsea structures are installations positioned on the ocean floor, serving as the foundational support and protective framework for the equipment used in underwater oil and gas processing. This includes wells, Christmas trees, connection systems, and the network of piping and valves essential for the operation. Frames form the skeleton of subsea structures. They are robust, geometric constructions that provide the necessary support and stability for the equipment and systems housed within or attached to them. These frames are designed to withstand the pressures and harsh conditions of the underwater environment, including corrosive saltwater and varying temperatures [9].



(a) Framework *(b) Pad eye*

Figure 2.1: Components of subsea structures

A pad eye is a lifting interface typically made from standard construction steels like S355 or similar. For subsea structures, pad eyes are used as an attachment point to ensure a safe connection between the structure and lifting equipment. Pad eyes allow for the attachment of shackles or hooks for various operations, including installation and recovery of equipment. The design of a pad eye, including its size and shape, is determined by its load-bearing requirements [10].

2.2 API SPEC 17D

API SPEC 17D is a standard OneSubsea uses to validate their structures, focusing on the design, testing, and maintenance of subsea wellhead and tree equipment, including essential lifting components like pad eyes. This standard ensures equipment meets stringent safety and performance criteria across various subsea operations. It is primarily used for structures with high-stress components and critical operational functions in the subsea environment. The standard is particularly applicable to heavier structures where a ballasted load test might be avoided for financial and logistical reasons in favor of validation through FEM analysis.

Single-point testing, as outlined in API SPEC 17D, mandates an overload of 2.5 times the maximum gross weight (MGW) to evaluate the load-bearing capabilities of individual components and their weld connections. While single-point testing is valuable, it cannot replace a broader testing regime to validate the structural integrity of subsea equipment fully.

Given the challenges and costs of full-scale physical testing, API SPEC 17D advocates for a blend of theoretical analyses and targeted physical tests. This integrated approach leverages simulations, such as finite element analysis (FEA), combined with single-point tests to ensure the reliability and safety of the equipment. This method is particularly crucial when comprehensive physical testing is impractical. This strategy ensures that subsea equipment meets stringent industry standards and functions effectively under demanding conditions by validating theoretical models and gaining practical insights into component behavior.

2.3 Hydraulic cylinder

A hydraulic cylinder's function is to transform hydraulic fluid energy into linear force and movement. Its components typically include a cylindrical barrel, a piston, and a piston rod. Pressurized hydraulic fluid is introduced into the cylinder, exerting force against the piston. This force generates linear motion, extending or retracting the piston rod accordingly [13]. There are many benefits to using a hydraulic cylinder. Hydraulic cylinders are highly versatile and available in various sizes and configurations to suit diverse industrial needs. They provide a significant force output, essential for heavy-duty applications. With controlled and stable operation, these cylinders ensure diligence and safety in tasks that require precise movement. Maintenance is straightforward, minimizing downtime and operational costs. Also, hydraulic cylinders are energy-efficient, effectively converting energy into linear motion with minimal energy loss [4].

Figure 2.2: Rendered image of a hollow Plunger Hydraulic Cylinder from Enerpac [7].

Hollow plunger hydraulic cylinders can be designated as single-acting or double-acting, meaning they can exert force in two directions if desired. *Figure 2.2* show a single-acting hollow hydraulic cylinder from Enerpac capable of exerting a force up to 100 tons [8]. The central hole in hollow hydraulic cylinders allows rods, cables, or bars to be passed through the cylinder. This flexibility offers a unique opportunity to center the force around the pad eye.

2.4 Load cell

A load cell is a transducer specifically designed to convert a force into an electrical signal. This conversion allows for the precise measurement of weight and force. The principle underlying this technology is when the force applied to the load cell increases, the electrical signal output changes proportionally. Frequent calibrations and maintenance are essential for accurate results, as load cells are prone to drift over time. The frequency of re-calibration procedures depends on the extent of usage and the required level of precision, normally once per year. [27].

Figure 2.3: Load cell from Vetek [26].

Shown in *Figure 2.3* is a thru-hole load cell from Vetek, characterized by its central hole. This design is handy in applications where the load to be measured is applied through a central point or when integrating the cell into existing mechanical structures without significantly altering the design. The load cell from Vetek is constructed with a high-accuracy alloy steel measurement inner core and protected with an outer stainless steel casing. This model has a maximum capacity of 100 tons [26].

2.5 S355G10+M/N

OneSubsea reported that steel alloys are mainly used in their subsea projects, with S355 being the primary material. The name S355 signifies the material's minimum yield strength of 355 MPa at room temperature, although this value will vary depending on the thickness. The "S" stands for structural steel. The specific characteristics of S355 can vary based on the treatments it undergoes. Consequently, the choice of the S355 variant is determined by the relevant standards and intended application. OneSubsea concluded that according to this project's EN10025 and EN10225 standards, S355G10+N/G10+M meets their specific requirements. Here, "G10" stands for mechanical performance test simulating post-weld heat treatment, "N" means normalizing, and "M" refers to controlled rolling [15].

Table 2.1: Mechanical Properties of S355G10+M/+N, yield strength

		t	$\leq$	16 mm	-	355MPa
16 mm	<	t	$\leq$	25 mm	-	355MPa
25 mm	<	t	$\leq$	40 mm	-	345MPa
40 mm	<	t	$\leq$	63 mm	-	335MPa
63 mm	<	t	$\leq$	100 mm	-	325MPa
100 mm	<	t	$\leq$	150 mm	-	320MPa

2.6 Calculations

The structure's weight and an enhancement factor (EF) determine the vertical load in a load test. OneSubsea provided that the maximum gross weight (MGW) of the HIPPS frame is approximately 80 tons, and the EF used in this case would be 2.5. With this information, the minimum vertical load for the HIPPS model can be calculated as outlined in *equation 2.1.*

$$F_v = MGW \cdot EF = 80t \cdot 2.5 = 200t \approx 1961.33kN \tag{2.1}$$

The minimum vertical load per pad eye will be approximately 50 tons or 490.33 kilonewtons, as shown in *equation 2.2.*

$$F_p = \frac{F_v}{4} = \frac{1961.33kN}{4} \approx 490.33kN \tag{2.2}$$

For loads applied at an angle *equation 2.3* is used. Where α ranges from 0-45 degrees, when using this equation, the load will vary from 50 to approximately 71 tons.

$$F_p = \frac{MGW \cdot EF}{cos(\alpha)} \tag{2.3}$$

Bearing stress is the pressure exerted on materials under load; in other words, bearing stress describes the pressure between two contact surfaces, such as when a bolt presses against a surface. It is an important consideration in machine design, ensuring that contact surfaces can endure the applied forces without failing. This type of stress is essential for assessing the structural integrity in various mechanical applications. The general formula to calculate bearing stress is where F refers to the applied load and A to the bearing area [23]. An additional area factor f_{cs} is added for subsea structures. The bearing stress denoted as σ_b can be calculated using *equation 2.4.*

$$\sigma_b = \frac{F}{A} = \frac{F_p}{D_h \cdot t \cdot f_{cs}} \tag{2.4}$$

Tear-out stress is a term used to describe the stress experienced by a material when subjected to forces that might cause it to tear. This type of stress is particularly relevant in scenarios involving mechanical fasteners such as bolts, screws, or rivets, where the force exerted by the fastener may cause the surrounding material to tear around the point of attachment. The calculation of tear-out stress is crucial in designing mechanical joints to ensure that the material's strength is sufficient to withstand the forces it will encounter during operation, thus preventing structural failure [14]. The tear-out stress, denoted as τ_{tear}, is given in *equation 2.5.*

$$\tau_{tear} = \frac{F_p}{(2 \cdot R_{pad} - D_h) \cdot t} \tag{2.5}$$

Calculations of the necessary bolt diameter can be used to verify the required classification of bolts. *Equation 2.6* show the connection between allowable shear stress, area, and external forces in a bolt. *Equation 2.7* further shows how to find the necessary bolt radius.

$$\tau_t = \frac{F}{A} \rightarrow A = \frac{F}{\tau_t} \tag{2.6}$$

$$\pi \cdot r^2 = \frac{F}{\tau_t} \rightarrow r = \sqrt{\frac{(\frac{F}{\tau_t})}{\pi}} \tag{2.7}$$

Typically, torque is applied to a bolt using a wrench to generate the necessary tightening force, known as preload. Preload refers to the tension generated in a fastener when it is tightened. Its primary purpose is to prevent slippage and the separation of construction components [12]. One can use *equation 2.8* to calculate the torque denoted as Γ, needed to tighten a bolt. The necessary information required will be a friction coefficient (k), the diameter of the bolt (D), and the desired tension in kilonewtons (F_0). For calculations done in this report, the friction coefficient is set to be 0.2.

$$\Gamma = k \cdot D \cdot F_0 \tag{2.8}$$

The maximum preload allowed for bolts that should be tightened and loosened is 70% of the yield stress multiplied by the tension area.

$$F_0 = 0.7 \cdot A_s \cdot \sigma_f \tag{2.9}$$

2.7 Sustainability

In 1972, the United Nations held its first environmental conference in Stockholm[16]. The concept of sustainable development gained further attention with the publication of the "Our Common Future" report by the World Commission on Environment and Development in 1987. The report defines sustainable development as satisfying current needs without hindering future generations from meeting their needs[11]. This concept emphasizes the necessity for society to devise methods for conducting industry and commerce that consider long-term impacts on future generations' ability to sustain similar economic activities and wealth generation. It is crucial to adopt a long-term perspective and a holistic approach. Therefore, carefully considering material selection, part design, and the product's operational environment is essential in product development to ensure sustainability and responsible resource use.

In 2015, the United Nations established the seventeen Sustainable Development Goals (SDGs). These 17 goals, depicted in *Figure 2.4*, aim to eradicate poverty, end hunger, ensure well-being, and foster global peace and prosperity by 2030[25]. By addressing all three dimensions of sustainability, the SDGs provide a comprehensive framework for achieving a balanced and equitable future for all. The SDGs build upon the foundation laid by the Millennium Development Goals (MDGs), which were in effect from 2000 to 2015. Although the MDGs significantly contributed to global development, they were often criticized for addressing only the symptoms of poverty and discrimination. In contrast, the SDGs strive to tackle the root causes of these issues with a heightened emphasis on environmental and ecological sustainability[20].

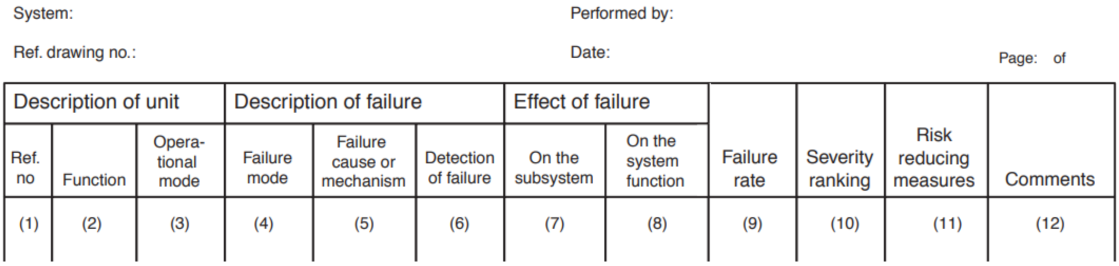
Figure 2.4: United Nations 17 sustainability goals [20]

2.8 Risk assessment

Failure Mode, Effects, and Criticality Analysis (FMECA) is a methodology employed to identify, analyze, rank, and prevent potential faults in a system before they reach the final users. This analysis offers a comprehensive system review, facilitating early detection of possible design improvements. It ranks potential failure modes using a consequence and frequency chart, scoring them based on their likelihood and the severity of likely events. The primary goals of FMECA are to detect potential failure sources, understand the causes of these failures, assess their impact on the system, and evaluate the severity of these impacts [19][3].

2.8.1 FMECA worksheet

A worksheet is used to present the results of an FMECA. The sheet should contain all the necessary elements for a detailed implementation. The worksheet used for this report is sourced from [19], illustrated in *Figure 2.5*. One modification is made to the table for this report's analysis. Under the reference number column, the component's name will be listed instead. This is because reference numbers are unknown at this point.



System:
Performed by:
Ref. drawing no.:
Date:
Page: of

Description of unit			Description of failure			Effect of failure		Failure rate	Severity ranking	Risk reducing measures	Comments
Ref. no	Function	Opera-tional mode	Failure mode	Failure cause or mechanism	Detection of failure	On the subsystem	On the system function				
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)	(11)	(12)

Figure 2.5: FMECA worksheet [19]

2.8.2 Failure rate and Severity ranking charts

The failure rate chart provides a quantitative analysis of the error rate for each component associated with specific failure modes, measuring the number of errors per unit of time. In the FMECA worksheet *Figure 2.5*, they are listed as (9) and (10). This data is crucial for understanding how often failures occur, and analyzing the frequency chart helps identify patterns and trends. Similarly, the severity ranking chart quantitatively assesses the impacts related to the identified failure modes. It outlines potential consequences for each component and evaluates their severity regarding system functionality, shedding light on how errors may affect system integrity. By reviewing these results, areas of high risk can be pinpointed, combining insights from both the frequency and consequence charts into a comprehensive analysis [19][3].

Chapter 3

Methodology

Designing and developing a structure for subsea components presented an unfamiliar challenge. Therefore, in the initial stage of the project, the group needed to research literature related to the subsea field. A considerable amount of time was also spent reviewing the various standards employed for load testing on subsea structures given by OneSubsea. In the early stages, OneSubsea intentionally withheld information about how they envisioned solving the problem. This allowed the group to think creatively and develop new ideas. As design ideas started to take shape, the group gradually received more feedback. The guidance and advice from OneSubsea have been extremely helpful, enabling the group to navigate and concentrate on relevant information within the subsea sector. This, in turn, contributed to the group gaining a clearer understanding of what would work as a good and realistic solution.

3.1 Sketching

In the project's initial phase, sketching was a key tool to promote idea development. Starting with sketches before progressing to 3D modeling allowed for efficient concept visualization. This approach ensured the team was not limited by the need for precise dimensions or detailed information early in the design process. Initially, team members worked independently to create and sketch solutions to prevent influence from other group members' ideas and promote innovation and fresh thinking. The drawing program Procreate was primarily used for this purpose. Further on in the process, these ideas were shared, allowing the group to merge various elements from the proposals into new and improved concepts.

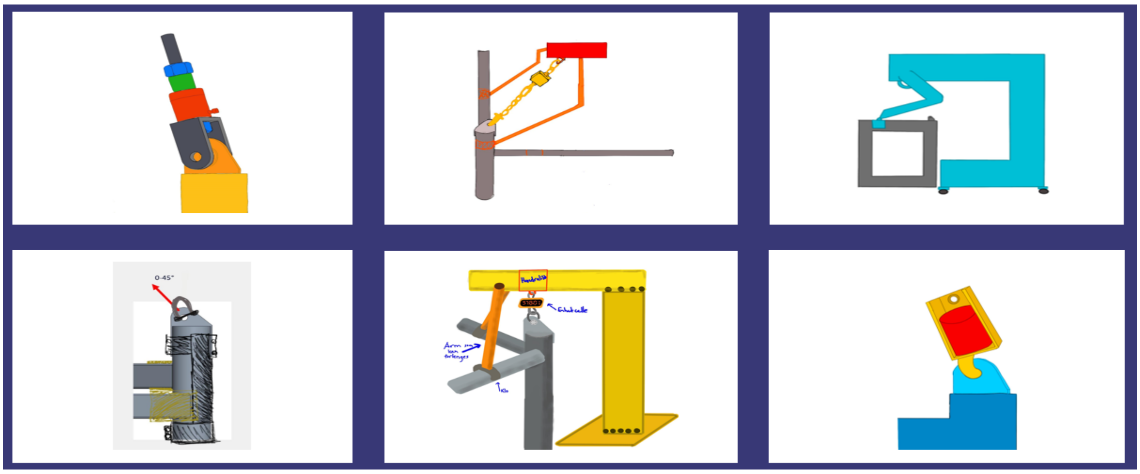


Figure 3.1: Sketches of different solutions for the load test

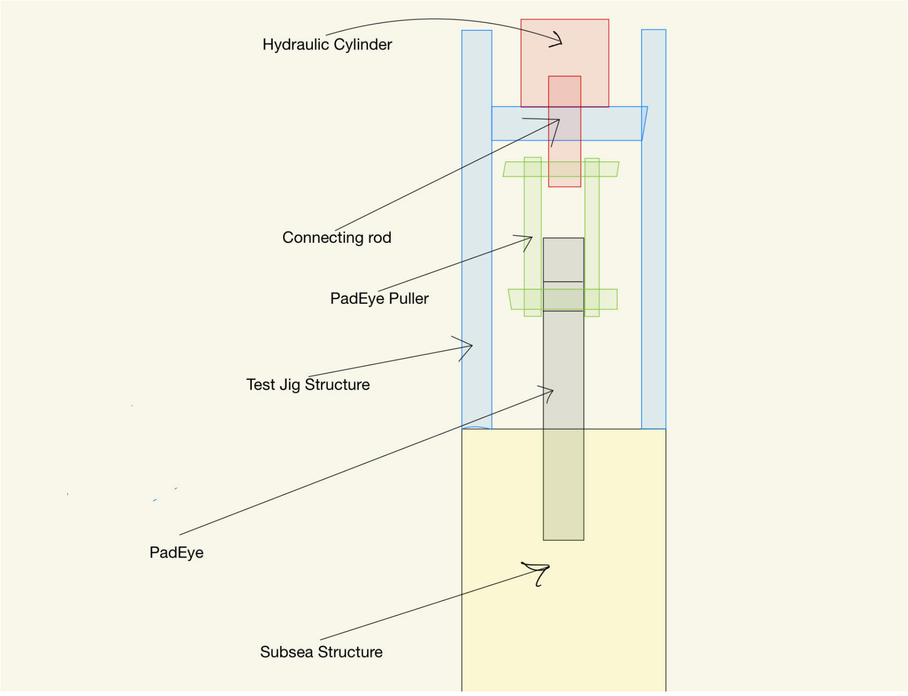


Figure 3.2: Idea sketch of a hydraulic pad-eye load tester

3.2 Computer Assisted Design in SolidWorks

After initially developing ideas in Procreate, the transition to SolidWorks, a Computer Assisted Design (CAD) tool, was implemented to create 3D models. SolidWorks enables the creation and integration of 3D models into functional assemblies, allowing for rapid and accurate modeling. The software allows Finite Element Analysis (FEA) on parts and assemblies created within the program. Making it easy to revise the design in search of the best solutions. Essential functions employed include:

- Creating accurate and quick models, both 3D and 2D sketches of different components.
- Debugging and integrating parametric functions.
- Mating components into complex assemblies and integrating moving parts.
- Analyzing and simulating loads and stresses on components.

SolidWorks enhanced the designs initiated in hand drawings, advancing the proposed solution by overcoming its constraints in multi-dimensional visualization and perspective viewing. This improvement aids in fine-tuning spacing and dimensions to ensure proper component interaction and to identify essential design adjustments. Consequently, SolidWorks is critical for crafting efficient solutions and delivering comprehensive models that align with the commissioners' specifications.

3.3 3D printed prototype

After presenting the ideas to OneSubsea, such as hand sketches and CAD models, the decision was made to focus on a locally mounted hydraulic load tester. Developing a prototype early in the process was a deliberate decision. Having all components physically present offers a distinctly different understanding of the concept than purely digital work, such as sketching and CAD models. The prototype was crucial in facilitating the exchange and comprehension of ideas for further design development among team members. It also greatly enhanced the team's ability to present the concept to the commissioner. Additionally, when the group initiated structural analysis, the prototype provided valuable insights into how the forces acted within the model, simplifying the analysis process. *Figure 3.3* shows a 3D-printed prototype of the initial concept model, which was later selected for further development.

