Numerical simulation of residual stresses in a weld seam

- An application of the Finite Element Method
Abstract

Articulated haulers are fundamental equipment to transport material. The load carrying structure on a hauler consists mainly of welded frames. During welding of the frames high residual stress will be introduced. These stresses may have a significant impact on the fatigue life of the frames. This is the reason for having good knowledge of the weld residual stresses. The finite element method was used to calculate the residual stress distributions in a butt weld and a T-join weld. Simulation of the welding process with thermal and mechanical analysis was prepared by means of welding GUI implemented in LS-PrePost.

The welding simulation is a computer intensive operation with high CPU time. That is why it is important to investigate which process factors that have the largest impact on welding simulation results. The aim of this thesis is to investigate the correlation between designed models in FEA software with published results of weld residual stress measurements and conclude which parameters should be mainly taken into consideration.

Keywords: Finite element method, residual stresses, temperature distribution, fatigue life, weld power, time step, T-joint weld model, Butt weld model.
Acknowledgement

I would like to offer sincerest thanks to my supervisor Andreas Linderholt. His help and knowledge was priceless and invaluable. Because of his advices, support and all constructive comments along with admirable patience, this thesis was possible to complete.

Furthermore, I would like to thank Volvo Construction Equipment in Braås for giving me the opportunity to conduct realizing my thesis. I am incredibly grateful to Magnus Andersson and Mehdi Ghanadi for never-ending support with plenty of advices. Additionally, I would like to thank Per-Olof Danielsson, the Department Manager, for providing me with a workspace and an access to utilize all the necessary equipment.

I would also like to thank Thomas Bader, Lecturer at Linnaeus University for vast and useful knowledge gained during Scientific Methodology and Planning course.

Maciej Maczugowski

Växjö 25th of May 2017
Table of contents

1. INTRODUCTION ............................................................................................................. 1
   1.1 BACKGROUND AND PROBLEM DESCRIPTION ...................................................... 1
   1.2 AIM AND PURPOSE ................................................................................................. 2
   1.3 HYPOTHESIS AND LIMITATIONS ........................................................................ 2
   1.4 RELIABILITY, VALIDITY AND OBJECTIVITY ...................................................... 3

2. THEORY AND LITERATURE REVIEW ......................................................................... 4
   2.1 RESIDUAL STRESS .................................................................................................. 4
   2.2 STRESS RELAXATION ............................................................................................. 4
   2.3 THERMAL ANALYSIS AND HEAT SOURCE ............................................................ 4
   2.4 CONTROLLING THE RESIDUAL STRESSES ........................................................... 7
   2.5 RESIDUAL STRESSES IN WELDING PROCESS .................................................... 8
   2.6 CORRELATION BETWEEN THE VARYING TEMPERATURE AND STRESSES ......... 9
   2.7 COOLING RATE ..................................................................................................... 9
   2.8 WELDING PARAMETER ....................................................................................... 10
   2.9 FATIGUE ............................................................................................................... 10
   2.10 FE-MODELING AND SIMULATION ..................................................................... 13
   2.11 MODELING PROCESS ........................................................................................... 14

2. METHOD ....................................................................................................................... 16
   3.1 FINITE ELEMENT METHOD .................................................................................... 16
   3.2 FEA MESHING ....................................................................................................... 16
   3.3 LS-PREPOST .......................................................................................................... 19
   3.4 ANSA PRE-PROCESSOR ....................................................................................... 22

4. RESULTS ...................................................................................................................... 25
   4.1 T-JOINT WELD MODEL ......................................................................................... 25
       4.1.1 Meshing density impact on the temperature and stress distribution .................. 25
       4.1.2 Influence of the type of the clamping and weld pool geometry on the temperature and stress distribution ................................. 40
       4.1.3 Influence of the step per element parameter on the temperature and stress distribution .......................................................... 51
       4.1.4 Correlation with published measurements ...................................................... 62
   4.2 BUTT WELD MODEL ............................................................................................. 82
       4.2.1 Weld stroke parameters impact on the temperature and stress distribution ....... 82

5. CONCLUSIONS ............................................................................................................ 92

REFERENCES ................................................................................................................... 93
1. Introduction

Articulated haulers are used to transport different kind of material, such as stones, sand or soil. These machines are made up by welded parts. Each of those parts is exposed to significant loads due to transported material by the haulers. Typically, residual stresses occur along with the loads being transmitted. The weld residual stresses may have suggestive impact on the fatigue life. Relaxation that occurs due to service loading may reduce that impact on fatigue life. This master thesis is carried out in cooperation with Volvo Construction Equipment, Braås, Sweden.

1.1 Background and problem description

Volvo Construction Equipment manufacturers, develops and trades construction equipment. The Volvo Construction Technology Department in Braås develops the load carrying structure in articulated haulers.

The articulated hauler invented by Volvo Construction Equipment is a part of the heavy equipment sector with the main objective to transport loads over rough terrain. The first machine of that kind was produced in 1966. The machine consists of the front, which was called the tractor and the rear, called the hauler or load unit containing the dump body. The first machines could handle loads up to 10 tons [1].

