Herborg Sørli

Examination of the Hot Spot Method and the Notch Stress Method for a Bracket Geometry

Master's thesis in Marine Structures
Supervisor: Sigmund Kyrre Ås
Co-supervisor: Arve Flesche
June 2023

Master's thesis

NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Marine Technology

NTNU
Norwegian University of
Science and Technology

Herborg Sørli

Examination of the Hot Spot Method and the Notch Stress Method for a Bracket Geometry

Master's thesis in Marine Structures
Supervisor: Sigmund Kyrre Ås
Co-supervisor: Arve Flesche
June 2023

Norwegian University of Science and Technology
Faculty of Engineering
Department of Marine Technology

NTNU
Norwegian University of
Science and Technology

Preface

This master's thesis is the result of the work conducted during the spring of 2023 at the Department of Marine Technology at the Norwegian University of Science and Technology (NTNU) in Trondheim. The thesis was written in collaboration with Aker Solutions. The focus in this thesis was to study the fatigue and fracture behavior of a specific bracket geometry manufactured by Aker Solutions. By closely examining the hot spot method and the notch stress method, this research aimed to provide a comprehensive understanding of the structural integrity assessment techniques applied to this particular bracket geometry. The main task of this work was to study the structure in Abaqus, using the finite element method.

This master's thesis builds on the thesis written by Mosbron (Mosbron 2022) last year, that also was carried out in corporation with Aker Solutions.

Acknowledgement

I would like to thank my supervisor, associate professor Sigmund Kyrre Ås, for regular guidance with this thesis. His expertise in the field of marine structures proved crucial in shaping the direction and quality of this work. His insightful feedback, encouragement, and scholarly mentorship have been vital in expanding my knowledge and refining my analytical skills.

Also co-supervisor Arve Flesche, senior engineer in Aker Solutions, has been of great help during the semester. I would also like to extend my sincere appreciation to Aker Solutions for proposing this topic and for their collaboration throughout the thesis. Their real-world expertise and involvement not only provided valuable insights into the practical aspects of the bracket geometry but also fostered a deeper understanding of the industry's challenges and requirements.

Furthermore, I am grateful to the Department of Marine Technology at NTNU for providing an enriching academic environment and access to relevant resources and facilities. Finally I would like to thank my fellow students at the office for a great year.

Abstract

This master's thesis focuses on the fatigue and fracture analysis of a bracket geometry provided by Aker Solutions. The assessment of fatigue in welded structures commonly employs three different approaches: the nominal stress method, the structural hot spot method, and the effective notch stress method. The applicability and differences among these approaches have been thoroughly discussed.

In this study, the hot spot stress at the weld toe of the bracket geometry was calculated using three different approaches. Remarkably, all three approaches yielded similar stress results. This indicates that the calculations are relatively robust and not heavily dependent on the specific approach used. However, it was observed that the approach employing extrapolation points at 0.5t and 1.5t (t being the thickness of the bracket) provided stress values closer to the D design S-N curve, indicating a potentially more conservative estimate of fatigue life. This suggests that increasing the extraction distance from the weld toe captures a broader stress field, resulting in a more representative hot spot stress value. Conversely, closer extraction points may be more influenced by stress concentrations, leading to higher stress values.

Furthermore, the notch stress, calculated at the weld toe, was found to be 2.4 times larger than the hot spot stress. This considerable discrepancy highlights a significant stress concentration caused by the geometric characteristics of the weld notch. The presence of such a high stress concentration increases the likelihood of localized stress concentrations, thereby promoting the initiation and propagation of fatigue cracks.

Sammendrag

Denne masteroppgaven fokuserer på utmatting- og bruddanalyse av en brakettgeometri levert av Aker Solutions. I vurderingen av utmatting i sveisede strukturer benytter man vanligvis tre forskjellige tilnærminger: Den nominelle spenningsmetoden, hot-spot metoden og notch stress metoden. Anvendeligheten og forskjellene mellom disse tilnærmingene har blitt grundig diskutert.

I denne studien ble hot spot-spenningen ved sveisetåen til brakettens geometri beregnet ved å bruke tre forskjellige tilnærminger. Bemerkelsesverdig nok ga alle tre tilnærmingene lignende spenningsresultater. Dette indikerer at beregningene er relativt robuste og ikke i stor grad avhengig av den spesifikke tilnærmingen som brukes. Imidlertid ble det observert at tilnærmingen med ekstrapolasjonspunkter ved 0.5t og 1.5t (t er tykkelsen på braketten) ga spenningsverdier nærmere D design S-N-kurven, noe som indikerer et potensielt mer konservativt estimat av utmattingslevetiden. Dette antyder at man ved å øke avstanden fra sveisetåen fanger opp et bredere spenningsfelt, noe som resulterer i en mer representativ hot spot-spenningsverdi. Omvendt kan nærmere ekstraksjonspunkter være mer påvirket av toppspenningsverdier, noe som fører til høyere spenningsresultater.

Videre ble notch stress beregnet ved sveisetåen funnet til å være 2.4 ganger større enn hot-spot spenningen. Dette avviket fremhever en betydelig spenningskonsentrasjon forårsaket av de geometriske egenskapene til sveisesnittet. Tilstedeværelsen av en så høy spenningskonsentrasjon øker sannsynligheten for lokaliserte spenningskonsentrasjoner, og fremmer derved initiering og forplantning av utmattelsessprekker.

Table of Contents

List of Figures

List of Tables

1 Introduction

Fatigue is one of the main causes of failures in mechanical and structural systems. Offshore installations, in particular, are susceptible to fatigue failure due to their exposure to the combination of wind loads, wave loads and currents. Marine structures are often subjected to significant dynamic wave loading during installation. Flare towers and bridges between platforms are subjected to wind loading. In these types of structures fatigue design is important, and it is necessary to assess the fatigue life of a component. It is crucial to design the different structural components such that they can withstand the effects of cyclic loading, because of the loads they are subjected to during their lifetime. There can be fatal consequences if a structural component fails due to fatigue and leads to a whole structure collapsing. Thus, it is important to use educated and experienced engineers in the development of new structures that should meet the long-term requirements for safe operation (Lotsberg 2016).

When designing a structure, such as a floating platform for drilling operations, the primary goals are to ensure its functionality, safety, and cost-effectiveness. The layout of the platform is determined based on the need for stability, a high deckload capacity, limited motions, mobility, and overall strength. To achieve the required structural strength, additional elements like bracings or other members are often introduced. The specific arrangement of these elements, such as the use of brackets, plate thicknesses, and material selection, is based on considerations of strength, fabrication feasibility, inspection requirements, and maintenance needs. Strength criteria play a crucial role in the design process. This includes assessing failure modes such as rupture due to excessive loads and fatigue failure of individual structural components or the progressive failure of the entire system. By considering these strength criteria, designers can ensure the structural integrity and longevity of the offshore structure. Overall, the design process involves a careful balance between meeting the functional requirements of the structure, optimizing its strength and durability, and considering practical aspects such as fabrication, inspection, and maintenance (Almar-Næss et al. 1985).

Brackets are typically joined by welding. Fatigue of welded joints is mainly a crack growth phenomenon. Fracture mechanics principles are commonly employed to analyze and calculate the crack growth, providing assessments of fatigue lives. However, performing a direct design calculation for a specific weld using fracture mechanics is not feasible for several reasons. One of the main challenges is the presence of numerous uncertainties in the calculation procedure, particularly concerning the initial stages of crack growth. Factors such as the size of initial defects, crack growth for short cracks, and threshold effects introduce uncertainties that make absolute calculations of endurance unreliable. Therefore, it is difficult to trust the results of absolute calculations of fatigue endurance on their own. However, in comparative studies, where multiple designs or scenarios are compared, the initiation problem of cracks can be eliminated or minimized. By focusing on relative assessments rather than absolute values, engineers can still gain valuable insights into the comparative fatigue performance of different welded joints or design variations (Almar-Næss et al. 1985).

1.1 Objectives

In this master's thesis the objectives are to evaluate the hot spot method and the notch stress method for a bracket geometry. Aker Solutions has provided a test specimen of a bracket geometry that is welded onto an 8 mm thick plate. Due to delays in relation with moving the lab from Tyholt to Tiller due to demolition, it was not possible to carry out lab tests this spring. But Mosbron, a master's student last year who wrote his thesis on the same subject, carried out a lab experiment last spring which will be used for comparison (Mosbron 2022). He carried out a fatigue test where the specimen was subjected to cyclic loading at different stress ranges to achieve failure, which gave a stress range and cycles until failure. Fatigue testing is an important procedure used by engineers to help predict the durability of a part or component under its operating conditions. This is often done in lab-experiments where manufactured specimens gets tested. Fatigue tests are performed to measure the reduction in stiffness and strength of materials under repeated loading and to determine the total number of load cycles to failure.

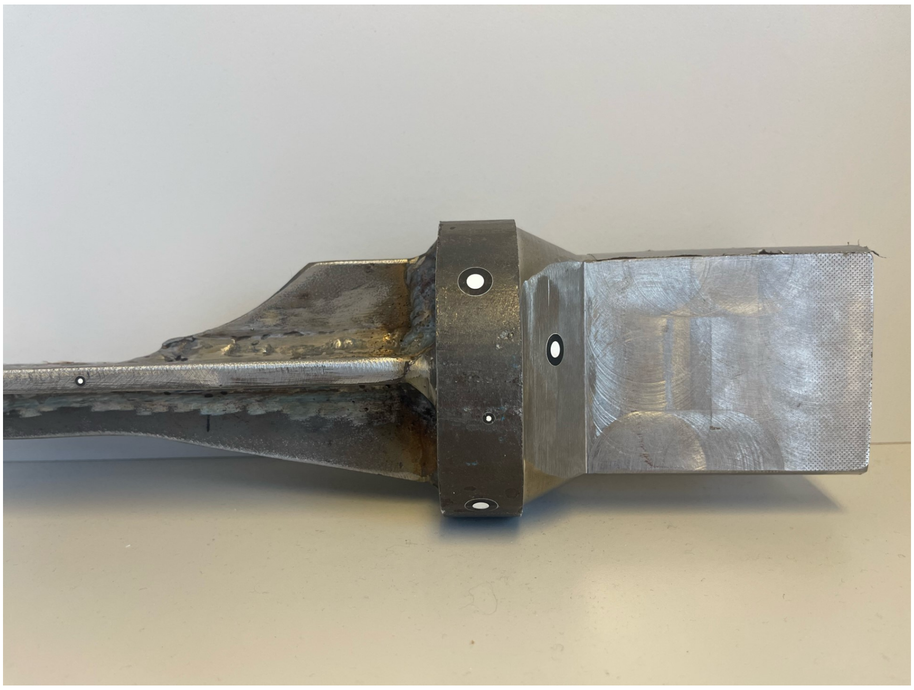


Figure 1: The bracket in the test specimen provided by Aker Solutions

Finite element analyses will be carried out in Abaqus for fatigue assessment. A global model will be used for calculation of the hot spot stress that represents the stress due to the weld toe. The results will be plotted into a hot spot stress S-N curve for calculation of fatigue damage. Also a submodel that accounts for the notch stress will be used for stress calculations. With a refined mesh study at the weld notch, the results converges towards a peak stress value. By combining the results from Abaqus analyses and lab experiments, it is possible to validate and refine the numerical models, ensure their accuracy, and gain confidence in the predictions made. The experimental data can also help in calibrating the numerical models for improved accuracy in future simulations. The synergy between numerical analysis and physical testing provides a more comprehensive and robust understanding of the bracket's fatigue behavior, leading to more reliable conclusions and recommendations for design optimization and maintenance planning.

1.2 Outline

In Section 2 the necessary background theory is presented. This includes a subsection about the thickness effect. The plate thickness of the test specimen is 8 mm, but in reality the plates can have a thickness of $60 - 100$ mm. Aker Solutions uses plates with a thickness up to 80 mm. Most of the background theory is based on the specialisation project written last autumn in preparation for this thesis. Section 3 presents the different fatigue assessment methods, and it is described how they are used in the calculations in this thesis. The finite element method and the modeling in Abaqus is explained in Section 4. The lab experiment and how a fatigue test is performed is described in Section 5. The results and a discussion of them is presented in Section 6 and Section 7. Lastly, a conclusion and recommendations for further work are given in Section 8 and Section 9.

2 Background theory

2.1 Fatigue failure

Fatigue can be described as a failure that occurs due to cyclic loading of damage in a material. The important note about fatigue is that the load is not large enough to cause immediate failure. The failure occurs after a certain number of load fluctuations have happened. The most important load effect parameter in fatigue is the fluctuating component of stress or strain. This is commonly referred to as stress or strain range, and is defined as the difference between a load peak and the subsequent valley. This range of stress is more important to study in fatigue cases, than the mean or peak levels of loading (Almar-Næss et al. 1985).

Frequently repeated loading of a material provokes plastic deformation, which decreases future ability to withstand stress and crack initiation. The cracks are initially microscopic, but when they propagate they eventually lead to a final fracture. Fatigue is the theory about the initiation and propagation of cracking. However, the stage involving stable propagation of cracking can include a major part of the total life (Radaj 1990).

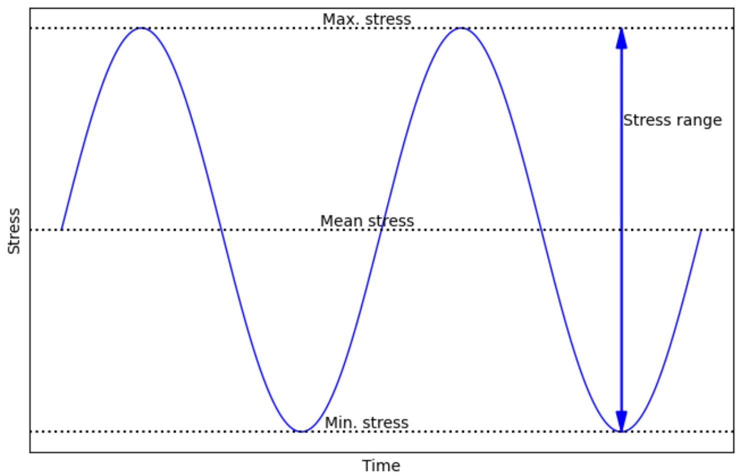


Figure 2: Fatigue load

The stress amplitude at which failure occurs for a given number of cycles is the fatigue strength. The stress range $\Delta S = S_{max} - S_{min}$ is defined in Figure 2. The stress amplitude and stress range are important concepts related to the endurance of a material, because it describes the cyclic loading conditions that a material can sustain before failure occurs (Q. Bai and Y. Bai 2014).

2.2 Crack growth

From the initial state of the material to final fracture, the fatigue process goes through three stages. These are described as:

- Stage I: Initiation or crack nucleation
- Stage II: Crack growth
- Stage III: Final fracture

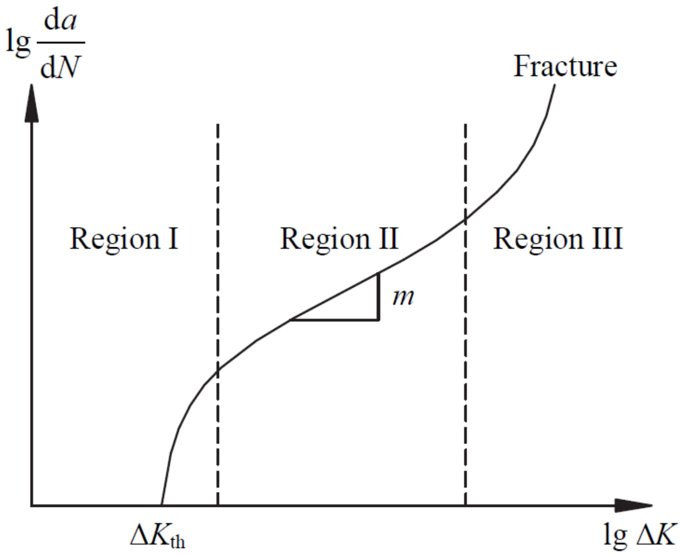


Figure 3: A fatigue crack growth curve that shows the different regions/stages of the fatigue process (Rege and Lemu 2017)

Stage I is a surface phenomenon. Fatigue cracks usually never initiates in the interior of a material. Stage II represents the phase where the crack will change mode and the growth direction will become perpendicular to the largest principal cyclic stress. The driving force in this stage is the maximum principal stress, not the shear stress. In stage III the crack growth is exhibiting a rapidly increasing crack growth rate, that either results in a ductile or brittle final fracture (Almar-Næss et al. 1985). Brittle fracture means a fracture of the material without a plastic deformation or with very small plastic deformation before the fracture. Ductile fracture is seen in materials that have extensive plastic deformation or necking that occurs before the material reaches a final fracture.

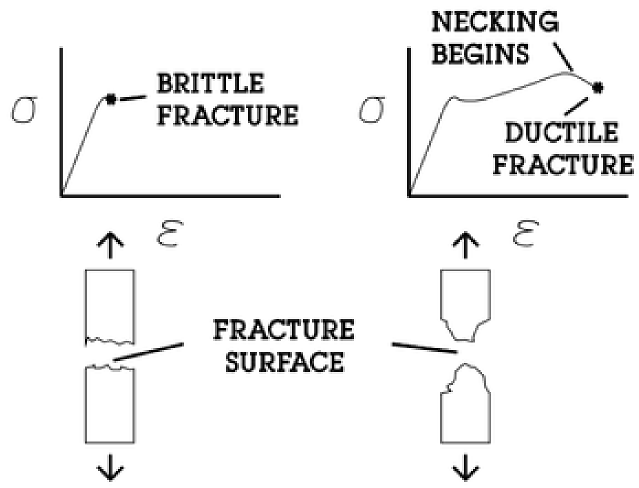


Figure 4: Brittle versus ductile fracture in a material (Dey 2021)

How much of the fatigue life that is spent in the two first stages, depends on the material, the geometry of the detail that is studied and the loading. The period of the first stage, the initiation of the crack in the base material without significant notches, is relatively long in comparison with the propagation period (Lotsberg 2016).