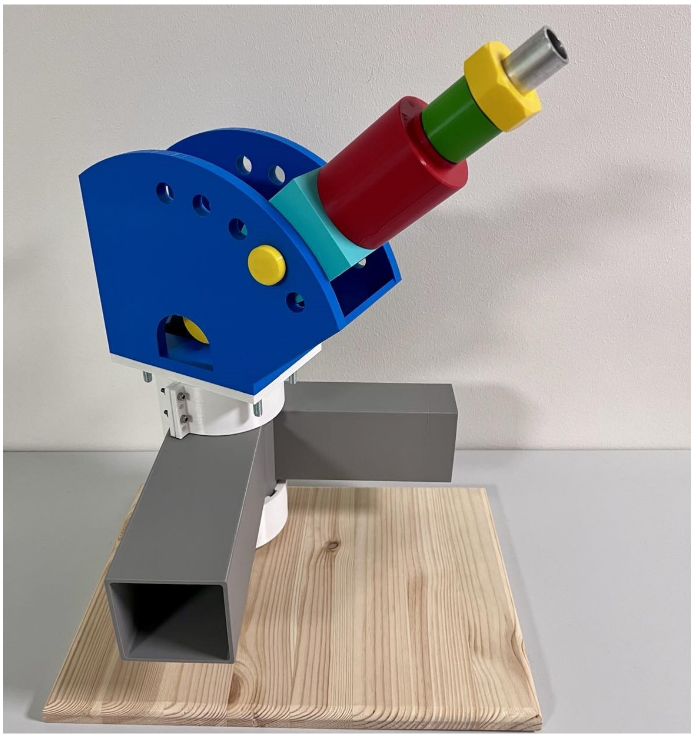


Figure 3.3: Image of 3D printed early concept of the LPALT

3.4 Finite Element Analysis

A crucial part of the task was creating a viable concept that the commissioner OneSubsea could choose to move further with. Therefore, the simulation of strength calculations became a significant part of the design process to verify the solution's integrity. Primarily, the programs SolidWorks and Abaqus were used. SolidWorks has an integrated Finite Element Analysis (FEA) tool within its software, which was extensively used when the group had arrived at a specific design but still had uncertainties about how some details should be shaped. Once the design and simulations in SolidWorks were completed, Abaqus was used to validate the results. Strength calculations are a complex field that requires extensive expertise to perform simulations of substantial value. The group attempted to compensate for the lack of knowledge by seeking assistance from a supervisor with vast experience in the Abaqus software. Still, the simulations conducted by the group serve only as an indication, and it will be necessary to perform actual, thorough strength calculations to validate the product's integrity fully.

3.4.1 Initial analysis with SolidWorks

The preliminary phase of the design study leverages SolidWorks, a decision based on its robust CAD and simulation capabilities. This section outlines the step-by-step workflow adopted to maximize SolidWorks' utility in the early stages of design.

1. **Design Conceptualization:** The process commences using SolidWorks' advanced CAD tools to construct detailed 3D models. This phase is crucial for translating conceptual ideas

into tangible designs that can be further explored and analyzed.

2. **Material Selection and Load Application:** Following model creation, materials are chosen in advance, then added into SolidWorks' comprehensive material library, and appropriate loads and boundary conditions are applied. This step is foundational for simulating real-world scenarios and assessing the structural integrity of the designs.
3. **Preliminary Structural Analysis:** Utilizing SolidWorks' simulation tools, preliminary structural analyses are conducted to evaluate the design under specified conditions. Key metrics such as stress distribution, deformation, and safety factors are calculated to identify potential weaknesses and areas for improvement.
4. **Iterative Design Refinement:** Insights gained from the initial analysis inform iterative refinements to the design. This cyclical analysis, feedback, and modification process is critical for evolving the design towards optimal structural integrity and functionality.
5. **Preparation for Advanced Analysis:** Upon achieving a refined design that meets preliminary criteria, preparations are made to transition the design to Abaqus for comprehensive analysis. This involves ensuring that the design is detailed and documented sufficiently to facilitate the subsequent in-depth analysis.

3.4.2 Advanced structural analysis with Abaqus

While this study's primary focus is on using SolidWorks for preliminary analysis, it is anticipated that a future phase of the project will involve advanced structural analysis using Abaqus. This transition aims to validate the design's performance under more complex conditions and optimize it for real-world applications. Using Abaqus represents a strategic deepening of the analytical rigor, addressing the limitations of preliminary analysis and ensuring that the final design adheres to the highest safety and performance standards. This methodology section explains the rationale behind choosing Abaqus for in-depth analysis, outlines the steps involved in this transition, and details the advanced analysis process to ensure the final design's integrity and performance.

1. **Transition Preparation:** The initial step involves preparing the SolidWorks-generated model for import into Abaqus. This includes ensuring that all geometries are accurately defined and that the model is compatible with Abaqus' requirements. Data such as material properties, load cases, and boundary conditions are thoroughly reviewed for consistency before the transition.
2. **Model Import and Setup in Abaqus:** Upon successful preparation, the model is imported into Abaqus. This stage may involve additional refinements to the mesh to suit the advanced analysis needs. Abaqus offers superior meshing capabilities that allow for a more precise simulation of complex behaviors and interactions within the model.
3. **Advanced Simulation Configurations:** Advanced analysis in Abaqus enables the application of more sophisticated simulation techniques not available in SolidWorks. This includes non-linear analysis, dynamic simulations, and exploring complex material behaviors. Configurations are carefully selected based on the specific requirements and challenges identified in the preliminary study.
4. **In-depth Analysis and Optimization:** Utilizing Abaqus' comprehensive analysis tools, in-depth simulations are conducted to scrutinize the design under a broader range of conditions and more accurately predict its performance. This phase is critical for identifying any final adjustments needed to optimize the design for safety, durability, and functionality.
5. **Final Design Validation:** The culmination of the advanced analysis phase is the validation of the design's readiness for real-world application. Results from Abaqus are thoroughly evaluated to ensure that the design meets all specified requirements and performance criteria. This validation is essential for confidently progressing to the production or implementation stage.

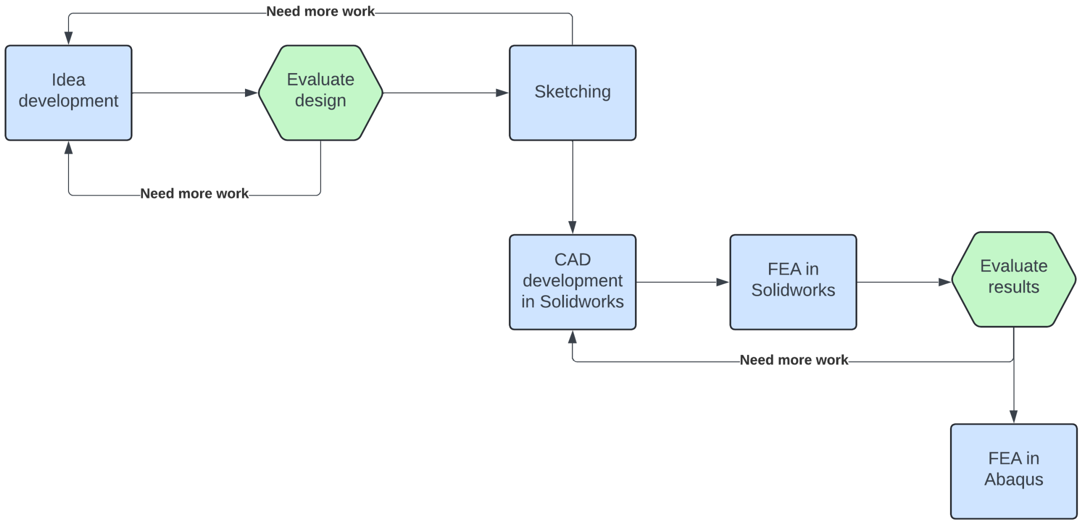


Figure 3.4: Flowchart of design development

3.4.3 FEA in Abaqus vs. SolidWorks

For several reasons, simulating components in SolidWorks is generally more straightforward and faster than in Abaqus. Firstly, SolidWorks is designed with an intuitive, user-friendly interface that allows users to set up and run simulations quickly. Its seamless integration with CAD modeling enables design modifications to be easily tested without switching between software environments. This streamlines the workflow and reduces the time required to perform simulations.

SolidWorks also offers built-in simulation tools sufficient for most standard engineering applications. These tools are accessible to users with varying levels of expertise, allowing engineers to perform basic to moderately complex simulations without needing specialized knowledge. Determining materials, loads, and constraints is straightforward, and the software provides guided steps to ensure accurate setup.

In contrast, Abaqus is a more advanced simulation tool known for its ability to handle highly complex and non-linear problems. It offers a wide range of capabilities, including sophisticated material models and advanced Finite Element Analysis (FEA) techniques. However, this complexity often necessitates specialized training and experience to utilize its features thoroughly. Setting up simulations in Abaqus can be time-consuming, requiring detailed input and a deep understanding of the underlying physics.

Additionally, Abaqus's learning curve is steeper due to its comprehensive and intricate functionalities. This makes it less accessible for general use and more suitable for specialized applications where high precision and advanced analysis are crucial. As a result, simulations in Abaqus are typically conducted by experts in the field who can leverage their deep understanding of high-stakes engineering challenges.

In summary, while SolidWorks offers a faster and easier solution for standard simulations with its user-friendly interface and integrated tools, Abaqus provides advanced capabilities for complex simulations, often requiring specialized expertise to operate effectively.

3.4.4 Revised strategy regarding Full System Integration Analysis

Initially, the plan involved conducting a comprehensive Finite Element Analysis (FEA) of an entire assembly designed using Abaqus. Access to a high-performance workstation at NTNU was anticipated, offering significantly greater computational capabilities than those available on the personal computers used during the design phase. Unfortunately, this workstation was not made available in time to perform a detailed FEA of the complete assembly. Additionally, the analysis was constrained by the limitations imposed by the student version of Abaqus, particularly in terms of mesh quality for larger assemblies. Consequently, the decision was made to switch to SolidWorks for this analysis. Despite the continued limitations due to lower computational power, the student version of SolidWorks does not impose as severe restrictions as Abaqus. Furthermore, the seamless integration between design and analytical functions in SolidWorks enhanced the efficiency of the process.

3.5 FMECA

One of the project's objectives was that the new jig could compete with the traditional method regarding risk and safety. Consequently, conducting a safety analysis like FMECA (Failure Mode, Effects, and Criticality Analysis) was found necessary. When performing an FMECA, thorough preparation is essential. Employing visual aids like mind maps and diagrams is important to understanding the objective. These tools help simplify mapping functions and faults in the product, thereby facilitating the evaluation and analysis of failure modes in the FMECA. This is also an essential step in clearly defining the scope of the study. The group used a system structure analysis to outline the jig and ensure a comprehensive grasp of the objective. This is achieved by organizing the primary system into smaller subsystems, and in some cases, it may be appropriate to break it down to the level of individual components. Even when limited to only the parts of the jig, the execution of a thorough FMECA is still a time-consuming process. Therefore, the group chose to limit the FMECA scope further to the most critical components. This is why some individual components have been examined more carefully than others [19][3].

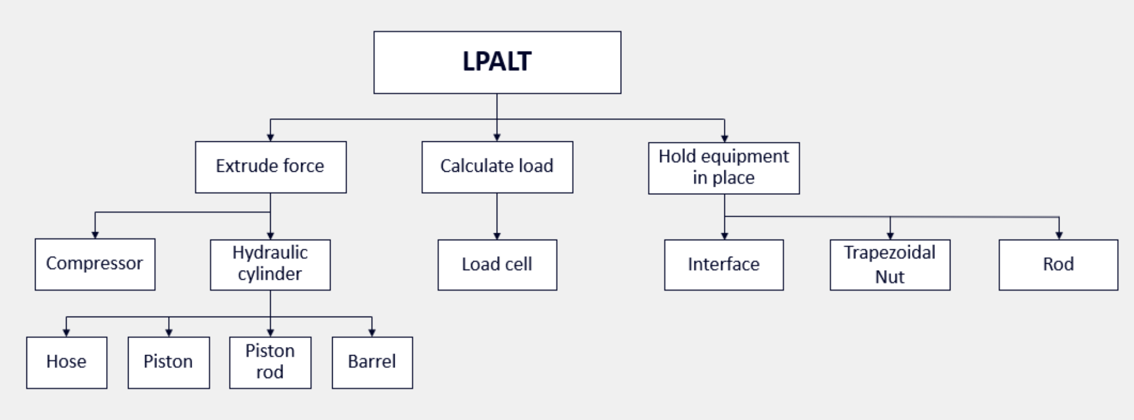


Figure 3.5: System structure analysis

A system can fail for various reasons, and different failure modes may emerge depending on the system's operation. The jig could sustain mechanical damage during transportation, installation, or misuse. Additionally, failure modes could arise during load testing or from defects in the production of subsea structure components. Defining ”normal operation” before conducting the FMECA is important to understand the specific scenarios being analyzed. For this study, ”normal operation” is defined as the conducting of a load test. The analysis will exclude ”out of the ordinary” events. Failures due to extreme weather or damages to the jig caused by defects in the subsea structure will not be considered.

3.5.1 Failure rate and severity ranking charts

During an FMECA, assessing the risk associated with potential failure modes is necessary. It is wise to use failure rate and severity ranking charts to make this assessment as unbiased as possible.

Both charts used were developed explicitly for the jig. The severity ranking chart was divided into five levels of consequences, limited to three categories. The safety category describes the risk of potential injuries to personnel performing the test. The category concerning material values describes the possible financial loss the company could face, while the environmental category details the ecological impact that could arise if a failure mode occurs. The frequency chart describes how often a failure mode might occur, ranging from once per 100 years or less to once per month or more. [19][3].

Table 3.1: Categorization of degree of probability

Class	**Probability**	**Frequency**
1	Very unlikely	Once per 100 years or less
2	Remote	Once per 10 years
3	Occasional	Once per 5 years
4	Probable	Once per year
5	Frequent	Once per month or more

Table 3.2: Categorization of degree of severity

Class	**Consequence**	**Humans**	**Material values**	**Environment**
1	Small	No injuries	Less then 200.000 kr	No environmental impact
2	Medium	Minor injuries	200.000-1 mil kr	Small degree, short recovery time
3	Big	Major injuries	1-10 mil kr	Big degree, short recovery time
4	Critical	1-2 dead	10-100 mil kr	Big degree, long recovery time
5	Catastrophic	3 or more dead	More than 100 mil kr	Big degree, long-term damage

Chapter 4

Results

This report aims to develop a design proposal for a jig capable of performing load tests on subsea structures. At the beginning of the project, the team considered numerous initial design concepts for potential refinement. The range of concepts discussed was extensive, including solutions focused on externally operated cranes and compact, locally mounted hydraulic load testing jigs, among various others.

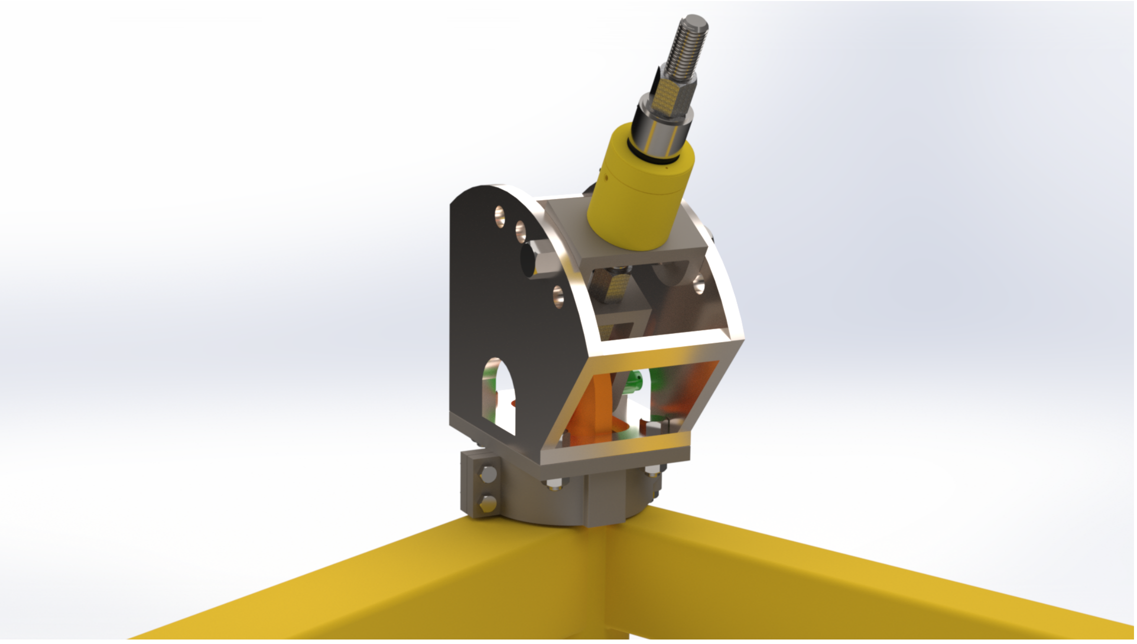


Figure 4.1: LPALT

The final design concept is: the Local Pad-eye Angular Load Tester (LPALT), a specialized device designed to improve safety and ensure compliance for pad eyes used in offshore heavy-duty lifting operations. The LPALT is a practical tool for verifying pad eyes' structural integrity and connecting welds. It is a valuable addition to safety and compliance protocols in operations requiring certified lifting points, featuring robust construction, precise load application capabilities, and user-friendly operation.

The compact design includes a hydraulic cylinder capable of exerting forces up to 100 tons at angles ranging from 0 to 45 degrees, closely simulating operational conditions to verify pad eyes' structural integrity and load capacity before their use. This design has the added benefit of yawing in the direction of the pad eye, so it will always test at the correct angles in both pitch and yaw. Constructed from S355 steel plates, the LPALT combines durability with portability. It utilizes

S355G10 steel, commonly used in OneSubsea's steel constructions, facilitating easy manufacture with accessible materials. Featuring standard components such as the hydraulic cylinder and load cell, the solution offers flexibility in procurement and simplifies maintenance. The LPALT is versatile, designed to accommodate various pad eye sizes and shapes, and tested at different angles, effectively ensuring functionality, safety, and convenience.

Summary of Key features:

- **Adaptive Interface:** Tests at variable angles from 0 to 45 degrees from the vertical, accommodating diverse operational requirements.
- **Digital Load Cell:** Equipped with a digital load cell for precise load measurement, capable of data logging for subsequent analysis and compliance checks.
- **Hydraulic Operation:** Can be operated either through a manually operated hydraulic pump or a pneumatically driven compressor, allowing for precise control over load application.
- **Sturdy Design:** Constructed to manage loads exceeding the hydraulic cylinder's maximum capacity, ensuring durability under typical operational conditions.

The LPALT is intuitive and straightforward in its operation. It is typically secured to a pad eye using an overhead crane or similar equipment. Load application is controlled through the hydraulic pump, with a digital gauge providing real-time force readings. This setup allows operators to carefully monitor and adjust the application of forces up to 100 tons, which is suitable for testing even the largest subsea structures produced by OneSubsea. With a focus on operational ease, the LPALT includes a long hose for the hydraulic pump and an extended cable or wireless connectivity for the load cell, allowing the operator to manage the equipment safely from a distance. The device is designed to be user-friendly, requiring minimal training for effective use.

