Throat Size Optimization for a Welded Boom Base

Puumalainen Harri

Thesis

Bachelor's degree

SAVONIA-AMMATTIKORKEAKOULU	**OPINNÄYTETYÖ** **Tiivistelmä**

Koulutusala Tekniikan ja liikenteen ala			
Koulutusohjelma Kone- ja tuotantotekniikan koulutusohjelma			
Työn tekijä(t) Harri Puumalainen			
Työn nimi Puomin rungon hitsaussauman a-mitan optimointi			
Päiväys	27.4.2012	Sivumäärä / Liitteet	49/2
Ohjaaja(t) T&K insinööri Pekka Holopainen, projektipäällikkö Tatu Westerholm			
Toimeksiantaja/Yhteistyökumppani(t) Normet Oy			

Tiivistelmä

Työn tarkoituksena oli tarkastella puomin rungon hitsaussauman a-mittaa kaivosajoneuvossa, joka on valmistettu räjähdysaineen panostukseen erilaisissa kaivoksissa. Nykyisin käytettävä a-mitta on 20 mm, jonka on epäilty olevan tarpeettoman suuri, ja mikä vaikuttaa suoranaisesti koneen valmistuskustannukseen.

Työ koostuu laskennallisesta ja soveltavasta osuudesta, joiden avulla tulosten paikkaansa pitävyys voidaan varmistaa. Laskennallisessa osiossa tarkasteltavasta kohteesta luotiin erillinen 3D-malli, jota tutkittiin elementtimenetelmän avulla. Soveltavassa osuudessa tarkasteltavalle kohteelle suoritettiin venymäliuskamittaus, jonka tuloksia verrattiin elementtimenetelmällä saatuihin tuloksiin. Venymäliuskamittausten tulosten perusteella oli myös mahdollista määrittää hitsaussauman kestoikä erilaisissa dynaamisissa rasitustilanteissa.

Tulosten perusteella on mahdollista laskea, kuinka suuria jännityskertymiä hitsaussaumaan muodostuu ja onko hitsausauman pienentäminen täten mahdollista.

Avainsanat
hitsaussauma, a-mitta, CAD, FEM, FEA, väsyminen, venymäliuska

Julkinen. Tämä versio ei sisällä yksityiskohtaisia tuloksia

SAVONIA UNIVERSITY OF APPLIED SCIENCES **THESIS Abstract**

Field of Study Technology, Communication and Transport			
Degree Programme Degree Programme in Mechanical Engineering			
Author(s) Harri Puumalainen			
Title of Thesis Throat Size Optimization for a Welded Boom Base			
Date	27 April 2012	Pages/Appendices	49/2
Supervisor(s) Mr. Pekka Holopainen, R&D Engineer Mr. Tatu Westerholm, Project Manager			
Client Organisation/Partners Normet Oy			

Abstract

The purpose of this thesis was to inspect the welding beam in a boom base, which is manufactured for explosive charging in different kind of mines. Nowadays the used throat size is 20 mm which is suspected to be unnecessarily thick and this has a straight influence on manufacturing costs.

The thesis consists of a theoretical and practical part with which the accuracy of the results can be ensured. In the theoretical phase a unique 3D-model was created which was analyzed with the finite element method. In the practical phase strain gage measurements were done on the part which was under examination. After measurements these results were compared to the finite element method results. With strain gage measurements it was also possible to determine the estimated life of the weld in different kind of loading conditions.

Based on these results it will be possible to determine what kinds of stresses are formed on the weld and possibility to decrease the throat size of the particular weld can be calculated.

Keywords
Welding Beam, Throat Size, CAD, FEM, FEA, Fatigue, Strain Gage

Public. This version does not include the detailed results.

FOREWORDS

I would like to express my gratitude to Mr. Asko Kettunen, R&D Manager, Mr. Pekka Holopainen, R&D Engineer, Mr. Tatu Westerholm, Project Manager and Mr. Mikko Huusko, Project Engineer for all support and supervision during the this project.

I also want to thank the personnel of Normet Oy and Savonia University of Applied Sciences for giving me great working environment and a possibility to do this work.

Kuopio 27 April 2012

TABLE OF CONTENTS

APPENDICES

Other appendix are not available on public version.

NOMENCLATURE

CAD	Computer Aid Drawing
CHARMEC 605	Charging machine of Normet Group
EN 280:2001+A2:2009	Standard of Mobile elevating work platforms
FEA	Finite Element Analysis
FEM	Finite Element Method
FOS = SF	Factor of Safety = Safety factor
LTC	Life Time Care
MEWP	Mobile elevating work platforms
PDM	Product Data Management
A	Surface area, mm^2
E	Young´s modulus, Pa
F	Force, N
G	Shear modulus, Pa
R	Electrical resistance, Ω
ReL	Lower yield point, MPa
S-N	Stress – number of cycles
$[K]$	Stiffness matrix of structure
$\{U\}$	Displacement vector of the structure
$\{F\}$	Total force vector of the structure
ρ	Resistivity, Ω·m
ε	Strain, m
σ	Stress, N / mm^2 = MPa
σ_x	Stress x-axis, MPa
σ_y	Stress y-axis, MPa
σ_z	Stress z-axis, MPa
σ_1	First principal stress, MPa
σ_2	Second principal stress, MPa
σ_3	Third principal stress, MPa
σ_{hs}	Calculated hot spot stress, MPa
σ_{eq}	Equivalent stress cycle
τ	Shear stress, MPa
τ_{xy}	Shear stress xy-plane, MPa
τ_{xz}	Shear stress xz-plane, MPa
τ_{yz}	Shear stress yz-plane, MPa
ν	Poisson´s ratio, -
$\perp$	Perpendicular direction
$\|\|$	Parallel direction

1 PREFACE

The purpose of this thesis is to determine the optimum size of the throat thickness between welded parts. These parts are used in Charmec 605 mining machine between a NT100 chassis and a boom mounting frame. Because Charmec 605 is a mobile elevating work platform, the design has to comply with the EN 280:2001 standard.

When a durable joint is needed, welding is one of the most common ways to attach steel parts together. When measuring a welding size the right term to use is throat thickness. Throat thickness tells the weld height from the root of the weld to the weld face in millimeters. The needed throat thickness is mostly determined by forces which have an impact on the parts.

The production department of Normet has suspected that the weld between these parts is too thick and in this way it is not as productive as possible. The throat thickness is now 20 mm but production department has estimated that a throat thickness of 12 mm would be enough. It has been evaluated that reducing the welding size by 8 mm can lower the welding time to the half and thus in practice cut down the welding time from two to one shift. In a year, these kind of chassis are manufactured around 100 pieces so if one hour of welding costs 40 €, the yearly saving would be around 15 000 to 30 000 €.

In this thesis, three kinds of steps (Figure 1) are used to solve optimum throat thickness. The first step is to make finite element method (FEM) analysis with the calculated forces and verify the created model with a strain gage verification. If this process is acceptable, the second step is to use actual forces in FEM and compare the results of different throat sizes. The final step is carry out a fatigue analysis of the welds. This is an important step because fatigue is a common reason for welded parts to get broken. If the results of these methods are accepted, then the calculation of throat thickness can be done with the FEM model.

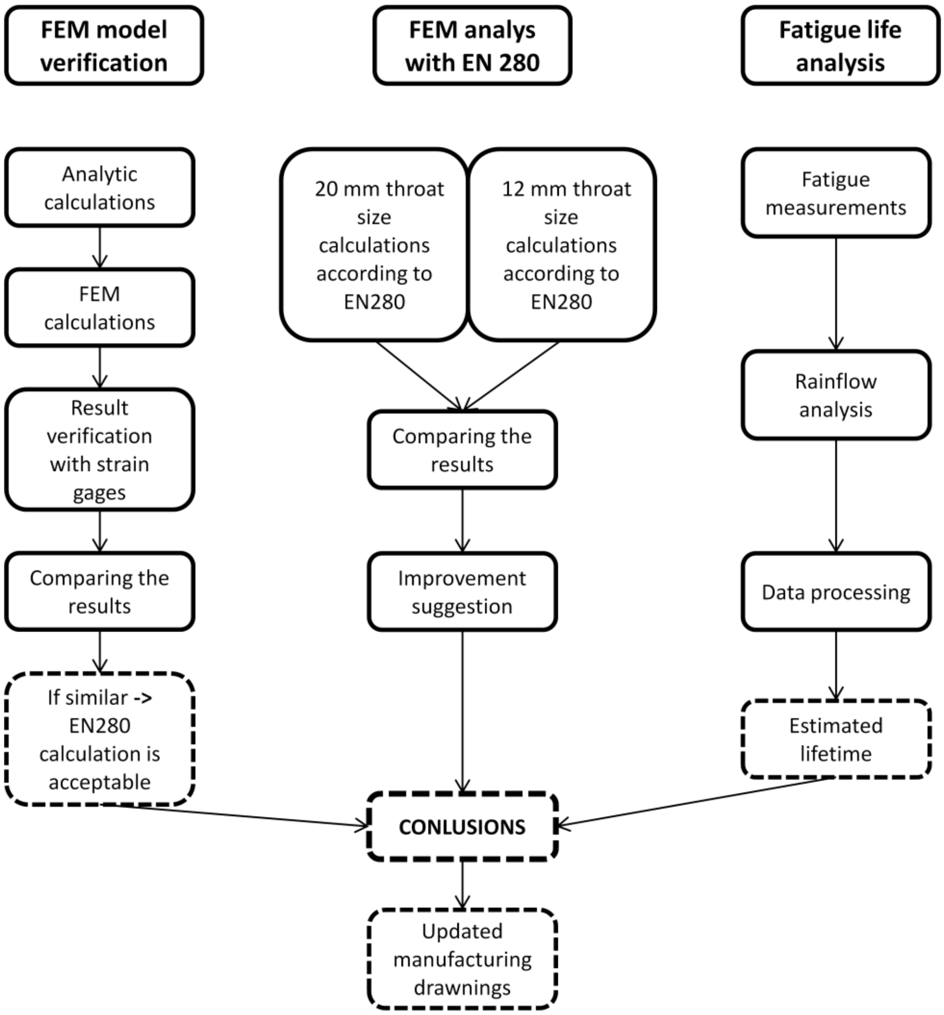


Figure 1. Thesis process

2 BACKGROUND

2.1 Normet Group

Normet provides different kind of solutions for underground mining and tunneling needs. The company has almost 50 years of experience in the development, production and sales of underground mining equipments. With over 7500 manufactured machines, Normet Oy has become one of the most significant manufacturers in its segment.

Number one priority in the company is to make customers satisfied by exceeding their expectations. Safety, quality of products, environment aspects and great co-operative network are the main priorities of working. These working aspects are combined to ISO 9001 *Quality standard*, ISO 14001 *Environment certification* and OH-SAS 18001:2007 *Safety certification*.

Main products of the company are concrete spraying, explosive charging, lifting, transport and scaling machines and also installations services. To keep these machines and processes running smoothly Normet also provides Life Time Care (LTC) which includes a full range of services.

The company’s head office, manufacturing and R&D functions are located in Iisalmi, Finland. Normet employs over 600 workers in 23 locations in 16 countries all over the world. Normet Group's turnover in 2010 was over EUR 115 million. (Normet Group)

2.2 Charging

Charging machines are used to blast soil in underground conditions. To ensure the best result it is very important to use good quality drilling, right kind of blasting operation and proper kind of bulk explosives (example ammonium nitrate-fuel oil, ANFO). To fill these above mentioned steps is the way to success. To maximize the benefits of the right kind of mining technique, it is important to use equipment which is designed to work in tough conditions. An example of this kind of machine is Charmec MC605 which is shown in Figure 2.

Figure 2. Charmec MC605. (Normet Group).

2.3 Standard for Mobile Elevating Work Platform

SFS-EN 280: 2001 + A2: 2009, *European standard for mobile elevating work platforms* includes important issues which has to be noted and verified during the design. These kinds of things are design calculations, stability criteria, construction, safety, examination and tests. Because Charmec 605 includes a lifting platform the design process has to pay attention to this standard.