2.3 Fatigue in welded structures

Fatigue is a weakest link process in the sense that the fatigue crack will initiate at the location along the weld where local stress, weld geometry, initial defects and material properties combine to give optimum conditions for crack nucleation and growth (Berge 1985). Welded joints can represent the weakest part of structures and are susceptible to failure. The quality and strength of welded joints depend on factors such as the design, dimensioning and welding processes undertaken during fabrication. It is these key factors which ultimately decrease the life of the structure and eventually lead to the formation of a fracture and subsequent structural failure. The correct design of welded joints requires factors, such as the loading of the joints, the steel type, welding process and the geometry of the structure to be taken into account (Thomas 2018).

In welded structures fatigue cracking from weld toes into the base material is a frequent failure mode. The fatigue crack is initiated at small defects or undercuts at the weld toe where the stress is highest due to the weld notch geometry (DNV-GL 2014). Welding defects, particularly poor fusion, undercuts, root notches, slag inclusions and weld cracks considerably reduce strength, the more so in joints free of other discontinuities and notches. Thus, weld defects are to be avoided, especially in high quality welds and it must be shown that they are not present by non-destructive testing (Radaj 1990).

The crack growth stage is dominating the fatigue life of welded joints. The crack initiation stage is not important in welded joints, because weld defects are always found. Any initiation of a crack growing from these defects is considered to be insignificant in relation to the Stage II crack growth period (Almar-Næss et al. 1985).

2.4 S-N curve for welded joints

An S-N curve gives a graphical presentation of the dependence of fatigue life (N) on fatigue strength (S). The fatigue design is based on use of S-N curves, which are obtained from fatigue tests (DNV-GL 2014). They give the number of cycles to failure, N, when a material is repeatedly cycled through a given stress range, S. The purpose of plotting fatigue results against an S-N curve is to determine the fatigue life of the component under the applied cyclic loading.

The basic design S-N curve is associated with 97.7% probability of survival, and is given as:

$$log\ N = log\ \overline{a} - m\ log\ \Delta S \tag{1}$$

This curve assumes that the test data is normally distributed on a log N scale. Log a is a slope of two standard deviations below the mean:

$$log\ \overline{a} = log\ a - 2s_{log\ N}, \tag{2}$$

and $s_{log\ N}$ is the standard deviation of log N.

For practical use of S-N curves, welded joints are divided into several classes. Each class has a corresponding design S-N curve. There are three main features of the welded joint that is considered when selecting a proper weld class (Lotsberg 2016):

- The geometry of the detail
- The direction of the loading
- The method of fabrication of the detail

The S-N curve is commonly divided into three regions; low-cycle, high-cycle and fatigue limit. The low-cycle range lays below approximately $5 \cdot 10^3$ cycles. The high-cycle range of fatigue is the range which is applicable for most offshore design problems. The fatigue limit region is the region where fatigue data may be fitted to a log-linear curve. This is consistent with a fracture mechanics analysis using the Paris-Erdogan crack growth law (Almar-Næss et al. 1985):

$$\frac{da}{dN} = C(\Delta K)^m \tag{3}$$

Here, $\frac{da}{dN}$ is the fatigue crack growth for a load cycle N. The material coefficients C and m are obtained experimentally and also depend on stress ratio, temperature, frequency and environment.

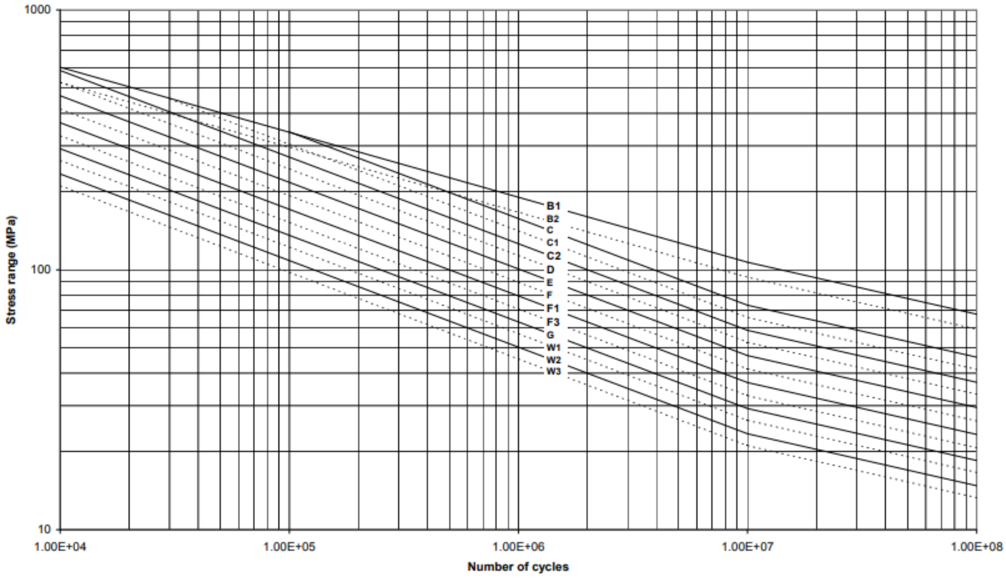


Figure 5: S-N curve in air (DNV-GL 2014)

Figure 5 shows S-N curves in air. When performing a hot spot stress analysis using the finite element method in Abaqus, it is generally recommended to compare the hot spot stress results with a D design S-N curve (DNV-GL 2014). If the stress level is below the corresponding fatigue strength coefficient specified by the D design S-N curve, the structure is considered to have an acceptable fatigue life within the given design requirements.

2.5 Thickness effect

In 1985 Stig Berge investigated the effect of plate thickness on the fatigue strength of transverse fillet welds in axial loading. He found that the thickness effect was found to follow a power law with decrease in fatigue strength by 40% when the plate thickness was increased from 12.5 to 80 mm. Increasing the size of a given type of specimen while maintaining all the other parameters will cause a decrease in fatigue strength (Berge 1985). Berge used the following assumptions for describing the thickness effect in weld toe fatigue failures:

1. Welded joints of the same type in various plate thicknesses are geometrically similar (typical for load-carrying welds)

2. Initial conditions of fatigue crack growth are independent of plate thickness (a_i = constant)

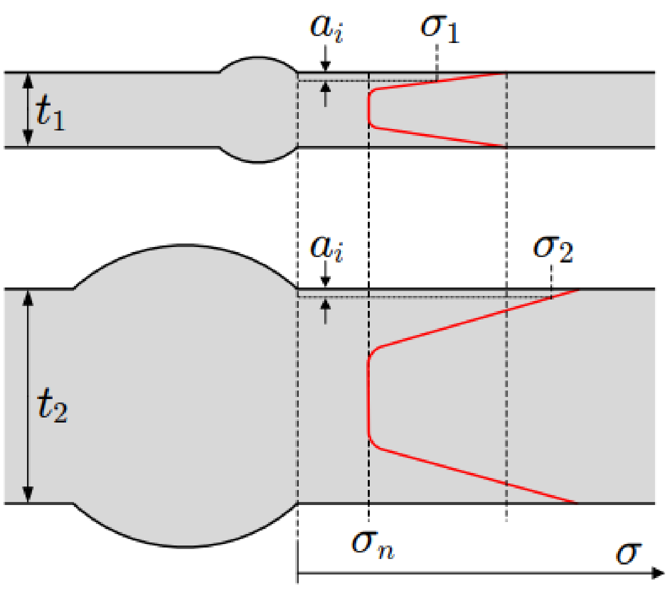


Figure 6: Illustration describing the geometric effect of thickness in fatigue failures developing from the weld toe (Berge 1985)

When considering these assumptions, the effect of plate thickness is understood from a simple geometric model as seen in Figure 6. The stress distribution across a load-carrying plate is affected by its thickness, with thinner plates having a steeper stress gradient near cracks. This means that the initial crack in a thinner plate will experience a smaller stress than the initial crack of the same length in a thicker plate. As a result, the initial crack growth rate in the thinner plate will be smaller, which can overcompensate for the difference in crack length and cause the thinner plate to have a longer fatigue life.

The thickness effect is thus caused by the following parameters (Berge 1985):

- The magnitude of the stress concentration at the weld toe, which is mainly determined by the local weld geometry
- The gradient of the stress in the plane of crack growth, which is mainly determined by the plate thickness
- The number of cycles in crack growth through the region of a steep stress gradient, relative to the total number of cycles to failure. This is mainly determined by the size of the initial crack and the crack ellipticity

In fatigue design, S-N curves are used to determine the fatigue life of a material based on its stress range. The reference plate thickness for S-N curves typically ranges from 16 to 25 mm, and the stress range is typically determined through calculations. To find the fatigue life from an S-N curve, the calculated stress range must be modified according to Equation 4, that takes into account the reference plate thickness. This allows to account for the thickness of the material in fatigue analyses and make more accurate predictions about the fatigue life. The formula is commonly used in fatigue analysis, especially in the hot spot stress method. The exponent k was set equal to 0.25 by Gurney. This is based on the notch effect of a typical weld bead (Berge and Ås 2017).

$$\Delta S_{ref} = \Delta S \left(\frac{t}{t_{ref}} \right)^{k} \tag{4}$$

The thickness effect is assumed to be accounted for in the calculated notch stress, such that a further reduction of fatigue strength for larger thicknesses is not required when using this method (Lotsberg 2016). The S-N curves for the notch stress method are presented in the standard format:

$$log\ N = log\ a_d - m\ log\ S \tag{5}$$

where a_d is the intercept of the design S-N curve with the log N axis, and m is the negative inverse slope of the S-N curve.

3 Fatigue Assessment Methods

Fatigue assessment methods are techniques used to evaluate the structural integrity and durability of components and structures subjected to cyclic loading conditions. They aim to predict the fatigue life and potential failure modes under repeated or fluctuating stresses.

3.1 Nominal stress approach

The most traditional method for predicting the fatigue life of a structural component is the nominal stress approach. Nominal stress is the stress calculated in the sectional area under consideration, disregarding the local stress raising effects of the welded joint. The nominal stress may vary over the section under consideration. For example at a beam-like component, the modified nominal stress and the variation over the section can be calculated using simple beam theory. Here, the effect of a welded detail is ignored (Hobbacher 2016).

Design rules for welded joints have been developed since welding became the predominant method of producing steel structures. Different geometries of welded specimens have been tested in laboratory to establish S-N curves for each type of joint. These were based on nominal stress as derived from the applied load in the laboratory. The nominal stress approach consists of the following steps (Berge and Ås 2017):

1. Identify critical welds in the structure and assign a weld class
2. Assign a nominal stress range or spectrum for each joint
3. Multiply the nominal stress with stress concentration factors if applicable
4. Find the fatigue life or damage sum for each joint using the SN curve determined in step 1

The nominal stress method can however be difficult to apply in real spot-welded structures because it is hard to determine the nominal stress in complicated structures. In this approach the local stress concentration due to the spot weld geometry and a circumferential notch is not considered. Therefore this method has less accuracy, and lends itself to simple calculations (Khanna and Long 2010).

3.2 Hot spot method

Because the stress concentration caused by a change in structural geometry is considered, the hot spot method is more accurate than the nominal stress approach.

The hot spot method is often used when there is no clearly defined nominal stress due to complex geometries, or where the structural discontinuity is not comparable to a classified structural detail.

According to DNV-GL in their *Recommended practice* when it comes to fatigue design of offshore structures, a hot spot is where a fatigue crack may initiate due to the combined effect of structural stress fluctuation and the weld geometry or a similar notch. The hot spot stress is the value of structural stress on the surface at the hot spot (DNV-GL 2014). The hot spot stress can also be called geometric stress or structural stress.

The hot spot stress approach takes as its starting point the idea that fatigue design stresses can be extrapolated directly at the weld toe from a linear stress distribution obtained by interpolating the stress states at two or three superficial reference points. These reference stress states are usually determined either by using strain gauges or by solving linear-elastic FE models with mesh size set according to specific geometrical rules (Zamzami and Susmel 2018). The reference point closest to the weld toe must be chosen to avoid any influence of the notch due to the weld itself. This will lead to a non-linear stress peak (Hobbacher 2016).

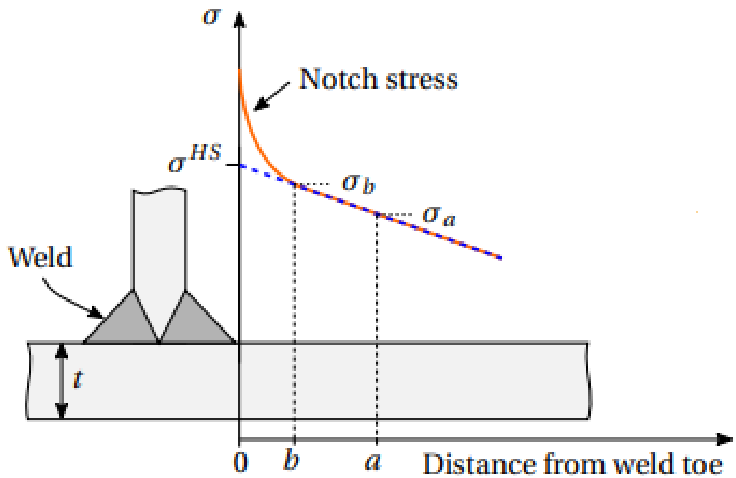


Figure 7: Hot spot stress found as an extrapolation of stresses outside the notch zone, between point a and b (Berge and Ås 2017)

3.2.1 Hot spot stress by FEA

In general, analysis of structural discontinuities and details to obtain the structural hot spot stress is not possible using analytical methods. Parametric formulae are rarely available. Thus, finite element analysis (FEA) is generally applied (Hobbacher 2016).

The extent of the model has to be chosen such that effects due to the boundaries and load application on the structural detail being considered are sufficiently small. Then reasonable boundary conditions can be formulated (Lotsberg 2016). According to DNV-GL it is important to have a continuous and not too steep change in the density of the element mesh in the areas where the hot spot stresses are to be calculated (DNV-GL 2014).

In plated structures, three types of hot spots at weld toes can be identified:

a) At the weld toe on the plate surface at an ending attachment

b) At the weld toe around the plate edge of an ending attachment

c) Along the weld of an attached plate

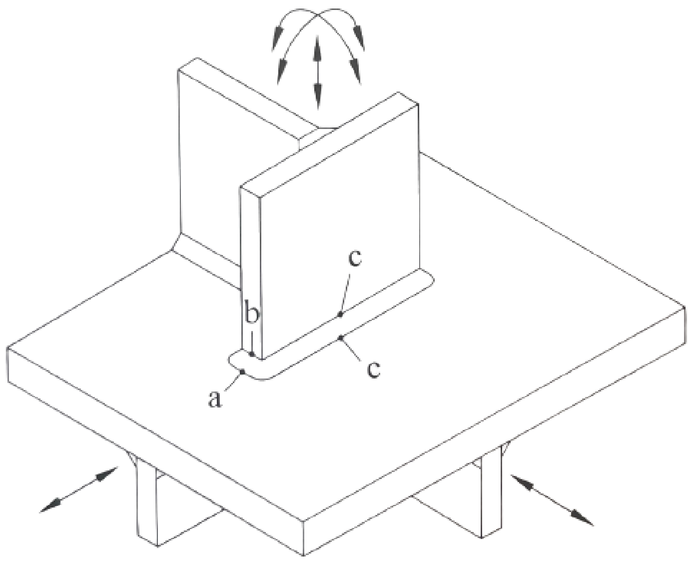


Figure 8: Different hot spot positions (Lotsberg 2016)

It is common practice in finite element analysis to model welds using shell elements with a thickness that is twice the thickness of the plates being joined. This helps to accurately capture the stresses and strains that occur in the weld region. Using a mesh size of $t\,x\,t$ at the hot spot region, where the stresses are expected to be the highest, is generally considered to be the most efficient way to obtain accurate results. Using a larger mesh size at the hot spot region may lead to non-conservative results, as it may not adequately capture the stresses and strains in the weld region (Lotsberg 2016).

In order to accurately capture the stresses and strains that occur in a solid element, the element must have a displacement function that can capture steep stress gradients as well as plate bending with linear stress distribution in the thickness direction. This is typically accomplished using iso-parametric 20-node elements, which only require one element in the thickness direction to capture the behavior of the solid material. By using reduced integration with only two integration points in the thickness direction, it is possible to easily evaluate the membrane and bending stress components of the solid element. This can provide more accurate results compared to other types of elements (Lotsberg 2016).

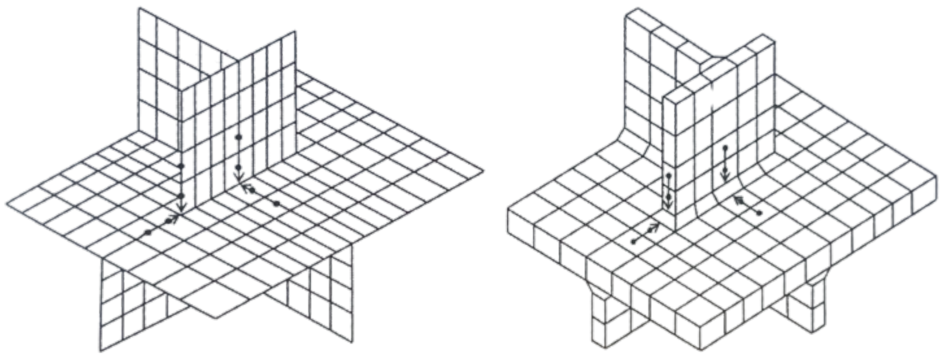


Figure 9: Shell element model and solid element model (Lotsberg 2016)

When evaluating the stress components on the plate surface, it is important to follow the recommended procedures in order to obtain accurate results. This typically involves evaluating the stress components along the paths shown in Figure 9, and extrapolating the results to the hot spot. The average stress components between adjacent elements can be used for this extrapolation. Recommended stress evaluation points are typically located at distances of $0.5t$ and $1.5t$ away from the hot spot, where t is the plate thickness at the weld toe. In cases of solid elements, the stress may first be extrapolated from the Gaussian points to the surface. See Figure 10. These stresses can then be interpolated linearly to the surface center or extrapolated to the edge of the elements, if this is the line for hot spot stress derivation (Lotsberg 2016).