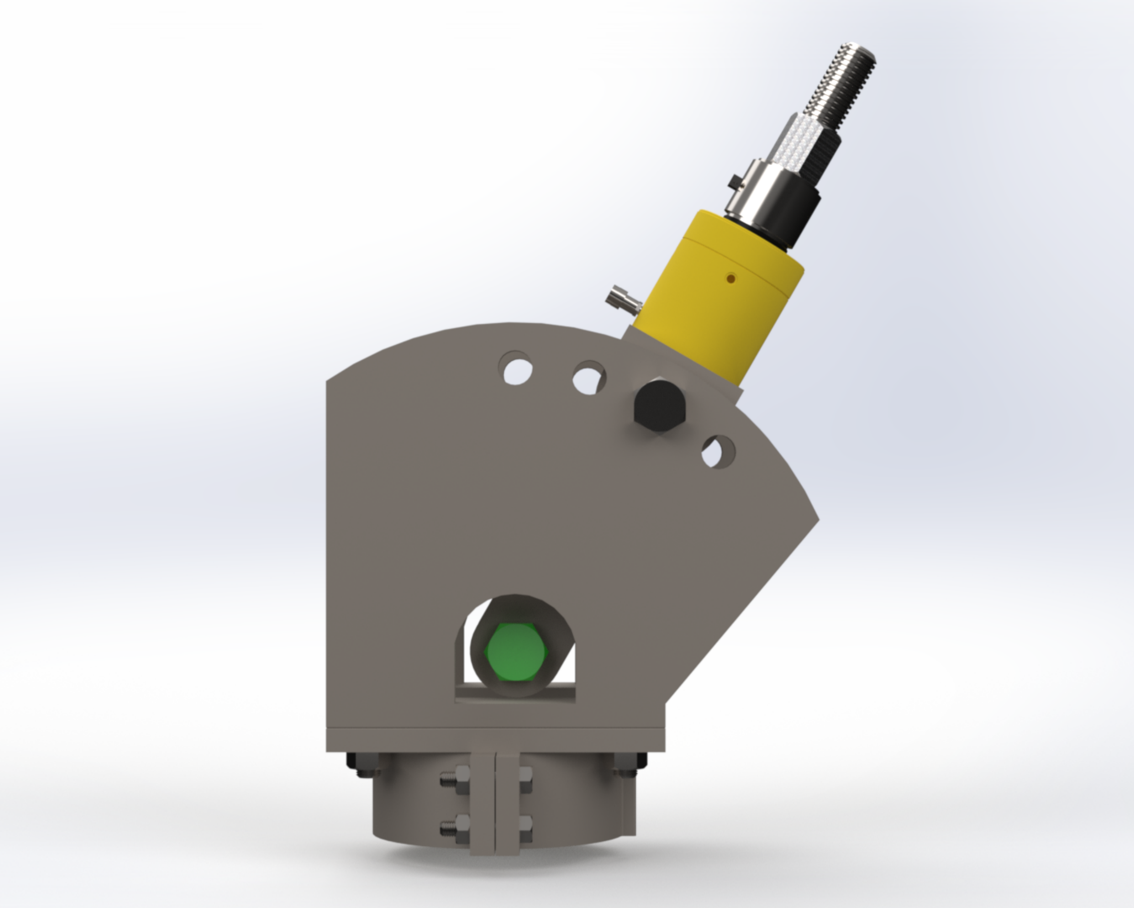


Figure 4.2: Side view of the LPALT

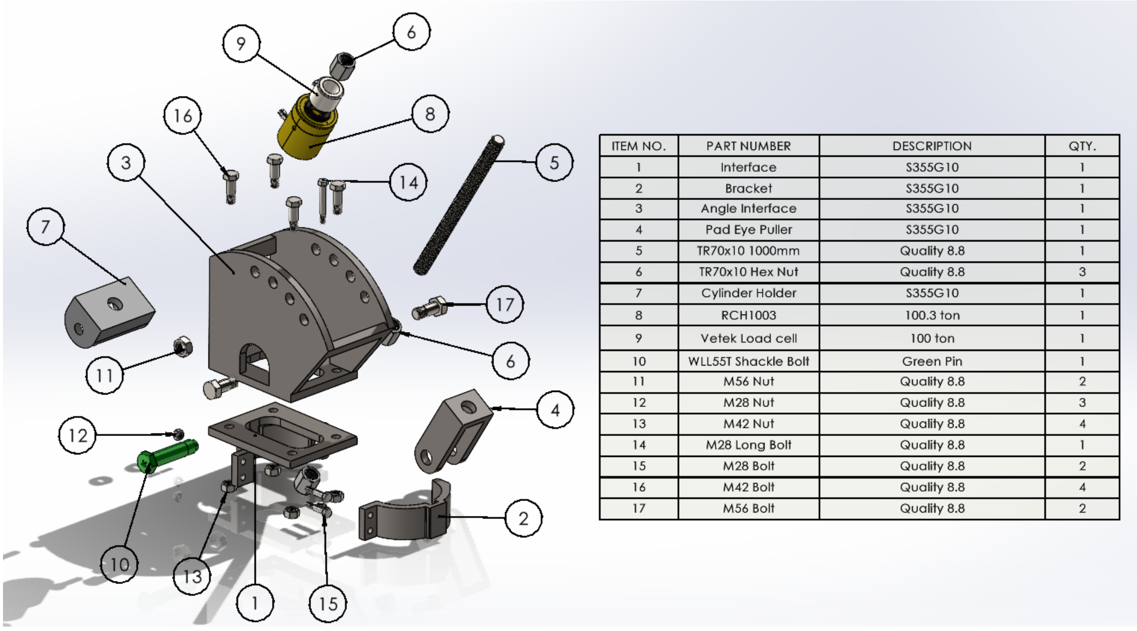
4.1 Settling on a basic design

After a collaborative decision with the commissioner at OneSubsea, our team agreed to proceed with the development of a local load-testing jig. The team engaged in thorough discussions to choose a name that best represents the project's ambitions, settling on "Local Pad-eye Angular Load Tester (LPALT)," which truly aligns with the project goals.

The design philosophy behind the LPALT emphasized cost-effectiveness while efficiently fulfilling its intended purpose. To achieve this, the team sourced as many components as possible from off-the-shelf parts, streamlining the manufacturing process and reducing costs. This strategy minimizes the need for complex machining and custom fabrication, utilizing standard, readily available parts unless necessary. The overarching goal was to develop an economically viable product that is functional and cost-effective, thereby increasing the likelihood of OneSubsea's adoption and aligning it with their need for practical and efficient solutions.

4.2 The components of the LPALT

The LPALT consists of 17 distinct parts, varying from basic off-the-shelf components to custom-engineered items. SolidWorks was utilized to generate a bill of materials (BOM), which is illustrated in *Figure 4.3*. The BOM briefly outlines the expected quality and materials of the parts, along with their quantities. The following sections will present and explain the various components of the LPALT in detail.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Interface	S355G10	1
2	Bracket	S355G10	1
3	Angle Interface	S355G10	1
4	Pad Eye Puller	S355G10	1
5	TR70x10 1000mm	Quality 8.8	1
6	TR70x10 Hex Nut	Quality 8.8	3
7	Cylinder Holder	S355G10	1
8	RCH1003	100.3 ton	1
9	Vetek Load cell	100 ton	1
10	WLL55T Shackle Bolt	Green Pin	1
11	M56 Nut	Quality 8.8	2
12	M28 Nut	Quality 8.8	3
13	M42 Nut	Quality 8.8	4
14	M28 Long Bolt	Quality 8.8	1
15	M28 Bolt	Quality 8.8	2
16	M42 Bolt	Quality 8.8	4
17	M56 Bolt	Quality 8.8	2

Figure 4.3: Bill of material

4.2.1 Off-the-shelf components

To advance with the design, the team had to make several decisions regarding off-the-shelf parts. This was to avoid unnecessary costs in designing and manufacturing components available for purchase. Using off-the-shelf parts dictates certain geometrical aspects of the design, which must be established before proceeding with any precise design iterations. Below is a list of the off-the-shelf parts selected for this project. Any subsequent design iterations must accommodate these components' specific dimensions and geometries.

Figure 4.4: Collection of selected component images [7] [26] [18]

- ENERPAC RCH1003 Hollow Plunger Hydraulic Cylinder (BOM ITEM NO.8) has a 103.1-ton Capacity, 3.00-inch stroke, Single-Acting mechanism. The RCH-Series Hollow Plunger Cylinders offer exceptional versatility for testing, maintenance, and tension applications. The hollow plunger design enables push, pull, or lifting operations. [7].
- VETEK Load cell thru-hole 100 ton (BOM ITEM NO.9) features a high-accuracy alloy steel measurement inner core encased in a protective outer stainless steel casing. The AR load cells are proper through-hole sensing elements incorporating top-quality bonded foil strain gauges. These load cells are sealed to protect against most industrial environments. The annular ring is designed for compression loading and is ideal for pull-through applications where the load is applied through the middle of the ring, typically using a bolt or cable. It is ideally suited for placement under a bolt or tie rod to measure force.[26].
- GREEN PIN G4163 55t shackle bolt (BOM ITEM NO.10) is grade 6 high tensile steel with a hot dipped galvanized finish. This specific bolt has a 6xWLL safety factor, putting it well inside the strength needed for testing purposes [18].

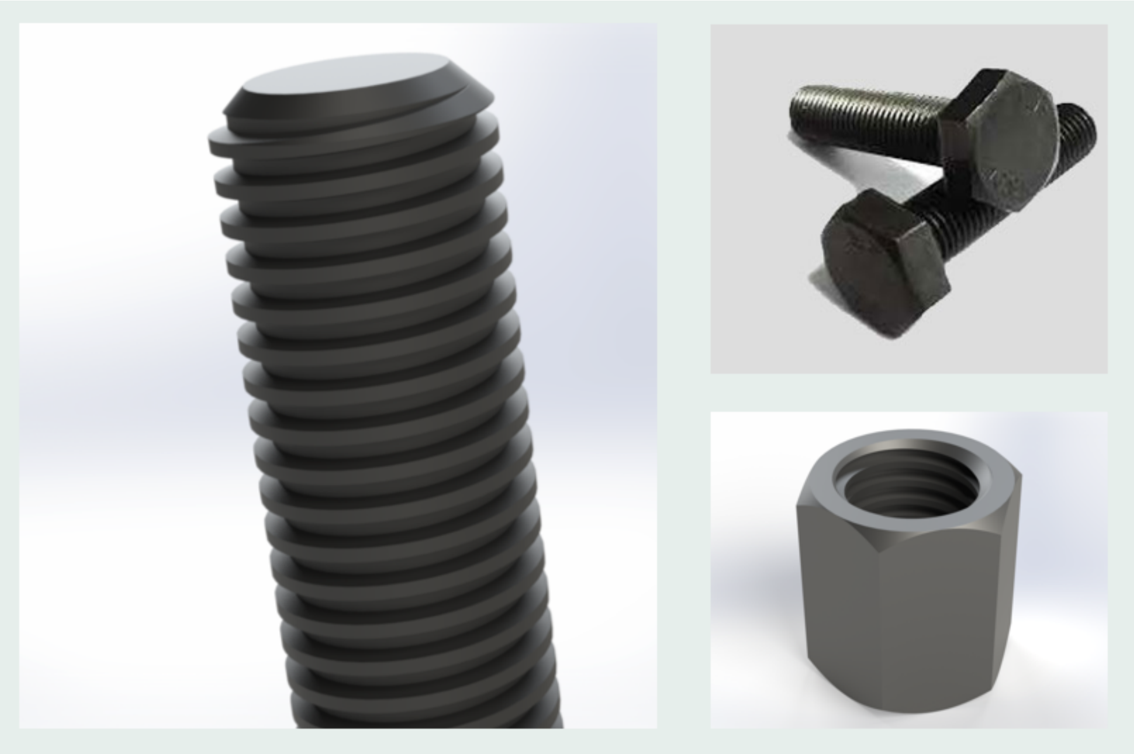


Figure 4.5: Collection of selected component images [24]

- 1000mm Tr70x10 Threaded rod (BOM ITEM NO.5) in 8.8 carbon steel with a yield strength of 640 MPa gives the rod an axial strength at a yield of 165 tons. Its yield exceeds the hydraulic cylinder's strength and is designed to take up axial loads.
- Threaded nuts in Tr70x10 in 8.8 carbon steel with a yield strength of 640 MPa (BOM ITEM NO.6) are for fixing the rod to pull the eye and load cell.
- Bolts and nuts to attach the different parts in a suitable quality, most likely 8.8 or 12.9 Black Carbon Steel, as they are robust and readily available. These parts are accessible from many different distributors, are cheap to replace, and come in various sizes. (BOM ITEM NO.11, 12, 13, 14, 15, 16, 17)

4.2.2 Interface and Bracket

The interface and Bracket (BOM ITEM NO.1 and 2) are designed to clamp onto the subsea structure, ensuring easy and safe mounting operations. OneSubsea designs and manufactures subsea structures featuring diverse geometrical configurations. Consequently, the LPALT is engineered to accommodate interchangeable interface plates, enabling its use for testing pad eyes across a wide range of subsea structures with varying geometrical characteristics. The bolt pattern on the Interface plates is sized to fit M42 bolts(16) and is matched to the bolt pattern on the bottom plate of the Angle Interface(3). The bolt pattern is made to avoid conflict with the Bracket and the vertical side walls of the Angle Interface(3). There is also spacing between the fastening faces of the Interface and the Bracket to allow a secure fit around the pipe through torquing of the M28 bolts(15).

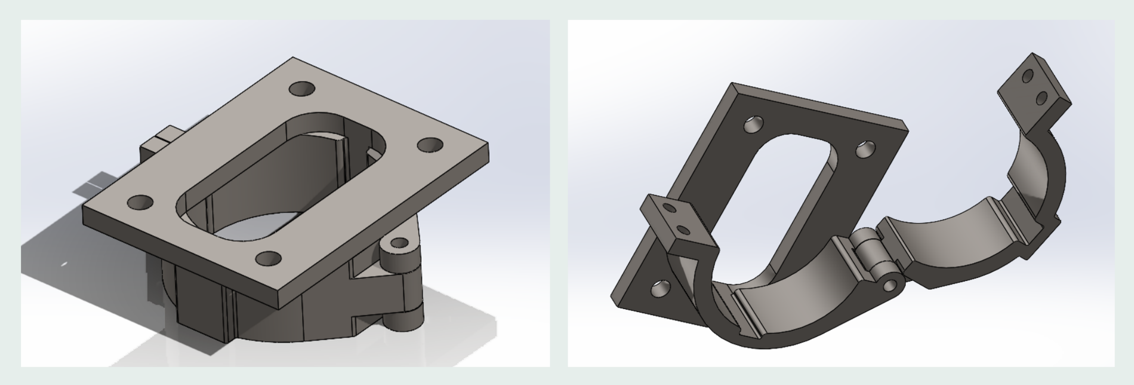


Figure 4.6: Interface designed to clamp onto the structure.

The concept involves OneSubsea designing the interface plate in parallel with the subsea structure designated for load testing, ensuring compatibility with the LPALT. In specific scenarios, it may be necessary to modify the design of the subsea structure around the pad eye to allow the interface plate to effectively connect the structure to the LPALT while facilitating the appropriate transfer of forces from the load test.*Figure 4.7* and *4.8* show two proposed solutions of how the Interface can be redesigned to accommodate different goals and geometries.

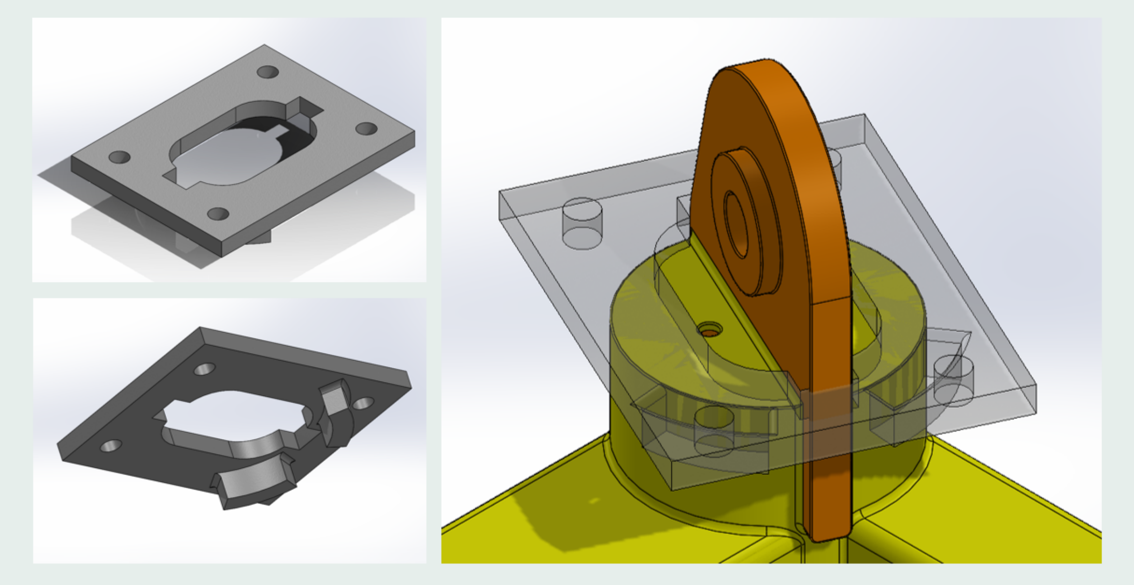


Figure 4.7: Passive interface designed with lower cost in mind, but with less stability during mounting operations

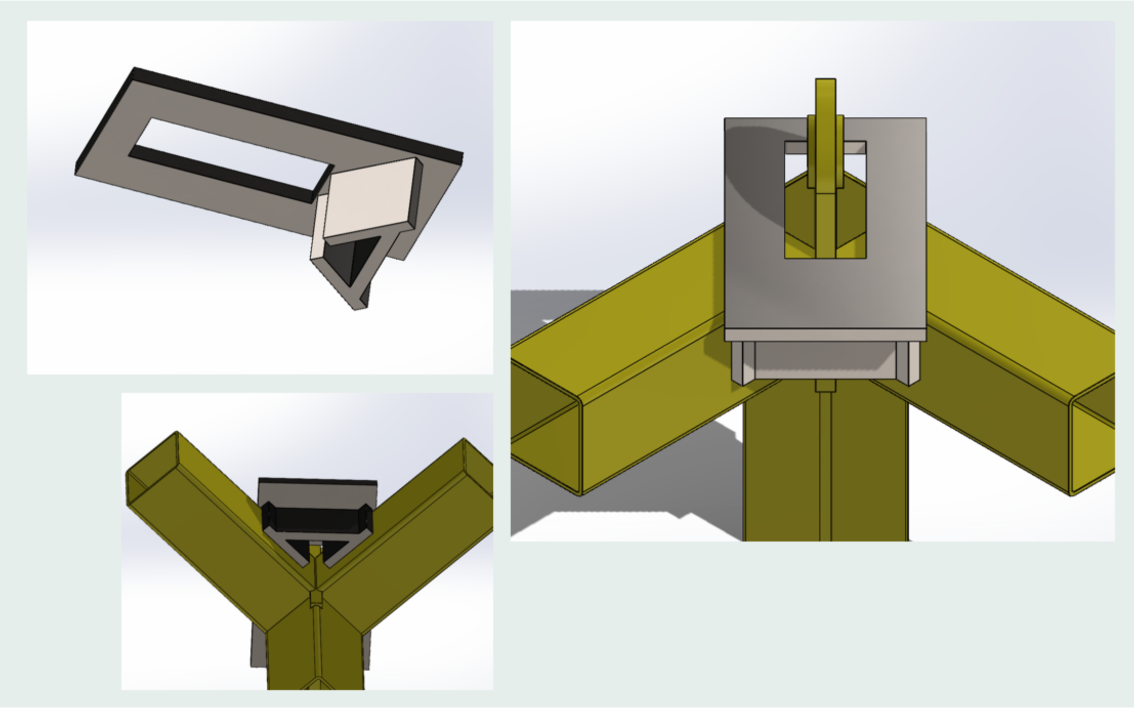


Figure 4.8: Example of interface designed to work with square tubing structure where horizontal tubes are flush with the top surface of vertical tube

4.2.3 Angle Interface

The Angle Interface (ANI) is carefully engineered to facilitate precise adjustments to the specific angles required during operational testing (BOM ITEM NO.3). ANI features slot holes calibrated at 0, 15, 30, and 45 degrees, in alignment with the specifications mandated by OneSubsea. These predefined angle slots ensure versatility and accuracy in testing various angular orientations and are sized to fit M56 bolts. The ANI incorporates access holes strategically positioned on its sides to enhance functionality. These access holes provide convenient reach to the shackle bolt and the pad eye, significantly improving ease of use and operational efficiency.