The meaning of this standard is to define safety introduction to peoples and their property against the risk of accident with the operation of Mobile Elevating Work Platforms (MEWP). Standard determinate technical safety requirements and measures to all types and sizes of MEWPs which are designed to lift persons to working position where they can carrying out work from the work platform. Because there are no previous acceptable national standard for explanation of dynamic factor in stability calculations, the test results of the CEN/TC 98/WG 1 workgroup determinate suitable factors and calculations to MEWPs. (SFS-EN 280: 2001 + A2: 2009, 2009)

When defining the safety factors for load and forces next 6 steps have to take account of:

- Rated load
- Structural loads
- Wind loads
- Manual forces
- Special loads and forces
- Dynamic factor

More precise information of the definitions can be found in the standard.

A primary design for all Normet machines with a mobile elevating work platform is developed and produced with SFS-EN 280 standard, but there are also some exceptions in the structure engineering. When comparing standards among different countries, the European standard is the most demanding in safety. Some countries may have some special requirements for certain features which need to be taken into account when the designed machines are delivered outside of the European area.

2.4 Machine Description

2.4.1 Technical Perspective

The Charmec 605 product family includes five different models which have the same purpose of usage but with some differences. These models are the MC 605 short, MC 605 long, LC 605 short, LC 605 long and LC 605 VE(C).

The Charmec 605 family is designed for charging in mines and tunnels with up to 65 m^2 cross sections where the maximum face height is 8.4 m. It weighs from 15 000 kg to 23 000 kg in full operating phase. For example 605 main dimensions are shown in Figure 3. The power source of these machines is a liquid cooled turbo charged Mercedes-Benz 904 LA diesel engine which produces its highest performance of 110 kW / 170 kW (MC / LC) at 2200 rpm.

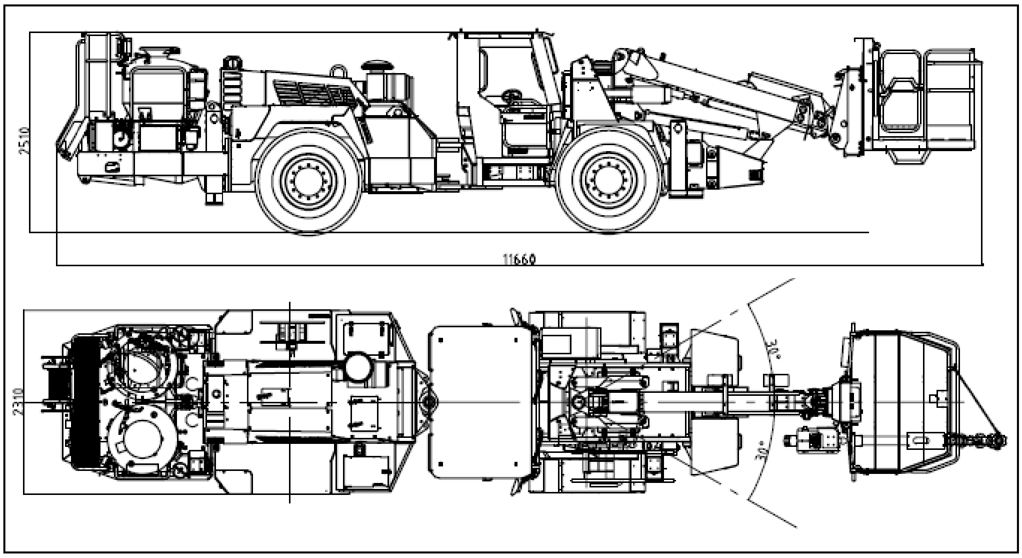


Figure 3. Main dimensions of LC 605 short end. (Normet Group)

The front chassis includes the cabin and lifting boom (Figure 4) with charging equipment. The lifting capacity of the platform is 500 kg and the boom can be lifted between -18° to 60° and slewing the boom is possible between ± 30° from the middle position.

Figure 4. Normet Basket Boom 3S. (Normet Group)

2.4.2 Operation Conditions

All Normet Group machines are designed to work in all kind of mining environment. Circumstances in underground work are difficult because humidity in the mines are always very high. As known raw metal combined with water will not last long without getting rusted if protection is done poorly. Some mines can also contain minerals which can corrode metal very fast so keeping the paint in good condition is important thing to increasing the lifecycle of the machines.

In normal use working the weight in a work platform is nowhere near the stress what is used in testing. Usually there is only one man with his equipments in the platform. However, calculation to the stability is made with 500 kilograms. Actual using loads which are measured with strain gages in tests are presented later in this final thesis.

Also a user size can influence usability of a machine and this issue has to be taken into account. Asian people usually have a smaller frame than for example people from South-America or Europe. Engineering of cabins and working platforms is made done in accordance ISO 3411:2001: *Earth-moving machinery - Physical dimensions of operators and minimum operator space envelope.* This standard take account 95 % of the people in the world so machine production done in accordance this standard guarantees products suitability to almost every people in the world.

3 THEORY

3.1 Hooke´s Law

In the elastic range of material the method of calculating the material stress from measured strains are based on Hooke´s Law. The name of this phenomenon was discovered by a British naturalist Robert Hooke who was the first person who experimentally proved linearity between stress and strain.

The most common construction material behaviour in the beginning of the stress-strain curve is usually linear until the offset yield strength point. Young’s module presents slope in the linear part of σε-curve as in Figure 5. (Outinen & Salmi, 38-39)

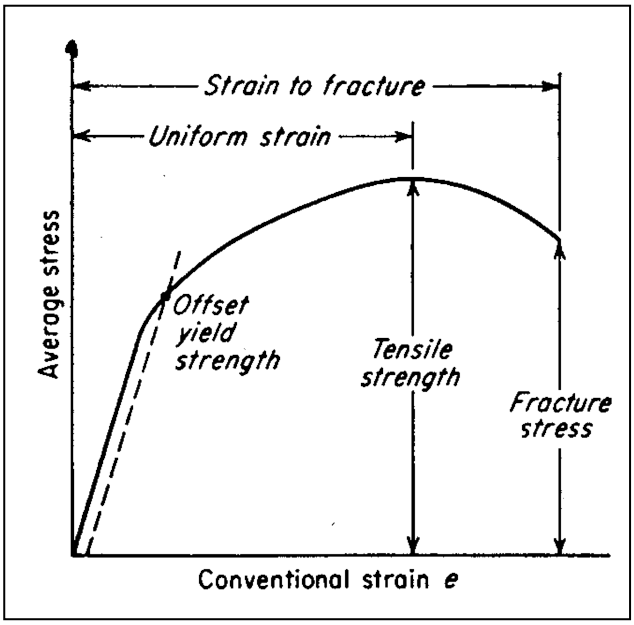


Figure 5. Stress strain curve to steel (Key to metals)

Structural engineering is usually accomplished with the assumption that material behavior is linear elastic. In these cases the link between stress and strain can be defined simply with equation 1.

$$\sigma = \varepsilon \cdot \mathrm{E} \qquad (1)$$

Where

σ	is material stress
ε	is material strain
E	is Young´s modulus

3.2 Stress

Normal stress presents dependence between force and area. This behavior can be described with equation 2. (Outinen & Salmi 2004, 25-26)

$$\sigma = \frac{F}{A} \tag{2}$$

Where

σ	is material stress
F	is force
A	is area

Result unit is in Pascal (Pa, N/mm^2), but because this unit is so small it is more natural to use units: kPa, MPa or GPa which:

kPa	10^3 Pa
MPa	10^6 Pa
GPa	10^9 Pa

This equation is very basic of material behavior in one dimensional force. When calculating behavior of multi axis stress state in viscous steel, the most accurate method for this is von-Mises hypothesis. According to this method, material damaged occurs in that point where distortion energy density reaches crucial point of material and damage type.

Distortion energy of certain material point can be describe with equation

$$U_{oD} = \frac{1}{12G}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2\right] + \frac{1}{2G}(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2) \tag{3}$$

Where

U_{oD}	is distortion energy
G	is shear modulus
σ_x	is stress x-axis
σ_y	is stress y-axis
σ_z	is stress z-axis
τ_{xy}	is shear stress xy-plane
τ_{yz}	is shear stress yz-plane
τ_{xz}	is shear stress xz-plane

On the other hand, in the catastrophic perspective of the axial stress, the equivalent distortion energy density is

$$U_{oD} = \frac{1}{12G}(\sigma_{ekv}^2 + \sigma_{ekv}^2) \qquad (4)$$

Where

σ_{ekv} is equivalent stress

When these equations (3) and (4) are marked equal, σ_{ekv} can be solved from:

$$\sigma_{ekv} = \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x\sigma_y - \sigma_y\sigma_z - \sigma_x\sigma_z + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)} \qquad (5)$$

This equation also works in main coordinate system, so

$$\sigma_{ekv} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_1\sigma_3} \qquad (6)$$

Where

σ_1 is first principal stress

σ_2 is second principal stress

σ_3 is third principal stress

The Von-Mises yield criterion gives a rather accurate result, because it takes into account all three shearing stress extreme values. A questioned hypothesis can be also used in the rainflow method to determine fatigue life. (Outinen & Salmi 2004, 349-351)

3.3 FEA / FEM fundamentals

Nowadays computers are developed and they have become one of the most important way to solve complicated mathematical problems. Forces in the structure which have many 3-dimensional parts are impossible to solve with simply hand calculation. To solve this kind of problems there have to be some kind of computer aid system. A method of this kind of procedure is the Finite Element Analysis process (FEA), also called the Finite Element Method (FEM).

Basically, in the finite element method the program first calculates force-displacement relations in each element of the structure and summarizes the calculations through each connecting point of the structure. These points are commonly called by nodes. From the result of these equations, unknown displacements can solved. This procedure can be describe with equation 7. (Mac Donald 2007, 73)

$$[K]\cdot\{U\}=\{F\} \tag{7}$$

where

$[K]$	is stiffness matrix of the structure
$\{U\}$	is displacement vector of the structure
$\{F\}$	is total force vector of the structure

Depending on the problem it is necessary to make some fair assumptions like ignoring small features which will not influence the results but can decrease calculation time significantly. When making this kind of assumptions it is very important to make the modification to the safer side. When adding the additional factor of safety (FOS) it can be ensured that catastrophic failure will not happen. This process is summarized in the chart which is presented in Figure 6.

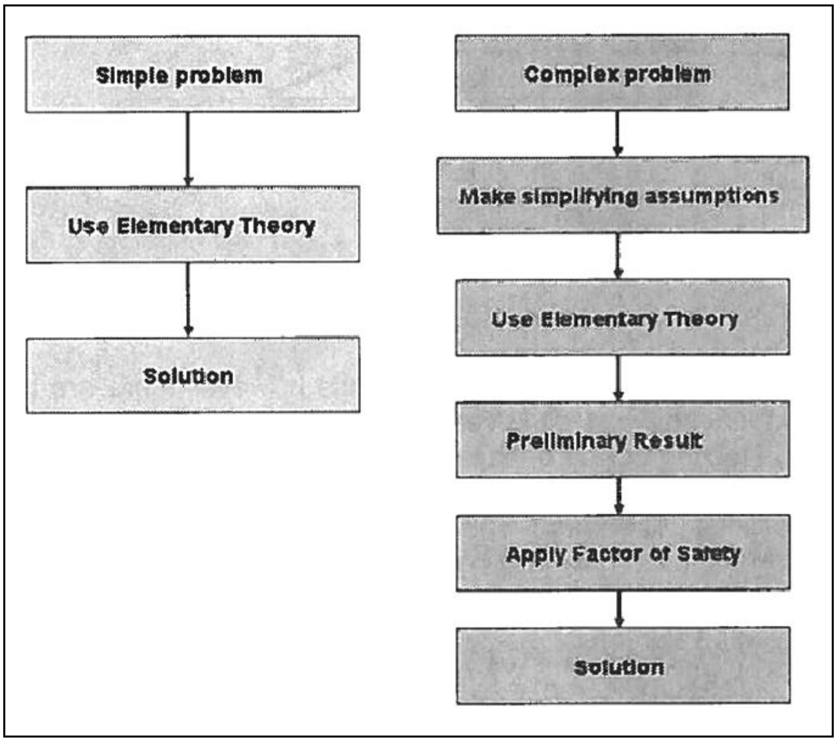


Figure 6. Solution method of simple and complex problems (Mac Donald 2007, 3)

The inner loop in the Figure 7 presents the building process of the finite element model, obtaining a solution for the nodal unknowns and post-processing the results. Nowadays there are many computer software´s like graphic interfaces and CAD modeling to help part processing. The outer loop represented the engineering decision making process which requires major of the time to perform the analysis.