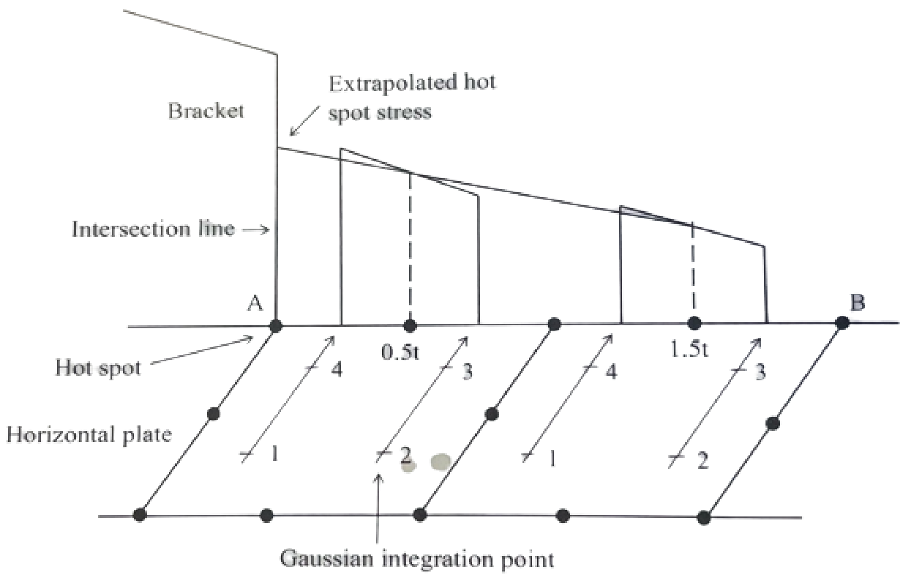


Figure 10: An example of derivation of hot spot stresses (Lotsberg 2016)

In the case of this study, the type "A" hot spot is considered. See Figure 8. For this case it is also possible to use reference points located at $0.4t$ and $1.0t$ away from the hot spot. This is recommended when using a relatively fine mesh with an element length not larger than $0.4t$ at the hot spot (Hobbacher 2016). Figure 11 shows the different recommendations for the different types of meshing.

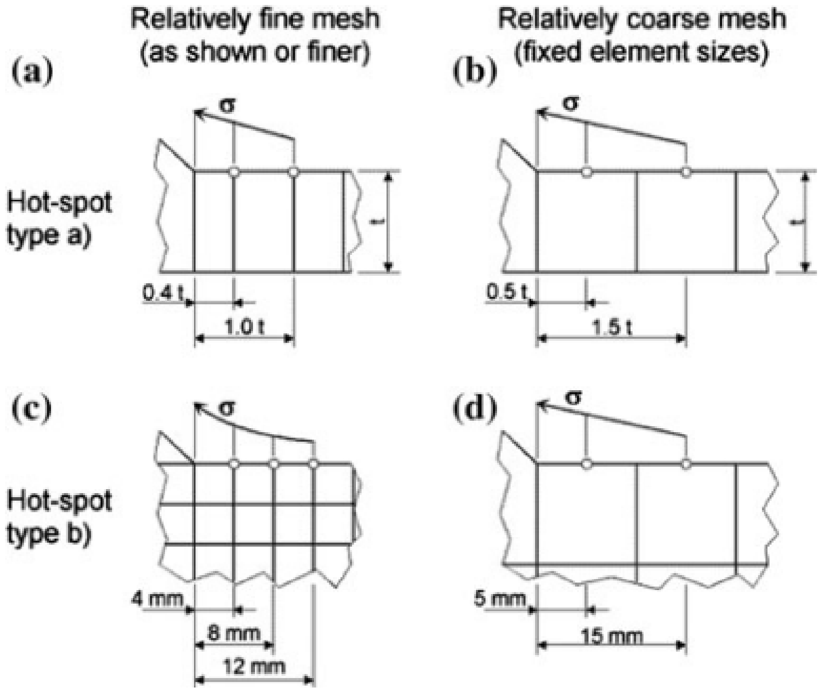


Figure 11: Reference points for different types of meshing (Hobbacher 2016)

In scenarios where the linear extrapolation technique may lead to underestimation of the structural hot spot stress, such as welded attachments to thick plates or symmetrically placed attachments, it is important to employ more accurate techniques to ensure a high safety margin for critical components. One recommended technique is the use of multi-grid strip gauges in fatigue testing of structural details with a pronounced nonlinear structural stress increase towards the hot spot. By measuring strains at positions $0.4t$, $0.9t$, and $1.4t$ along the structure, a quadratic extrapolation can be performed to derive the hot spot strain at the weld toe. This strain information can then be used to calculate the corresponding hot spot stress.

This allows for a more accurate estimation of the hot spot stress. These alternative methods, involving quadratic extrapolation and strain/stress measurements at specific positions along the structure, are particularly useful for cases involving thick-walled structures or situations where the stress distribution exhibits significant nonlinearity towards the hot spot. By employing these techniques, a more precise assessment of the hot spot stress can be achieved, enhancing the safety and reliability of critical components (DNV-GL 2021). Equation 6 can be used to calculate the hot spot stress using this technique.

$$\sigma_{hotspot} = 2.52 \cdot \sigma_{0.4t} - 2.24 \cdot \sigma_{0.9t} + 0.72 \cdot \sigma_{1.4t} \tag{6}$$

3.3 Notch stress method

The notch stress is defined as the total stress resulting from the geometry of the detail and the non-linear stress field due to the notch at the weld toe (DNV-GL 2014). The local notch stress approach mainly focuses on the stress concentration in the notch area in order to obtain the effective stress in the notch area. The notch stress method is restricted to welded joints that are expected to fail from the weld toe or from the weld root. Other causes of failure is not covered by the method, such as crack growth from surface roughness or embedded defects. The method is also not applicable if there is a significant stress component parallel to the weld.

The notch stress approach also needs to be used with S–N curves and it is most effective for high cycle fatigue assessment because the load amplitude is low and stress is the dominant control factor in the fatigue process (Khanna and Long 2010).

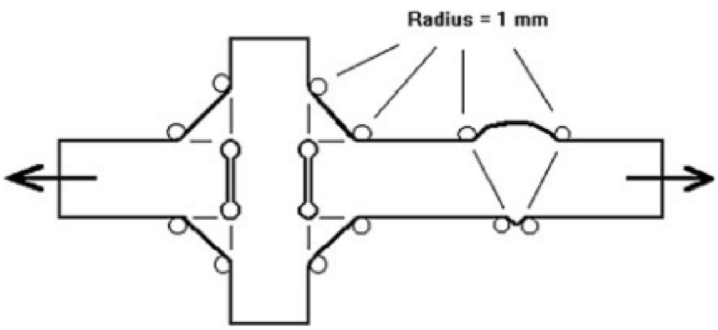


Figure 12: Fictitious rounding of weld toes and roots (Hobbacher 2016)

In the notch stress method linear-elastic material behaviour is assumed. The weld shape parameters may vary, so to take account for this variation the actual weld contour is replaced by an effective one. This takes also account for the non-linear material behaviour at the notch root. An effective notch root radius of $r = 1$ mm has been verified to give consistent results for structural steels. For the assessment of fatigue, the effective notch stress is compared with a single fatigue resistance curve. But it is important to check that the fatigue resistance curve for the parent metal not is exceeded in the direct vicinity of the weld (Hobbacher 2016).

The notch stress method is well suited to the comparison of alternative weld geometries. It is suggested that welds should be modelled with flank angles of 30° for butt welds and 45° for fillet welds. The method is limited to thicknesses t $\geq$ 5 mm, since the method has not yet been verified for smaller wall thicknesses. Weld toes, machined or ground to a specified profile, shall be assessed using the notch stress of the actual profile in conjunction with the nominal stress based fatigue resistance curve for a butt weld ground flush to plate (Hobbacher 2016).

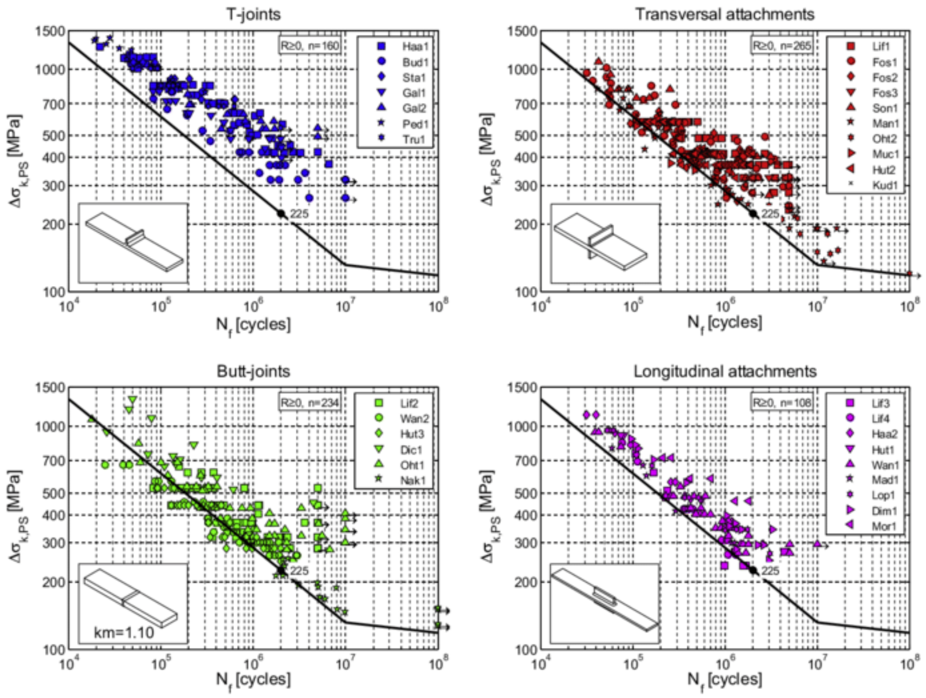


Figure 13: Fatigue data converted to the notch stress system (Pedersen et al. 2010)

Figure 13 shows a plot where a notch-element analysis has been run for four different types of geometries. The results are plotted against lifetime. This shows that the drawn line gives conservative lifetimes, corresponding to the design S-N curve in DNV for nominal stress. FAT 225 is drawn, and the line with $m = 3$ goes through the point $\Delta S_{notch} = 225$ and $N = 2 \cdot 10^6$. Pedersen et al. concluded with that for most fillet-welded joints, the experimental fatigue data agrees reasonably well with the current IIW guidance, i.e. using a FAT 225 S–N curve, except for a few data points (Pedersen et al. 2010).

Using the notch stress approach, the stress concentration factor (SCF) of an arbitrary welded joint can be determined using finite element analysis. The geometric SCF K_t corresponds to the fatigue effective SCF K_f due to the fictitious rounding of the notch.

$$K_f = K_t(r_{ref} = 1\,mm) \tag{7}$$

The notch approach uses an idealized geometry, which makes the above statement true. Conversion of the extracted fatigue data in the nominal stress system $\Delta\sigma_n$ to the notch stress system $\Delta\sigma_k$ for a given specimen can thus be accomplished as follows:

$$\Delta\sigma_k = K_f \cdot \Delta\sigma_n \tag{8}$$

K_t is determined according to the IIW recommendations for fatigue assessment using the notch stress approach (Fricke 2012). A tensile nominal stress of 1 MPa was applied to the specimen, such that the maximum principal stress observed in the notch corresponds to the SCF (Pedersen et al. 2010).

3.3.1 Notch stress by FEA

Calculation of effective notch stress by the finite element method requires the use of a fine element mesh around the notch region. The maximum calculated surface stress in the notch is the effective notch stress to be used, together with the recommended S-N curve. The maximum surface stress can be found directly from the nodal stress calculated at the surface, or from extrapolation of element stresses to the surface. Required mesh size depends on the number of elements used. The mesh should be made with regular elements without transition to reduce refinement within the first three element layers from the notch surface (Lotsberg 2016).

Table 1: Recommended sizes of elements on surface (Hobbacher 2016)

Element type	Relative size	Absolute size [mm]
Quadratic with mid-side nodes	$\leq$ r/4	$\leq$ 0.25
Linear	$\leq$ r/6	$\leq$ 0.15

When using FEA to determine effective notch stress, it is important to consider the size of the elements used in the analysis. In general, it is recommended to use elements with sizes no larger than 1/6 of the radius of the notch for linear elements, and no larger than 1/4 of the radius for higher order elements. This is to ensure that the FEA model accurately captures the stress concentration at the tip of the notch and provides reliable results. Additionally, it is important to use sufficiently small elements in the curved parts of the notch, as well as in the beginning of the straight part of the notch surfaces in both the tangential and normal directions. This is to ensure that the stress concentration at the tip of the notch is accurately captured in all parts of the model (Hobbacher 2016).

To calculate the effective notch stress in a welded notch using the finite element method in Abaqus, it is then possible to carry out the following steps:

- Create a 3D model of the welded notch using the geometry and material properties of the component
- Use the appropriate element type and size to accurately capture the stress concentration at the tip of the notch
- Apply the appropriate loading conditions to the model, such as external forces
- Use the Abaqus solver to solve the finite element model and calculate the stress distribution within the notch

- Extract the maximum stress at the tip of the notch, which is the effective notch stress
- Compare the calculated effective notch stress with the allowable stress for the material to determine if the component is safe to use

3.4 Comparison of the Numeric Models of Hot-Spot and Effective Notch Stress Approaches

The effective notch stress approach is generally considered to be more favorable than the hot spot stress method for analyzing structural components, because it can be applied to a wider range of geometric shapes and does not have any restrictions on the geometry of the component being investigated. This makes the effective notch stress approach more versatile and applicable to a wider range of problems.

However, one disadvantage of the effective notch stress approach is that it can be more difficult to build a numerical model of the component using this method, compared to the hot spot method. This is because the effective notch stress approach requires the complex geometry of the component to be accurately modeled in order to accurately calculate the stress concentration at the tip of the notch. In contrast, the hot spot method does not require detailed modeling of the geometry and can be applied more easily to simple shapes. Overall, the effective notch stress approach offers many advantages over the hot spot method, but it can be more challenging to implement in some cases due to the need for detailed modeling of the component geometry (Mecséri and Kövesdi 2020).

4 Modeling and Finite Element Analysis

In this chapter the outline of the finite element method will be explained. The method will be used to perform both a hot spot and a notch stress analysis using the Abaqus software. A global model will be used to calculate the hot spot stress. For the notch stress analysis the submodeling technique will be used, and therefore a local model is created for this purpose. This technique will be discussed in Section 4.2.6.

4.1 Finite Element Method

The finite element method (FEM) is a numerical procedure for analysing complicated structures, such as brackets. Finite element analysis (FEA) is performed on the bracket geometry, which is modelled in Abaqus using solid elements. Though results from a finite element analysis rarely are exact, errors can be decreased by processing more equations to obtain more accurate results. FEM is a much used tool when the problem is too complicated to be solved satisfactorily by solving e.g. the differential equations by classical analytical methods.

A finite element model of a structural system is a discrete piece of that system. The finite element method is used to solve problems governed by a set of partial differential equations. First, the problem is discretized into a set of small elements. Discretization is important because it has a significant effect on the accuracy of the results. The equations obtained for each element are then assembled together with adjoining elements to form the global finite element equation for the whole problem domain. Then the equations created for the global problem domain can be solved for the entire displacement field (Liu and Quek 2014).

4.2 Abaqus

Abaqus is a software suited for finite element analysis. It is widely used in engineering and science for simulating and analyzing complex mechanical systems and structures. Abaqus offers a range of capabilities, including nonlinear analysis. When it comes to fatigue analysis, Abaqus offers several capabilities for simulating and predicting the behavior of structures subjected to cyclic loading. Abaqus can predict the fatigue life of a structure or component based on the loading conditions and material properties (Smith 2009).

4.2.1 Creating the model

The global model is created based on the technical drawing shown in Appendix A. The weld geometry is created based on the dimensions shown in Figure 14. This is however the geometry for a plate with 50 mm thickness. The plate provided by Aker Solutions that is studied in this thesis is 8 mm thick. The dimensions therefore needs to be scaled. First you find the ratio between the two plates:

$$\frac{8\,mm}{50\,mm} = 0.16 \tag{9}$$

Then it is possible to find the new dimensions for the horizontal and vertical weld legs:

$$H = 30\,mm \cdot 0.16 = 4.80\,mm \tag{10}$$

$$V = 35\,mm \cdot 0.16 = 5.60\,mm \tag{11}$$

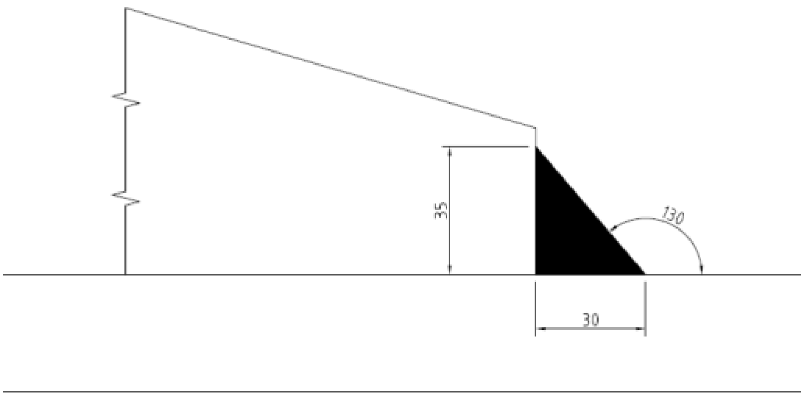


Figure 14: Weld geometry for a 50 mm thick plate provided by Aker Solutions

It is recommended to include the geometry of the weld in the Abaqus solid model. This is because the weld toe geometry can significantly affect the stress distribution and hence the stresses at the weld toe. By including the geometry of the weld, one can accurately capture the stress concentration effects at the weld toe and obtain a more realistic estimate of the stresses.

Table 2: Linear-elastic material properties

Young's modulus	Poisson's ratio
210 000 MPa	0.3

Linear-elastic material properties can be appropriate for the solid model in Abaqus in certain cases. Linear-elastic behavior assumes that the material response is linearly proportional to the applied stresses within the elastic range, and there is no plastic deformation. The linear-elastic assumption is often valid for materials that are not expected to undergo yielding or plastic deformation under the loading conditions of interest. The values given in Table 2 was used for the solid model in

Abaqus.