Sustainable and increasing sales were the reasonable arguments for continuous development of the vehicle industry. Customer demands of transporting heavier loads resulted in large stresses acting on each element. Those factors encouraged Volvo to develop manufacturing and improve the welding process.

Welding techniques have been known since the industrial revolution in the 1800s. The first, fully arranged joining process by welding in commercial use took place in the 1920s. During World War II welding was pushed even further. Market needs of faster production at lower cost contributed to improvements of welding techniques.

Nowadays, welding has a significant impact on the production of durable products due to its low cost. Various types of welding processes are still under development. Expected outcome of this thesis is correlate with measurement from published in T-joint and butt welding procedures data [23,24] using already known methods such as, tack weld application or changes in the shape of the model.
The most important goal during this thesis is to expand the knowledge of modeling a welding process using the FEA software. Relaxations due to service loading may reduce the impact on fatigue life. Three major fields should be considered in a welding analysis. Material simulation as a thermal and mechanical material properties, process simulation (heat input and heat source parameters) and structural simulation (temperature distribution and residual stress taken into an account).

The influential parameters of welding analysis like temperature dependence, thermal and mechanical material properties, welding parameters, weld types and sequence, heat source parameters, geometry of component will be studied during this thesis. Additionally, the temperature distribution has an explicit impact on weld residual stresses. Furthermore, the used type of weld also changes the strength of the material which could lead to a decrease of the fatigue life.

1.2 Aim and purpose

The aim of this thesis is to model the welding process. Additionally, the aim is to investigate the correlation between the results from a finite element model with published results of real weld residual stress measurements.

The purpose of this thesis is to contribute to the development of a design process which allow to verify correctness of proposed improvements in weld joint regarding to given result of measurements. Furthermore, the purpose is to determine the welding residual stresses using a thermo elasto plastic analysis.

1.3 Hypothesis and limitations

The master thesis suggests that by using improved welding sequence the weld residual stresses will be reduced. It is possible to apply the heat input simultaneously from two sides or separately one after another. Furthermore, welding from one side to the end of the model for both sides at the same time will reduce unwanted weld residual stresses. Moreover, using two clamps instead of one, could increase efficiency of welding process.

This thesis is limited by the already proposed, by Volvo, two welding models. Thus, comparison of residual stresses from the models with given results of measurements will be studied along with welding methods. Additionally, the FEA software such as ANSA Pre-Processor, LS-PrePost and META Post-Processor should be utilized for the assessment of the models.
The investigated aspects are limited by different variations of meshing used in the models and weld seam dimensions. Moreover, the welding aspects are also limited by the material properties. Different types of material have significant impact on cracking resistance and shrinkage so that is why computational modeling will take into consideration those factors.

1.4 Reliability, validity and objectivity

To attain valuable and reliable results, it is critical to create a suitable model devoid of errors. The main measuring equipment in this study is computer software, mainly ANSA LS-PrePost and META Post-Processor. Those aspects are needed to get the results which would fulfill the aim and purpose of the thesis.

Input data used in simulation will be provided by Volvo CE. The company collected the data its within years of experience. In addition, cooperation with Volvo engineers will be an effective way to perform relevant simulations.
2. Theory and Literature Review

2.1 Residual stresses

Residual stresses are derived from the temperature gradient as a function of time within the material. They also can occur through other changing mechanics such as plastic deformations or phase transformation. Residual stress is mostly described as existing stresses inside the component without external load in absence of external loads [5]. Magnitude of residual stresses can arise, when a material is exposed to machining, heat treatment or coating. Because of plastic deformation risk the greatest value of residual stresses never overstep the elastic metal limits. If that value appears, the stress will be used as a distortion of component [6]. Residual stresses are divided into compressive and tensile one. Residual stresses might be tensile or compressive depending on location and volumetric change which arises from differential cooling and heating during processes such as welding or heat treatment.

2.2 Stress relaxation

Stress relaxation is expressed as a stress reduction in function of a time at constant total strain. The relaxed stress is the difference between the initial and remaining stress and is defined as percentage of the initial stress. Commonly, relaxation characteristic is described for constant structure deflection, for external loading and deflection variation or for cases concerning initial residual stresses.

Stress relaxation investigations indicate performance of residual stress relaxation during fatigue for peak cyclic stresses at or near the endurance limit. Moreover, stress concentration could provide the mechanism of relaxation in certain regions, i.e. in low fatigue region cycle, residual stress is influenced by the direction and magnitude of loading. Besides that, it is important to remember about compressive residual stresses dependence in increasing fatigue life and additionally it is highly effects on their stability, since relaxation could take place early in the fatigue process [7].