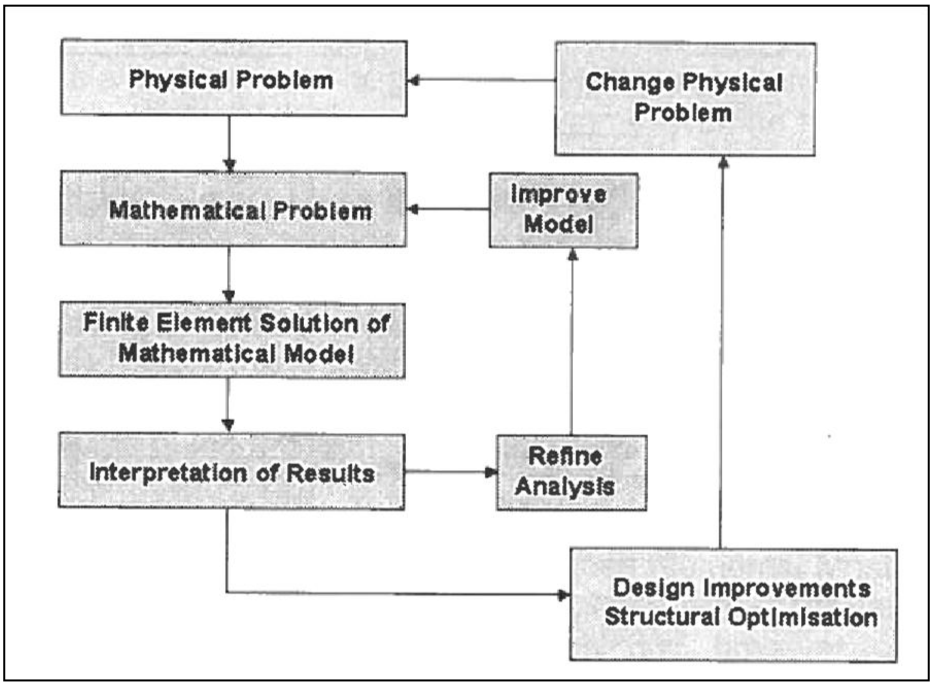


Figure 7. Basis procedure of FEA. (Mac Donald 2007, 47)

Simply, engineers have to make decisions how to transform a physical problem to mathematical model for the FEM analysis. To do this step it is necessary to make some assumptions that enable to bring a real life problem to the computer. These kinds of assumptions are usually related to geometry, loading, forces or material types. (Mac Donald 2007, 47-50)

Undisputed, the FEM solution calculates precisely the problem that the user inserts to the software. It is impossible to expect any accuracy information that the mathematical model contains. This is the reason why the most of the time is spend on doing proper FEM model and the actual calculation with the FEM software can be only a small part of the whole process. (Mac Donald 2007, 47-50)

3.4 Making of a FEM Model

3.4.1 Model Combining

The company usually makes welding assemblies in which there is free space between the each part. These gaps are reserved to welds to reach the right kind of final structure. With these kinds of structures the FEM model can be created but then contacts between each part have to be done manually with certain laws.

The final problem comes when trying to make a common mesh to the structure which consists of separate parts. The mesh can be created but the mesh will not be continuous between the parts. This feature causes transition discontinuity of nodes and mathematical solutions are not as accurate as they could be. To avoid this problem it is recommendable that the whole structure is combined to be a one part. Because the whole model is now one part there have to be small gaps between the parts and only the modelled weld keeps the parts together like in real life (Figure 8).

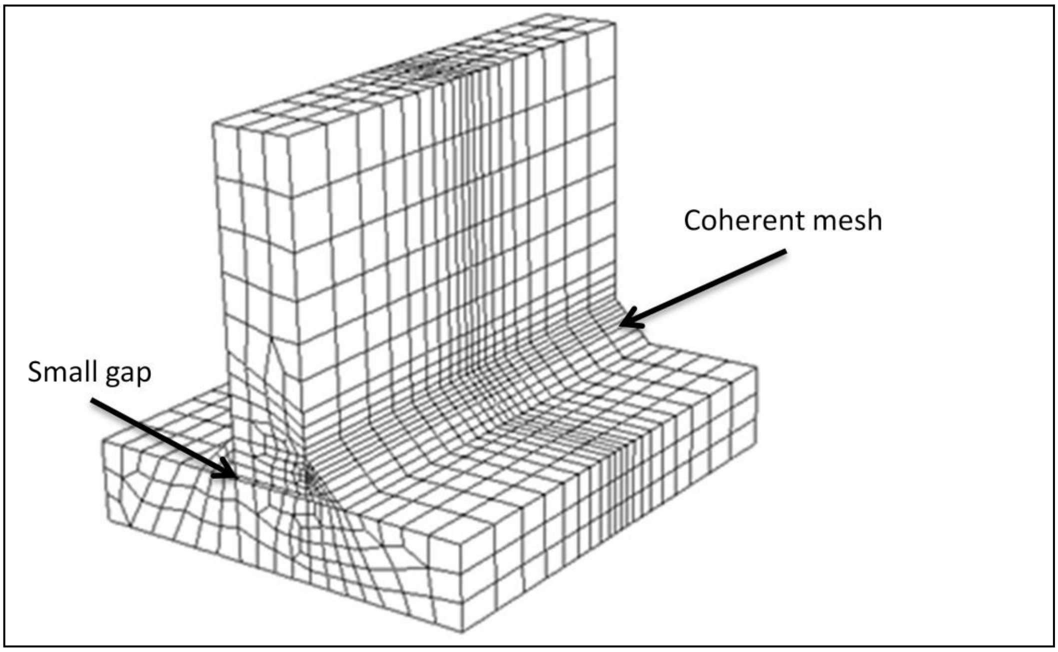


Figure 8. Proper kind of combining. (PHLEXcrack: A Meshless Code for Dynamic Fracture Propagation)

This method takes a serious amount of time and this is why the company usually makes only a simplified model and use ready-made connections to spare time. With current technology of the software and accuracy of the modelling the result usually are very near to the reality.

3.4.2 Defeaturing the Model

Components usually have some features which are important to manufacturing or aesthetic point of view, but which have no influence to mechanical behavior. This kind of features make FE-model complicated and may do meshing almost impossible, whereas removing or suppressing these things will not affect much to the result.

This kind of features can be small holes, fillet, chamfers, screw threads etc.

When removing unnecessary attributes you have to know exactly what you are doing. Especially when defeaturing is focused to areas where are huge forces, you have to understood structural behavior in order to make competent decisions how much of details can be deleted.

Defeaturing the pointless features can lower amount of tetrahedron significantly and that way lower calculation time and usage need of memory. Great example of this is shown in Figure 8.

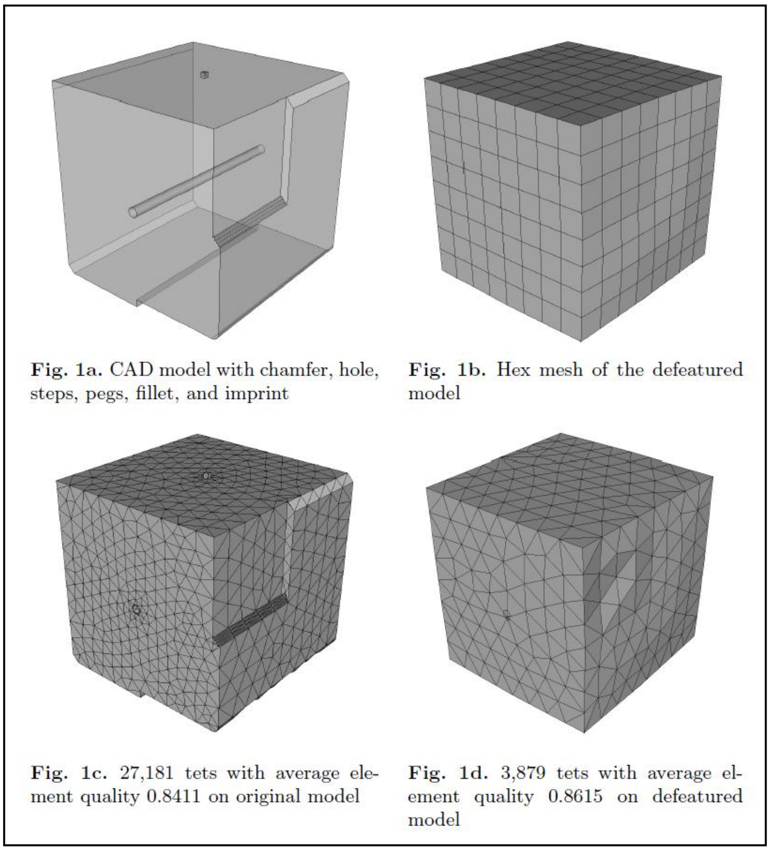


Figure 9. Example of defeatured part. (William & Owen 2010, 302).

3.4.3 Mesh Optimizing

When the mechanical perspective of a structure is optimized the next very important step is to make a proper mesh that mathematical problems can be solved without heavy need of the calculation time. Obviously finer mesh need put to the places where strain or stress is changing rapidly (Figure 10) and especially places where investigation is focused. If mesh is not fine enough forces can divide to the wrong places which can cause inaccuracy to the results.

To define how fine the mesh need to be is almost impossible to say. Situations are always different and one rule cannot be used in all cases. Nevertheless there are some basic rules for determining how thick mesh should be. This rule is called by five percent rule. It means that if results of calculation differs less than five percent with finer mesh the coarser mesh should be enough. (Mac Donald 2007, 204-208)

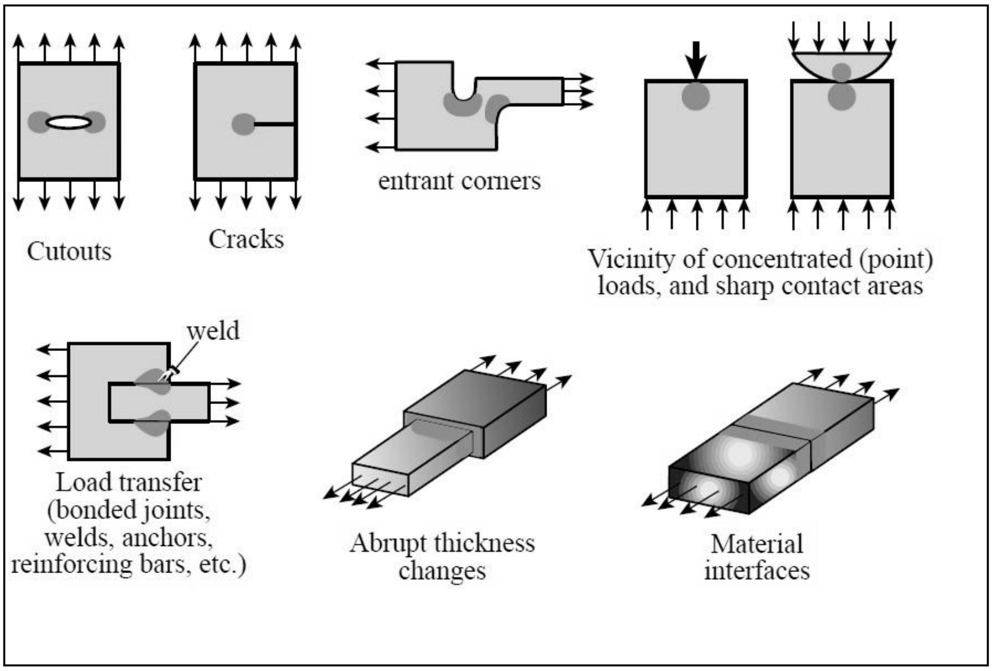


Figure 10. Points where finer mesh should be used. (FEM Modeling:Mesh, Loadsand BCs)