4.2.2 Symmetry and boundary conditions

The boundary conditions should be set up correctly to accurately represent the loading conditions on the model. For simplicity only 1/4 of the full size structure is modeled in Abaqus. This can be a smart choice for several reasons:

- Computational efficiency: Modeling only a quarter of the full size model reduces the number of elements, nodes, and equations that the solver has to solve, which can significantly reduce the computational time and resources required to perform the analysis.
- Reduced modeling efforts: Modeling only a quarter of the full size model can reduce the modeling efforts and the complexity of the analysis.
- Symmetry assumption: The use of symmetry assumptions can be beneficial when the geometry, loads, and boundary conditions exhibit symmetry. The symmetry modeling technique allows the analysis of one quarter of the model, with the other three quarters being assumed to have identical conditions (Smith 2009).

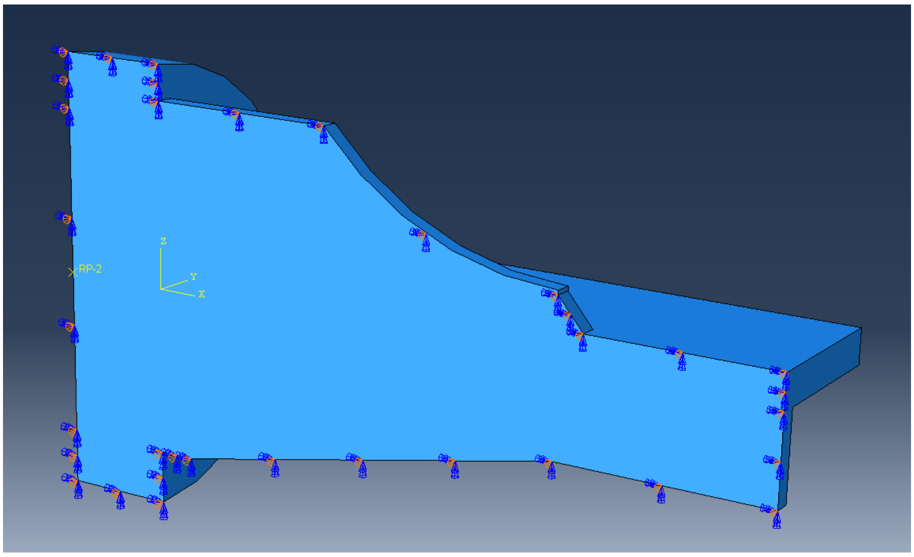


Figure 15: The boundary conditions applied to the model in Abaqus

As shown in Figure 15 there are applied symmetry boundary conditions. There are applied symmetry boundary conditions in both X- and Y-direction which enforce that the displacement and rotation of the nodes on one side of the model (for example, the left side) are equal to the displacement and rotation of the nodes on the opposite side (for example, the right side).

Specifically, X-symmetry enforces symmetry with respect to the Y-axis, meaning that the displacement and rotation in the Y-direction are zero on the Y-axis. Similarly, Y-symmetry enforces symmetry with respect to the X-axis, meaning that the displacement and rotation in the X-direction are zero on the X-axis. These boundary conditions are appropriate when the geometry, loads, and boundary conditions exhibit symmetry along the respective axes.

4.2.3 Applying load

In an actual lab experiment the test specimen is attached between to grips in a static test machine, and then tensile load is applied at the top grip on the machine while the bottom grip is held steady and still. To simulate this in Abaqus, a concentrated force is applied at the grip section of the model, with the force pointing away from the model. This is the same as a tensile load. It is also important to ensure that the bottom grip is constrained using appropriate boundary conditions, such as fixed support or restrained nodes, to simulate the fixed bottom grip in the actual test.

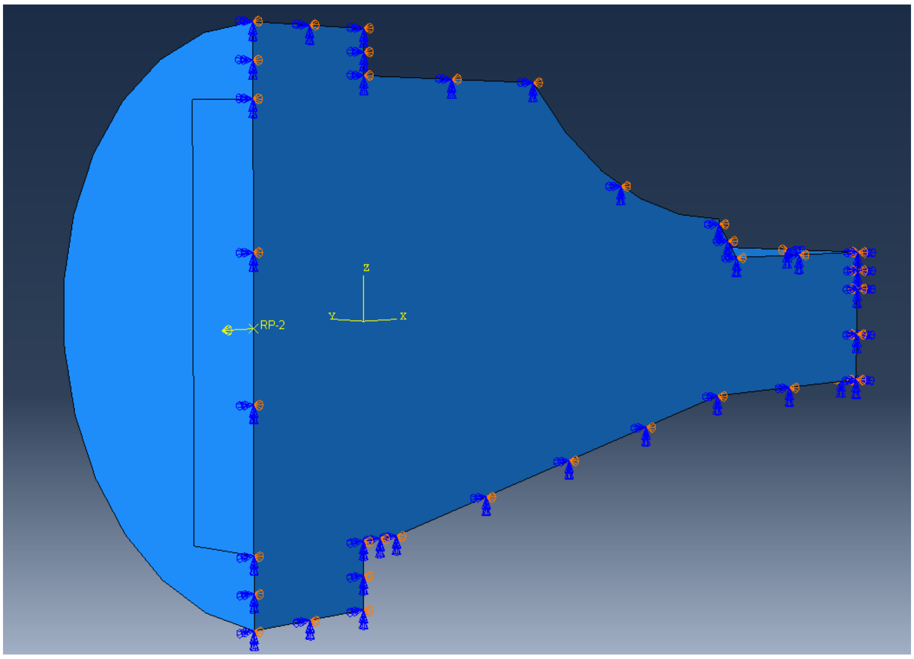


Figure 16: The load applied to the model in Abaqus

Since the model in Abaqus is half the size of the full size model, the force applied to the model will have to be doubled. So, if the force applied to the model in Abaqus is *1000 N*, the actual load is the double, *2000 N*.

Table 3: The force applied to the model in Abaqus

Solid model	Full size model
1 000 N	2 000 N
2 500 N	5 000 N
5 000 N	10 000 N

4.2.4 Applying mesh

For the hot spot analysis the model consisted of quadratic tetrahedral elements of type C3D10. These elements have a higher order shape function than the linear tetrahedral elements, resulting in a more accurate representation of the displacement and stress fields. In addition, they are efficient in terms of computational cost and can handle complex geometries with curved boundaries. The global size of the elements in the mesh were set to 4 mm. Mesh refinement in the regions of high stress gradients, such as the weld toe and hot spot locations, is often necessary to obtain accurate results. The mesh along the weld toe and along the hot spot path is refined into smaller elements. Along the hot spot path, away from the weld toe, the elements increases gradually from 1 to 4 mm. Along the weld the elements are 1.33 mm.

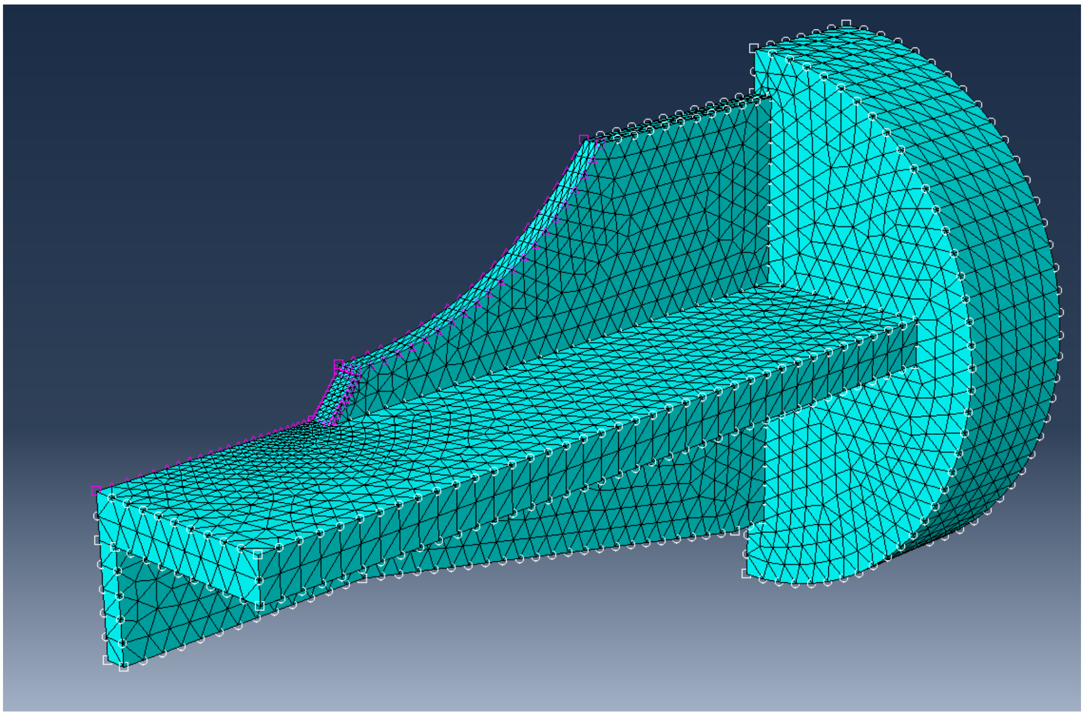


Figure 17: The mesh applied to the solid model for the hot spot analysis

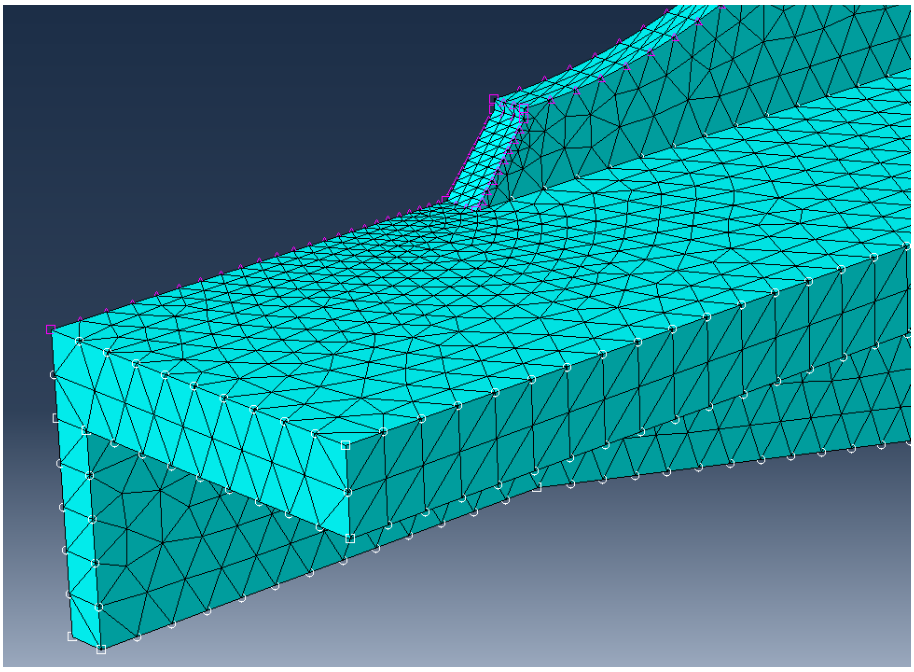


Figure 18: The purple colored local seeds are the regions where the mesh is refined

For the notch stress analysis the model consists of a submodel. The submodeling technique uses higher dense mesh in the areas of interest and a coarser mesh in the other parts of the structure. This is explained in Section 4.2.6. For the submodel, that is the notch root, the model consists of quadratic hexahedral elements of type C3D20R. This is because the notch stress method is based on the nonlinear elastic-plastic calculation of the notch stress at the location where local yielding occurs, such as the weld root. Quadratic hexahedral elements can model large deformations and nonlinear material behavior better than tetrahedral elements, and they also have less numerical integration errors than fully integrated elements. The point of the submodel is that it is easy to change the mesh without taking the whole model into account. To calculate the notch stress the mesh was made smaller and smaller until the first principal stress converged towards a given value. It is also this technique that is used in industry where there are often large global models. As you refine the mesh, the solution should converge towards a stable value, which is less sensitive to further mesh refinement. This means that the results become more accurate as the mesh gets finer, but eventually reach a point where additional refinement does not significantly improve the accuracy. To get very small elements in the relevant areas you have to use a submodel. The global size of the elements in the mesh were set to 0.4 mm. The mesh in the curvature area was refined from 10 elements to 55. At this point the results converged towards a value.

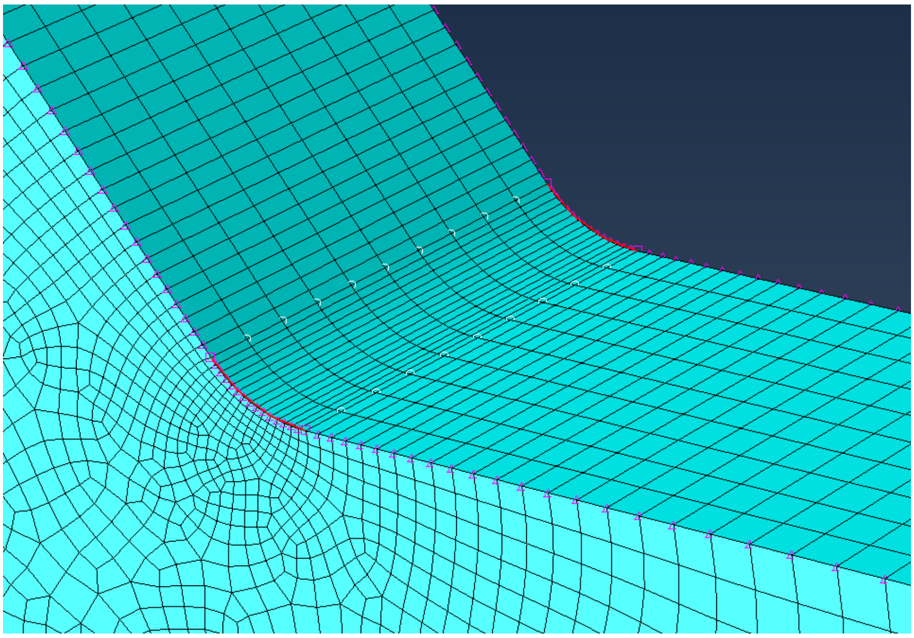


Figure 19: The purple colored local seeds shows the refined mesh at the notch root radius in the submodel

Figure 19 shows the notch root radius where the mesh was refined. The notch was given a radius of 1, as explained previously in Section 3.3. The angle between the weld and the plate is approximately 130°.

4.2.5 Shell and solid elements

Shell elements are often used in large constructions where there are many elements, but there is a greater uncertainty around local stresses in or during welding. Shell element is often used as input to a solid submodel to achieve accurate local analyses. To analyse a three-dimensional stress problem, solid elements are often used. These types of problems are often related to design of solid structures or structures with irregular geometry where larger strength or better utilisation of the materials is required. For joints between structural parts, it may be necessary to perform a three-dimensional analysis (Moan 2003).

The main difference between a solid and a shell model is the level of detail. A solid model represents a three-dimensional object, whereas a shell model represents a thin-walled structure, typically with a thickness much smaller than its other dimensions. Shell models are useful for analyzing structures such as plates, shells, and membranes. Solid models are useful for analyzing more complex structures that require a higher level of detail. The benefits of making a solid model include the ability to accurately capture the details of the geometry and loading conditions, and the ability to analyze complex structures. Solid models also tend to be more accurate than shell models. However, solid models require more computational resources and may take longer to solve than shell models. Shell models, on the other hand, are faster and require less computational resources, but they may not be suitable for all types of analyses (Smith 2009).

4.2.6 Submodeling

The submodeling technique is used in Abaqus for the notch stress analysis. Submodeling is a technique that involves studying a local part of a model using a refined mesh, based on interpolating the solution from an initial, global model onto the nodes on the appropriate parts of the boundary of the submodel. This allows for a more detailed analysis of the stresses and strains in a specific region of the model, without significantly affecting the overall solution. The submodeling technique uses higher dense mesh in the areas of interest and a coarser mesh in the other parts of the structure. The response at the boundary of the local region is defined by the solution for the global model, and this, along with any loads applied to the local region, determines the solution in the submodel. The technique relies on the global model accurately defining the boundary response of the submodel in order to provide accurate results (Smith 2009).

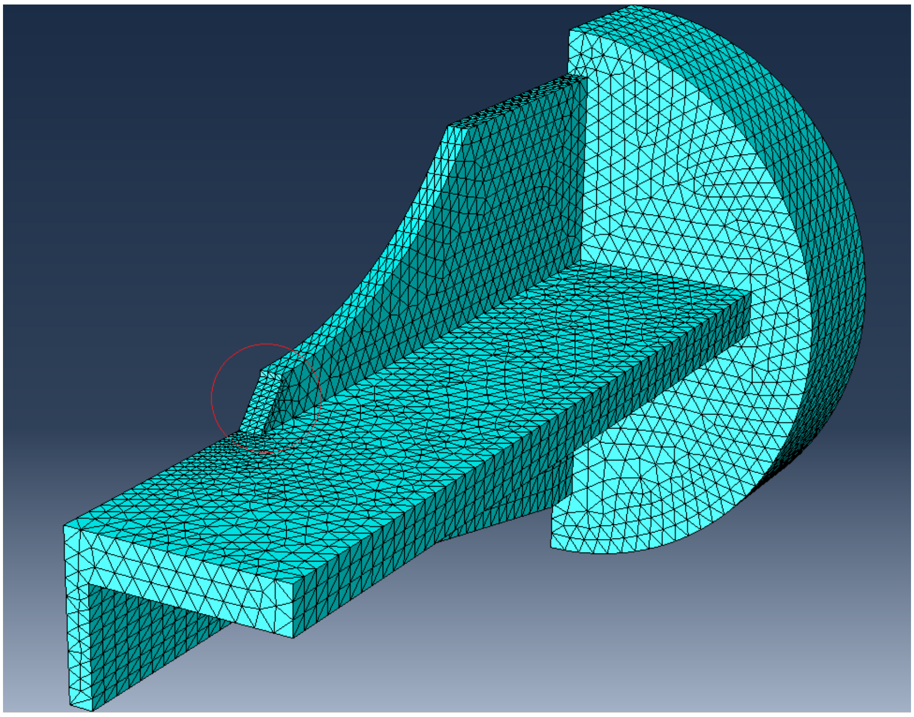
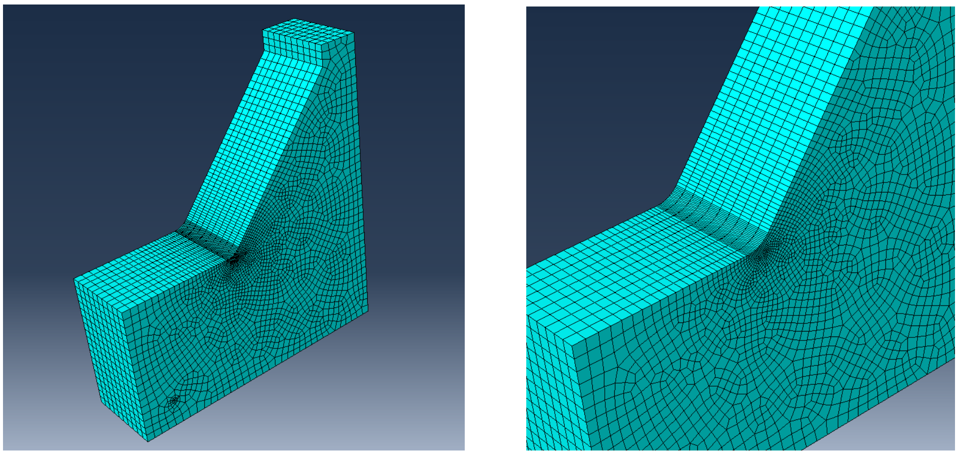


Figure 20: The notch root in the global model

For the notch stress analysis in Abaqus, the notch root is modeled as a submodel. The mesh at the notch root is modeled with a finer mesh, while the other parts of the structure keeps a coarser mesh. This is useful for a bracket geometry because notches can introduce high stress concentrations in the material that can be difficult to accurately capture with a full model. The notch root is created in Abaqus as a submodel, and then incorporated into the full model. Then, when running the analyses for the submodel, this results in a more accurate prediction of the overall stress distribution in the bracket.