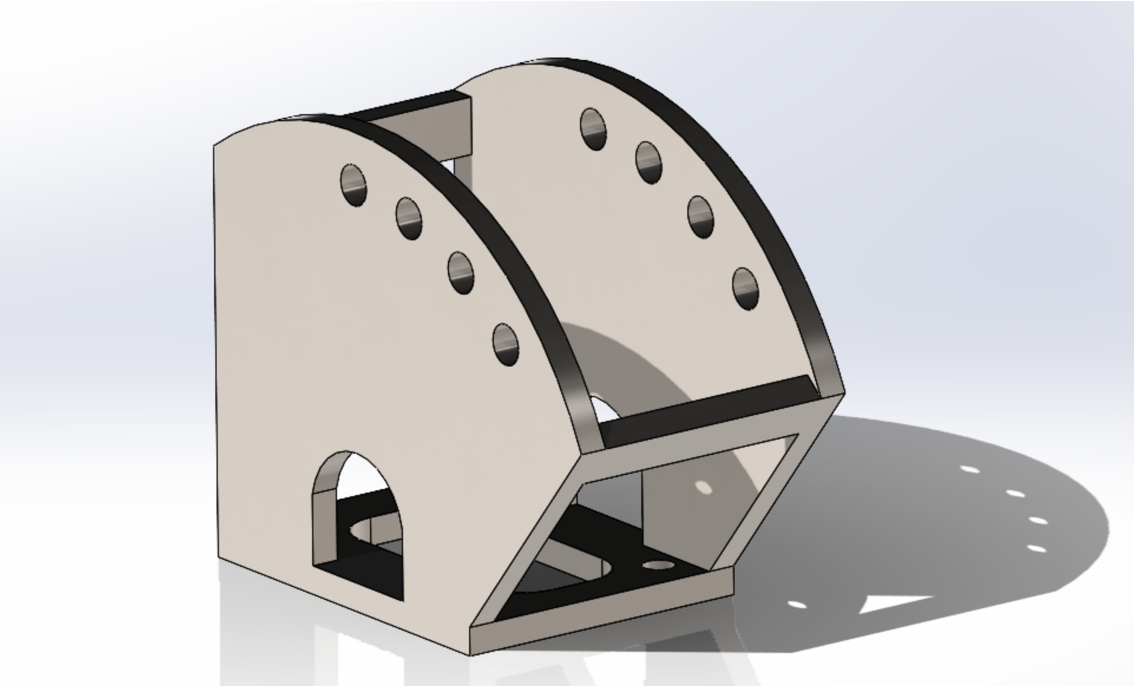


Figure 4.9: Angle Interface

Structural integrity and robustness are paramount in the design of the Angle Interface. At the ends, two support brackets have been integrated into the structure; one positioned at the front and the other at the back, between the two sides. These brackets play a critical role in preventing deformation and enhancing the overall stiffness of the component. By reducing potential warping, these reinforcements ensure the Angle Interface maintains its geometric precision and functional reliability under various operational conditions.

4.2.4 Cylinder Holder

The Cylinder Holder (BOM ITEM NO.7), in short (CH) is engineered to withstand a load of 100 tons exerted by the hydraulic cylinder (8). The robust design focuses on two critical areas: the surface upon which the cylinder rests and the bolt holes responsible for bearing the load.

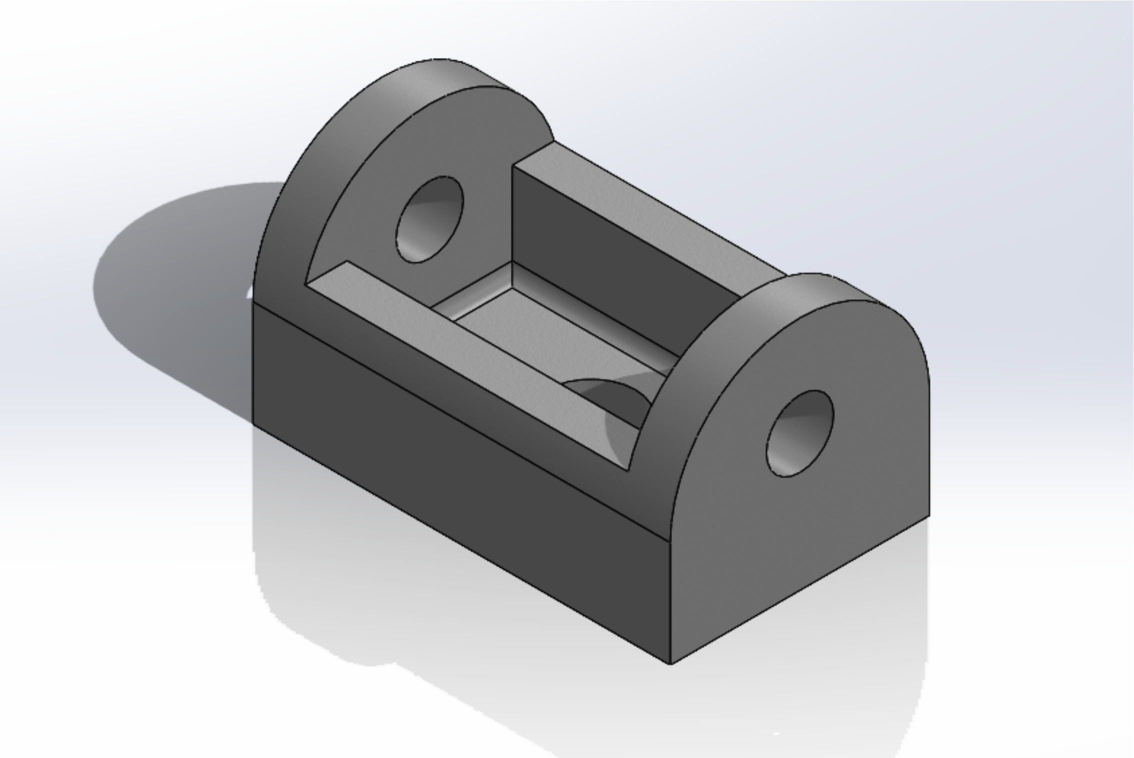


Figure 4.10: Cylinder Holder

The support brackets on the side of the CH ensure a reliable and durable solution, allowing the part to handle extreme loads. These brackets are strategically placed to reinforce the component, thereby reducing the risk of bending under high stress. Each hole on the sides of the CH is 57 mm sized to fit M56 bolts (17). The 75 mm hole in the center of the CH is designed to accommodate the TR70x10 threaded rod (5), allowing the forces to be concentrated in the center of the part. The width of the Cylinder Holder matches the inside width of the ANI (3) to prevent any unnecessary movement or space, which is later tightened down to the correct torque setting to lock it in place.

4.2.5 Pad Eye Puller

The Pad Eye Puller (PEP) is designed to endure a load of up to 100 tons (BOM ITEM NO.4). The design of the PEP ensures that both the surface engaged by the TR70x10 nuts (6) and the holes accommodating the shackle bolt (10) can reliably sustain these significant forces.

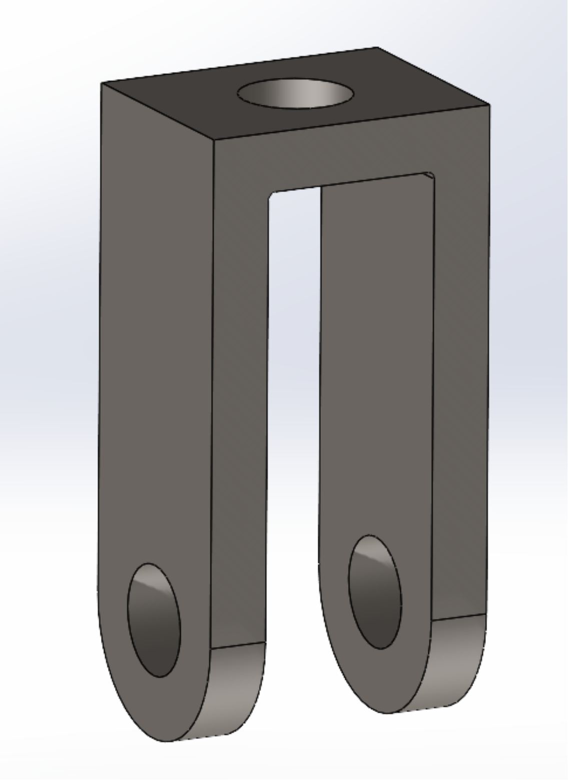
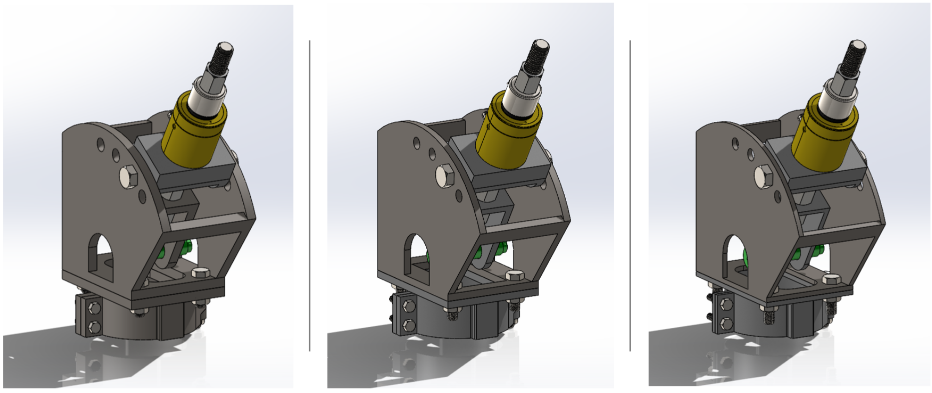


Figure 4.11: Ped Eye Puller

The PEP is set to match the requirements of the HIPPS module. The bolt holes are 74mm in diameter to match the pad eye and the shackle bolt with enough material on the bearing face (underside of the bolt holes) to withstand the loads it needs to hold against. Like the Cylinder Holder (7), the Pad Eye Puller has a centered 75mm hole at its top base plate. The fillets on the inside are 5mm in radius but can change depending on the size of the PEP. These fillets are to reduce stresses at the corners and potential welds. The PEP is designed to be scaled up and down depending on the pad eye it needs to test. It can also be switched out for other designs and solutions, such as shackles or hooks with connection interfaces to the TR70x10 threaded rod (5).

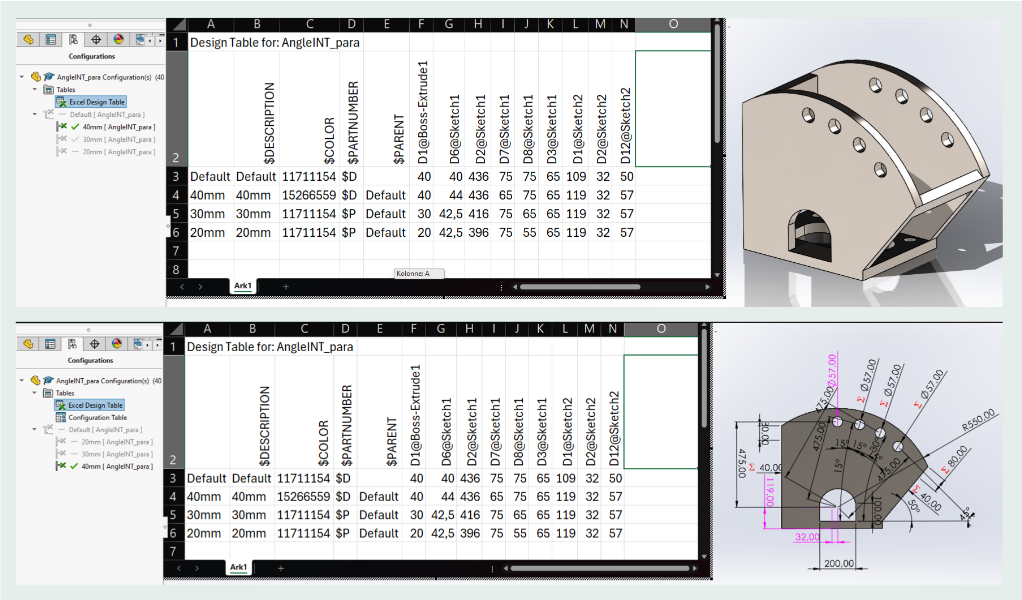
4.3 Parametric design of model

After finalizing the foundational design concept for the LPALT, the team developed a parametric model in SolidWorks. This approach allows straightforward adjustments to the steel plate thickness without compromising the model's integrity. The primary advantage of this method is that it streamlines the Finite Element Analysis (FEA) by eliminating the need to recreate the model for each desired thickness. The team examined steel plates of 20 mm, 30 mm, and 40 mm, composed of S355G10. Additionally, by knowing the thickness of the steel plates, the design can be more effectively refined. This ensures that even minor modifications are unlikely to undermine the structural strength of the design. When looking at *Figure 4.13*, one can see how the incorporation of a design table in SolidWorks can create a parametric model by referring to different annotations and sketches in the model. Also, by using equations and formulas in both the sketching and in the excel format, it is possible to create a fully parametric model that can increase efficiency when testing different models and sizes.



40mm **30mm** **20mm**

Figure 4.12: The LPALT modeled in 40 mm, 30 mm, and 20 mm plate thickness



Design Table for: AngleINT_para

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
2		$DESCRIPTION	$COLOR	$PARTNUMBER	$PARENT	D1@Boss-Extrude1	D6@Sketch1	D2@Sketch1	D7@Sketch1	D8@Sketch1	D3@Sketch1	D1@Sketch2	D2@Sketch2	D12@Sketch2
3	Default	Default	11711154	$D		40	40	436	75	75	65	109	32	50
4	40mm	40mm	15266559	$D	Default	40	44	436	65	75	65	119	32	57
5	30mm	30mm	11711154	$P	Default	30	42,5	416	75	65	65	119	32	57
6	20mm	20mm	11711154	$P	Default	20	42,5	396	75	55	65	119	32	57

Design Table for: AngleINT_para

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
2		$DESCRIPTION	$COLOR	$PARTNUMBER	$PARENT	D1@Boss-Extrude1	D6@Sketch1	D2@Sketch1	D7@Sketch1	D8@Sketch1	D3@Sketch1	D1@Sketch2	D2@Sketch2	D12@Sketch2
3	Default	Default	11711154	$D		40	40	436	75	75	65	109	32	50
4	40mm	40mm	15266559	$D	Default	40	44	436	65	75	65	119	32	57
5	30mm	30mm	11711154	$P	Default	30	42,5	416	75	65	65	119	32	57
6	20mm	20mm	11711154	$P	Default	20	42,5	396	75	55	65	119	32	57

Figure 4.13: Design table used for parametric modeling in SolidWorks

4.4 Explanation of Forces and Reaction Forces

The Local Pad-Eye Angular Load Tester (LPALT) tests the integrity and strength of pad eyes by applying controlled loads through a hydraulic system. Here's a detailed explanation of how forces and reaction forces travel through the system:

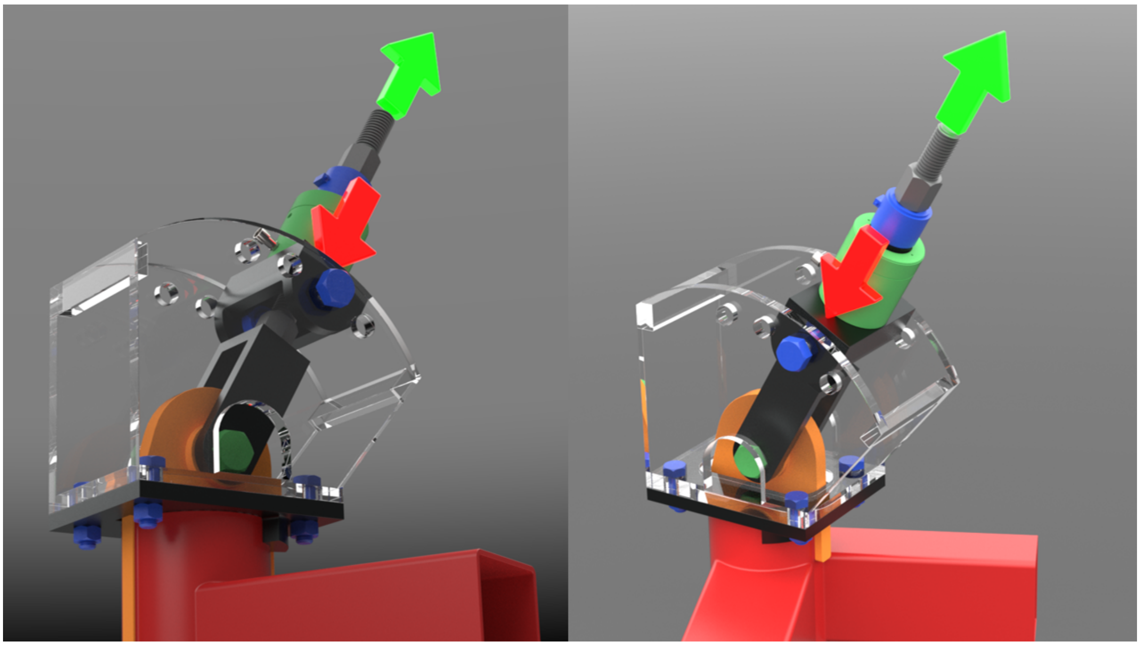


Figure 4.14: Rendering showing the reaction forces through a clear structure from two angles.

The hydraulic cylinder (8) is connected to a pump that applies force. When activated, the hydraulic pump pushes the cylinder outward, exerting force through the system. The force generated by the hydraulic cylinder is transmitted through the load cell (9), which measures the applied force and provides real-time readings via a digital gauge. The force continues from the load cell to the fastener at the top of the TR70x10 rod (5). Then, the force is transmitted through the rod to the pad eye puller (4), which is connected to the shackle bolt (10) and ultimately to the pad eye.

The reaction force from the hydraulic cylinder (8) is counteracted by the cylinder holder (7). The cylinder holder is secured by two high-strength M56 bolts (17), which absorb and distribute the reaction forces. The forces are further transmitted through the angle interface (3) to the Interface (1). Finally, the Interface (1) is held in place by the Bracket (2), which is mounted around a pipe. This bracket provides stability and ensures all forces are appropriately countered.

Summary of Force Dynamics:

- *Applied Force:* Initiated by the hydraulic cylinder and transmitted through the load cell and rod to the pad eye puller.
- *Reaction Force:* Counteracted by the cylinder holder and bolts, transmitted through the angle interface and main interface, and stabilized by the bracket around the pipe.

This setup ensures the LPALT functions correctly, providing accurate and reliable testing for subsea structures.

4.5 FEA in SolidWorks

Each component has undergone Finite Element Analysis in SolidWorks, verifying their stand-alone structural integrity at 100 tons or 980 665 Newtons. Throughout the iteration process of the component design, a load of 100 tons was used to ensure they could handle the loads applied from the hydraulic cylinder. In general, the group decided in consultation with OneSubsea that 80 % of the materials yield would be a desirable amount of stress. The yield strength of 40 mm thick steel plates in S355 is 345 MPa, making the acceptable amount 276 MPa in theory. After further discussion with OneSubsea, it became clear that in some cases, it is acceptable that the stresses can exceed this theoretical number depending on its size and location. In general, the stresses should not exceed 310 MPa, which is equal to 90% of the material yield stress when testing with a load of 100 tons. From the results, the team could reasonably assume that none of the components would fail during a full system integration analysis later on. This also helped avoid switching back and forth between SolidWorks and Abaqus.

It is worth mentioning that abnormal peak stresses can occur when simulating single parts or components with fixed orientations, points, vertexes, or planes. The lack of degree of freedom can cause the simulation to show abnormally high stresses in these areas. It is important to discuss and investigate the results when this happens. In this manner, one can determine if it is abnormal peak stress or the component needs more reinforcement. After discussion in meetings between the group, OneSubsea, and the supervisor, the decision was made to look beyond the most significant stresses in some instances and focus on where the ”real” stresses occur. This is also backed upon different studies by Digital Engineering, COMSOL, and Apollo Engineering Design Group [1] [2] [22] [5].