3.5 Fundamentals of Material Fatigue

Various types of failure have to be avoided through relevant mechanisms design, structural dimensions and in the materials choices. Criteria limits to the designs are set by yielding, buckling, creeping, corrosion and especially the fatigue life. Point is that, the welded joints are very vulnerable to the fatigue damage because joints are subjected to variable loadings. The fatigue crack may even slowly grow even if a dynamic stress to weld is much below yield strength. The amount of how much fatigue joints will last depends of very much of the range of stress and what is the amount of the loading cycles. These are the reasons why the fatigue inspection is one of the most important points of the design. (Lassen & Réche 2006, 3)

Fatigue process can usually split to three phases (Figure 11):

- Picture A: Crack initiation
- Picture B & C: Crack growth
- Picture D: Final fracture

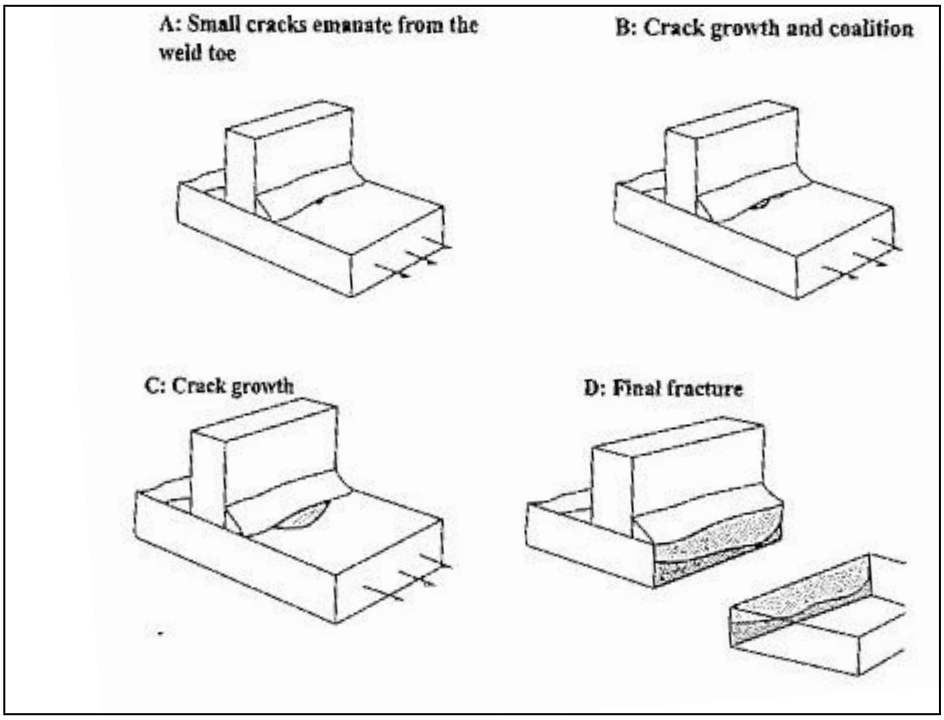


Figure 11. Various stages of fatigue crack. (Lassen & Réche 2006, 28)

For the fatigue life calculation there is usually a need for the long time stress history. One way to get the needed information of the stress is to use a Hot Spot method. This method is available when the critical point of the structure is known. Usually this point locates in the root of the weld. When this stress histogram is combined to Palmgren-Miner calculations it is possible to obtain the fatigue life.

3.5.1 Hot Spot Method

In this approach the fatigue strength is generally based on strain measurements in specific spots near to critical crack initiation. One huge advantage of the hot spot stress approach is the possibility of predicting fatigue behaviour of many types of joint only by using one S-N curve. More S-N curves may be needed if there is a variation of welding types, material thickness effects or if environmental effects have to be taken into account.

Structural Hot Spot stresses are measured with the strain gages which are usually installed near of the weld root. FEM analysis has revealed that, the notch effect is practically gone from the distance of 0.4 times plate thickness. Test result can be obtained with two strain gages which are placed to the certain place from the weld toe. Defined places for the gages are shown in Figure 12. (Niemi 1994, 100)

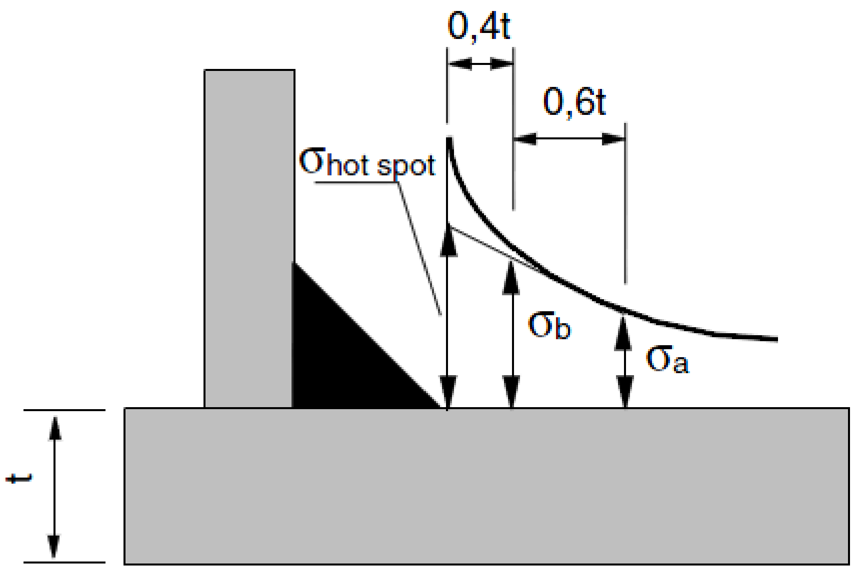


Figure 12. Gage Places in Hot Spot Method.

It is recommended that the Hot Spot measuring gages are fitted parallel to the principal stress direction. Assuming that, the shear strain near the weld is inconsiderable and the Hot Spot stress can be calculated with lineal extrapolation to the weld root by equation 8. (Niemi 1994, 20)

$$y - y_1 = \frac{y_2 - y_1}{x_2 - x_1}(x - x_1) \tag{8}$$

$y = \sigma_{hs}$	is calculated hot spot stress
x	is extrapolation point from the weld root in mm
x_1	is dimension from the weld root to closer hot spot gage
x_2	is dimension from the weld root to further hot spot gage
y_1	is closer stress to the weld
y_2	is further stress to the weld

3.5.2 Rainflow

In practice, stress levels on machine components are always irregular and random. If there is need to measure the possibility of the fatigue under irregular stress there has to use some kind of method to calculate this variety. It is important to take account all stress levels and not only the maximum amplitude. One of the most common methods to do this is a method called the rainflow counting. This algorithm was developed by Tatsuo Endo and M. Matsuihi in 1968. The simplified rainflow method calculates how many cycles there are at certain stress levels. Figure 13 presents a typical rainflow histogram which follows logarithmical curve.

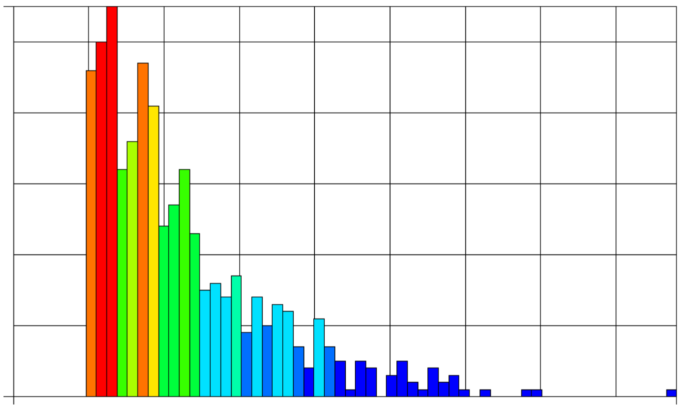


Figure 13: Rainflow Histogram

This method is especially used in long time period experiments. The main reason to this is that this particular method does not require a lot of memory to be logged. The data which a machine collects is basically histogram of different cycles of stress.

3.5.3 Equivalent Stress Cycle

The equivalent stress cycle (σ_{eq}) describes variations of stress in different kind of load cases. This stable amplitude curve has the same fatigue effect than the original stress curve. The equivalent stress cycle can be used to determine fatigue life estimation from the rainflow data with the Equation 9 (Westerholm 2000, 19). This cycle counting method suits best to long testing periods which last from couple days to weeks. In shorter time period's σ_{eq} would not work properly and because of that there may be huge variances in the results.

$$\sigma_{eq} = \left(\frac{\sum_{i=i_0}^{k} \Delta\sigma_i^{m_1} n_i}{N_d} \right)^{\frac{1}{m_1}} \tag{9}$$

Where

i	is number of stress level
i_0	is number of stress level when cut-off
m_1	is slope of S-N curve
n_i	is number of stress cycle in stress level i
σ_i	is amplitude of stress cycle in stress level i
N_d	is time which is determine from the stress period

3.6 Strain Gages

Strain gages are the most common devise to measure strain from the object. The principal idea is that measured strain transfers from measurable surface to the gage without any loss lose of strain value. This is why preparation of the measurements has to do properly to ensure as good results as possible. This means that surface of material have to be very smooth when attaching the gages.

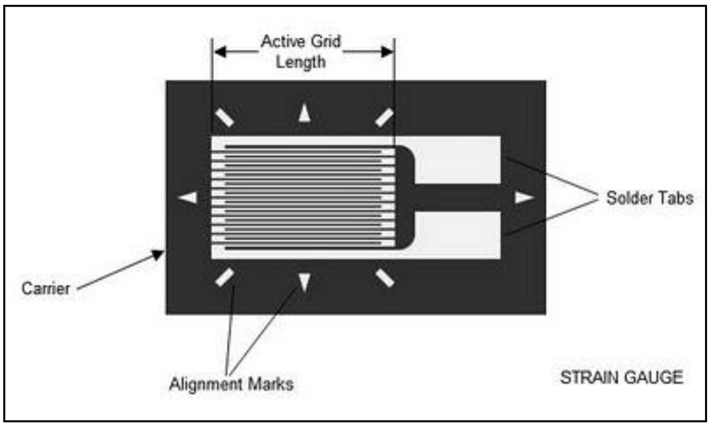


Figure 14. Metallic strain gauge (Strain Gauge - how does it work)

The operating principle of the strain gage is based on the consistency of the strain and resistance of electrical conductors. This means that the electrical conductor resistance changes by mechanical stress. When the microstructure of material transforms, it changes the resistivity of the conductor. This phenomenon can be described with the Equation 10 (Hoffmann 1989, 2-13)

$$\frac{dR}{R_0} = \varepsilon(1+2\nu) + \frac{d\rho}{\rho} \tag{10}$$

Where

R	is electrical resistance
ε	is train
ν	is Poisson´s ratio
ρ	is resistivity

Strain gages (Figure 14) are connected to Wheatstone Bridge (Figure 15) and when stain in the particle changes, resistivity of the gauge also changes. This causes that voltage between power supply (U) and gauges (V) differs from the original.