(a) Submodel (b) Notch radius

Figure 21: The submodel

4.3 Calculation

4.3.1 Hot spot stress

To calculate the hot spot stress at the weld toe, the path tool in Abaqus is used. A path is drawn from the weld toe and away from the bracket. Then the S11 stress component is plotted against the distance along the path. This is the relevant stress component due to the applied uniaxial load in the x-direction. The XY-data points are then extracted, because it is not possible to get the hot spot stress values directly from Abaqus. A Python code is used to calculate the values. First the stress values at the hot spot extrapolation points $X = 0.5t$ and $X = 1.5t$ are found. This corresponds to $X = 4\,mm$ and $X = 12\,mm$. Then a linear extrapolation function is used to draw a straight line that goes through the hot spot extrapolation points. Then the hot spot stress is the value at $X = 0$. The same procedure is done for the two other approaches with extrapolation points at $X = 0.4t$ and $X = 1.0t$, and $X = 0.4t$, $X = 0.9t$ and $X = 1.4t$.

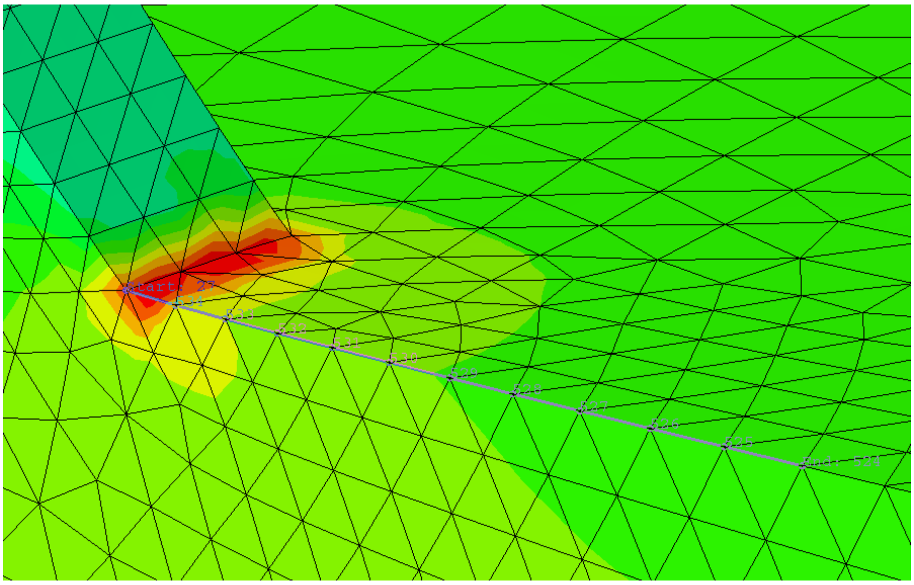


Figure 22: The path drawn away from the weld toe

4.3.2 Notch stress

First the region of interest around the notch where the maximum principal stress is going to be calculated is defined. This is at the notch root radius. When the submodel is meshed, the job analysis is submitted. The results are shown in the "Visualization" tab. The maximal principal stress is then read from the plot shown when the "Plot Contours on Deformed Shape" is chosen. The analysis is then repeated with finer mesh densities until the maximum principal stress converges towards a specific value. This value is then the notch stress.

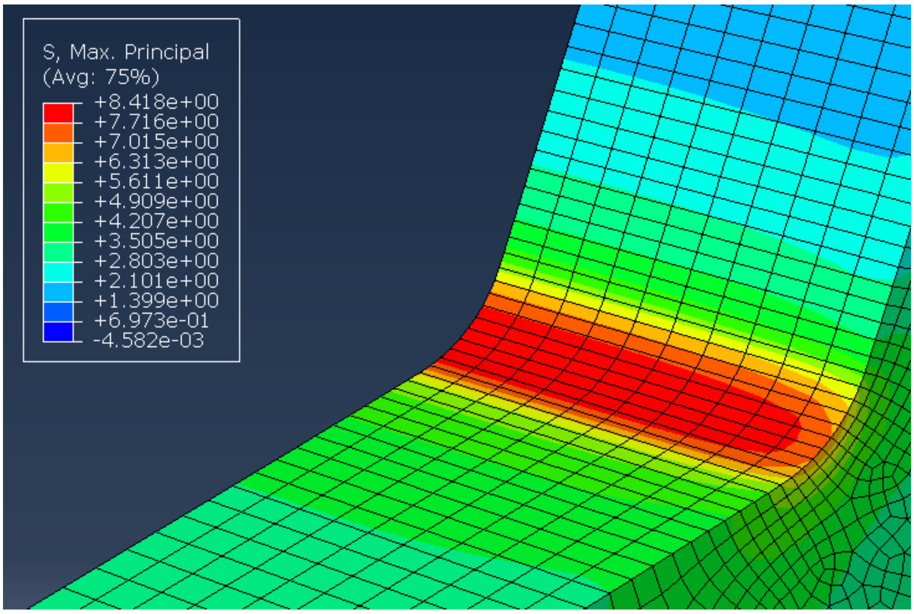


Figure 23: The Max. Principal stress with 10 elements in the curvature area

5 Lab experiment

For this thesis a lab experiment was not carried out, because it got cancelled. It will however be explained in this chapter how it is performed, and what the purpose of the experiment is.

The purpose of the lab experiment is to test a bracket geometry delivered by Aker Solutions in the Marine Structures Laboratory at NTNU, and measure the hot spot stress and notch stress around the weld toe. Also a long-life fatigue test was going to be performed to predict the fatigue life of the structure. Bracket geometries are typically joined by welding. Fatigue cracks have a higher likelihood of developing in welds, and therefore it is necessary to assess the fatigue life in these regions (Radaj 1990).

5.1 Lab facilities

The lab experiment was going to be conducted in the Marine Structures Laboratory at NTNU. The Marine Structures Laboratory is operated by the division of structural engineering at SINTEF Ocean in cooperation with the Marine Structures group at the Department of Marine Technology. The main activities in the laboratory are the testing of structures, structural components and materials. Typical problems involve fatigue testing, ultimate strength and collapse testing, testing for serviceability, and advanced materials testing.

A wide range of structural analysis programs are available for linear and non-linear static and dynamic analyses. These include special programs for fatigue and fracture mechanics analysis, and the collapse behaviour of intact and damaged structures. High performance dynamic actuators with load capacities up to 2000 kN with hydraulic grips are available in the lab, and was going to be used for the fatigue life test. A load program, including randomized load sequences with variable or constant rms levels and variable signal bandwidth is used in these kind of experiments.

5.2 Test specimen

It is important that the test specimen is designed and fabricated in a way similar to how the detail is going to look when it will be used in a structure. Then it is possible to use the fatigue test data for the considered component and say something about the predicted fatigue life. A technical drawing of the test specimen provided by Aker Solutions is shown in Appendix A. It is important that the boundary conditions and load application provides a similar stress distribution at the hot spot region as should be expected in the actual structure. It is also important that the designed test specimen will fail at the considered hot spot region, rather than at another region that is created for test purposes only, such as at the connection to the testing machine. It is also important to have in mind that there might be a difference in residual stresses in a scaled test specimen compared with a real structure component. Therefore, measurements of residual stresses at welded connections in the test specimen is recommended in order to provide a better assessment of the results (Lotsberg 2016).

Figure 24: A manufactured test specimen (Mosbron 2022)

The grip section on the test specimen is shown in Figure 24, at each end of the component. The component has two brackets to provide symmetry.

Figure 25: The grip section on the test specimen (Mosbron 2022)

5.3 Strain gauges

To measure the strain in the test specimen, strain gauges are used. Since it is only necessary to measure the strain fields in one direction for this experiment, uniaxial strain gauges are sufficient. Then the stress is simply derived from the measured strain multiplied by Young's modulus (Lotsberg 2016).

Figure 26: Strain gauges placed on a test specimen near the weld toes (Mosbron 2022)

To get a direct derivation of the measured hot spot stress, the first strain gauge is placed as close as possible to the weld toe. This is because it is normally assumed that the local stress increase due to the weld toe is accounted for in the hot spot stress S-N curve (Lotsberg 2016). The first strain gauge is placed $0.5t$ from the weld toe, where t is the plate thickness, which is 8 mm in this case. The second strain gauge is placed $1.5t$ from the weld toe. Then the stress at the weld toe can be derived from a linear extrapolation of the stresses at these two measured points. In total there is placed four strain gauges on the test specimen, two at each weld toe.

5.4 Static test

The static tests are performed using INSTRON model 1342, which is a servo-hydraulic test machine that has a maximum load capacity of 100 kN. See Figure 27 to see what the test machine looks like in an example picture. The test specimen is attached between the two grips in the test machine, and then tensile load is applied at the top grip on the machine while the bottom grip is held steady and still.

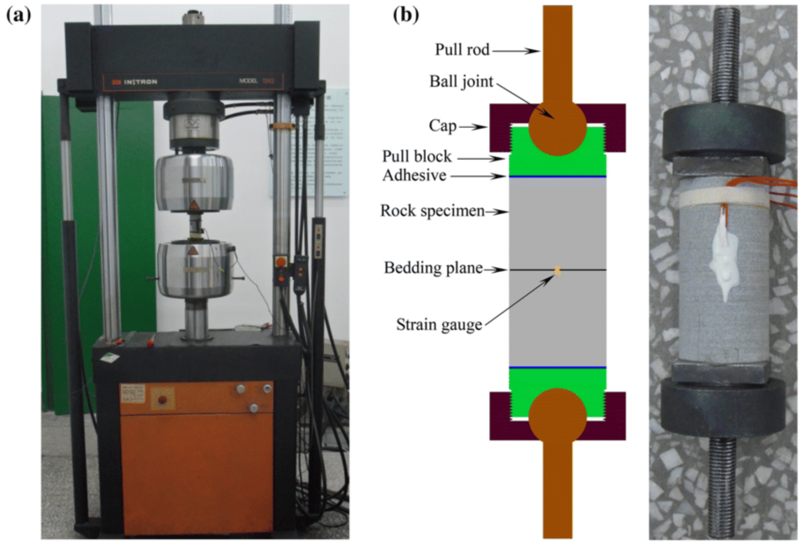


Figure 27: a) The INSTRON test machine, b) The connection device for an example direct tensile testing of rocks (Cen et al. 2020)

The test specimen is subjected to static, tensile loads with different magnitude. The loads are first applied in ascending order, and then in descending order. This is shown in Figure 28. The loads Mosbron applied in the lab-experiment was 2 kN, 5 kN and 10 kN.

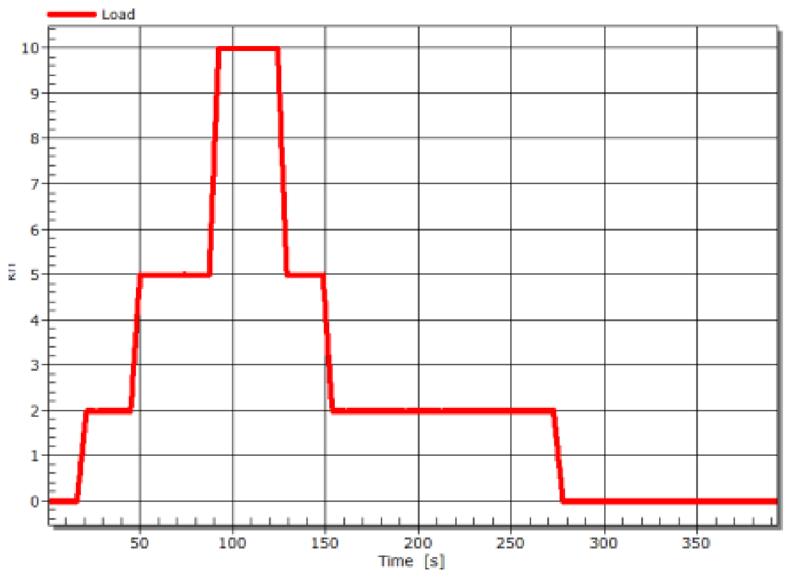


Figure 28: Load application on a test specimen (Mosbron 2022)

5.5 Fatigue test

Before executing the fatigue test it is necessary to classify the welded detail on the bracket geometry. The weld toe is located at the end of the bracket which is transverse of the stress direction. See Figure 29. The crack will most likely initiate at the weld toe on the stressed surface. This places the welded detail in class E, according to Table A-7 in DNV-GL (DNV-GL 2014).

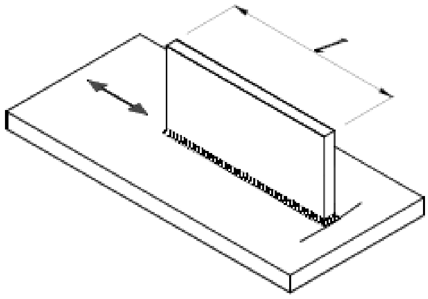


Figure 29: The welded longitudinal attachment places the weld detail in class E (DNV-GL 2014)

The S-N curve, and the values for $log\,\overline{a}$ and m_1 is found to be the following:

Table 4: Values for class E detail in air

S-N curve	$N \leq 10^7$
$log\,\overline{a}$	12.010
m_1	3.0

It is also necessary to calculate the cross-sectional area of the test specimen. This value will be used in the calculations when the load levels for the fatigue test is being determined. The cross-sectional area of a T-profile is calculated using this formula:

$$A = W \cdot t_1 + (H - t_1) \cdot t_2 \tag{12}$$

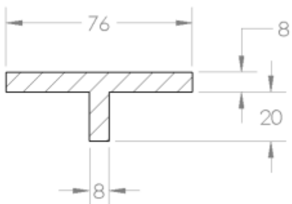


Figure 30: The cross-sectional area of a test specimen below the weld toe (Mosbron 2022)

Based on Equation 12 the cross-sectional area is $A = 768\,mm^2$. The fatigue test is performed using the high performance dynamic actuators with load capacities up to 2000 kN with hydraulic grips. Based on the calculations shown in Section 5.6, the assumed number of cycles will be predicted, along with the predicted stress range and load range. These calculations are done to make sure that the test specimen will fracture within a reasonable time.

5.6 Fatigue calculations

Before one can perform the fatigue test, the loading cycles needs to be established using some general fatigue-formulas. They will be presented in this subchapter. The formulas are obtained from the compendium *Fatigue and Fracture Design of Marine Structures* (Berge and Ås 2017).

Table 5: Nomenclature

N	Number of cycles to failure
a	Crack length
R	Stress ratio
A	Cross-sectional area
F_{min}	Minimum load level
F_{max}	Maximum load level
Δ S	Stress range
S_{min}	Minimum stress in a cycle
S_{max}	Maximum stress in a cycle

The general design S-N curve is formulated as:

$$log\, N = log\, \overline{a} - m\, log\, \Delta S \tag{13}$$

This can be reformulated with respect to $log\, \Delta S$:

$$log\, \Delta S = \frac{log\, \overline{a} - log\, N}{m_1} \tag{14}$$

The stress range, ΔS is then found to be:

$$\Delta S = 10^{log\, \Delta S} \tag{15}$$

The stress ratio R, is defined as the ratio between the minimum stress in a cycle and the maximum stress in a cycle:

$$R = \frac{S_{min}}{S_{max}} \tag{16}$$

The mean stress, S_m, is calculated using the following formula:

$$S_m = \frac{\Delta S}{2} \cdot \frac{(1+R)}{(1-R)} \tag{17}$$

From the previous formula, S_{min} and S_{max}, can be derived as:

$$S_{min} = S_m - \frac{\Delta S}{2}, \qquad S_{max} = S_m + \frac{\Delta S}{2} \tag{18}$$

Lastly, the corresponding loads are found using the cross-sectional area:

$$F_{min} = S_{min} \cdot A, \qquad F_{max} = S_{max} \cdot A \tag{19}$$

Now it is possible to calculate the mean load S_m that is needed to reach the yield point in the weld, but not yielding on the entire cross-section. Typical yield strength for this steel is 355 MPa. The calculations are done for three different load cases. Then, the three load levels can be used to check the dispersion in relation to the S-N curve when the fatigue testing is finished.

Table 6: Calculations for three different load cases

Cycles (N)	S_m [MPa]	S_{min} [MPa]	S_{max} [MPa]	F_m [kN]	F_{min} [kN]	F_{max} [kN]
$10.5 \cdot 10^4$	410	274	547	315	210	420
$4 \cdot 10^5$	205	137	274	157	105	210
$8 \cdot 10^5$	163	109	218	125	84	167

For the calculations presented in Table 6, an R-ratio of 0.5 was used. The values for the first case are obtained from Mosbron's master's thesis (Mosbron 2022).