4.5.1 Cylinder Holder

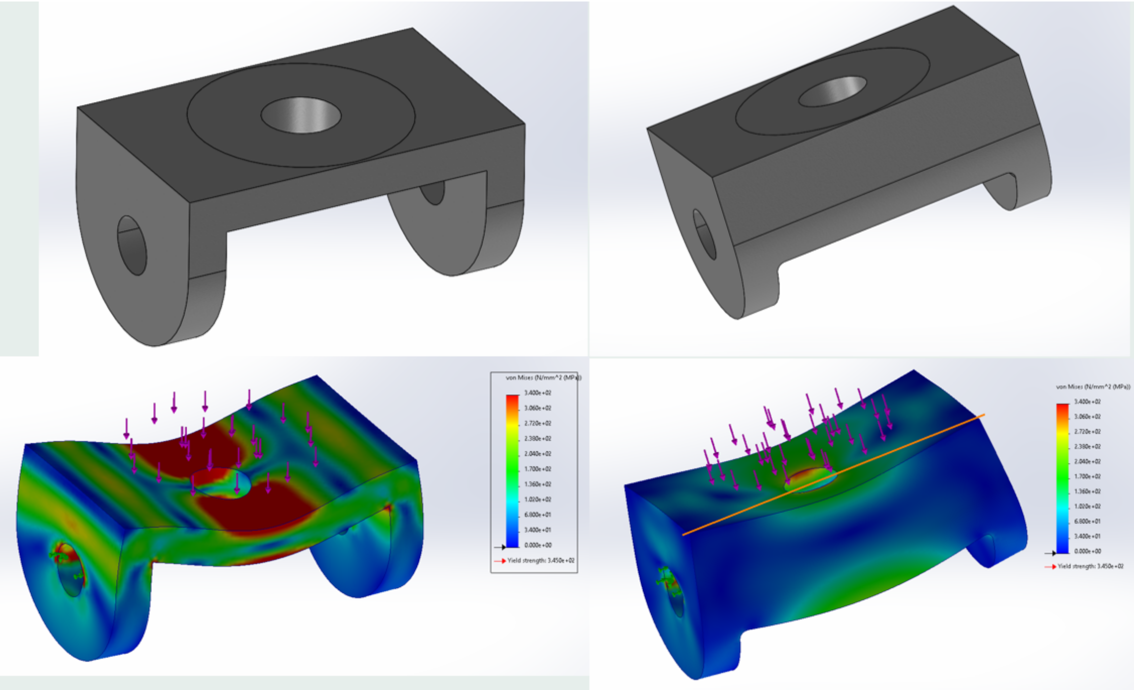


Figure 4.15: Cylinder Holder

In *Figure 4.15*, shows the FEA-based iterative process on the Cylinder Holder (CH) utilizing SolidWorks. The two left images show an earlier design proposal of the CH. When loads of 100 tons were applied the analysis revealed that the component did not possess enough rigidity to withstand the forces. Specifically, the bridge holding the hydraulic cylinder goes beyond what was deemed as allowable stress of 276 MPa. The color map in the image above with red equates to 340 MPa. The CH was redesigned by adding extra bracing on the sides. The results from the second analysis shows a significant drop in stress, reducing the value to approximately 204 MPa. Indicating that the bracing successfully bolstered the rigidity of the component to withstand the maximum loads the LPALT can operate under. Even with the addition of extra bracing peak, loads were still detected in the center and bolt holes. In consultation with OneSubsea, these loads were deemed abnormally high and explained by the lack of degree of freedom. From these assumptions and FEA results, it was concluded that the Cylinder Holder is sufficiently strong enough.

4.5.2 Pad Eye Puller

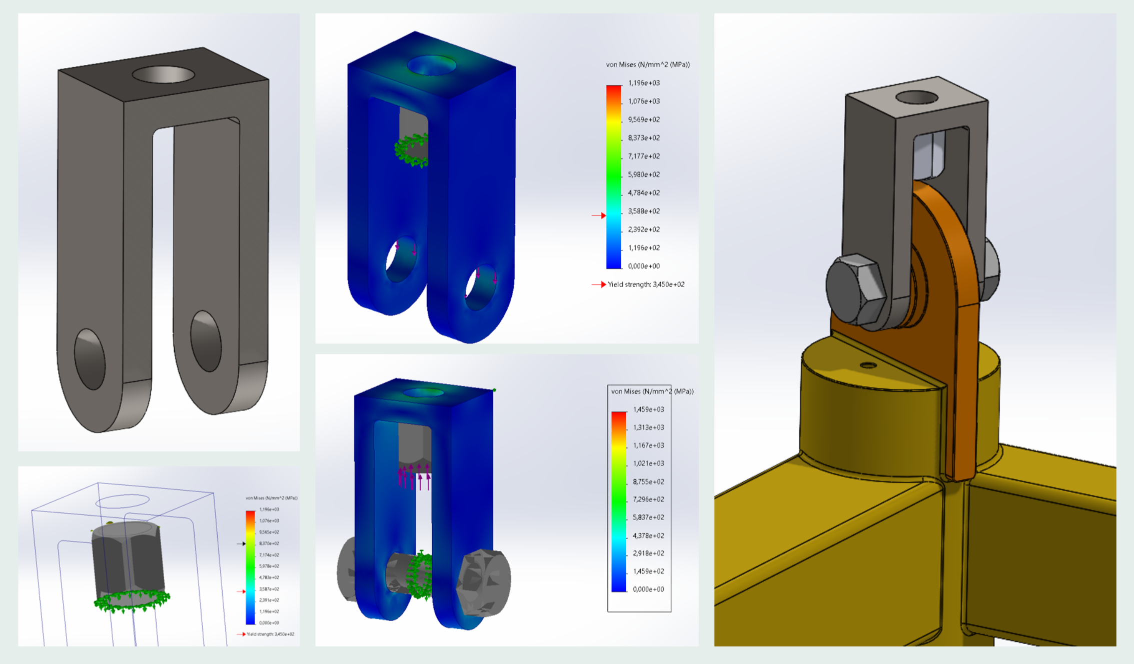
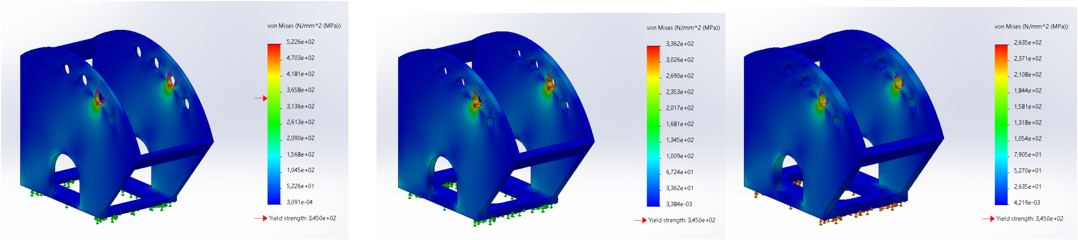


Figure 4.16: Pad Eye Puller Simulation

Figure 4.16 presents a simulation of the Pad Eye Puller (PEP) under bearing load, conducted using SolidWorks. Multiple approaches and methods were employed to ensure the simulations closely represented real-world conditions. By experimenting with various constraints and load types, the results provide a basis for comparing the effectiveness of different simulation techniques. A force of 980 665 Netwons (100 tons) was applied to simulate the forces on the components. As mentioned earlier, the lack of degree of freedom can lead to unrealistic stress concentrations. Additionally, interactions with rigid (non-destructible) parts can contribute to these unrealistic peaks at concentrated points. However, the stress distribution around the bolt holes is likely realistic, which is backed up by hand calculations. Section 4.6 gives an in-depth explanation of how the bearing and tear out stress was calculated for the PEP. By this dual approach, the team identified the need for additional material on the underside of the shackle bolt hole. Increasing the distance from 42 mm to 52 mm. This modification was crucial to enhance the component's strength and prevent yielding under high stress. After extensive testing and different approaches, estimates from ISO clippings show that the realistic peak stresses in these areas are around 200-300 MPa. In combination with hand calculations, the PEP was deemed to be sufficiently strong.

4.5.3 Parametric thickness analyzis

The parametric study analyzed steel plates of varying thicknesses (20mm, 30mm, and 40mm) under a bearing load of 100 tons applied to the M56 bolt holes at 30 degrees. The simulation results revealed peak stresses of 522.6 MPa, 336.2 MPa, and 263.5 MPa for the 20mm, 30mm, and 40mm thick plates, respectively. Based on these results, it is evident that the 40mm steel plate offers the optimal thickness for the LPALT, as it maintains the stress well below the yield strength of the material.



(a) 20mm *(b) 30mm* *(c) 40mm*

Figure 4.17: Parametric thickness of Angle Interface

4.5.4 Angle Interface with Cylinder Holder

Simulating stresses within an assembly yields more realistic results compared to single-part simulations. This is because assemblies eliminate the need to fix or constrain specific points, allowing for a more accurate representation of how components interact under load. Consequently, stress distributions in an assembly are more representative of real-world conditions.

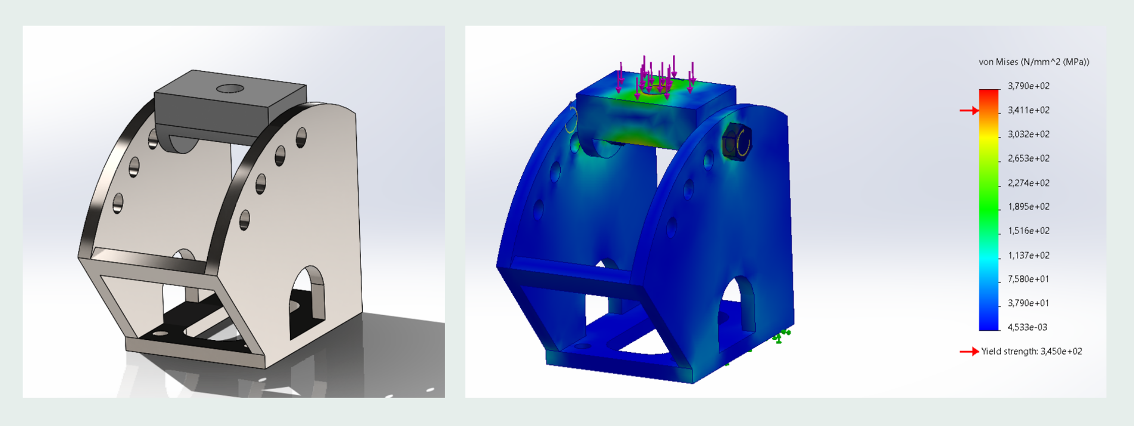


Figure 4.18: Angle Interface with Cylinder Holder Simulation

By incorporating multiple parts into an assembly, it is possible to simulate how different components interact. Using contact with friction during the simulation enhances the realism of the result. As expected, the highest stress concentrations are observed around the bolt holes and within the friction zones between the parts. Some peak stresses are detected in the model at 479 MPa. Excluding this, the principal stresses are determined to be between 265 and 302 MPa. With these results, the team decided to continue further with simulations in Abaqus.

4.5.5 System Integration Finite Element Analysis of the LPALT

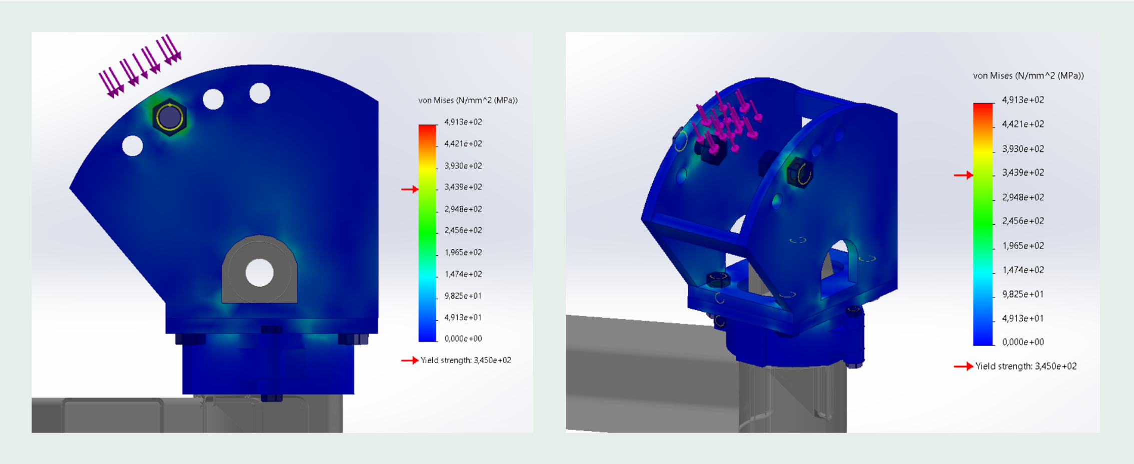


Figure 4.19: LPALT Simulation

The last simulation was conducted with all components except for the Pad Eye Puller, Trapezoid rod and nut, Enerpac cylinder, and Vetek load cell. These components either consist of off-the-shelf parts designed to withstand loads significantly exceeding the system's capabilities or, in the case of the Pad Eye Puller, do not physically interact with the parts analyzed here, so it is better analyzed by itself. Referring to *Figure 4.19*, it is evident that the peak stresses drop when compared to the single-part simulation. This reduction in peak stress results from the fixation of the degree of freedom. Although some abnormally high peak stresses at 491 MPa still exist, these can be neglected in certain areas and vertex points, as explained earlier. Therefore, the principal "peak" stresses are observed to be between 250 to 300 MPa. After presenting and discussing the results with the supervisor from OneSubsea, it was confirmed that the amount of stress is acceptable.

The reason for the difference between the different simulations likely stems from the different methods used and the number of components included in the simulation. In *Figure 4.18*, the simulation includes only two parts, excluding the bolts. In contrast, *Figure 4.19* contains four parts, with the Cylinder Holder hidden to better display the stresses. During the discussion on the differences in peak loads, our supervisor from OneSubsea concluded that very small volumes of local peak stresses can be considered artifacts and overlooked, as is the case here.

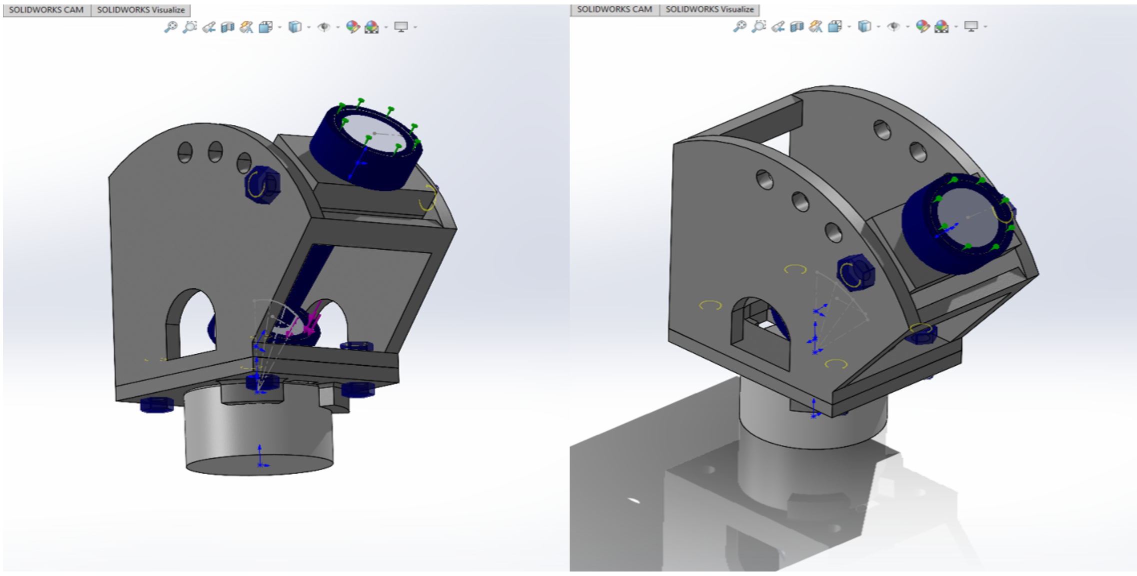


Figure 4.20: Model used in Full Integration Finite Element Analysis

An earlier simulation shows that steel plates need a 40 mm wall thickness for structural integrity. The hydraulic ram analytical simulator is remotely connected through an analytical linkage rod to move loading away from the hydraulic cylinder plate. This was done to remove any rotation of the cylinder mounting plate during the simulation. The interface plate is mated to a rigid dummy representing the vertical pipe of the HIPPS subsea module.

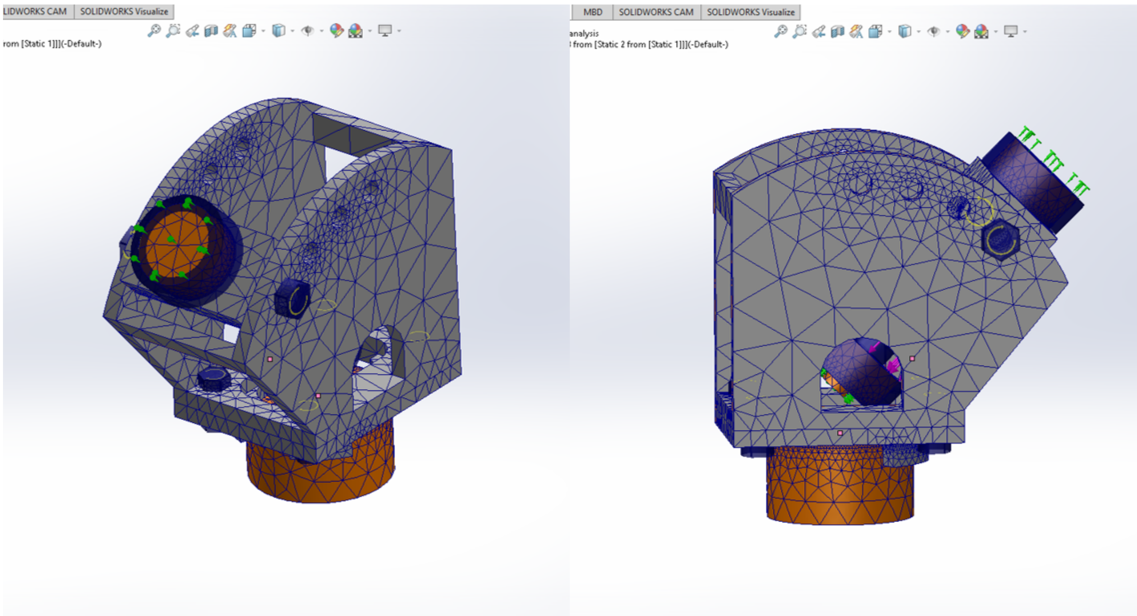


Figure 4.21: Mesh used in Full Integration Finite Element Analysis

Automatic meshing in SolidWorks is set to its most course settings. The mesh is not as detailed as the team would prefer, but increasing mesh fidelity increases computation time to more than 24 hours and often fails due to memory usage.

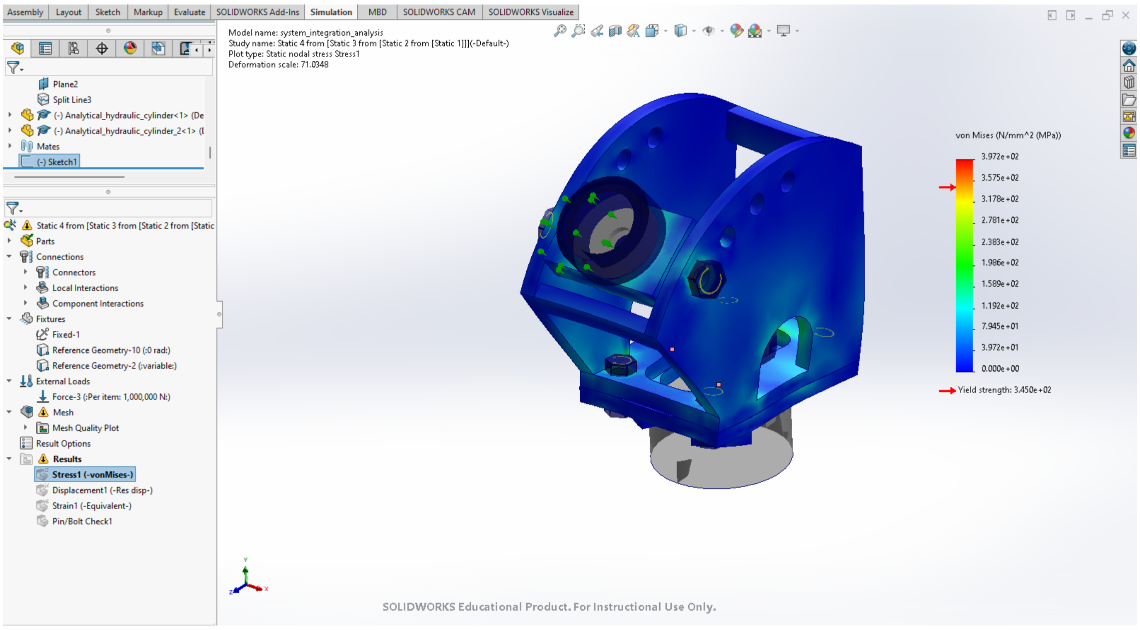

Model name: system_integration_analysis
Study name: Static 4 from [Static 3 from [Static 2 from [Static 1]]](-Default-)
Plot type: Static nodal stress Stress1
Deformation scale: 71.0348

Figure 4.22: vonMises stress under 100 ton load

Figure 4.22 shows the von Mises stresses in the model at 100 ton load and 71x deformation scale. Except for a few local stress areas, the model demonstrates enough structural rigidity to withstand

100 tons of loading from the hydraulic cylinder.