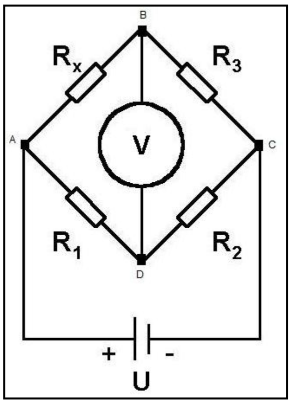


Figure 15. Wheatstone Bridge

Where

U	is Power Supply
V	is Potential difference

There are possible to arrange strain gages in three different kinds of setups depending on what kind of phenomenon is intention to inspect. In these situations gauges are installed to R1 - R4 in groups of one, two or four. Names of these setting are ¼, ½ and full-bridge configuration. (Strain Gage Measurements, 3-4)

4 CREATING OF THE FEA MODEL

The purpose of making the FEM model was to create a consistent part where mesh can be divided smoothly through the parts. In fact the built model won´t include separated parts in critical area. When making this kind of model there were some difficulties to create wanted individual model. Program what was used to do this part of the work was Autodesk Inventor professional 2010. Appropriate software have certain kind of methods how model can be created, this is why model cannot be done without many steps. Progress chart of this procedure is described in Figure 16.

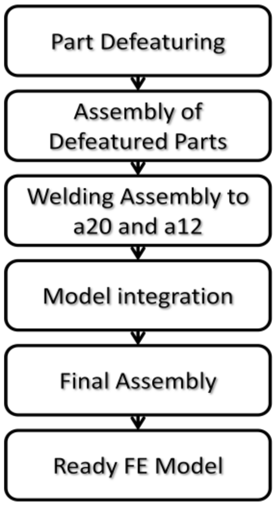


Figure 16. CAD modelling process

4.1 CAD Modeling

Normet Oy already has a finished model of welding assembly (Figure 17). This model includes gaps between parts and this is why the whole model had to be remade. In conversion work, assembly was divided in three parts: Frame, boom base and extension. To do proper kind of model which can be used wanted way in FEM calculations, there have to make following five steps in Inventor.

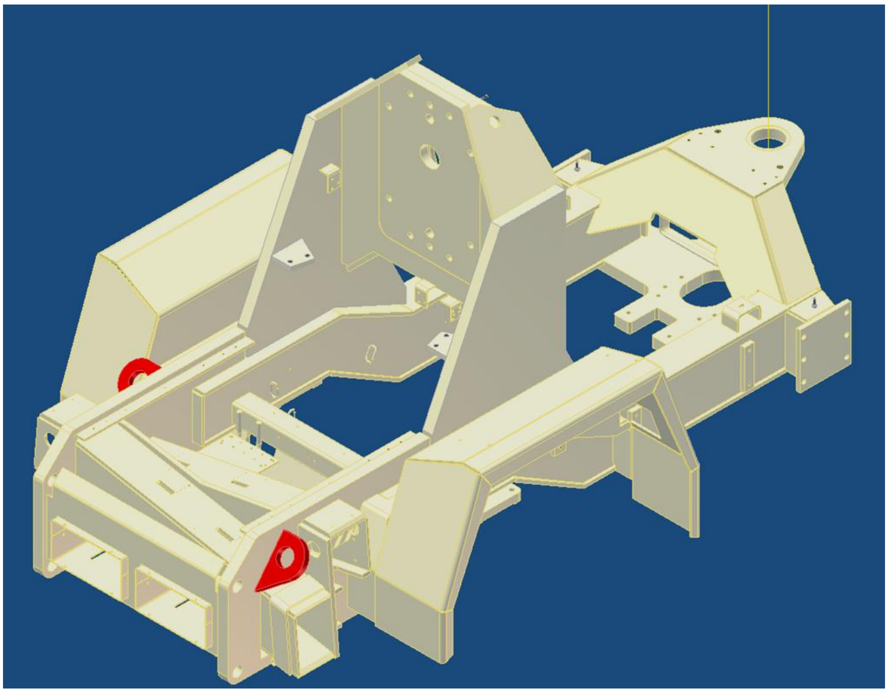


Figure 17. Original frame assembly

1. Part defeaturing: Aim of this step was to make three solid parts which can be used in FEM analysis. This phase is base of whole process and it have to be precisely. All dimensions and connections are dependence from this step. First the model was divided to the parts. Then all unnecessary features are suppressed and gaps were filled.

2. Assembly of Defeatured parts: When separated parts are defeatured then upcoming phase is to assemble part to the right places where are small gap between the frame and boom base. This gap reveals the space where are no welding in final model and in this way parts are connected only by the welding beams. More detailed picture of this is shown in Figure 8.

3. Welding assembly to a20: These separated parts are connected with weld which throat size is 20 mm. This weld presents existing weld in real machine.

4. Model integration: To make parts homogeneous, Inventor have tool named derive which converts assemblies to the one part. This step connects parts together through welding beam. After this step mesh can be divided smoothly through the whole part.

5. Final assembly: In the last phase, the boom bracket and bolts will be attached to the assembly. The reason why these features was not added earlier is that the bolt connections cannot make in properly in FEM. Besides forces which locate near the bolts are not interested in calculations.

After these five steps model is ready and it can be transferred to FEM software. This model is shown in Figure 18. Final assembly

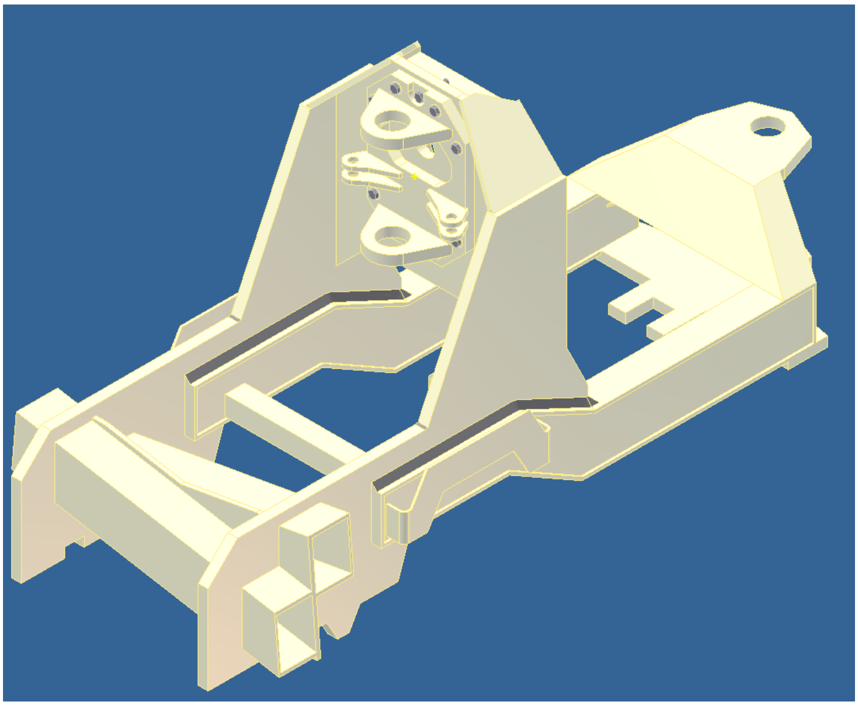


Figure 18. Final assembly

4.2 Making of the FEM Model

When the final assembly of the model is ready the structure can transferred to FEM program. In this analysis used program is called by Ansys 14. Inventor 2010 pro includes an interface to Ansys so model can be transferred straight to the software.

When opening project in the Ansys workbench, the program automatically creates link between these software's. This is a huge assist if there were need to edit model later. The model can be updated in the Inventor and then only refresh the geometry in the Ansys 14.

After the geometry is finished Ansys needs four basic definitions to solve the wanted mathematical problem:

- Meshing
- Contacts
- Supports
- Forces

When these features are defined the program can solve the problem if there are no conflicts between the certain features.

4.2.1 Meshing

The mesh was generated by using an automatic mesh tool which generates the mesh around the model with defined accuracy. The common mesh was generated with medium relevance, but if this kind of mesh is used in the inspected area the results could be remarkably divergent. This is why the model with thicker mesh is needed in the surrounding area of the weld and in the critical points of the structure. Thicker mesh was generated with 12 mm element size and mesh near critical points with 3 mm element size. If thicker mesh is used in the whole part then calculation time would be excessively longer and the overall advantage of the finer mesh would be quite small.

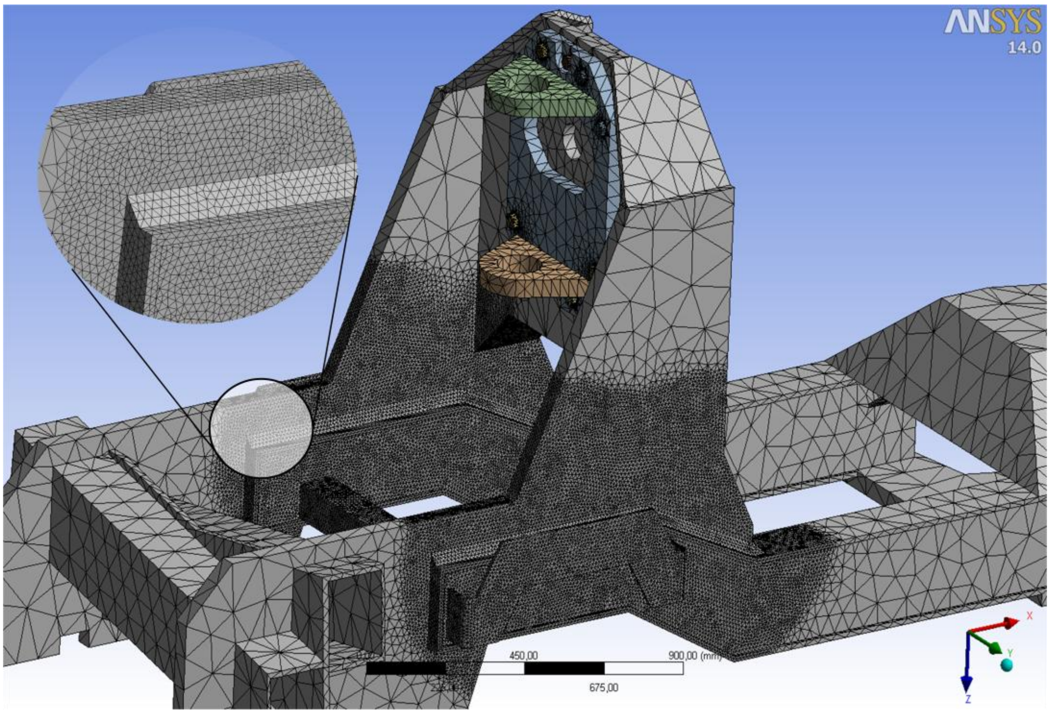


Figure 19. Part meshing

In the strain gage installation places it is useful to use mapped mesh where gage locations are placed to nodes like in Figure 20. This gives a benefit when want to find the principal stress directions or the stress levels in these points. After modifications, model contains around 1.4 million nodes and 950 000 elements