5.7 Calibration and uncertainties

It is important to calibrate the test specimen before applying a load. Skewness can occur in the test specimen during installation, and therefore one must calibrate to make sure that the strain gauges do not have different tension before the experiment starts. When the test specimen is mounted into the test machine, the force must therefore be set to zero before the load gets applied.

To reduce the uncertainties in the experiment, each load in the static test is maintained for at least 10 seconds. Loading is performed several times prior to recording the results. It will also be confirmed that the registered strain acted proportionally to the applied load.

In the static test, one will strive to achieve yield stress in the weld toe, and then adjust the force down after this has been achieved. If yield stress is achieved over a larger area than this, it can lead to uncertainties in the measurements.

6 Results

In this chapter the results will be presented. Both the hot spot stress and notch stress results from the solid model in Abaqus will be given and compared to Mosbron's results. In his master's thesis he used both solid and shell elements to calculate the hot spot stresses in Abaqus. Since the lab experiment was canceled due to delays related to moving the lab to a new location, only Mosbron's lab result is available for comparison.

6.1 Hot spot method

6.1.1 Hot spot stress from the solid model

The hot spot stress was calculated using three different approaches. The results from the method where the extrapolation points was located $0.5t$ and $1.5t$ away from the weld toe is presented in Table 7. Then the hot spot stress was calculated using extrapolation points at $0.4t$ and $1.0t$ away from the weld toe. The results are shown in Table 8. In the last approach three extrapolation points was used, at $0.4t$, $0.9t$ and $1.4t$. The resulting hot spot stress for this method was calculated using Equation 6. The resulting stress is presented in Table 9. The stress at the extrapolation points are also shown in the tables.

Table 7: Hot spot stress with extrapolation points at $0.5t$ and $1.5t$ away from the weld toe

Load [kN]	$\sigma_{hotspot}$ [MPa]	$\sigma_{0.5t}$ [MPa]	$\sigma_{1.5t}$ [MPa]
2	3.0454	2.8094	2.3376

Table 8: Hot spot stress with extrapolation points at $0.4t$ and $1.0t$ away from the weld toe

Load [kN]	$\sigma_{hotspot}$ [MPa]	$\sigma_{0.4t}$ [MPa]	$\sigma_{1.0t}$ [MPa]
2	3.4008	3.0283	2.4694

Table 9: Hot spot stress with extrapolation points at $0.4t$, $0.9t$ and $1.4t$ away from the weld toe

Load [kN]	$\sigma_{hotspot}$ [MPa]	$\sigma_{0.4t}$ [MPa]	$\sigma_{0.9t}$ [MPa]	$\sigma_{1.4t}$ [MPa]
2	3.4899	2.9357	2.5010	2.3529

In Figure 31 and Figure 32 a plot of the data points extracted from Abaqus is shown for the first two approaches. The linear extrapolation that calculates the hot spot stress at the weld toe is also plotted.

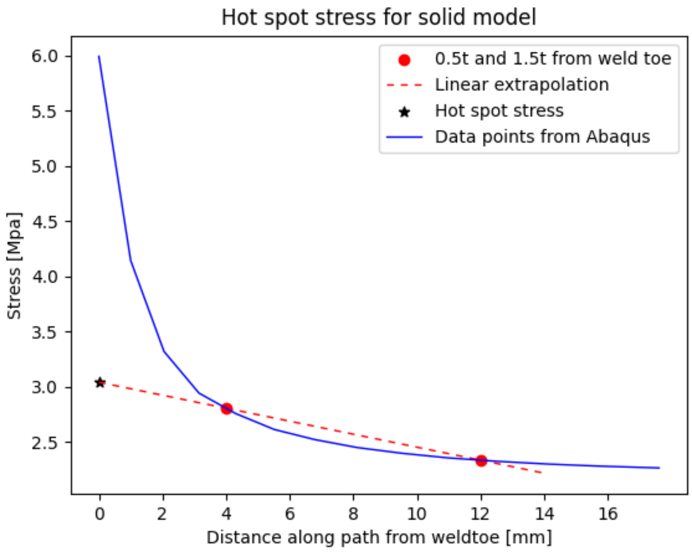


Figure 31: The hot spot stress at the weld toe for the solid model in Abaqus with extrapolation points at $0.5t$ and $1.5t$ away from the weld toe, with a load level of $2\,kN$

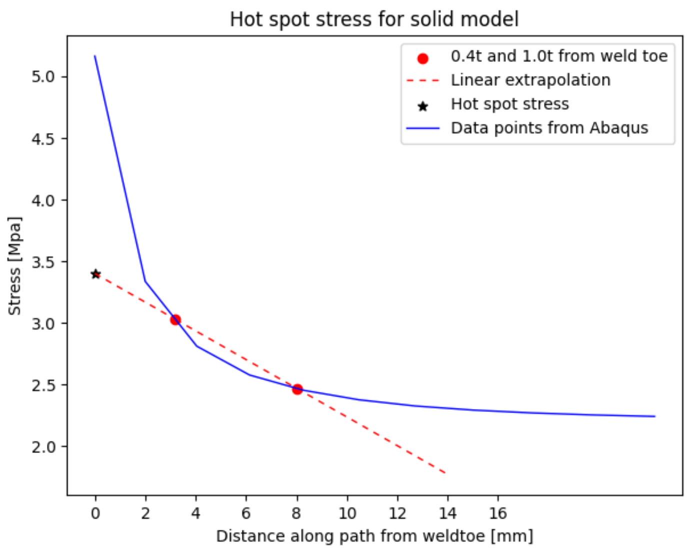


Figure 32: The hot spot stress at the weld toe for the solid model in Abaqus with extrapolation points at $0.4t$ and $1.0t$ away from the weld toe, with a load level of $2\,kN$

6.1.2 Evaluation of the hot spot method

It is important to evaluate whether the hot spot method is applicable for predicting the failure of the structure. To assess the applicability of the hot spot method for predicting fatigue failure, it is necessary to ensure that the hot spot stress calculated at the weld toe corresponds to the load range applied during the fatigue test. This is because the fatigue life of a structure is affected by the maximum cyclic stress range applied to it. If the hot spot stress calculated at the weld toe is not representative of the maximum cyclic stress range applied during the fatigue test, then the hot spot method may not be applicable in predicting the fatigue failure of the structure. This is because the fatigue failure is governed by the cyclic stress range, and if the hot spot stress is not representative of the cyclic stress range, it may not provide an accurate prediction of the fatigue life (Mecséri and Kövesdi 2020).

During the fatigue test that Mosbron performed during his master's thesis the load range was (Mosbron 2022):

$$\Delta F = F_{max} - F_{min} = 210\ kN \tag{20}$$

Because of the linear-elastic material properties, there is a proportional relationship between the load range ΔF and the corresponding hot spot stress $\Delta\sigma_{hs}$. This can be formulated as:

$$\Delta\sigma_{hs} = \Delta F \cdot x \tag{21}$$

The constant x can be calculated for each hot spot stress, and is the proportionality constant:

$$x = \frac{\sigma_{hs}(F_i)}{F_i} \tag{22}$$

The stress $\sigma_{hs}(F_i)$ is the hot spot stress resulting from the applied load, F_i. The hot spot stresses computed using Equation 21, and the thickness corrected values, are presented in Table 10. Also Mosbron's values are listed for comparison (Mosbron 2022).

Table 10: Hot spot stresses corresponding to the load range

	$\Delta\sigma_{hs}$ [MPa]	$\Delta\sigma_{hs,ref}$ [MPa]
Solid, Sørli, 0.5t and 1.5t	319.8	240.5
Solid, Sørli, 0.4t and 1.0t	357.1	268.6
Solid, Sørli, 0.4t, 0.9t and 1.4t	366.4	275.6
Solid, Mosbron	568.4	427.5
Shell, Mosbron	554.4	417.0
Specimen 1, SG1 and SG2	425.2	319.8
Specimen 1, SG3 and SG4	425.5	320.1
Specimen 2, SG1 and SG2	460.5	346.4
Specimen 2, SG3 and SG4	400.6	301.3

To determine the fatigue life of the component under the applied cyclic loading, the results are plotted against the D design S-N curve. In Figure 33 the results from the solid model in Abaqus are plotted against Mosbron's Abaqus results (Mosbron 2022). In Figure 34 the Abaqus results are plotted against Mosbron's lab experiment results for specimen 1. In Figure 35 the Abaqus results are plotted against Mosbron's lab experiment results for specimen 2. The hot spot stresses that are calculated represents the maximum stress in the critical region of the component, which is likely to experience the most significant cyclic loading. By comparing the hot spot stress against the S-N curve for the material, it is possible to estimate the number of cycles to failure of the component under the applied loading conditions.

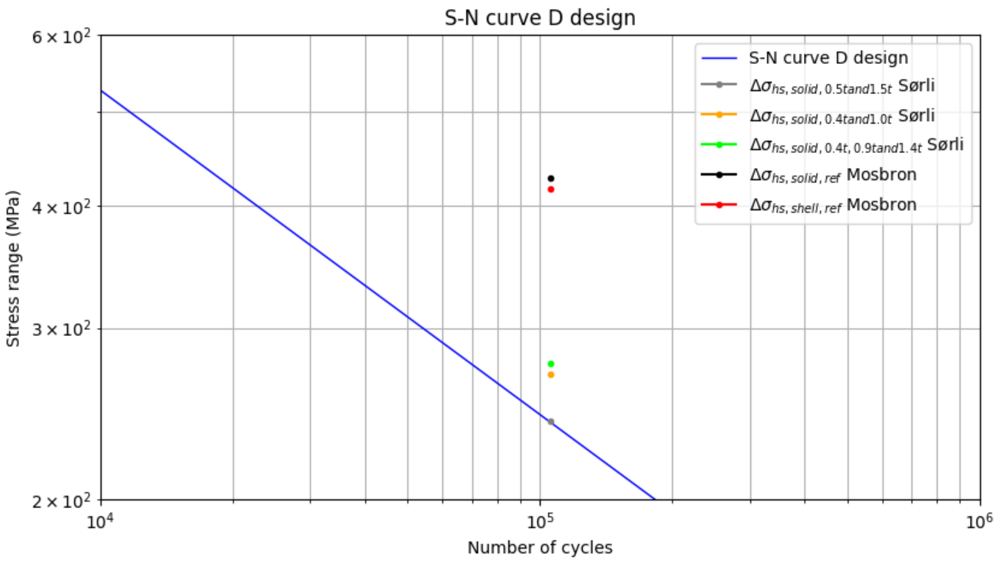


Figure 33: The results from the hot spot method in Abaqus compared to Mosbron's Abaqus results

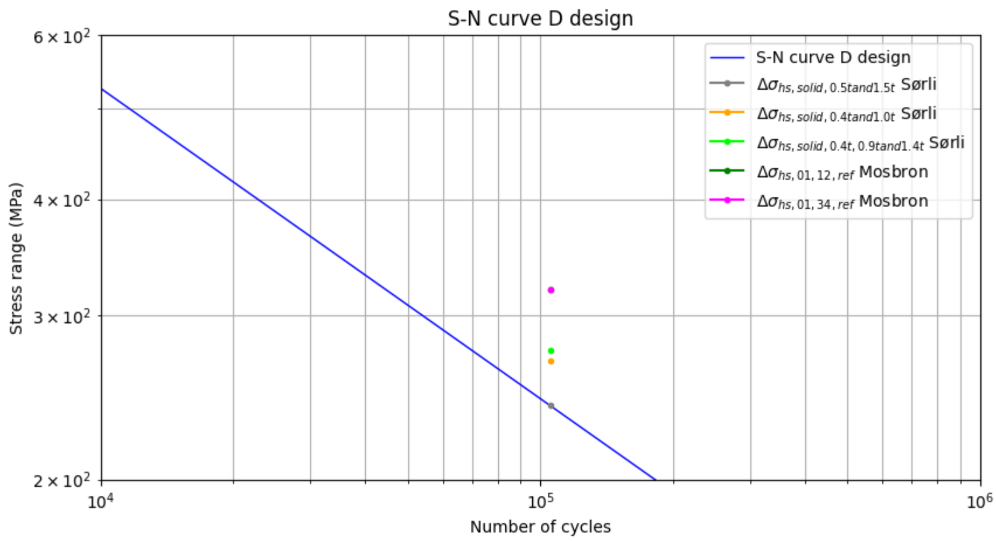


Figure 34: The results from the hot spot method in Abaqus compared to Mosbron's lab experiment results for specimen 1

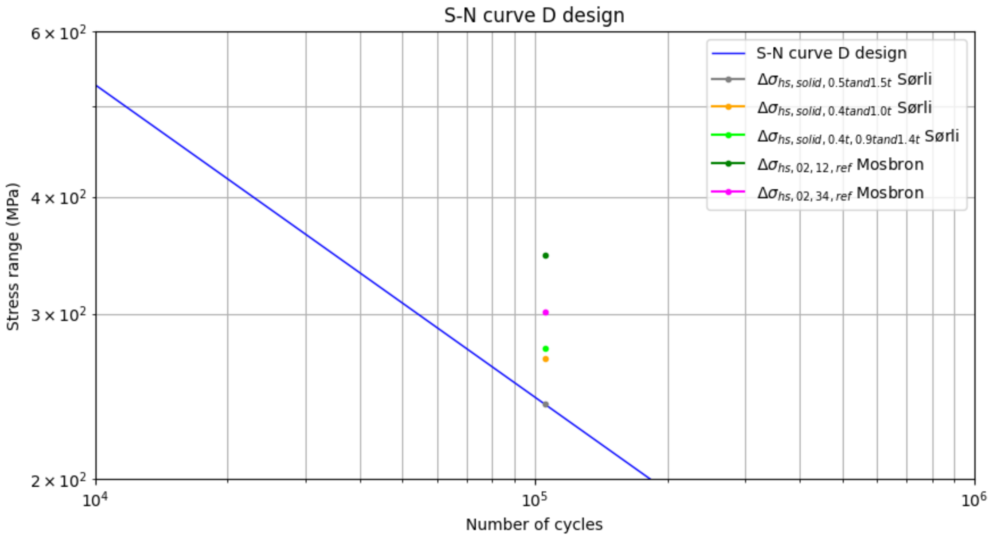


Figure 35: The results from the hot spot method in Abaqus compared to Mosbron's lab experiment results for specimen 2

6.2 Notch stress

6.2.1 Notch stress from the solid model

The notch stress was found in the notch radius, and converged towards a value when the mesh was gradually refined. The values from the convergence study are presented in Table 11, where the mesh in the curvature at the notch has been adjusted.

Table 11: Results for converging notch stress using a submodel

Global mesh	Curvature mesh	Stress (Max. principal)
0.4	8	8.415
0.4	10	8.416
0.4	15	8.418
0.4	18	8.422
0.4	22	8.426
0.4	28	8.433

A plot of the results from Table 11 is shown in Figure 36. The standard deviation, calculated using Equation 23, of the notch stress values is approximately 0.00632 MPa. A small standard deviation suggests that the stress values are consistent and do not exhibit significant fluctuations with further mesh refinement. This is an indicator that the analysis has reached a converged solution, as the stress values are relatively stable and not highly dependent on the mesh density.

$$\sigma = \sqrt{\frac{1}{N}\Sigma_{i=1}^{N}(x_i - \overline{x})^2} \tag{23}$$

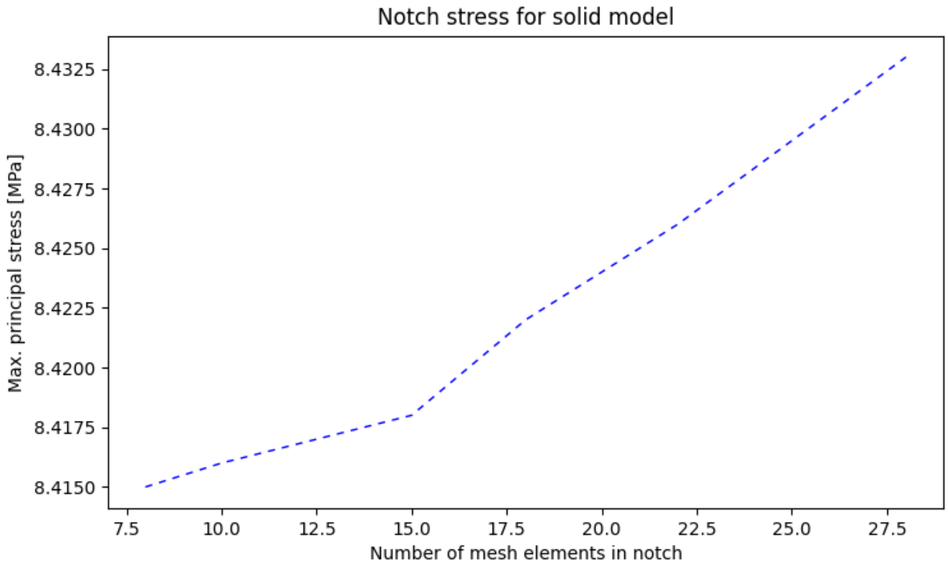


Figure 36: The results from the notch stress convergence study

The resulting notch stress is in Figure 37 plotted into the FAT 225 S-N curve. The result is within the range specified by the S-N curve. Being at or close to the S-N curve implies that the stress level at the notch is within an acceptable range and that the component can withstand the applied cyclic loading for a substantial number of cycles before potential fatigue failure.