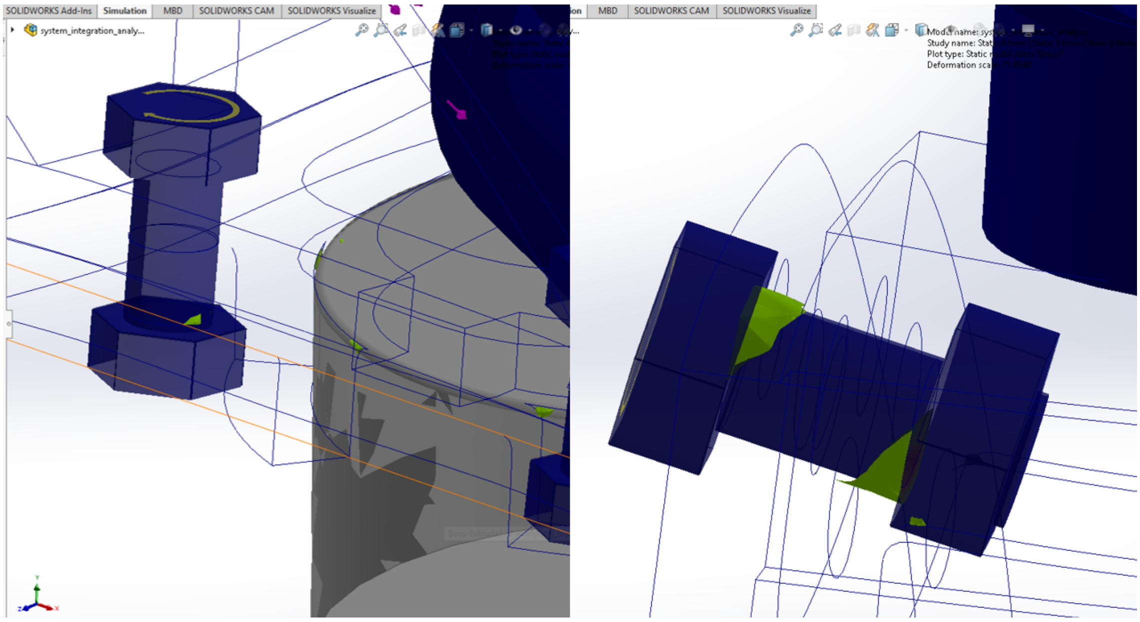
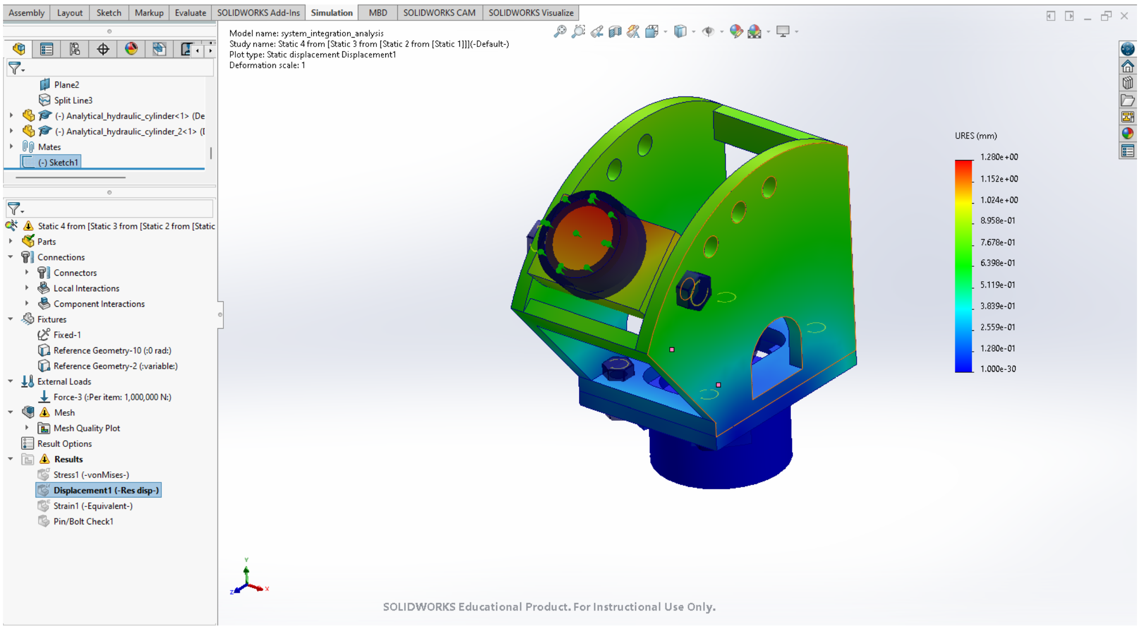


Figure 4.23: vonMises IsoClippings at 80% yield

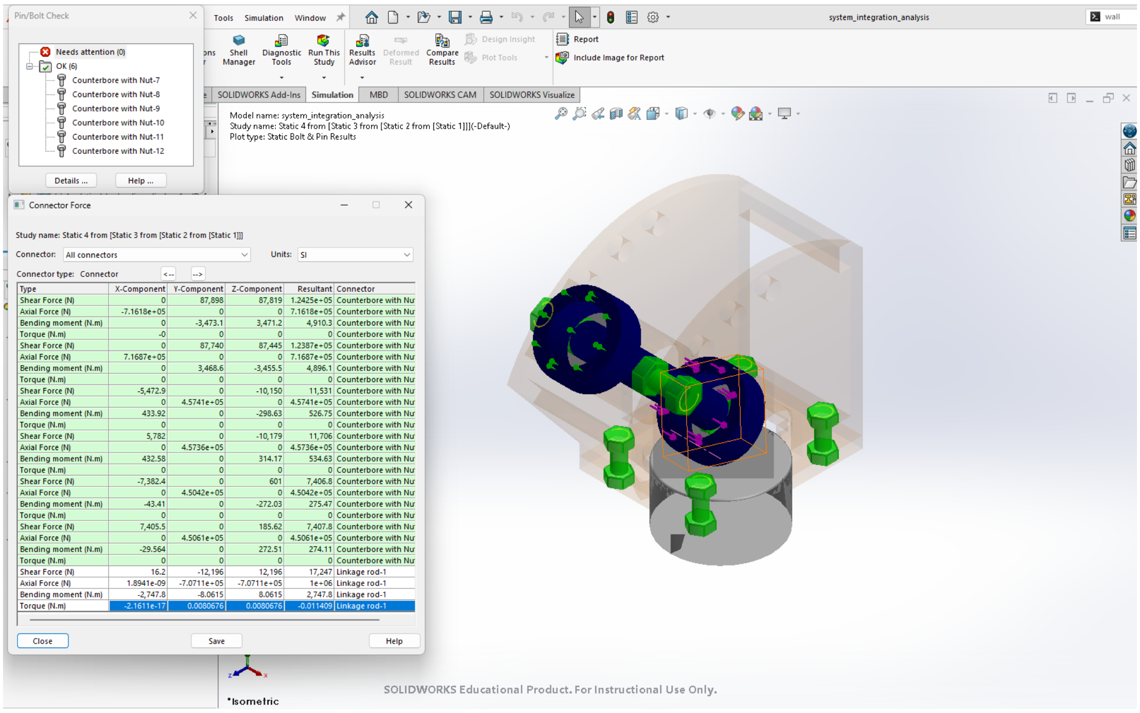
Figure 4.23 shows ISO Clippings with a cutoff at 80% of yield. Here, it shows that forces will somewhat rotate the bolts, putting stress on the structure in the contact points in its rotational direction. But when showing only stress above 80% of yield, it is fairly certain that the stress-induced is not more than the structure can withstand. By further investigation with ISO clipping, it can be concluded that the stresses around the M56 bolts are around 280 to 315 MPa.



Model name: system_integration_analysis
Study name: Static 4 from [Static 3 from [Static 2 from [Static 1]]](-Default-)
Plot type: Static displacement Displacement1
Deformation scale: 1

Figure 4.24: Model displacement at 100 ton load

Figure 4.24 shows the structure's total displacement of 1.28mm at a 100-ton load.



Model name: system_integration_analysis
Study name: Static 4 from [Static 3 from [Static 2 from [Static 1]]](-Default-)
Plot type: Static Bolt & Pin Results

Study name: Static 4 from [Static 3 from [Static 2 from [Static 1]]]

Connector: All connectors

Units: SI

Connector type: Connector

Type	X-Component	Y-Component	Z-Component	Resultant	Connector
Shear Force (N)	0	87,898	87,819	1.2425e+05	Counterbore with Nu
Axial Force (N)	-7.1618e+05	0	0	7.1618e+05	Counterbore with Nu
Bending moment (N.m)	0	-3,473.1	3,471.2	4,910.3	Counterbore with Nu
Torque (N.m)	-0	0	0	0	Counterbore with Nu
Shear Force (N)	0	87,740	87,445	1.2387e+05	Counterbore with Nu
Axial Force (N)	7.1687e+05	0	0	7.1687e+05	Counterbore with Nu
Bending moment (N.m)	0	3,468.6	-3,455.5	4,896.1	Counterbore with Nu
Torque (N.m)	0	0	0	0	Counterbore with Nu
Shear Force (N)	-5,472.9	0	-10,150	11,531	Counterbore with Nu
Axial Force (N)	0	4.5741e+05	0	4.5741e+05	Counterbore with Nu
Bending moment (N.m)	433.92	0	-298.63	526.75	Counterbore with Nu
Torque (N.m)	0	0	0	0	Counterbore with Nu
Shear Force (N)	5,782	0	-10,179	11,706	Counterbore with Nu
Axial Force (N)	0	4.5736e+05	0	4.5736e+05	Counterbore with Nu
Bending moment (N.m)	432.58	0	314.17	534.63	Counterbore with Nu
Torque (N.m)	0	0	0	0	Counterbore with Nu
Shear Force (N)	-7,382.4	0	601	7,406.8	Counterbore with Nu
Axial Force (N)	0	4.5042e+05	0	4.5042e+05	Counterbore with Nu
Bending moment (N.m)	-43.41	0	-272.03	275.47	Counterbore with Nu
Torque (N.m)	0	0	0	0	Counterbore with Nu
Shear Force (N)	7,405.5	0	185.62	7,407.8	Counterbore with Nu
Axial Force (N)	0	4.5061e+05	0	4.5061e+05	Counterbore with Nu
Bending moment (N.m)	-29.564	0	272.51	274.11	Counterbore with Nu
Torque (N.m)	0	0	0	0	Counterbore with Nu
Shear Force (N)	16.2	-12,196	12,196	17,247	Linkage rod-1
Axial Force (N)	1.8941e-09	-7.0711e+05	-7.0711e+05	1e+06	Linkage rod-1
Bending moment (N.m)	-2,747.8	-8.0615	8.0615	2,747.8	Linkage rod-1
Torque (N.m)	-2.1611e-17	0.0080676	0.0080676	-0.011409	Linkage rod-1

Figure 4.25: Pin & Bolt Check

Pin & Bolt Check is a native function in SolidWorks that evaluates the safety of bolts based on their material quality, the requested safety factor, and the applied forces in the simulation. The function provides a Go/No-Go result, with green bolts indicating a pass and red bolts indicating a fail. In this case, the simulation passed with six green bolts and a minimum safety factor of 1.9 at a 100-ton load using 12.9 steel bolts. Given that the LPALT will only experience a maximum load of 80 tons, the team determined through hand calculations that material quality 8.8 is sufficient for safe operation.

4.6 Hand calculations

Finite Element Analysis (FEA) is a powerful tool for predicting how structures will react to forces, vibrations, heat, and other physical effects. However, performing hand calculations when analyzing assemblies and components can also be an important tool for verifying the integrity of structures, especially when unexpected peak stress occurs.

4.6.1 Pad Eye Puller

Hand calculations were used to verify the strength of the Pad Eye Puller (PEP). The FEA simulations revealed high-stress concentrations around the bolt holes, and it was challenging to ascertain whether these stresses were realistic. Therefore, hand calculations of the bearing and tear-out stress were conducted.

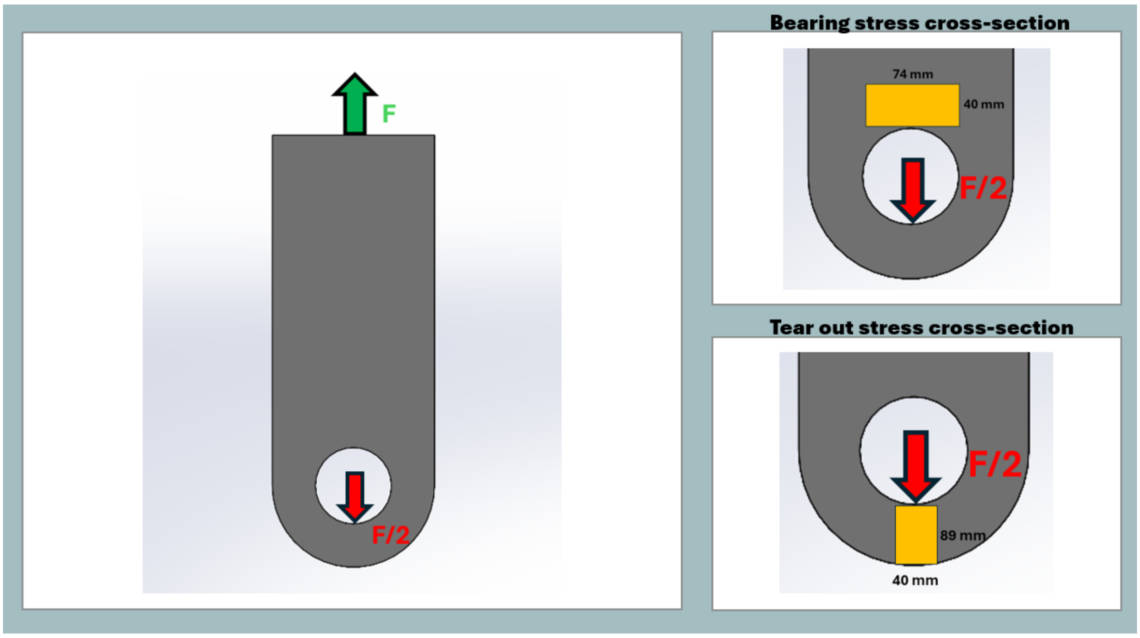


Figure 4.26: Cross section of bearing and tear out stress

The cross-section for calculating the bearing stress is illustrated in *Figure 4.26*. Here, the diameter of the hole (D_h) is 74 mm, the thickness of the plate (t) is 40 mm and the area factor f_{cs} equals 75%. The load applied to the PEP is 100 tons. Consequently, the load (F) per bolt hole is 50 tons, equivalent to 490330 Newtons. The bearing stress, denoted as σ_b, is calculated using *equation 2.4*.

$$\sigma_b = \frac{F}{D_h \cdot t \cdot f_{cs}} = \frac{490330\,N}{74\,mm \cdot 40\,mm \cdot 0.75} \approx \underline{220.87\,MPa}$$

The bearing stress (σ_b) is determined to be 220.87 MPa from the calculations. Allowable stresses are set at 80% of yield strength, which amounts to 276 MPa. This indicates that the PEP will be sufficiently strong to handle the design load.

The tear out stress for the PEP was calculated using *equation 2.5*. The parameters F, t, and D_h will be the same when calculating the bearing stress. R_{pad} is the distance from the bolt hole's center to the PEP's outside radius. In the earlier PEP design, the distance from the center of the hole down to the outside radius was 79 mm. With this geometry, the tear-out stress was found to be 253 MPa. An allowable value in terms of theoretically acceptable stress, but still somewhat higher than the value first expected. After a discussion with OneSubsea, the team decided to increase the outside radius of the PEP. Because of the high stresses found in this area in the FEA done in SolidWorks, in combination with a relatively high tear-out stress, the radius was increased to 89 mm.

$$\tau_{tear} = \frac{F}{(2 \cdot R_{pad} - D_h) \cdot t} = \frac{490330\,N}{(2 \cdot 89\,mm - 74\,mm) \cdot 40\,mm} \approx \underline{117.87\,MPa}$$

In these calculations, the tear out stress is assumed to be pure shear. To make the results comparable with Von Mises stresses one must multiply the tear out stress with $\sqrt{3}$.

$$\sigma_{tear} = \tau_t \cdot \sqrt{3} = 117.87MPa \cdot \sqrt{3} \approx \underline{204.16\,MPa} \tag{4.1}$$

The tear-out stress is determined to be approximately 204.16 MPa. Adding 10 mm to the outside radius shows a big decrease in stress. These results indicate that the PEP will be sufficiently strong to handle a load of 100 tons.

4.6.2 Angel Interface

Hand calculations were also used to back up the angel interface (ANI) design. The most challenging part of the design involved determining the appropriate size and classification of the bolts. This area also exhibited the highest stress levels in FEA conducted in SolidWorks. To help determine the size and classification of the bolts, *equation 2.7* and *2.6* were used. The load applied to the ANI is 100 tons. Consequently, the load (F) per bolt hole is 50 tons, equivalent to 490330 Newtons. *Table 4.1* shows the values used to calculate required bolt diameter. As mentioned in section 2.6 the allowable stress and shear in a bolt that is supposed to be tightened and loosened is 70% of the yield stress, shown as σ_t and τ_t in the table. From *Table 4.1*, one can see that the classification can be as low as 8.8, validating the use of an M56 bolt.

Class	**12.9**	**10.9**	**8.8**
σ_f	1080 MPa	900 MPa	640 MPa
τ_f	623.54 MPa	519.6 MPa	369.5 MPa
σ_t	756 MPa	630 MPa	448 MPa
τ_t	436.48 MPa	363.73 MPa	213.33 MPa
A	1123.37 mm^2	1348.06 mm^2	2298.46 mm^2
r	18.91 mm	20.71 mm	27.05 mm
d	37.82 mm	41.43 mm	54.1 mm
D	38 mm	42 mm	55 mm

Table 4.1: Required bolt diameter

The torque formula determines the minimum tightening needed for the bolts to withstand the force exerted by the hydraulic cylinder. For these calculations *equations 2.8* and *2.9* were used. Typically, a safety factor would be added to the applied load. But because the jig is already tested with a force above the assumed test load it was dropped in the calculations.

$$\Gamma = k \cdot D \cdot F_0 = 0.2 \cdot 0.056m \cdot 490.33kN \approx 5492Nm \tag{4.2}$$

Class	**12.9**	**10.9**	**8.8**
F_0	1534.68 kN	1278.9 kN	909.44 kN
Γ	17 188.42 Nm	14 323.68 Nm	10 185.73 Nm

Table 4.2: Required torque and preload

The minimum required torque was found to be 5492 Nm. The results from *Table 4.2* show that the M56 bolt's 8.8 classification can handle the needed amount of torque without problems.