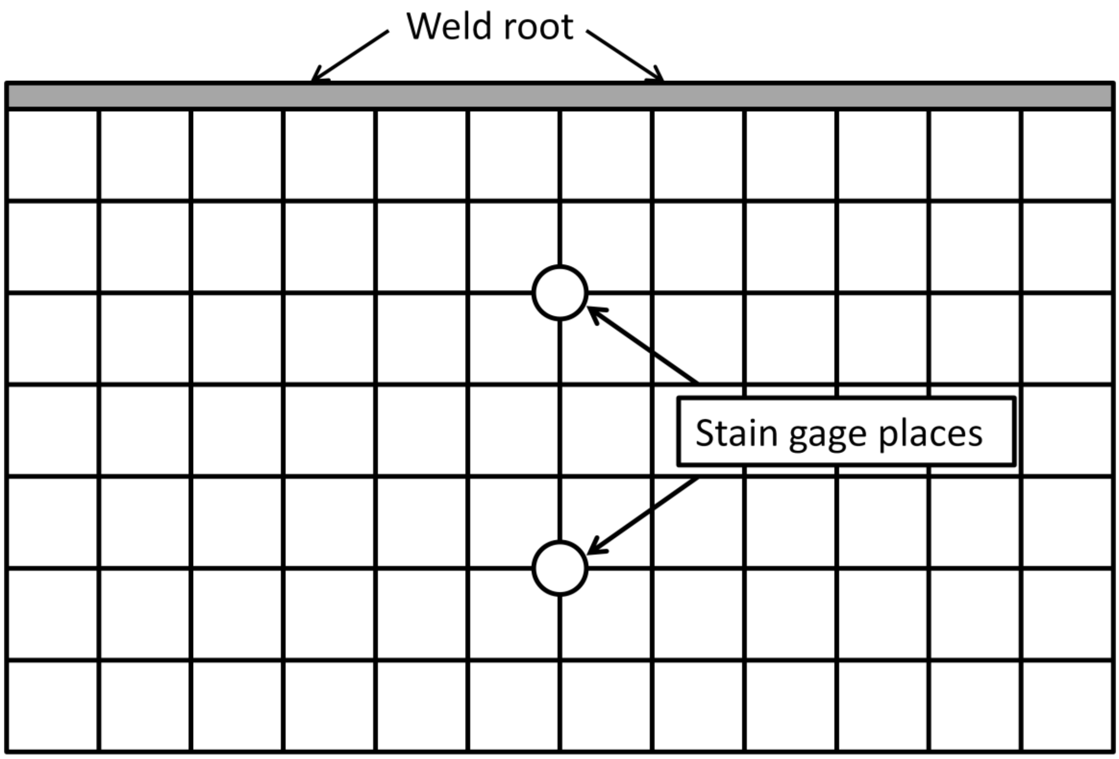


Figure 20. Mapped mesh

4.2.2 Contacts

In this situation there are no needs of the contacts in critical area because the part is consistent. Only possible place for contacts is located in the bolts which connects the boom clamp to the boom base. In Ansys, there were a way to make proper kind of bolt fastening between parts; this feature is called by bolt pretension. Required pretension forces for M24 bolts is 188 kN for on each. (Valtanen 2007, 565)

Nevertheless, bolt fastening is not necessary because the bolt joints area is not under inspection. Instead of bolt connections, it is possible to use bounded connection between the boom clamp and the boom base.

4.2.3 Forces

Forces which are used in an analysis are calculated earlier to the NBB3S boom. Particular boom is the heaviest which is mounted to the boom base so calculations with the NBB3S covers all lighter boom models. These calculations already includes safety factors and it also takes notice of the dynamic loadings which are defined in the standard EN 280:2001.

Dynamic calculations of the boom base are made in position where the boom is fully out in horizontal position. Impacting forces in this position are XXX kN vertically in upper bracket and XXX kN horizontally in both brackets but separate directions. Impacting directions and places can be found in Figure 21.

Other interesting situation in dynamic load aspect is the situation when driving vehicle and the boom is on the driving support. This kind of use may impact frame with very large but short lasting force. This kind of data cannot be confirmed by FEM calculations easily. So this is the situation where the practical strain gage measurements present significant part.

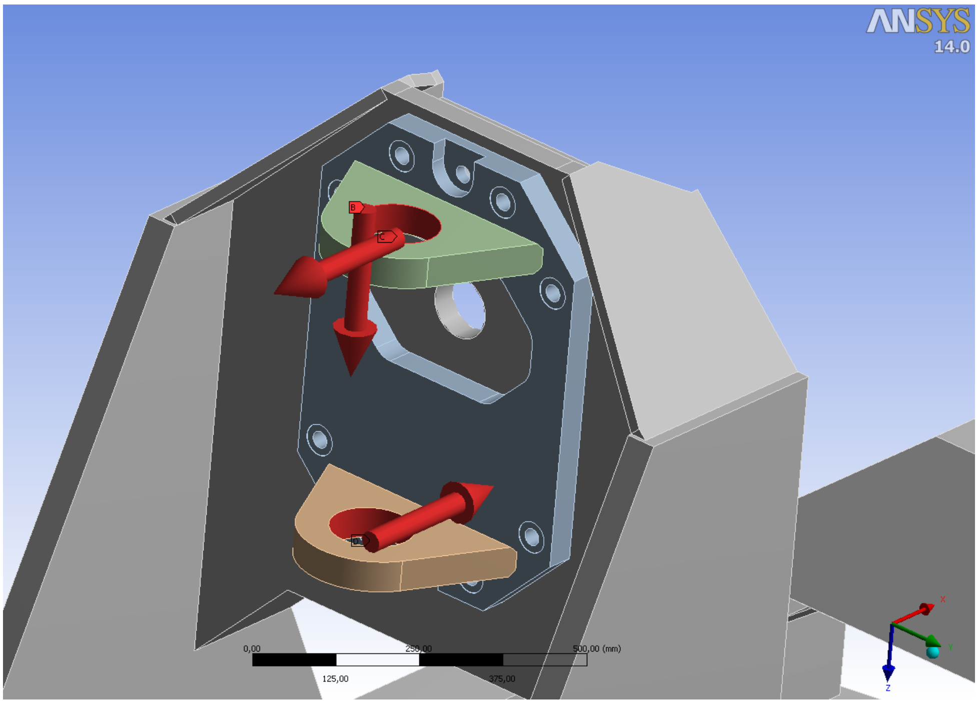


Figure 21.Impacting forces

4.2.4 Supports

Supports in the model are accomplished with the remote displacement tool. In both supports, movements to xyz- direction are bound but rotations around xyz- axes are possible. Supports are shown in yellow color in Figure 22. In the left picture (a), offset to support point locate 1850 mm along x-axis and 400 mm along z-axis. In the right picture (b), support locations are 600 mm to z-axis and 1010 mm to outside from the center of the frame. Figure 23 present these supports in actual use.

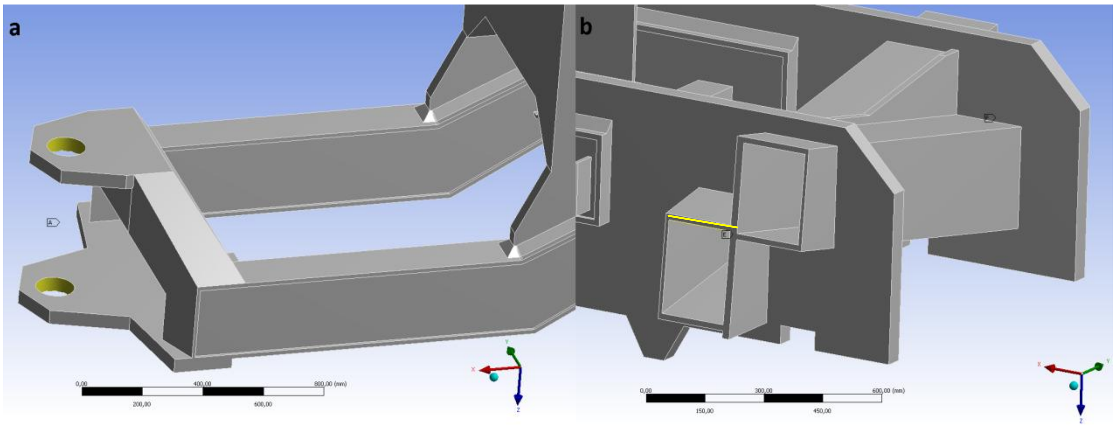


Figure 22. Supports

Figure 23. Supports in actual use

4.3 FEM Results

To analyze the results the most relevant way is to use the von-Mises stress, which indicates the highest equivalent stress in the part. Different kind of steel sustains different size of stress without any change in shape; this point is also called lower yield point (ReL). If this point exceed there will be plastic deformation in the material and this is highly restricted.

The NT100 frame includes two types of steel which are under observation. These materials can be found in Table 1 and strain gage places in Figure 28.

Table 1. Material types in the model

Part	Part nro in Appendix 1	Strain gages	Material	ReL (Mpa)
Flat iron (frame)	17	1,2,6,7	S235JRG2	235
Boom base	2	8,9,10	S355K2+N	355
Extension	4	3,4,5	S355K2+N	355

Where in material column: (European structural steel standard EN 10025:2004)

S...	is structural steel
.235...	is lower strength (ReL) in MPa @ 16mm
...JR..	is longitudinal Charpy V-notch impacts 27 J @ +20°C
...K2..	is longitudinal Charpy V-notch impacts 40 J @ -20°C
...+N	is supply condition normalized or normalized rolled

4.3.1 Results with 20 mm Throat Thickness

In case one, where the welding throat thickness is 20 mm (Figure 24) stress levels stay around XXX MPa which is low enough and there is no plastic deformation in material. This is obvious because the existing machine is similar to the created FEM model. The highest stress value locates on the tension of the boom base. The strain gage measurements are planned to be done in this area so that FEM calculations will supports the preliminary measuring plan.

As earlier mentioned, the main reason to do FEM calculations to a 20 mm throat thickness is to verify results between the mathematical model and a real life case by using strain gages. Strain gage places and directions are defined by the result of the FEM model and this is why the model should be as accurate as possible.

Figure is not available on public version

Figure 24. Von-Mises stress with throat size of 20 mm

Total displacements of model are shown in Figure 25. As figure presents transition of parts are very small in examination point. In real life the structure will be even stiffer because part includes additional welded parts.

Figure is not available on public version

Figure 25. Displacements with throat size of 20 mm

4.3.2 Results with 12 mm Throat Thickness

In case two where the welding throat thickness is 12 mm (Figure 26) stress levels near the welding in flat iron are little larger than in the case one. Stress is about XXX MPa which stays in acceptable range because lower yield limit of material is 235 MPa. Besides these values are located in very sharp geometry discontinuation point and real values of these points have to be calculated by the hot spot method. After measurements and results comparing it is possible to say how much deviance is between the FEM model and the actual machine.

Figure is not available on public version

Figure 26.Von-Mises stress with throat size of 12 mm

As Figure 27. Displacements with throat size of 12 mm presents displacements stays around the same with both throat size values. This indicates that welding between boom base and frame will not present the demanding part in the structural stiffness.

Figure is not available on public version

Figure 27. Displacements with throat size of 12 mm

4.4 Conclusions

At the first look of the result it seems that there are two critical points in the welding. These spots are located in tensile and compressions side of the boom base. These locations will be the main interest during strain gage measurements. As Figure 24 and Figure 26 there are no plastic deformations in the model and stress levels in these spots stays in acceptable area. Best way to more accurate measurements of stress in these places is type "A" hot spot method which gives the real stress level in the root of the weld.

After investigation of the structure there were two simple developing spots in the structure. First possible thing to do is to replace flat iron which is S235JRG2 to more durable steel like S355K2+N. S355 in nowadays more common structural steel than S235 and it is even slightly more durable. But in fatigue perspective changing to the S355 will not increase crack grows speed but it will effect to initial crack formation in the root of the weld. (Niemi 2003, 95)

Another issue locates in the compression side of the boom base where filling weld is located. In this area, there are two high risks of fatigue in one place in one place. First one is fast geometrical change and secondly there is even welding at the same spot. These features cause quite high stress peak in the corner. This problem can be fixed very easy by changing the geometry of boom base near corner and add around 30 mm fillet before welding. By doing this stress can divide smoothly through fillet and there are no longer so high stress peaks in the welding area. Nevertheless stress levels in this area are so small that this change is not necessary to make.

These assumptions are purely made by using the FEM model with static load and the results have to be verified by using the strain gage measurements. These measurements also produce information of the metal fatigue and this way the life time of the structure can be calculated.

In summary, structure will last throat size changing from 20 mm to 12 mm in static use without any plastic deformation. These forces which are used in the FEM analysis include safety factors according to the EN 280:2001.

5 MEASURING

Strain gage measurement is one of the most significant parts of this thesis and this is why the test must be carefully planned. The main purpose is to verify result of the FEM calculation near the weld but with few extra gages it is possible to ensure whole model correction.