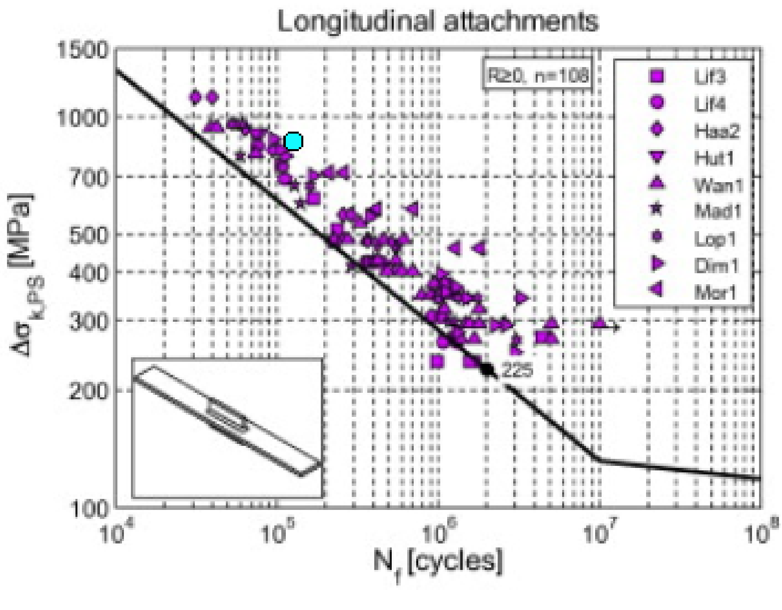


Figure 37: The notch stress plotted in turquoise into the FAT 225 S-N curve

6.2.2 Evaluation of the notch stress method

The stress concentration factor, K_t, for the longitudinal attachments studied by Pedersen et al. lays in the range between 2.69-4.01. These was found when a tensile nominal stress of 1 MPa was applied to the specimen, such that the maximum principle stress observed in the notch corresponds to the SCF. This was explained in Section 3.3. The bracket geometry has a cross-sectional area of $608\,mm^2$ if the girder is neglected from the area under the weld toe. To achieve a nominal stress of 1 MPa in the bracket geometry, the following load in Newton should be applied to the model:

$$F = 608\,mm^2 \cdot 1\,N/mm^2 = 608\,N \tag{24}$$

The resulting maximum principal stress observed in the notch is then 4.96, which is the stress concentration factor. This is then multiplied with the notch stress corresponding to the load range of $210\,kN$:

$$\Delta\sigma_k = K_f \cdot \Delta\sigma_n = 4.96 \cdot 884.31\,MPa = 4386.18\,MPa \tag{25}$$

Table 12: Notch stress corresponding to the load range

	$\Delta\sigma_n$ [MPa]	$\Delta\sigma_{n,ref}$ [MPa]
Solid, Sørli	884.31	4386.18

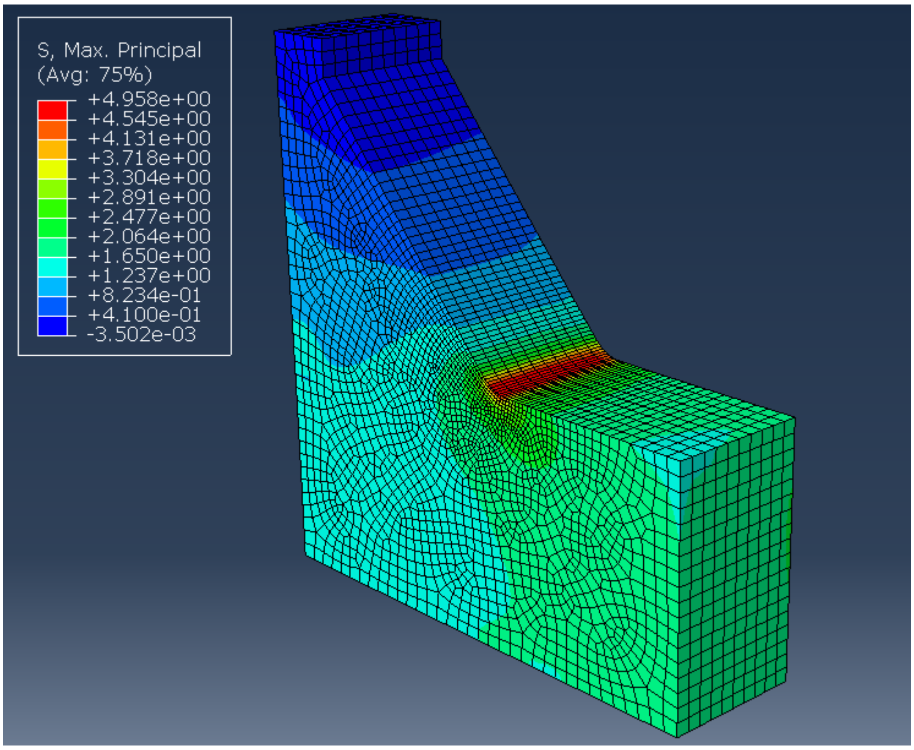


Figure 38: The SCF of the specimen is determined by applying a nominal stress of 1 MPa to the FE model

6.3 Thickness effect

Aker Solutions uses plates with a thickness up to 80 mm. It is possible to calculate the scale factor for lifetime when going from 8 mm to 80 mm using the standard thickness correction formula (Equation 4), with a factor k of 0.25:

$$Scale\ factor = \frac{\Delta S_{80}}{\Delta S_{8}} = \frac{(\frac{80}{25})^{0.25}}{(\frac{8}{25})^{0.25}} = 1.78 \tag{26}$$

When going from a thickness of 8 mm to 80 mm the scale factor is approximately 1.78. This means that the stress range for the 80 mm thickness is about 1.78 times larger than the stress range for the 8 mm thickness when considering the thickness effect.

7 Discussion

7.1 Hot spot method

7.1.1 Comparing the different results

Three different approaches was used to calculate the hot spot stress at the weld toe. The hot spot stresses corresponding to the load range of 210 kN, are in the range of $240.5-275.6$ MPa. Based on the results it appears that all three approaches for calculating the hot spot stress gives somewhat similar values. This indicates some consistency and reliability in the results obtained from these approaches. The two approaches with a point closer to the weld toe at $0.4t$ gives a higher stress range than the case with the first point at $0.5t$. This is because the stress is higher near the weld toe, and the line drawn between the extrapolation point gets steeper, and therefore gives a higher resulting hot spot stress value at the weld toe.

7.1.2 S-N curve

Mosbron's fatigue test failed after $N = 105544$ cycles with a stress range of $\Delta S = 273.5$ MPa. The three different hot spot stress values all lay above the D design S-N curve for $N = 105544$. When the results of a hot spot analysis lay above the relevant S-N curve, it indicates that the calculated stress amplitudes at the hot spot are lower than the fatigue strength predicted by the S-N curve. This suggests that the specimen can withstand the applied stress range for the given number of cycles without experiencing fatigue failure. It suggests that the specimen has a larger safety margin against fatigue failure, and its actual fatigue life may be longer than what is estimated by the S-N curve. But the lowest point plotted against the D design S-N curve lies directly on the S-N curve. It is important to note that while the hot spot stress analysis provides valuable information about the stress distribution and potential fatigue risks, the S-N curve is based on empirical data and may not capture all the complexities of the real-life behavior. Therefore, it is recommended to use conservative estimates and consider other factors such as material variability, operational conditions, and other potential sources of fatigue damage when assessing the fatigue life of the specimen (Q. Bai and Y. Bai 2014).

The results from the hot spot analyses lay significantly below Mosbron's results when plotting them against the D design S-N curve. This indicates that Mosbron's results are more conservative. The reason for this could be the use of different meshing. Mosbron used hexahedral shaped elements and element type 20-node quadratic, solid elements with reduced integration, denoted C3D20R. While in this thesis the model consisted of quadratic tetrahedral elements of type C3D10. Also the refinement of the mesh is different, which could affect the result.

7.1.3 Different hot spots

There are different ways of classifying hot spots, but mainly two ways are used. For type "A", the hot spot is located at the weld toe on a plate surface at an ending attachment. For type "B", the hot spot is located at the weld toe around the plate edge of an ending attachment. See Figure 8. In this study, the type "A" hot spot was considered. Type "B" hot spots take into account geometrical notch effects in the vicinity of the welded joint, as well as unequal stress distribution due to various reasons. In contrast to type "A" hot spots, type "B" hot spots do not depend on the plate thickness for stress extrapolation. They focus on specific locations where stress concentrations occur, such as plate edges or regions with uneven stress distribution. When analyzing welded joint structures, the fatigue strength of these structures may be underestimated when using structural stress concentrations. This means that if the analysis only considers the nominal stress or structural stress without accounting for the localized stress concentrations at type "B" hot spots, the predicted fatigue strength of the welded joint may be lower than the actual performance.

To address this issue, it is important to consider type "B" hot spots and account for the stress concentrations they introduce. This can be achieved through appropriate stress extrapolation techniques and by incorporating the effects of geometrical notches and unequal stress distribution in the analysis. However, it is important to note that the fatigue strength estimated based on type "A" hot spot stresses may still be conservative and may underestimate the actual performance of the welded joint structures. In further work, it may be beneficial to include type "B" hot spots and investigate their influence on the fatigue strength in this case. Comparing the results obtained from both type "A" and "B" approaches can provide a more comprehensive understanding of the structural behavior and help in refining the fatigue assessment methodology for welded joint structures (Lee et al. 2010).

7.2 Notch stress method

7.2.1 Comparing with hot spot stress

The notch stress in Abaqus for a load of 2 kN was calculated to be 8.422 MPa. The highest hot spot stress was calculated as 3.5 MPa for the similar load in Abaqus. Hobbacher states that at weld toes, an effective notch stress of at least 1.6 times the structural hot spot stress should be assumed (Hobbacher 2016). This condition is usually given at welded roots. In this thesis the notch stress is approximately 2.4 times larger than the hot spot stress. This suggests that there is a significant stress concentration and a higher stress concentration factor at the notch locations compared to the overall structural hot spot. This difference indicates that the weld toe regions are experiencing higher stress intensities than what is typically assumed, and may be more susceptible to fatigue failure.

7.2.2 S-N curve

The notch stress corresponding to the load range was calculated to be 884.3 MPa, when the stress concentration factor was not taken into account. When plotting this value into the FAT 225 S-N curve, the point ended up above the curve, in the area with the points furthest away from the curve. The results from the "Mor1" attachment lay in the same lane. This is a longitudinal attachment with the following parameters (Pedersen et al. 2010):

Table 13: Parameters for the "Mor1" detail

Detail	S_y [MPa]	t [mm]	K_t	R	Loading
Mor1	417	12	4.01	0.0	Tension

The thickness of the "Mor1" detail is 12 mm. The specimen studied in this thesis is 8 mm. Five of the other longitudinal attachments plotted into the FAT 225 S-N curve has a thickness of 8 mm, but are closer to the curve and further away from the turquoise point plotted in Figure 37.

7.2.3 Stress concentration factor

When the stress concentration factor is accounted for, the notch stress corresponding to the load range was calculated to be 4396.2 MPa. It is important to note that the SCF is an approximation and assumes that the stress distribution in the vicinity of the notch follows a certain pattern. However, in practical scenarios, the actual stress distribution can be more complex, and the SCF may not fully capture all the local effects. It is possible that the chosen SCF value is not representative of the actual stress concentration at the weld toe. The SCF can vary depending on factors such as geometry, loading conditions, and material properties. Therefore, it's crucial to ensure that the selected SCF is appropriate for the specific geometry and loading scenario being analyzed. This result should therefore be studied in more detail.

The stress concentration phenomena is always a matter of concern to structural engineers engaged in performing the fatigue assessment. This is the reason why the notch fatigue problem has extensively been investigated over the last century, so that nowadays there exists a variety of criteria suitable for successfully taking into account the damaging effect of stress raisers when subjected to fatigue loading. Susmel investigated notches that had a root radius ranging between 0.01 mm and 4 mm, resulting in K_t values varying from 1.6 up to 20 (Susmel 2009).

7.2.4 Mesh convergence study

For the notch stress method, a mesh convergence study was performed. The mesh convergence study was performed in the weld toe area only. During the analysis, it was observed that changing the mesh in other areas had only minor effects on the weld root stress. This observation suggests that the stress distribution in those regions is less influenced by mesh refinement. This implies that the critical stress concentrations primarily occur at the weld toe, and it is crucial to accurately capture the stress behavior in these specific locations.

7.3 Meshing

For the solid model in this study, tetrahedral elements was used to mesh the geometry. Fricke et al. 1997 recommend to use 20-node solid elements that have a side length of the plate thickness at the hot spot. They also recommend at least three elements of equal length in the area where the stress increases.

In general, hexahedral elements tend to provide better accuracy than tetrahedral elements for structural analyses. Hexahedral elements can better capture the stress and strain distributions within the model due to their regular shape and reduced distortion. This is particularly important when analyzing stress concentration regions like weld toes, where accurate stress prediction is crucial. However, quadratic tetrahedral elements can also provide reasonable accuracy for many applications, especially when the mesh is sufficiently refined.

Tetrahedral elements, such as the quadratic tetrahedral elements (C3D10), are commonly used due to their flexibility in handling complex geometries, including irregular shapes and curved surfaces. They require relatively simple mesh generation, and their reduced element count can lead to computational efficiency, especially for large and complex models. However, tetrahedral elements may exhibit certain limitations, such as lower accuracy in representing bending and volumetric behavior, as well as potential issues with element distortion and mesh convergence.

On the other hand, hexahedral elements, like the quadratic hexahedral elements (C3D20R), are known for their superior accuracy in representing bending and volumetric effects. They are more suitable for modeling structures with predominantly bending or volumetric behavior, and they provide better convergence properties in certain cases. However, generating hexahedral meshes can be more complex, especially for irregular geometries or regions with curved surfaces (Smith 2009).

In the context of the hot spot and notch stress methods, both tetrahedral and hexahedral elements can be used effectively. The most critical aspect is to ensure that the mesh is adequately refined near the weld toe and other regions of interest to capture the stress concentrations accurately. Performing a mesh sensitivity analysis by comparing the results obtained from different element types and mesh densities can help assess the convergence and accuracy of the results.

7.4 Lab experiment

Performing a fatigue test in a laboratory is an important step in validating and verifying the accuracy of numerical models and predictions, such as the calculation of the hot spot stress at the weld toe. It was supposed to be done during this semester for this thesis, but unfortunately got cancelled. By comparing the numerical results with experimental data, you can assess the agreement between the two and ensure that the model accurately represents the real-world behavior. This validation process enhances confidence in the numerical analysis and its ability to predict the response of the bracket geometry.

Lab experiments allow you to replicate the actual loading and environmental conditions experienced by the bracket geometry. This includes factors such as material properties, weld quality, stress distribution, and loading conditions. By conducting experiments, you can capture the intricacies and complexities of the physical system that may not be fully captured in the numerical model. This helps to ensure that the numerical analysis reflects the real-world behavior and improves the reliability of the results.

Comparing numerical results with experimental data can reveal any limitations or deficiencies in the numerical model. If discrepancies exist between the numerical and experimental results, it indicates areas where the model may be inaccurate or incomplete. This insight can lead to model refinements, parameter adjustments, or further investigations to improve the accuracy of the numerical predictions.

Lab experiments provide valuable information on the fatigue life, failure mechanisms, and structural integrity of the bracket geometry. By comparing numerical results with experimental data, you can ensure that the design meets safety requirements and predict the remaining life of the structure accurately. This is particularly important when the failure of the bracket geometry can have severe consequences.

7.5 Fracture surface

When Mosbron conducted a lab experiment on a test specimen in the spring of 2022, the test specimen experienced yielding before fracture, and a reduction in the cross-sectional area was observed. See Figure 40. Experiencing yielding before fracture in the lab experiment is not necessarily a problem, especially if the material in the bracket geometry is expected to exhibit ductile behavior. Yielding indicates that the material has reached its yield strength and has undergone plastic deformation.

The ductile fracture observed in the specimen, characterized by plastic deformation before fracture, suggests that the material has the ability to undergo large deformation before failure. This indicates a ductile failure mode, which is generally desirable as it provides warning signs such as plastic deformation, giving time for inspection and maintenance before catastrophic failure.

To increase confidence in the accuracy of the lab experiment and its applicability to the specific bracket geometry, it is beneficial to conduct multiple experiments. Performing tests with varying parameters, such as different load levels or different specimen geometries, can provide a broader understanding of the material's behavior and fracture characteristics. It allows for a more comprehensive analysis and assessment of the fatigue life and failure modes.

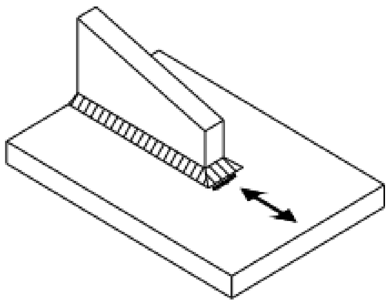


Figure 39: Fatigue crack growth from the weld toe into the base material (DNV-GL 2014)

It should be noted that any element or member of the structure, every welded joint and attachment or other form of stress concentration, is potentially a source of fatigue cracking and should be individually considered (DNV-GL 2021).

Figure 40: The fatigue crack in the test specimen after Mosbron's fatigue test

The fatigue analysis should be based on S-N data, determined by fatigue testing of the considered welded detail, and the linear damage hypothesis. When appropriate, the fatigue analysis may alternatively be based on fracture mechanics. If the fatigue life estimate based on S-N data is short for a component where a failure may lead to severe consequences, a more accurate investigation considering a larger portion of the structure, or a fracture mechanics analysis, should be performed. For calculations based on fracture mechanics, it should be documented that there is a sufficient time interval between time of crack detection during in-service inspection and the time of unstable fracture (DNV-GL 2021).

7.6 Thickness effect

The stress range for the 80 mm thickness is about 1.78 times larger than the stress range for the 8 mm thickness when considering the thickness effect. This means that the stress concentration at critical locations, such as weld toes or notches, also increases. This increased stress concentration can lead to higher stress amplitudes, which can potentially result in reduced fatigue life and an increased likelihood of failure. In fatigue analysis, it is crucial to consider the thickness effect as thicker components tend to experience higher stress concentrations due to geometrical changes or load redistribution.

The thickness effect can be accounted for by multiplying a factor of $\left(\frac{t}{t_{ref}}^{k}\right)$ by the stress range. In general, the thickness correction to the design equation for the S–N curve is required when the plate thickness is thicker than the reference thickness. To some extent, the thickness correction also accounts for the size of the weld and its attachments. However, it does not account for the weld length or the length of a component that is different from the tested component (Y. Bai and Jin 2016).

An increased plate thickness generally leads to a higher stress concentration factor if the local weld toe geometry remains the same. This means that the stress is more concentrated at the weld toe, potentially leading to higher stress levels and reduced fatigue life. The stress gradient, which represents the rate of stress change along the plate thickness, becomes less steep with increased thickness. This can result in higher crack tip stresses at a given crack depth in a thick plate compared to a thin plate (Berge 1985).