4.7 FEA in Abaqus

After extended testing in SolidWorks, the next step was moving to Abaqus to compare and confirm the simulations. As mentioned earlier in *section 3.4.4*, the team needed to change the FEA strategy. Abaqus is still an excellent tool for FEA, even with the limited student version. Therefore, the bachelor group ran some simple simulations in Abaqus to mimic load tests on the Cylinder Holder, Angel Interface, and the Pad Eye Puller. As in the SolidWorks simulations, a force of 100 tons, equivalent to 980,665 Newtons, was applied to the models.

4.7.1 Cylinder Holder

Similar to the SolidWorks simulation, the Cylinder Holder (CH) also experiences unexplainable high peak stresses at single points. This results from fixing surfaces and using rigid bodies with tie constraints in Abaqus, similar to SolidWorks. Excluding this peak stress as shown in *Figure 4.27*, the stresses around the bolt holes reach around 265 MPa and around 290 MPa at the surface where the hydraulic cylinder goes, from the SolidWorks simulations of the CH the stress. From these results, one can argue that the previous tests from SolidWorks are reliable.

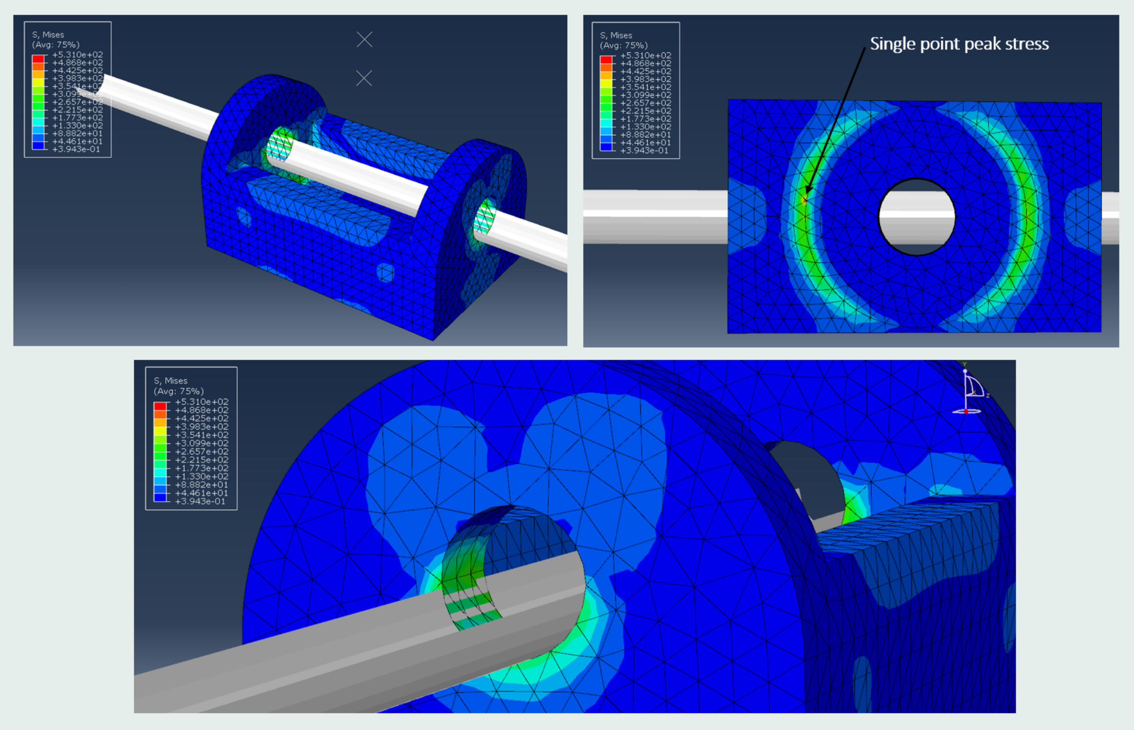


Figure 4.27: Cylinder Holder Abaqus simulation

4.7.2 Angle Interface

The Abaqus simulation of the Angle Interface resulted in expected peak stresses around M56 bolt holes. The peak stress of 272 MPa is concentrated at the edges where the bolt is positioned and propagates through the plate down to the base.

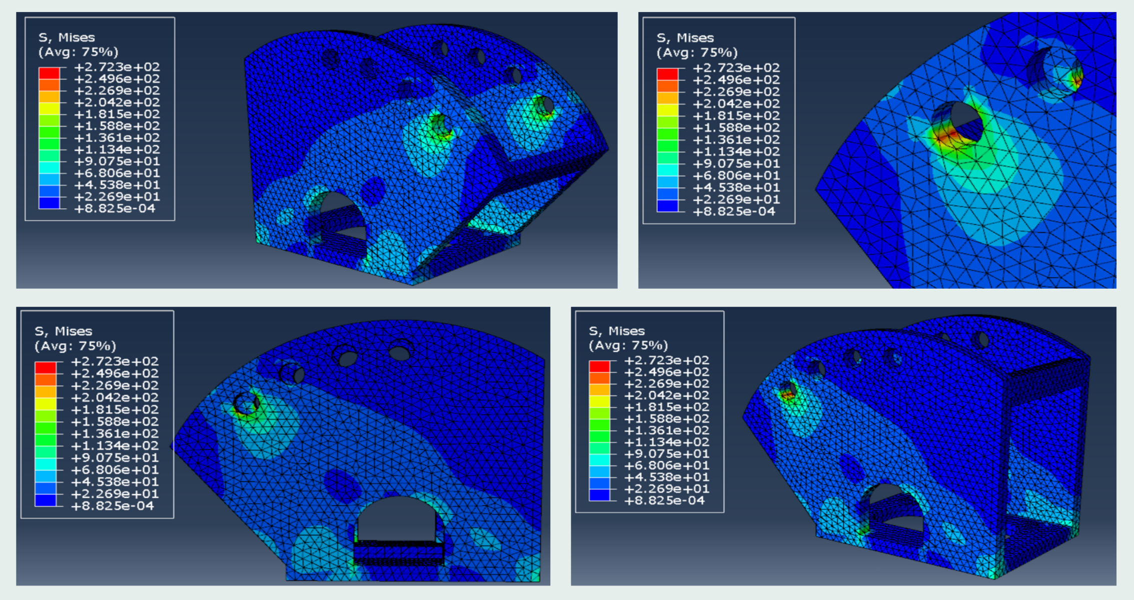


Figure 4.28: Angle Interface Abaqus simulation

4.7.3 Pad Eye Puller

Similar to the SolidWorks simulation, the PEP model experiences high peak stresses of 472 MPa at the side of the shackle bolt holes. Therefore, even with the Abaqus simulation, it is concluded that the best method of validating the Pad Eye Puller is both with hand calculations and extensive simulations in an assembly and not as a single part. By excluding the degree of freedom, it constructs higher stresses than expected.

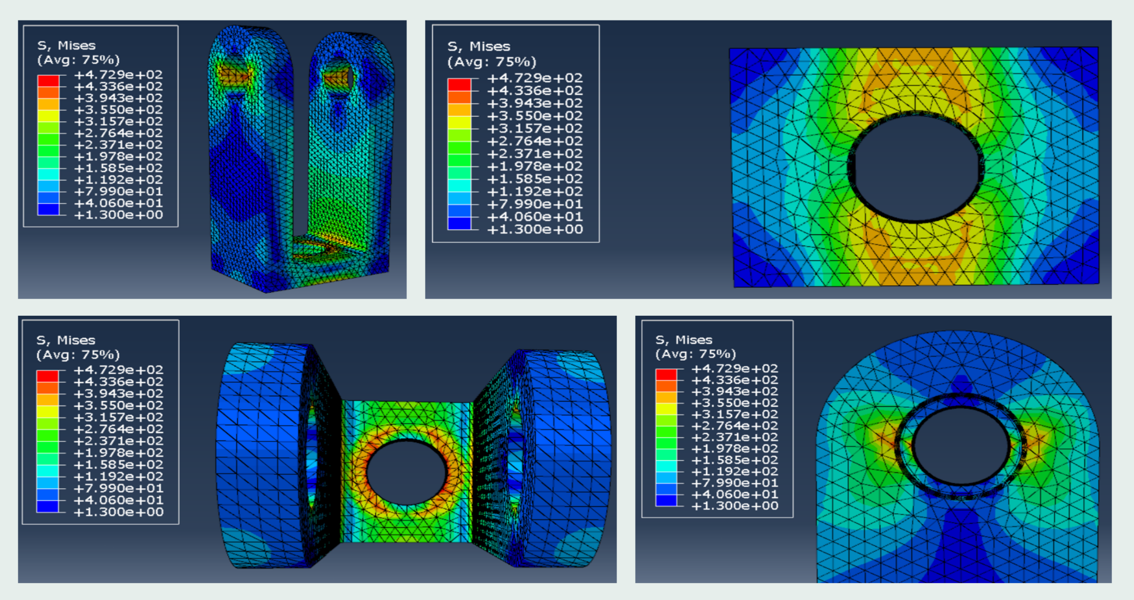


Figure 4.29: Pad Eye Puller Abaqus simulation

4.8 FMECA

System/Equipment:			LPALT	Date:	15.04.2024	Preformed by: Sigmund Linn, Bjarte Ullebust Almklow, Åse Helgeland Pedersen					
Description of unit			**Description of failure**			**Effect of failure**		**Risk**		**Risk reducing measures**	**Comment**
Component	Function	Operational mode	Failure mode	Failure cause	Detection of failure	Local	System	Failure rate	Severity ranking		
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)	(11)	(12)
Hydraulic cylinder Barrel	Hold the pressure and prevent leaking	Normal operation	Leakage	Incorrect installation of components, damaged seals	Visual inspection, monitoring system performance	Not holding pressure in cylinder	Reduced efficiency and effectiveness	3	1	Regular inspection	
		Normal operation	Cracks	Mechanical damage to the cylinder wall	Visual inspection	Not holding pressure in cylinder	Full stop, significant downtime	2	1	Regular inspection	
Hydraulic cylinder piston	Separate the two chambers of the barrel. Move when beeing pushed by the hydraulic fluid	Normal operation	Difficulty in moving	Inadequate maintenance, contaminants in hydraulic fluid	Monitoring system performance, listening for unusual sounds	Inconsistent movement	Reduced efficiency and effectiveness	3	1	Regular inspection	
		Normal operation	Complete failure to move	Inadequate maintenance, contaminants in hydraulic fluid	Monitoring system performance	No movement	Full stop, significant downtime, stops all dependent operations	1	1	Regular inspection	
Hydraulic cylinder piston rod	Utilize the forces of the hydraulic fluid to retract or extend	Normal operation	Rod bending	Excessive load, improper alignment	Visual inspection, monitoring system performance	Exerts reduced force	Reduced efficiency and effectiveness	2	1	Regular inspection	
		Normal operation	Rod breaking	Excessive load, improper alignment	Visual inspection	Exerts no force	Full stop, significant downtime, risk of damage on other components	1	2	Regular inspection	
Hose	Connect the compressor to hydraulic cylinder	Normal operation	Leakage	Cracks, material fatigue	Visual inspection, monitoring system performance	Not holding pressure	Reduced efficiency and effectiveness	4	1	Regular inspection, use of parts with high quality	
Load cell	Measure the true amount of load	Normal operation	Does not produce accurate output	Wear and tear over time, inadequate maintenance	Precision tests	Loss of accauracy	Compromise product quality, critical error	3	2	Regular inspection and calibration	
		Normal operation	Produces no output at all	Excessive load, electrical failure	Visual inspection	Loss of function	Full stop, significant downtime	3	1	Regular inspection and calibration	
Interface	Hold the LPALT in place on the frame/ pad eye	Normal operation	Does not hold correct position	Incorrect installation	Visual inspection	Loss of stability	Full stop, significant downtime, damage to other components	1	2	Proper training in how to install the interface	
Compressor	Provide pressure to the hydraulic cylinder	Normal operation	Does not provide adequate pressure	Wear and tear over time, leakage, blockage	Monitoring system performance	Reduced efficiency	Significant downtime, increased operational cost	1	1	Regular inspection	

Figure 4.30: FMECA

Chapter 5

Discussion

5.1 Self-assessment of solution

The project's initial aim was to emulate a full-scale load test of subsea structures manufactured by OneSubsea. However, it quickly became apparent that reproducing the load conditions from an entire system integration load test was unachievable with the limited resources, time frame, and knowledge available to the group. In collaboration with OneSubsea, the project's scope was redefined to focus on local load testing of pad eyes—essential for lifting the structures—and the welds attaching the pad eyes to the structure itself.

To independently evaluate the bachelor group's solution to the objective, it is essential to begin with the initial scope of the task. The report should present a robust design capable of providing a test load of up to 80 tons per lifting point in angles varying from 0-45 degrees. The strength of the components should be verified with Finite Element Analysis (FEA). The final solution should be versatile, working on subsea structures with a range of geometry. Having undergone and successfully passed a comprehensive FEA, the LPALT, in theory, should exhibit robust structural integrity. Accommodating various subsea structure geometries is possible by interchanging the interface plate that adjoins the LPALT to the subsea structures. The Angle Interface features slot holes from 0-45 degrees, ensuring accurate testing in various angular orientations. Thus, the team infers that, at least in theory, the LPALT is poised to meet its intended design objectives successfully.

Beyond meeting these initial expectations, the LPALT has demonstrated capabilities surpassing those anticipated by OneSubsea. The LPALT has been engineered to manage loads of up to 100 tons, which provides a 20% safety margin beyond the maximum loads OneSubsea plans to apply to their equipment. The LPALT's innovative design offers a compact, efficient solution for localized testing of pad eyes and their surrounding welds, an approach not previously considered by OneSubsea. Its small size greatly enhances transportability and storage efficiency, as it can be easily accommodated on a standard European pallet. This compactness theoretically enables rapid and efficient testing, as the LPALT can be quickly relocated between testing points using an overhead crane, minimizing downtime between tests.

The LPALT is designed to meet the specific requirements set by OneSubsea. However, it is critical to understand the LPALT's limitations. It does not replace a full-scale system integration test but is intended for localized assessments. The LPALT's scope is confined to specific component testing, leaving most of the structure untested under these conditions. This necessitates continued reliance on Finite Element Analysis (FEA) by OneSubsea to verify the overall structural integrity of their systems when broader tests are not feasible. The capabilities of the LPALT, at this stage, remain theoretical as it has not yet been produced, and its performance can only be discussed in hypothetical terms. Moreover, preliminary FEA is crucial before employing the LPALT for localized testing. This step ensures the entire structure meets safety and integrity standards, a process indispensable for confirming structural integrity and compliance under operational stresses, which the LPALT alone cannot fully assess. This approach highlights the necessity of combining the LPALT with other testing methods to evaluate subsea structures comprehensively. Clear communication about these limitations is essential to manage expectations and maintain transparency about what the LPALT can and cannot achieve within the broader context of subsea structural testing.

5.2 Further work

This thesis primarily conceptualizes the LPALT and does not thoroughly explore its production, treating it as a conceptual model rather than a finalized product. Consequently, a logical progression for future work would involve a comprehensive redesign of all steel components into modular assemblies, coupled with the creation of detailed manufacturing drawings aimed at facilitating straightforward production. The team proposes adopting a "tab in slot" design approach to simplify assembly and enhance structural integrity. This method would utilize the inherent strength of the steel plates, minimizing reliance on welds to bear forces, thus promoting more precise assembly and reducing welding-induced thermal distortion. By implementing this design philosophy, it is anticipated that assembly could be streamlined in a fail-safe ("poka yoke") manner while also reducing the extent of welding required and thereby reducing troublesome thermal warping.

The interface plates currently designed are tailored specifically for the HIPPS subsea structure. Given that OneSubsea produces a variety of structures with diverse geometries, it would be prudent to develop additional interface plates to accommodate these different forms. Concurrently, ensuring that the LPALT's geometry is versatile enough to support a broad range of subsea structures is essential. This approach will prevent the limitations akin to fitting a square peg in a round hole, enhancing the LPALT's adaptability and utility across OneSubsea's product line.

Once a future design freeze is achieved, it would be advisable to 3D print a comprehensive, finished model based on the production drawings to confirm that all components fit together as intended. Additionally, 3D printing various pad eye structures would provide an opportunity to test the interface plate designs, ensuring they function correctly and as expected. This step would be a crucial validation phase, leveraging rapid prototyping to enhance design accuracy and effectiveness.

5.3 Feedback from OneSubsea

As highlighted in the introduction of this report, it is paramount to ensure that the final product is both manufacturable and capable of further development by OneSubsea. This focus on practicality and future scalability underscores our commitment to delivering a solution that meets our client's high standards and specific needs. In light of this, we sought and received feedback from our supervisor at OneSubsea, which reflects our work's overall progress and quality.

> "I think the project has gone very well. To me, it seems like you have worked steadily throughout and brought up new questions and suggestions at every meeting. You have had ideas and solutions that we hadn't thought of, which we believe will work well. We think the final result has turned out well, and we believe that we can produce and use it." - Ole Petter Saxrud

5.4 Reflection of finite element analysis

The team initially planned a two-stage Finite Element Analysis (FEA): a preliminary analysis using SolidWorks during the iterative design process, followed by a detailed system integration analysis in Abaqus. This approach was intended to utilize the high-performance computational resources the Norwegian University of Science and Technology (NTNU) provided. However, unforeseen delays in the availability of these resources necessitated a revision of the planned analysis.

The only accessible student version of Abaqus posed many limitations. Making the program lack the necessary robustness for a comprehensive FEA. Slow processing times of the available computers, frequently resulting in erroneous outputs, significantly constrained our capabilities. These challenges forced the team to reassess and adapt our FEA strategy.

SolidWorks was then selected for the complete system integration analysis due to its greater accessibility and the team's familiarity with the software. Despite its limitations, the student version of SolidWorks proved more functional than the Abaqus student version, especially for handling larger assemblies, thus supporting extensive system-level analysis. Simultaneously, using Abaqus for individual component analysis allowed the team to capitalize on its advanced simulation capabilities for targeted tasks, thereby complementing the broader system analysis conducted in SolidWorks. This bifurcated approach facilitated a more detailed and comparative evaluation of component behaviors and system dynamics, enhancing the overall validation of the design.

This adjusted methodology maintained the project's continuity and adhered to the original timeline despite the computational setbacks. The experience underscored the importance of flexibility in project management and the ability to pivot strategies effectively in response to technical and resource-related challenges. The insights gained from using two different software environments enriched our understanding of the system's performance and the reliability of our design under varied conditions, ultimately contributing to a more robust product.

It is important to clarify that the simulations done in the report are only meant to serve as an indication of the structural integrity of the LPALT. As mentioned earlier, FEA is a subject requiring years of experience to fully master. Even though the group has received guidance and feedback through the process, it is still important to recognize the team's limitations. In most of the simulations done on the LPALT, the peak stresses located were deemed as a result of a lack of degree of freedom. There will be some uncertainty linked to these assumptions which will be a potential source of error. It is also important to mention that the simulations done are set in perfect conditions, which exclude real-life human error and impurities. Therefore, it will be vital for OneSubsea to conduct their own analysis before a potential production of the LPALT. Only this way one can truly conclude whether or not the results are trustworthy.

5.5 Financial viability

Financial viability refers to the ability of a company to sustain its operations financially over the long term by generating sufficient revenue to meet its expenses and debt obligations. This ensures that the company can continue to function, invest in growth, and remain stable during economic fluctuations [17][21].

When assessing the economic viability of the LPALT system, it is essential to evaluate the initial and ongoing costs against the potential revenue it can generate. The initial costs, detailed in *Table 5.1*, represent the direct investment needed to purchase components and assemble the system, establishing the initial capital outlay. These costs range from 100,000 to 150,000 kr, not including manufacturing expenses. Operational expenses for the system cover maintenance, staffing, transportation, and consumables. The system's revenue potential, vital for assessing its viability, stems from direct payments for tests or ancillary services, which help offset costs and demonstrate the system's value.

Depreciation is a significant financial consideration, allocating the initial costs of the LPALT system across its estimated usable life. It was decided to use a lifespan of 10 years to determine the depreciation of the LPALT. Assuming a straight-line depreciation method, the annual depreciation expense would be calculated as $\frac{\text{initial cost}}{10}$. For instance, with an initial cost range from 100,000 to 150,000 kr, the annual depreciation would be between 10,000 to 15,000 kr per year. This method provides a systematic approach to accounting for the asset's cost over its productive life, thus impacting the annual financial performance of the system.