To protect the gages from out coming risk, there have to add some protections for the gages. Gages are covered with silicon and sealing compound which keep humidity away from the electrical circuit. Wiring and the central unit have to also put places where are no possibility to get damaged.

5.1 Purpose of Strain Gage Measurements

In phase one, intention is to ensure the results of the FEM. Point of this verification ensure that stress levels in the FEM model and the real live machine are equal. In this case it is important to think how is possible to imitate stress calculation as accuracy as possible in a test course. Stresses in the FEM calculations and the laboratory tests need to be the same that results are acceptable. It is obvious that boom position have to be in horizontal direction and fully out as in calculations, but more demanding part is to solve how lateral direction of the forces can be accomplished in the tests. There were also some extra equipments in the platform which need to be take account when measuring verification loads.

In phase two, point was collect data from the test drive where machine is under heavy driving. From these results it is possible calculate due life time of welding in hot spot places as mentioned earlier. When main priority of phase one is only check FEM calculation results seconds step concentrate fatigue life in driving situations. Effort of this operation stage can be remarkably different and this is why both situations have take in account with high priority.

5.2 Measuring Equipments

Measurements were accomplished with ten strain gages which were connected to the eDAQ plus base processor. Four gages measured hot spot stresses from two different locations and other gages measured one dimension stresses. Gage properties are shown in Table 2 and eDAQ specification can be found in a appendix 7**.** Real values of the gage resistivity was measured before testing and updated to the software. A data processing program to hot spot stresses and estimated life calculation are done with GlyphWorks software which is developed for signal processing, data visualization and metal fatigue analysis.

The measuring of the data was done with the quarter bridge connection with a frequency of 2500 Hz. This frequency is quite high, but there was enough memory capacity to accomplish the test such a high data collecting frequency. High collecting frequency make possible to collect rapid load changes in gages. This feature comes useful when there is a striking load on the frame during the use.

Table 2. Strain gage properties

ONE DIMENSION STRESS

Type	KFG-5-120-C1-11L1M2R
Gage factor	2.08 ± 1 %
Gage length	5 mm
Gage resistivity	120.4 ± 0.4 Ω

HOT SPOT

Type	KFG-1-120-D9-11N10C2
Gage factor	2.08 ± 1 %
Gage length	1 mm
Gage resistivity	120.4 ± 0.8 Ω

5.3 Locations of the Strain Gages

After the highest values of the stress are calculated in the FEM it is possible to determine locations for the strain gages. Gage locations and directions are depending from the material thickness and the action lines of principal stresses. When using only one direction gages in hot spot method, gages should be placed exactly parallel to principal stress direction. These directions can be added to model by stress tool named by vector principal stress. Because the plate where gages are installed is 15 mm thick the strain gage positions are 7 mm and 15 mm from the root of the weld. The reason why ten gages are installed instead of only few was the meaning to ensure the match of the FEM and real life properly

Other interesting place for measurements is located in boom side of the boom base where the stress is compression unlike in hot spot location one. Gages in this location were also installed 7 mm and 15 mm from the weld root. Approximate locations of the strain gages are shown in Figure 28. More accurate locations for strain gages can be found from appendix 1**.**

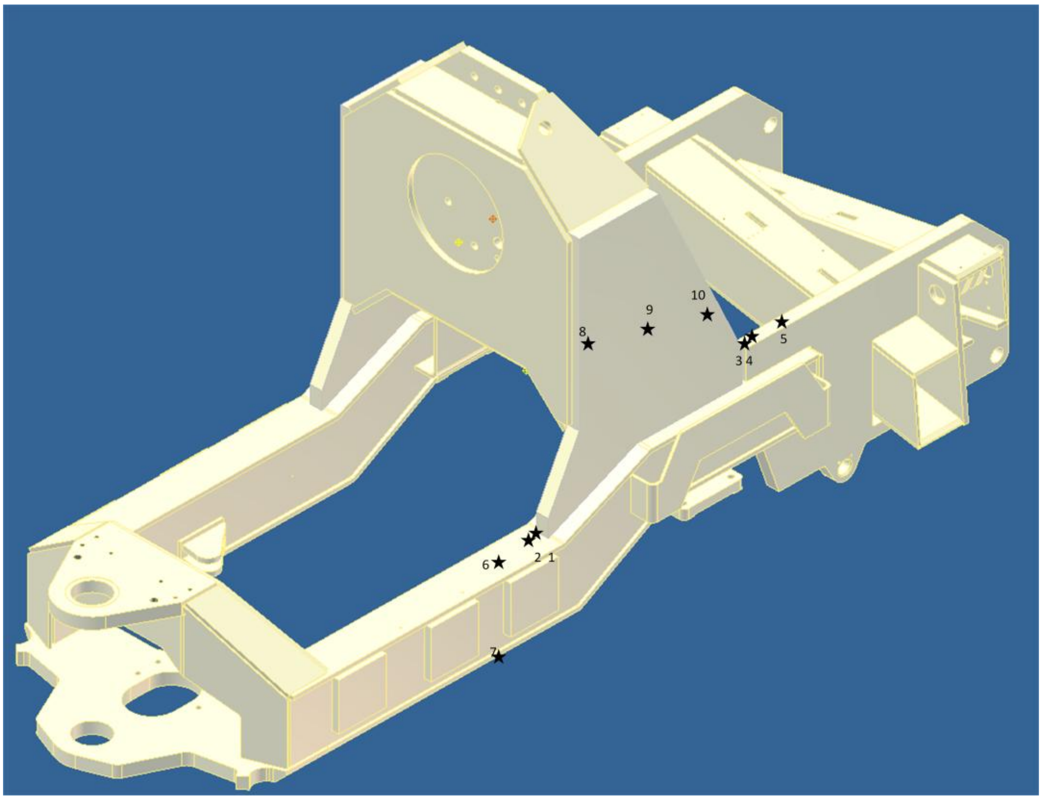


Figure 28. Strain gage places

5.4 Measuring Plan

Well designed measuring would provide accurate result with only one test. If measuring need to do again on the same place that would be aimless and expensive.

Before this test can be started strain gages have to glued to the frame. Calibration position of the boom was accomplished with a crane where is a scale which located between the crane and the lifting chains. In this way it is possible to light up the boom influence to boom base when boom mass and it´s center of gravity in known. The boom mass with all additional parts was 2600 kg and the center of the gravity locates near the lifting bracket. The best possible situation to lighten up the influence of the boom would had been detach the whole boom from the boom base by loosen up the connecting bolts, but this operation was not possible because there were lots of cables and bolts already tighten up finally. Purpose of this calibration was to set known zero point to the strain gages in a certain position where are no outside forces impacting to the parts which are under investigation.

Figure 29. Calibration of the strain gages

5.4.1 FEM Verification

Laboratory test were done in Normet Oy proto hall. The purpose of this test was to run some basic tests and verify the specially planned FEM calculation in the particular position.

Calculations were based on the basic situation where are some certain equipments. But in this test machine, there are additional parts in the boom which cause more weight and in this way more stress to the structure. Because of that, it is purposely to do some recalculations where all extra equipments are take account. Safety factors to all loads are irrelevant because they are only additional forces to the calculations. In this calculation there was also need to plan how to load the platform to right direction that calculation forces which are used in FEM model are equal. In this situation best way was to placed 140 kg extra load on the lifting platform which indicates maximum working load as in calculations.

In this verification, it was not necessary take account wind loads and other forces which would make gage measurements harder to accomplish. If this kind of situation is available it is much easier to adjust the FEM model forces than make hard cable installation for the test event. Figure 30 present the outline of the test how verification testing are planned to complete.

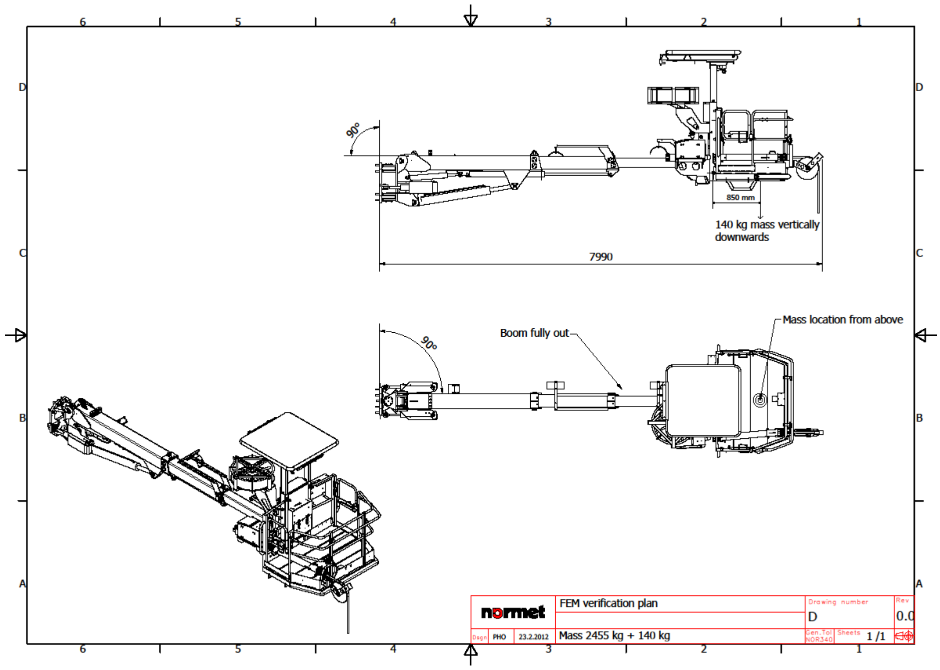


Figure 30. Verification position

5.4.2 Fatigue Calculations

There was purpose to accomplish fields test in Pyhäsalmi mine, but the schedule of tests changed and this is why test was impossible to include in the thesis. A real test environment would have been really vital to the work, but now the test had to be completed in a proto hall where boom is moved around the limit points. Fatigue calculations also include driving test which was droved on the test course. In this condition the working operation can be imitated, but the test time is much shorter which can cause unreliability especially in fatigue calculations. If wanted to define precisely fatigue life it is supposed to arrange around one week lasting test in real environment. In this thesis that was not possible, so it have to be satisfied with short time results.

In practice most of the time is spend to charging where the boom makes slow and controlled movements, opposite to this operation is driving the boom down on its support which causes striking load to the frame.

Another important part was to calculate the life cycle in a driving situation in two different boom positions. On the first position the boom is on its support and on the second position the boom is lifted around 20 centimeters up from the support. After these tests it is possible to say rough assumption of the life time in a driving situation. Of course driving conditions change a lot between different kind of mines and it is impossible to cover all possible driving conditions.

Savonia has a premade excel chart to the life cycle calculations which is made by using the SFS-EN 1993-1-9: *Eurocode 3: Design of steel structures. Part 1-9: Fatigue* standard. This chart gives an estimated life time for the structure when rainflow data is known. This data can be collected from the hot spot gages which are installed to the gage position one to four.

6 RESULTS

Half day of measuring from 10 strain gages provided around 40 hours of data which needed to be post processed to the wanted type. The hot spot gage results needed to combine to hot spot stress (σ_{hs}) by lineal extrapolation which is explained in chapter 3.5.1. The results from the other gages express how accurate stress levels were between the FEM model and the actual machine.

The results are split to three sections which divergence each others quite much. A static chapter is mainly determination of lowering the throat size with EN 280 standard but it also operate base to fatigue analysis. Rest two chapters imitate the main usage function in the mine and present estimated life cycle in these operations.

Processing of data was done in the office with program called GlyphWorks which can calculate the hot spot stress and rainflow analysis from the measured data. Main view of the program can be found from the Figure 31. This flowchart procedure shows how data are extracted and then processed to the wanted form.