Pedersen concludes with the following negative influences of plate thickness (Pedersen 2019):

- The stress concentration factor increases with increased thickness if the local weld toe geometry remains the same.
- The stress gradient becomes less steep with increased thickness, thus giving rise to higher crack tip stresses at a given crack depth in a tick plate compared to a thin.

To account for the thickness effect and obtain accurate fatigue life predictions, it is recommended to validate the analysis results against experimental data or well-established standards. This can help assess the reliability of the thickness correction factor and ensure that the chosen approach adequately accounts for the influence of plate thickness on the fatigue behavior of the welded component.

8 Conclusion

The three different approaches for calculating the hot spot stress provided quite similar results. The fact that the different approaches provided similar stress results is encouraging. It suggests that the hot spot stress calculations are relatively robust and not heavily dependent on the specific approach used. This consistency is important for ensuring the reliability and repeatability of the analysis. The approach where the extrapolation points were set to $0.5t$ and $1.5t$ ended up closer to the S-N curve suggesting that it may provide a more conservative estimate of the fatigue life. This can be beneficial from a design perspective as it allows for a more conservative assessment of the structural integrity. Moving further away from the weld toe (increasing the extraction distance) likely captures a broader region of the stress field, leading to a more representative hot spot stress value. The extraction points closer to the weld toe may be more influenced by stress concentrations, resulting in higher stress values.

The results for the hot spot analysis lay above the relevant S-N curve. This indicates that the calculated stress amplitudes at the hot spot are lower than the fatigue strength predicted by the S-N curve. This suggests that the specimen can withstand the applied stress range for the given number of cycles without experiencing fatigue failure. In this case, it would be considered a conservative estimation, indicating a higher fatigue life for the structure.

The notch stress was calculated to be 2.4 times larger than the hot spot stress. This indicates a significant stress concentration at the weld toe due to the geometry of the weld notch. This high stress concentration can result in localized stress concentrations and increase the likelihood of fatigue crack initiation and propagation.

When it comes to the thickness effect, it has some impact on the results. The increase in thickness from 8 mm to 80 mm introduces a substantial increase in stress concentration and stress amplification, which should be considered when assessing the fatigue performance and durability of the component. By accounting for the thickness effect using appropriate correction factors or equations, it is possible to estimate the actual stress levels and evaluate the fatigue life of the component more accurately.

9 Further work

The numerical analysis provides valuable insights, but it is essential to validate the results through experimental testing. Conducting lab experiments on test specimens, as mentioned before, can help verify the accuracy of the numerical predictions and provide additional data points for comparison. It would be necessary to perform more experiments for this specimen to get more insight.

It could be interesting to study the type ”B” hot spots in an analysis. Type ”B” hot spots take into account geometrical notch effects in the vicinity of the welded joint, as well as unequal stress distribution due to various reasons. Studying both ”A” type and ”B” type hot spots can provide a more comprehensive understanding of the behavior and failure mechanisms in the welded bracket geometry. Considering the ”B” type hot spot can offer additional insights and help evaluate the overall structural integrity.

There is a great uncertainty related to the stress concentration factor found for the notch stress. It is possible that the chosen stress concentration factor value is not representative of the actual stress concentration at the weld toe. Determining the accurate stress concentration factor is crucial in calculating the notch stress more accurately using the notch stress method. While the obtained SCF has a value of 4.96 in the analysis, it is important to evaluate its reliability and consider additional steps to obtain a more accurate result. To assess the accuracy of the SCF obtained, it is possible to compare it with experimental data or reference values available for similar geometries and loading conditions. If such data is not available, it may be necessary to rely on established guidelines, literature references, or expert judgment. It is essential to ensure that the SCF value aligns reasonably well with established practices to instill confidence in its accuracy.

Bibliography

Almar-Næss, A. et al. (1985). *Fatigue Handbook - Offshore Steel Structures*. Tapir, Norges Tekniske Høgskole. ISBN: 82-519-0662-8.

Bai, Qiang and Yong Bai (2014). '12 - Fatigue and Fracture'. In: ed. by Qiang Bai and Yong Bai, pp. 283–318. DOI: https://doi.org/10.1016/B978-0-12-386888-6.00012-2. URL: https://www.sciencedirect.com/science/article/pii/B9780123868886000122.

Bai, Yong and Wei-Liang Jin (2016). 'Chapter 25 - Fatigue Capacity'. In: *Marine Structural Design (Second Edition)*. Ed. by Yong Bai and Wei-Liang Jin. Second Edition. Oxford: Butterworth-Heinemann, pp. 489–507. ISBN: 978-0-08-099997-5. URL: https://www.sciencedirect.com/science/article/pii/B9780080999975000253.

Berge, Stig (1985). 'On the effect of plate thickness in fatigue of welds'. In: *Engineering Fracture Mechanics*. ISSN: 0013-7944. DOI: https://doi.org/10.1016/0013-7944(85)90030-X. URL: https://www.sciencedirect.com/science/article/pii/001379448590030X.

Berge, Stig and Sigmund Kyrre Ås (2017). *Compendium - Fatigue and Fracture Design of Marine Structures*. 3rd Edition. NTNU - Faculty of Engineering Science and Technology.

Cen, Duofeng et al. (2020). 'Direct Tensile Behavior of Limestone and Sandstone with Bedding Planes at Different Strain Rates'. In: *Rock Mechanics and Rock Engineering*. DOI: 10.1007/s00603-020-02070-x. URL: https://doi.org/10.1007/s00603-020-02070-x.

Dey, Anup Kumar (2021). 'Brittle Fracture and Ductile Fracture: Definition, Mechanism, Differences'. In: *What is piping*. URL: https://whatispiping.com/brittle-fracture-ductile-fracture/.

DNV-GL (2014). 'Recommended practice: Fatigue design of offshore steel structures'. In: *Det Norske Veritas*.

— (2021). 'Recommended practice: Fatigue design of offshore steel structures'. In: *Det Norske Veritas*.

Fricke, Wolfgang (2012). 'IIW Recommendations for the Fatigue Assessment of Welded Structures by Notch Stress Analysis: IIW-2006-09'. In: *IIW Recommendations for the Fatigue Assessment of Welded Structures by Notch Stress Analysis*. Woodhead Publishing Series in Welding and Other Joining Technologies. Woodhead Publishing. ISBN: 978-0-85709-855-9. DOI: https://doi.org/10.1533/9780857098566.3. URL: https://www.sciencedirect.com/science/article/pii/B978085709855950001X.

Fricke, Wolfgang, Hansjörg Petershagen and Hans Paetzold (1997). 'Fatigue Strength of Ship Structures'. In.

Hobbacher, A. F. (2016). *Recommendations for Fatigue Design of Welded Joints and Components.* Springer International Publishing AG. ISBN: 978-3-319-23756-5. DOI: 10.1007/978-3-319-23757-2.

Khanna, S.K. and X. Long (2010). 'Failure Mechanisms of Advanced Welding Processes'. In: DOI: https://doi.org/10.1533/9781845699765.frontmatter. URL: https://www.sciencedirect.com/science/article/pii/B9781845695361500123.

Lee, Jae-Myung et al. (2010). 'Comparison of hot spot stress evaluation methods for welded structures'. In: *International Journal of Naval Architecture and Ocean Engineering.* ISSN: 2092-6782. DOI: https://doi.org/10.2478/IJNAOE-2013-0037. URL: https://www.sciencedirect.com/science/article/pii/S2092678216302497.

Liu, G. R. and S. S. Quek (2014). 'Fundamentals for Finite Element Method'. In: *The Finite Element Method*, pp. 43–79. DOI: 10.1016/B978-0-08-098356-1.00003-5.

Lotsberg, Inge (2016). *Fatigue Design of Marine Structures.* Cambridge University Press. ISBN: 9781316343982. DOI: 10.1017/CBO9781316343982.003.

Mecséri, B. J. and B. Kövesdi (2020). 'Discussion on the Hot-Spot and Notch Stress Based Fatigue Assessment Methods Based on Test Results'. In: *International Journal of Steel Structures.* DOI: 10.1007/s13296-020-00345-1. URL: https://doi.org/10.1007/s13296-020-00345-1.

Moan, Torgeir (2003). *Compendium - Finite Element Modelling and Analysis of Marine Structures.* NTNU - Faculty of Engineering Science and Technology.

Mosbron, Erlend (2022). 'Master's thesis: Evaluation of the Hot-Spot Method for a Bracket Geometry'. In.

Pedersen, Mikkel Melters et al. (2010). 'Re-analysis of fatigue data for welded joints using the notch stress approach'. In: *International Journal of Fatigue.* ISSN: 0142-1123. DOI: https://doi.org/10.1016/j.ijfatigue.2010.03.001. URL: https://www.sciencedirect.com/science/article/pii/S0142112310000575.

Pedersen, Mikkel Melters (2019). 'Thickness Effect in Fatigue of Welded Butt Joints'. In: *International Journal of Steel Structures.* DOI: 10.1007/s13296-019-00254-y. URL: https://doi.org/10.1007/s13296-019-00254-y.

Radaj, Dieter (1990). *Design and Analysis of Fatigue Resistant Welded Structures.* Woodhead Publishing. ISBN: 978-1-85573-004-5. DOI: https://doi.org/10.1533/9781845698751.frontmatter. URL: https://www.sciencedirect.com/science/article/pii/B9781855730045500015.

Rege, K. and H. Lemu (2017). 'A review of fatigue crack propagation modelling techniques using FEM and XFEM'. In: *IOP Conference Series Materials Science and Engineering.* DOI: 10.1088/1757-899X/276/1/012027.

Smith, Michael (2009). *ABAQUS/Standard User's Manual, Version 6.9.* English. United States: Dassault Systemes Simulia Corp.

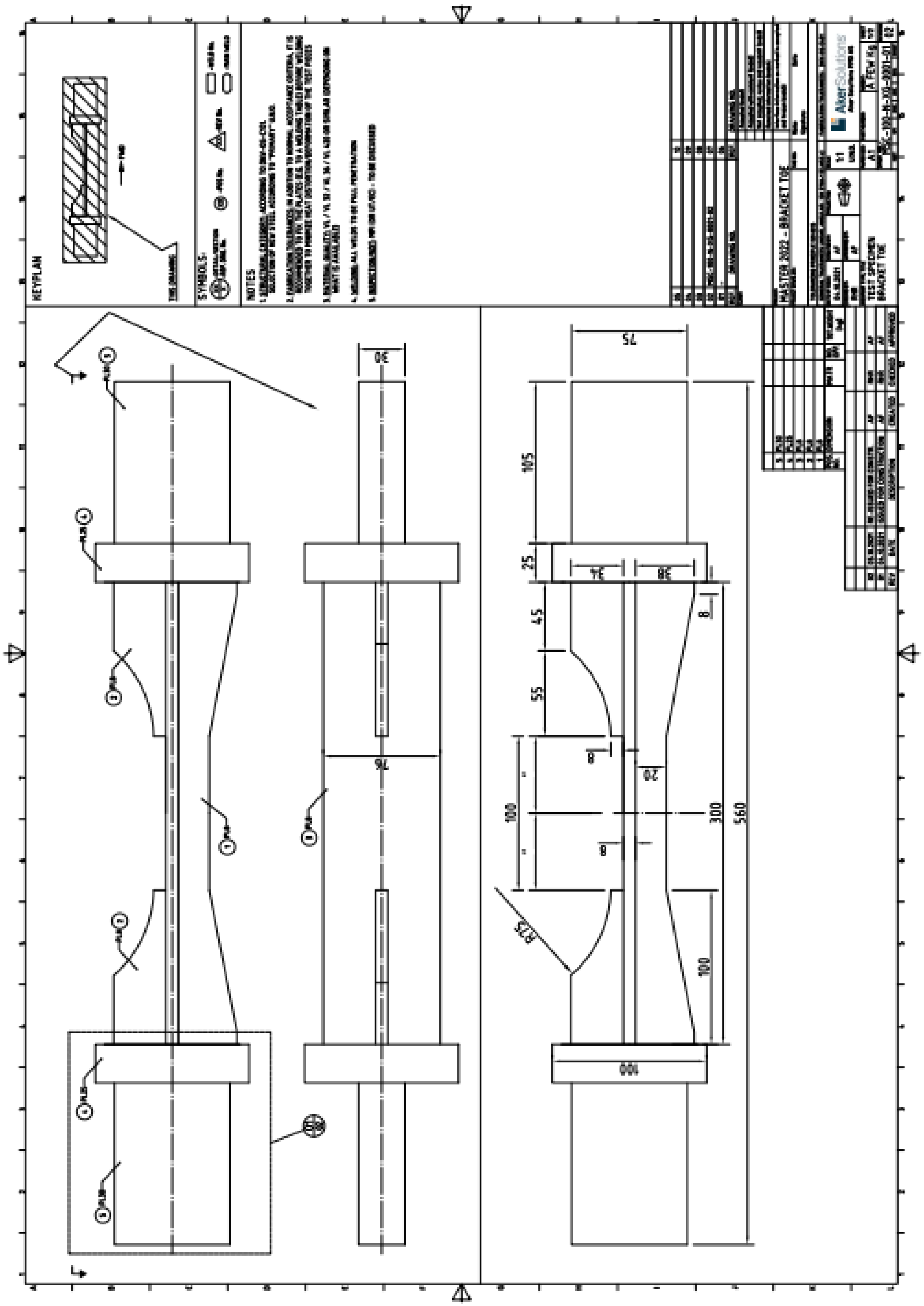
Susmel, Luca (2009). *Multiaxial notch fatigue - From nominal to local stress/strain quantities.* Woodhead Publishing Limited.

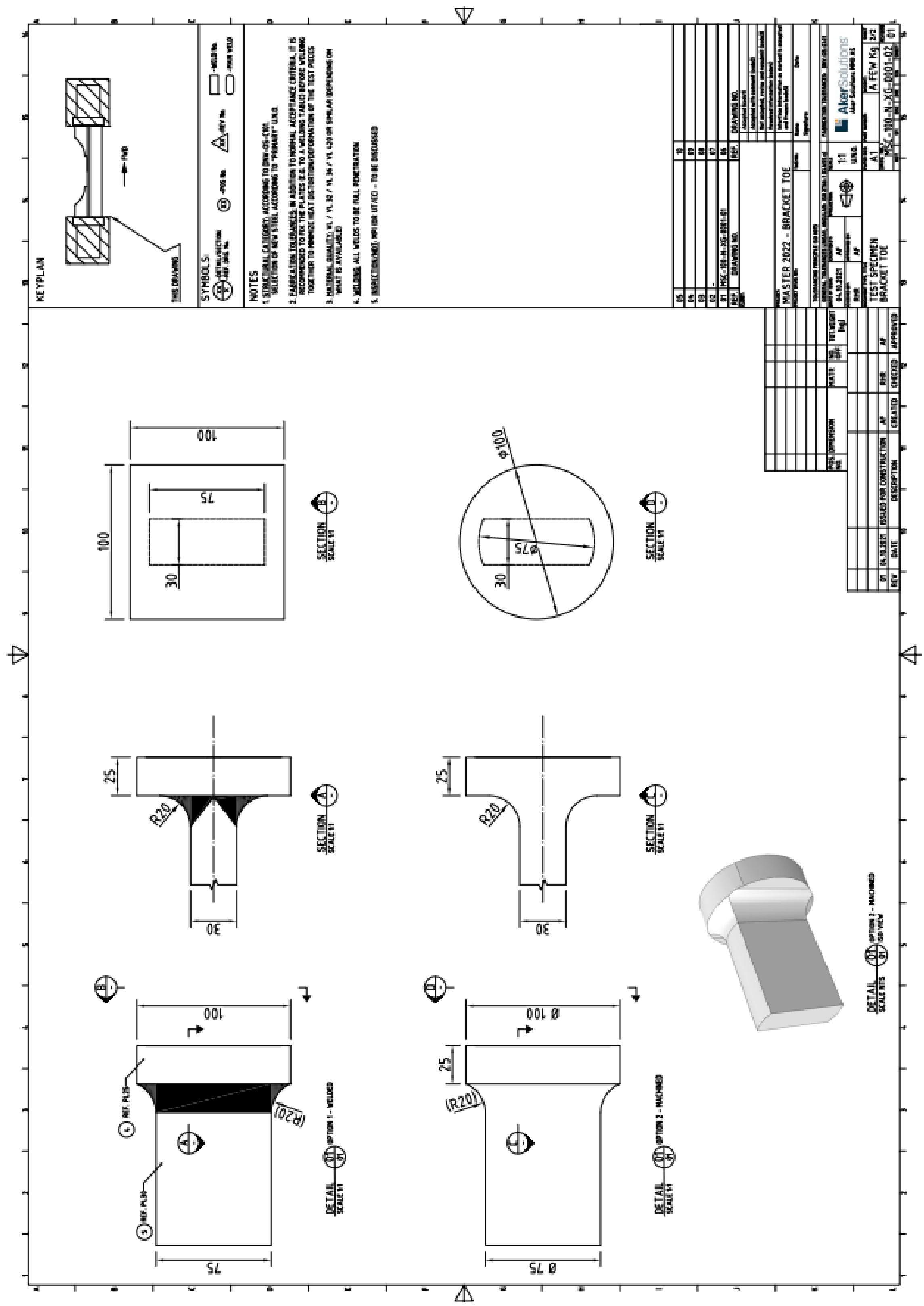
Thomas, Daniel J. (2018). 'Analyzing the Failure of Welded Steel Components in Construction Systems'. In: *Journal of Failure Analysis and Prevention.* DOI: 10.1007/s11668-018-0392-x. URL: https://doi.org/10.1007/s11668-018-0392-x.

Zamzami, Ibrahim Al and Luca Susmel (2018). 'On the use of hot-spot stresses, effective notch stresses and the Point Method to estimate lifetime of inclined welds subjected to uniaxial fatigue loading'. In: *International Journal of Fatigue.* ISSN: 0142-1123. DOI: https://doi.org/10.1016/j.ijfatigue.2018.08.032. URL: https://www.sciencedirect.com/science/article/pii/S0142112318304390.

Appendix

A Technical drawing of bracket toe




KEYPLAN
SYMBOLS
NOTES
MASTER 2022 - BRACKET TOE
TEST SPECIMEN
BRACKET TOE
AkerSolutions
SECTION
DETAIL
100
75
30
25
R20
ø100
ø75

NTNU

Norwegian University of
Science and Technology