Furthermore, the residual value of the system at the end of its life also plays a crucial role in its overall financial return. Assuming a conservative residual value of 20% of the initial cost, the system could be resold or recycled for 20,000 to 30,000 kr. This residual value is essential as it represents a recovery of some of the initial outlay at the end of the asset's useful life, thereby enhancing the total return on investment. Incorporating such concrete financial figures provides a clearer picture of how depreciation and residual value contribute to the economic sustainability of the LPALT system.

Table 5.1: Cost Analysis of Parts (NOK)

Prod.nr	Name	Manufacturer	Cost per unit	Units	Sum
1	RCH1003 Cylinder	Enerpack	78165.22	1.00	78165.22
2	Hydraulic Hand Pump	Enerpack	11824.85	1.00	11824.85
3	ATEX Hose	Enerpack	9150.44	1.00	9150.44
4	Pressure Gauge	Enerpack	5525.63	1.00	5525.63
5	Load Cell	Vetek	8718.44	1.00	8718.44
6	1000mm Screw	CNCshop	9554.35	1.00	9554.35
7	Screw Nut	CNCshop	3145.56	3.00	9436.68
8	M42 Nut	MiSUMi	274.84	10.9	1099.36
9	M42 Nut	MiSUMi	476.11	4.00	1904.44
10	M56 Bolt	MiSUMi	735.02	2.00	1470.04
11	M56 Bolt	MiSUMi	436.57	4.00	1746.28
12	M30 Bolt	MiSUMi	107.04	2.00	214.08
13	M30 Nut	MiSUMi	134.15	3.00	402.45
14	M30 Bolt	MiSUMi	301.02	1.00	301.02
15	Steel Plate	Norsk Stål	8610.26	0.62	5372.80
				Total	144 886.08

Based on conversations with OneSubsea, the estimated cost of performing a load test without overload would be around 50,000 to 80,000 kr depending on the size of the structure, and prices could increase depending on numerous factors like transportation, what equipment they have available and rental cost. Even though LPALT's initial cost exceeds the standard testing cost, it can not be compared. LPALT is not directly replacing an existing testing practice but creating a new one. As mentioned earlier, even though strongly encouraged by API and DNV standards, performing a load test for structures like the HIPPS is not required. However, customers of OneSubsea often express the desire to perform one anyway. Therefore, the LPALT is creating a new testing opportunity for OneSubsea and its clients.

The goal of LPALT is not to create and generate cash flow but to give customers, for a price, a way to increase safety by testing the pad eyes locally without using cranes and ballast. A comprehensive cost-revenue analysis incorporating these factors provides insights into LPALT's profitability and long-term financial sustainability. This evaluation underscores LPALT's dual contribution to safety and economic viability, highlighting its strategic value to OneSubsea and its clients.

5.6 Risk Assessment

One of the goals for LPALT was to match or surpass the traditional method in terms of risk and safety. To identify the most critical components during the deployment and operation of the LPALT, a failure mode, effects, and criticality analysis (FMECA) was conducted. In addition to the FMECA, the group discussed risk reduction measurements regarding production and safety protocol design.

5.6.1 FMECA Results

The results from the FMECA highlight that regular inspections are the most critical risk reduction measure. The LPALT is designed with easy access to most of its components, making visual inspections essential for maintaining the optimal condition of all parts. Additional routine maintenance checks, such as calibration and monitoring system performance, will help detect errors early. Another risk reduction measure implemented by the group was to prepare for overloads. Although the intended maximum force produced by the LPALT is set to be 80 tonnes, the load cell and other components can handle up to 100 tons—equivalent to the maximum force the hydraulic cylinder can generate. This precaution ensures that different components of the LPALT are not damaged if the cylinder exerts its full power for any reason.

The FMECA identified the hose connecting the hydraulic cylinder and compressor as the most critical component in terms of frequency. A measurement to reduce the failure frequency can be done by utilizing high-quality parts. In terms of severity, none of the components were ranked highly, with the highest ranking being only a 2. Despite this, the group still identified the load cell as the most critical component due to its relatively high-frequency score of 3 and severity ranking of 2. Regular load cell calibration will be the most crucial risk reduction measure.

The results from the FMECA will inherently carry some uncertainty. FMECA is a complex and time-consuming process with several potential sources of error. The most significant source of error is the lack of knowledge among the individuals who conducted the FMECA. Despite group members striving to understand the underlying theory of the components used and seeking guidance to fill knowledge gaps, it cannot substitute for years of experience in the subsea sector. The analysis heavily relies on internal group discussions, which have led to numerous assumptions without any certainty of their accuracy. Parts of the FMECA process may also have been improperly executed due to lack of experience, resulting in incomplete or incorrect findings. Additionally, components were excluded from the analysis based on the assumption that they were unlikely to fail under normal operating conditions or were deemed overly detailed. This might have been an erroneous assumption, potentially causing the report to overlook crucial aspects of the analysis.

5.6.2 Additional Precautions

The FMECA only analyses the impact of failure modes on individual LPALT components when operational. The production of the LPALT also involves inherent risks that must be carefully managed. The LPALT is a heavy-duty piece of equipment, with individual steel components weighing over 100 kg each. There is a substantial risk of injury to personnel or damage to surroundings if these components are mishandled. Because of this, the group discussed measurements to reduce the risk of injuries in addition to the FMECA. The manufacturing team will adhere to industry standards and employ robust techniques to minimize these risks. This includes using cranes and other lifting devices to maneuver parts into position safely, as well as welding clamps and tables to secure them during assembly. Such measures are crucial not only during the initial fabrication of parts but also throughout the assembly of the system, including heavy components like the Interface, Angle Interface, Pad Eye Puller, and the Cylinder Holder.

Emergency procedures must also be in place in case of sudden equipment failure. Standard health and safety procedures must be rigorously followed to prevent accidents. Despite the LPALT being engineered to withstand loads well beyond those expected during normal operations, all personnel must maintain a safe distance from the equipment while it is pressurized. During testing, protective shielding should be installed around the LPALT and the pad eye to guard against the mechanical failure of any part of the LPALT, the subsea structure, or the hydraulic system, which could potentially eject oil under high pressure. No personnel must remain near or approach the LPALT during testing to ensure safety.

5.6.3 Comparison between LPALT and a traditional load test

Previous approaches to load testing relied heavily on adding multiple weights to the frame to achieve the necessary safety factor or physically securing the structure to the ground and then lifting it. These methods posed significant risks, including the potential failure of fastenings and the danger of weights detaching during the process, which could lead to severe accidents or even fatalities.

The LPALT system eliminates the need to accumulate large weights or employ substantial lifting equipment such as large cranes. This enhances safety by reducing the risks of handling heavy weights and simplifies logistics and setup at testing sites.

Furthermore, the design is overbuilt, with components that handle loads over 100 tons. OneSubsea intends never to load past 80 tons, ensuring that no component is subjected to loads exceeding its capacity. This comes in addition to already built-in safety factors in all off-the-shelf components. This preventive measure minimizes the risk of stress-related failures and contributes to the overall durability of the system. By integrating these maintenance and protection strategies, the LPALT is engineered to perform reliably under demanding conditions, in theory making the LPALT capable of competing with a traditional load test in terms of risk and safety.

5.7 Sustainability

The LPALT is predominantly constructed from S355 steel, a material chosen for its mechanical strength and suitability for underwater applications. While sustainability was not the primary criterion, it was considered in the context of end-of-life management. The LPALT predominantly comprises various steel grades, a material known for its recyclability. Thus, despite the significant carbon footprint associated with steel production, the end-of-life recyclability of the LPALT contributes positively to its environmental impact. As the operational life of the LPALT concludes, we are committed to adhering to responsible and sustainable disposal practices. It is anticipated that most of the steel components will be recycled following industry standards such as ISO14001 [6]. This ensures that these materials re-enter the manufacturing cycle, reducing the need for raw material extraction and minimizing environmental impact.

The oil industry has long been a primary source of income in Norway. Due to this, there has been significant support for the sector regarding technological expertise. In recent years, a considerable debate has been about transitioning from oil to prioritizing more environmentally friendly industries that generate fewer emissions. The resources dedicated to developing technology for the oil industry will not be wasted, even as Norway gradually phases out oil production. This expertise and technology can be redirected towards new sources of revenue for the country, such as water and wind energy. Although the project focuses on subsea structures utilized in the oil industry, the LPALT has broader applications. It can be employed in any industry that requires the lifting and verifying pad eyes on heavy structures.

5.8 Innovation

At the beginning of the project, OneSubsea deliberately refrained from sharing their vision for the new load test, encouraging the team to think creatively and generate innovative ideas. Although the team was given free rein during the initial development phase, numerous constraints were involved in the design. The jig must accommodate a structure with pre-established parameters such as the pad eye, frame size, and shape. It is wise not to unnecessarily overcomplicate the design in a design process with fixed parameters. Reinventing the wheel is rarely required. To adhere to this principle, the group reviewed previous pad eye tester designs, considering their strengths and limitations. Working closely with the specific requirements provided by the client, the group identified an innovative approach early in the process. The goal was to develop a solution that could perform a load test at various angles, not just vertically. The LPALT system allows the load test to be conducted at multiple pre-set angles with varying loads. This capability distinguishes the group's design from prior models. Most pad eye testers operate only in a vertical orientation. This versatility is further enhanced by the ability of the design to interchange the interface plate to accommodate different geometries and types of pipes, making it highly adaptable to various operational needs.

Another feature that distinguishes the LPALT from previous solutions is how it is designed for practicality. It can be easily assembled and disassembled, allowing for high flexibility in use.

Consumable components such as nuts and bolts are designed for quick replacement, promoting maintenance ease and operational continuity. The structure is designed around readily available shelf items and adaptable frameworks, enabling quick substitutions and modifications based on specific project requirements or frame types. This design philosophy ensures that LPALT can be easily tailored to meet a wide range of testing scenarios, making it an indispensable tool in modern testing environments.

5.9 Ethical and environmental challenges

The construction of LPALT primarily utilizes S355 steel, a material chosen for its strength and durability, which is essential for the structural and safety requirements of subsea equipment. However, steel production is resource-intensive, with significant energy consumption and carbon emissions posing substantial environmental challenges. These aspects are particularly relevant to SDG 12, Responsible Consumption and Production, which encourages resource and energy efficiency, sustainable infrastructure, providing access to essential services, green and decent jobs, and a better quality of life.

It is also essential to manage the testing sites with minimal local environmental disruption, such as avoiding soil contamination or disturbance to local wildlife. Implementing comprehensive impact assessments and adopting measures to minimize ecological disruptions are crucial. These practices align with SDG 15, Life on Land, which focuses on managing forests sustainably, combating desertification, halting and reversing land degradation, and halting biodiversity loss [25].

Recognizing the importance of steel's recyclability helps mitigate some environmental concerns associated with its production. Although a Life Cycle Assessment (LCA) was not performed for LPALT, the recognition of LCA's value underscores the importance of such evaluations in future projects. This consideration is vital for SDG 13, Climate Action, which calls for urgent action to combat climate change and its impacts by regulating emissions and promoting developments in renewable energy.

Chapter 6

Conclusion

Developing the Local Pad-Eye Angular Load Tester (LPALT) represents a significant advancement in subsea testing equipment. This project addressed OneSubsea's need for a practical, efficient, and cost-effective solution for localized load testing of pad eyes on subsea structures. Through rigorous design, analysis, and theoretical validation, the LPALT has shown the potential to enhance safety, performance, and operational efficiency.

The LPALT project has achieved several key milestones. Its innovative design features a compact and transportable form, allowing for efficient onsite testing. By utilizing off-the-shelf components, the LPALT minimizes custom fabrication needs, reducing costs and complexity in assembly. In terms of safety, traditional load testing methods often involve lifting entire structures with significant ballast, posing various risks. The LPALT reduces these risks by applying localized loads directly to pad eyes and their welds, thereby enhancing safety for operators and reducing the likelihood of structural failures during testing.

Regarding performance and cost-effectiveness, the LPALT can test loads up to 100 metric tons, which is 20% above OneSubsea's highest load criteria as of today. This capability provides a robust and reliable means of validating the structural integrity of subsea pad eyes. Additionally, the design's reliance on standard components ensures a cost-effective solution compared to more comprehensive testing systems. From an environmental perspective, the LPALT is constructed primarily from recyclable S355 steel, aligning with sustainable engineering practices. The design and operational protocols aim to minimize environmental impact, supporting responsible resource consumption and production.

Despite its many advantages, the LPALT has some limitations. The LPALT is designed for localized testing of pad eyes and their connecting welds, and it cannot replace complete system integration tests or comprehensive Finite Element Analysis (FEA) required for validating entire subsea structures. Therefore, it should be used in conjunction with other testing methods, specifically comprehensive FEM analysis, to ensure the overall structural integrity of subsea installations. The LPALT has not yet been physically produced or empirically tested, so all performance evaluations remain theoretical. Actual physical testing will be necessary to confirm the LPALT's capabilities and address any unforeseen issues during practical use.

To further develop and validate the LPALT, several steps should be taken in future work. Manufacturing a prototype of the LPALT will be essential for empirical validation, and detailed manufacturing plans should be developed to optimize the assembly process and ensure component reliability. Comprehensive physical testing should be conducted to validate theoretical performance predictions, including load testing under various conditions to ensure robustness and reliability in real-world scenarios. Based on empirical testing results, further design refinements, such as modifications to enhance durability, ease of use, and adaptability to different subsea structures, may be necessary. Considering potential modular redesigns to facilitate easier assembly and reduce reliance on welding could improve manufacturability and maintenance. Additionally, a more detailed analysis of the environmental impact of the LPALT's manufacturing and operational processes should be conducted to enhance its sustainability profile.

In conclusion, the LPALT project successfully integrates theoretical and practical engineering approaches to solve a real-world problem in subsea operations. By providing OneSubsea with a valuable tool for enhancing the safety and efficiency of their structural assessments, the LPALT represents a significant step forward in subsea engineering practices. This innovative solution addresses current industry challenges and sets the stage for future advancements in subsea load testing technology.

Bibliography

[1] Toby Abbey. *Dealing with Stress Concentrations and Singularities.* Last updated: 01.05.2017. 2017. URL: https://www.digitalengineering247.com/article/dealing-stress-concentrations-singularities/ (visited on 10th May 2024).

[2] Toby Abbey. *Stress in FEA: Part 3.* Last updated: 01.07.2016. 2016. URL: https://www.digitalengineering247.com/article/stress-in-fea-part-3/ (visited on 10th May 2024).

[3] Eget arbeid. *Upublisert semesteroppgave i MAST2003- FMECA av landeveissykkel.* NTNU, 2023.

[4] Golden Asia. *What Are the Advantages and Disadvantages of Hydraulic Cylinders?* 2023. URL: https://www.airhydraulic.com/news/What-Are-The-Advantages-And-Disadvantages-Of-Hydraulic-Cylinders.html (visited on 4th Apr. 2024).

[5] Parker Dahl. *Modeling Welds for Finite Element Analysis (FEA).* Last updated: 19.07.2018. 2018. URL: https://apolloedge.com/2018/07/19/model-welds-for-finite-element-analysis-fea/ (visited on 10th May 2024).

[6] DNV. *ISO 14001 - miljøledelse.* URL: https://www.dnv.no/services/iso-14001-miljoledelse-33649?gad_source=1&gclid=CjwKCAjw5v2wBhBrEiwAXDDoJWw8i%20VtBrUvgWW0dYhYxeULGod_Bu_jbZNft7pLWKuoiKoaRwwbp6%20hoCwyEQAvD_BwE (visited on 17th Apr. 2024).

[7] Enerpac. *RCH1003, 103.1 ton Capacity, 3.00 in Stroke, Single-Acting, Hollow Plunger Hydraulic Cylinder.* URL: https://www.enerpac.com/en-us/rch-series-cylinders/hollow-plunger-cylinder/RCH1003 (visited on 3rd Apr. 2024).

[8] Enerpac. *Types of Hydraulic Cylinders.* 2022. URL: https://blog.enerpac.com/types-of-hydraulic-cylinders/ (visited on 3rd Apr. 2024).

[9] Esubsea. *Subsea Structures.* URL: https://www.esubsea.com/subsea-structures/ (visited on 4th Apr. 2024).

[10] esubsea. *Padeye og Løftepunkter.* URL: https://www.esubsea.no/padeye-og-loftepunkter/ (visited on 4th Apr. 2024).

[11] FN. *Bærekraftig utvikling.* Last updated: 28.06.2023. 2023. URL: https://fn.no/tema/baerekraftig-utvikling-fattigdom-og-befolkning/baerekraftig-utvikling (visited on 9th Apr. 2024).

[12] Nord-Lock Groupe. *What happens to preload and clamp force when external loads are applied?* URL: https://www.nord-lock.com/insights/bolting-tips/2021/what-happens-to-preload-and-clamp-force-when-external-loads-are-applied/ (visited on 10th May 2024).

[13] Hydroline. *How does a hydraulic cylinder work?* URL: https://hydroline.fi/blogs/how-does-a-hydraulic-cylinder-work/ (visited on 4th Apr. 2024).

[14] Jeff Kerns. *What's the Difference Between Bearing, Shear, and Tear-Out Stress?* 2016. URL: https://www.machinedesign.com/fastening-joining/article/21834800/whats-the-difference-between-bearing-shear-and-tear-out-stress (visited on 6th May 2024).

[15] Masteel. *S355G10+M/+N.* Last updated: 01.31.2017. 2017. URL: https://masteel.co.uk/s355g10mn/ (visited on 5th Feb. 2024).

[16] Kåre Olerud, Andreas Tjernshaugen and Gisle Andersen. *Bærekraftig utvikling.* Last updated: 13.01.2023. 2023. URL: https://snl.no/b%C3%A6rekraftig_utvikling (visited on 9th Apr. 2024).

[17] onEntrepreneur. *What is financial viability? Meaning, Importance and Assessment.* Last updated: 25.08.2023. URL: https://onentrepreneur.com/financial-viability/ (visited on 17th Apr. 2024).

[18] Green Pin. *Green Pin® G4163 Bow Shackle BN.* URL: https://www.greenpin.com/en/product/green-pinr-bow-shackle-bn (visited on 6th May 2024).

[19] Marvin Rausand. *Chapter 3 FMECA.* URL: https://www.ntnu.edu/documents/624876/1277590549/chapt03-fmeca.pdf/ecf0c289-bc19-492f-88ef-6a197ad4a9f1 (visited on 15th Apr. 2024).

[20] Ellen Jenny Ravndal and Jo Halvard Halleraker. *FNs bærekraftsmål.* Last updated: 13.01.2023. 2023. URL: https://snl.no/FNs_b%C3%A6rekraftsm%C3%A5l (visited on 9th Apr. 2024).

[21] Hydropower Sustainability Secretariat. *Financial Viability.* URL: https://www.hydrosustainability.org/financial-viability (visited on 17th Apr. 2024).

[22] Henrik Sönnerlind. *Singularities in Finite Element Models: Dealing with Red Spots.* Last updated: 03.06.2015. 2015. URL: https://www.comsol.com/blogs/singularities-in-finite-element-models-dealing-with-red-spots/ (visited on 10th May 2024).

[23] StudySmarter. *Bearing Stress.* URL: https://www.studysmarter.co.uk/explanations/engineering/solid-mechanics/bearing-stress/ (visited on 6th May 2024).

[24] Suraj. *12.9 Carbon Steel Bolts.* URL: https://www.surajmetal.com/high-tensile-bolts-12-9-grade.html (visited on 6th May 2024).

[25] UN. *Sustainable development goals.* 2024. URL: https://www.undp.org/sustainable-development-goals (visited on 9th Apr. 2024).

[26] Vetek. *Load cell thru hole 100 ton.* URL: https://www.vetek.com/en/article/load-cell-thru-hole-100-tonnes-ar-100ton (visited on 9th Apr. 2024).

[27] Wikipedia. *Load cell.* URL: https://en.wikipedia.org/wiki/Load_cell (visited on 9th Apr. 2024).

NTNU

Norwegian University of
Science and Technology