Results are not available on the public version

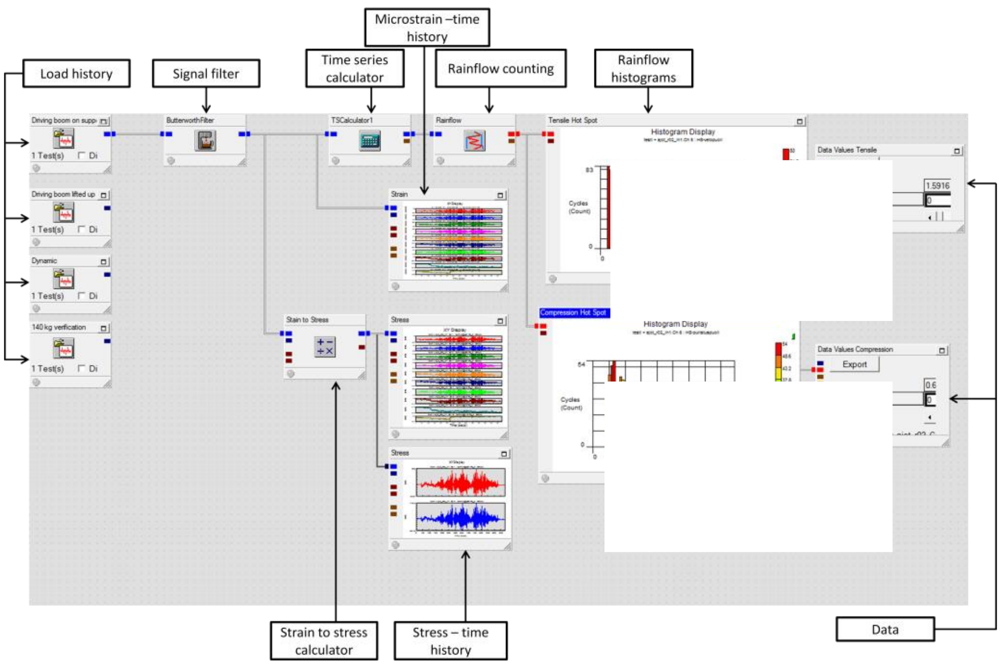


Figure 31. Flow diagram of GlyphWorks process

7 CONCLUSIONS

Overall doing this thesis was pleasant because the project object was interesting and there a possible to apply studied information to in practice. This project produced an examination of weld strength calculation which can be used for later needs. The project goal was clear even from the very beginning of the work and this make possible to divide the whole work to certain parts. This helped to plan working schedule and it was easy to follow which part had to be done.

From the point of view of the thesis it was very unfortunate that the mine test was delayed and it was not possible to include it in to the work. This would have been a great addition from the experimental and calculation perspective. Otherwise all issues proceeded as planned and there were no remarkable problems which effected the schedule. Especially comparing the FEA model and measured strain gage data was illustrative on how two types of method produce almost the same results. This fortifies information that FEA models correspond to the real life structures. Working schedule was fitted well to the thesis and work load was suitable to the whole working time.

From the educational perspective the work was instructive on how all systems and methods are linked together. Although there are plenty of software available nowadays the user may still have to define certain phases step by step because some features may not work together. To achieve the final results there was a need to get familia with programs like Sovelia PDM, Autodesk Inventor 2010 pro, Ansys Workbench 14 and GlyphWorks.

In the end as an conclusion it can be said that modern Finite Element Method software and CAD programs function extremely well together and they are indispensable to engineering.

REFERENCES

Ariduru, S. 2004. *Fatigue life calculation by rainflow cycle counting method* [web publication] Middle east technical university. [accessed 5 February 2012] Available from: http://wind.nrel.gov/designcodes/papers/FatLifeCalcByRFCycleCountingMeth_Ariduru.pdf

Corus Construction & Industrial. European structural steel standard EN 10025 : 2004. *Explanation and comparison to previous standards* [web publication]. [accessed 14 February 2012] Available from: http://www.tatasteeleurope.com

FEM Modelling: Mesh, Loads and BCs [web publication]. [accessed 26 January 2012]. Available from: http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/IFEM.Ch07.d/IFEM.Ch07.Slides.d/IFEM.Ch07.Slides.pdf

Hoffmann, K.1989. *An Introduction to Measurements Using Strain Gages*. Darmstad: Hottier Baldwin Messtechinik GmbH.

Key to metals: *Engineering Stress-strain curve* [webpage]. [accessed 30 January 2012]. Available from: http://www.keytometals.com/Article107.htm

Lassen, T & Réche, N. 2006. *Fatigue Life Analyses of Welded Structures*. Chippenham: Antony Rowe Ltd.

Mac Donald, B J.2007. *Practical Stress Analysis with Finite Elements.* Ireland: Glasnevin Publishing.

Mäkelä, M., Soininen, L., Tuomola, S & Öistämö, J. 2002. *Tekniikan kaavasto*. Jyväskylä: Gummerlus kirjapaino Oy

Measuring Strain with Strain Gages [web publication] [accessed 26 January 2012]. Available from: http://zone.ni.com/devzone/cda/tut/p/id/3642

Niemi, E. 1994. *Recommendation Concerning Stress Determination For Fatigue Analysis Of Welded Components*. Lappeenranta: University of Technology

Niemi, E. 2003. *Levyrakenteiden suunnittelu*. Teknologiateollisuus: Tekninen tiedotus 2/2003.

Normet Group. introduction. [webpage]. [accessed 4 January 2012]. Available from: http://www.normet.fi/normet/normet_en

Normet Group. technical date sheet [webpage]. [accessed 18January 2012]. Available from: http://www.normet.fi/products/Charmec_MCLC_605_DA

Outinen, H & Salmi, T. 2004. *Lujuusopin perusteet*. Tampere: Pressus Oy

PHLEX crack: *A Meshless Code for Dynamic Fracture Propagation* [web publication]. [accessed 25
January 2012]. Available from: http://www.tx.altair.com/products/phlexcrack.html

SFS-EN 1993-1-9. Eurocode3: *Design of steel structures*. Part 1-9: Fatigue. Helsinki: Finnish standards association SFS

SFS-EN 280: 2001 + A2: 2009. *Mobile elevatig work platforms*. 2009. Helsinki. Finnish standards association SFS

Strain Gauge - how does it work [web blog]. [accessed 26 January 2012] Available from: http://digitalweighing.blogspot.com/2010/09/strain-gauge-how-does-it-work.html

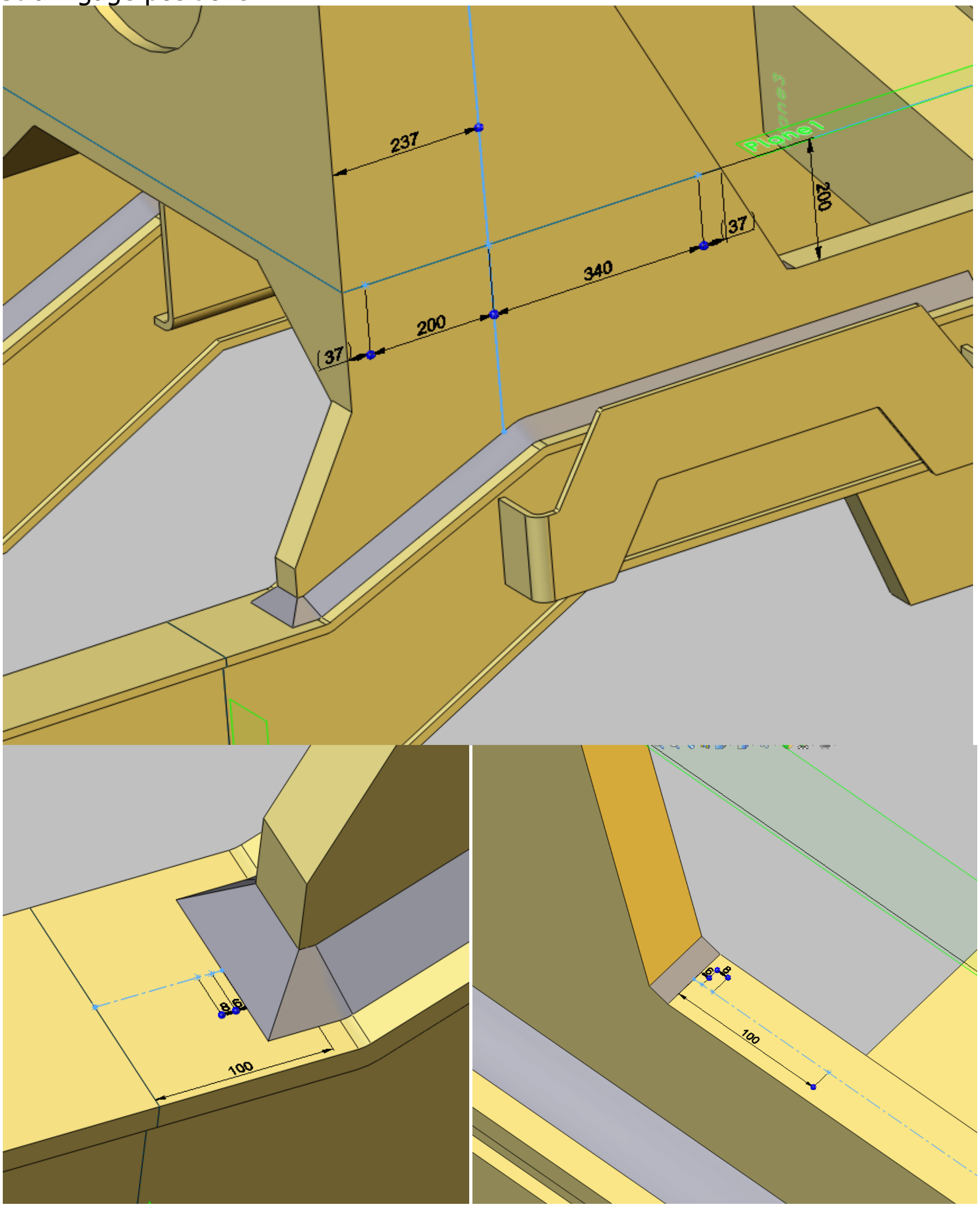
Strain Gauge Measurement – A Tutorial [web document]. [accessed 26 January 2012]. Available from: http://www.eidactics.com/Downloads/Refs-ethods/NI_Strain_Gauge_tutorial.pdf

Valtanen, E. 2007. *Tekniikan taulukkokirja*. Jyväskylä: Genesis-kirjat Oy

Westerholm, T. 2000. *Measurements of Operating Loads in Forwarder Use.* Tampere: University of Technology

William, R.Q & Owen, S.J. 2010. *Defeaturing CAD Models Using a Geometry-Based Size Field and Facet-Based Reduction Operators* [web document]. [accessed 25 January 2012]. Available from: http://www.imr.sandia.gov/papers/imr18/Quadros.pdf

Strain gage positions



eDAQ Base processor product specification

Physical

Size: 23cmW x 27.6cmL x 6.6cmH
Weight: 8.32lbs (3.78kg)
Temperature: -20° to 65° C
Connectors:
Power - 15 Pin D-Sub
Communications - 26 Pin High Density D-Sub
HSS - SoMat M8 Female Bulkhead Connector
Digital I/O - 44 Pin High Density D-Sub

System

Input Power: 10-55 VDC
Fuses: 10A, Automotive Mini-blade
Internal Back-up Battery
Sample Rates:
Master Clock Rates
100 kHz = 0.1Hz to 100 kHz
98.3 kHz = 0.1 Hz to 98,304Hz

Communications

Ethernet: 100 BaseT
Serial: RS232 up to 115,200 baud

Memory

Internal Flash:
Standard: 1GB
Upgrades: 4GB, 8GB, 16GB, 32GB
External Memory: 4GB
Internal DRAM:
Standard: 64MB
Upgrade: 256MB

Inputs

Digital I/O:
Minimum: -0.3V
Maximum: 5.5V
Pulse Counters
Number of Inputs: 8
Pulse Counter Modes: Pulse Time Period, Pulse Frequency, Duty Cycle and Pulse Rate