2.3 Thermal analysis and heat source

To investigate the distribution of transient temperature during the welding process thermal analysis is also required. The dependence between temperature distribution in a function of time \( t \) and spatial coordinates \((x, y, z)\) is described by using nonlinear heat transfer equation posted below:
\[ c \rho \frac{\partial T}{\partial t} = k \left( \frac{\partial^2 T}{\partial x^2} \right) + k \left( \frac{\partial^2 T}{\partial y^2} \right) + k \left( \frac{\partial^2 T}{\partial z^2} \right) + Q \]  

(1)

where: \( c \) - heat capacity (\( J/kg\,^\circ C \)), \( \rho \) - density (\( kg/m^3 \)), \( T \) - temperature (\(^\circ C\)), \( t \) - time (s), \( Q \) - internal heat generation rate (\( W/m^3 \)), \( k \) - thermal conductivity of isotropic material (\( W/m\,^\circ C \)).

During the welding process the total heat input to the seam is modeled as a volumetric heat flux \( q \, (W/m^3) \) and can be described by equation:

\[ q = \frac{\eta UI}{V} \]  

(2)

where: \( \eta \) – arc efficiency, \( U \) – voltage (V), \( I \) – current (A), \( V \) - volume (m³).

First equation clearly indicates material properties, such as heat capacity, thermal conductivity and density have significant impact on the thermal analysis. Heat capacity and thermal conductivity are strongly dependent on temperature than on density [18, 19]. Nevertheless, with lowering thermal conductivity more amount of heat may be accumulated. Additionally, the sample will take more time to conduct heat to material.

In this consideration, the heat loss that occurs due to convection should be also included with an exception of radiation heat losses, as it has no significant effect on the result. Basing on Newton’s law with constant film coefficient heat loss due to convection is expressed by equation

\[ q_c = -h \, (T_s - T_T) \]  

(3)

where, \( T_s \) – surface temperature (\(^\circ C\)), \( T_T \) – ambient temperature room (\(^\circ C\)), \( h \) – film coefficient (\( W/m^2\,^\circ C \)).

In this thesis Gas Metal Arc Welding method will be applied to determine heat source parameters. This approach is popular in industrial applications because the metal
deposition transfer can be controlled by modulation of the current [21]. To obtain high thermal accuracy it is required to correct heat input. The most common method for Gas Metal Arc Welding is Double Ellipsoid Goldak method. That solution was provided by John Goldak in 1980s. The weld pool shape is performed by double ellipsoid, which uses the welding power input Q:

$$ Q = \eta UI $$

where: $\eta$ – arc efficiency, mostly between 70 to 90%, U – voltage (V), I – current (A)

Goldak double ellipsoidal model distributes the arc power as a Gaussian function from a radial distance from the center to the interior of the double ellipsoid [18]. It is defined with six parameters that match six dimensions of the weld pool shape.

$$ q(x, y, z, t) = \frac{6\sqrt{3Q}}{abc\pi\sqrt{\pi}} e^{-\frac{3x^2}{a^2}} e^{-\frac{3y^2}{b^2}} e^{-3[z+v+(\tau-t)]^2} c^2 $$

where: x, y, z – the coordinate system; t – time (s); Q – heat input rate (W); a, b, c – ellipsoid dimensional parameters (m); v – velocity of the torch (m/s); $\tau$ – a lag factor. Dimensional parameters are shown below in the figure 2.1.

![Figure 1: Sketch of the double ellipsoidal heat source [22].](image)
Parameters $a_r$, $a_f$ corresponds to the length of the rear and front halves of the weld pool. Parameter $b$ is equal to half the face of the weld and $c$ is equal to the depth of the weld.

The Goldak double ellipsoidal model is commonly used and has relationship to the weld pool dimensions. On the other hand model does not have relation with the welding parameters, such as weld power and weld speed. Nevertheless, work continues to increase the accuracy of the heat source model. Method invented by Goldak has the quickest computing time of operation for industrial use in residual stress and calculations of the distortion [18].

2.4 Controlling the residual stresses

The applications demand relieving residual stresses of weld seams by mechanical and thermal methods. Residual stress relaxation is based on releasing the locked-in strain by improving conditions to facilitate plastic flow to relieve stresses moment.

Mechanical method involves applying the external load beyond yield strength to cause plastic deformation to the moment of release locked-in strain [8]. Normally, external load is applied in an area where the peak of residual stresses is expected.

Thermal method refers to the decrease of yield strength and hardness of the metals with increasing temperature which in turn facilitates the release of locked-in strain which allow on relieves the residual stresses. Therefore, the higher is temperature of thermal treatment of the weld seam the greater will be reduction in residual stresses [12].
2.5 Residual stresses in welding process

Residual stresses are mostly developed owing to differential thermal cycle which consists of heating to attain the peak of the temperature and later on cooling during welding. In general, magnitude and type of residual stresses continuously change in dependence on the stage of welding, such as cooling or heating. Heating of the base metal leads to melting of the sample due to thermal expansion and is limited by the low temperature of the surrounding metal. When the temperature reaches its peak, compressive residual stress rapidly and significantly decreases which is caused by the softening of previously heated metal. Then it is diminishing till to the surface and achieve zero. Nevertheless, during the cooling process magnitude of tensile residual stresses increase while metal shrinks until it reaches the room temperature [8].

![Temperature distribution during welding](image)

*Figure 2: Temperature distribution during a welding a) in studied locations A, B and C and b) in temperature dependence versus time related to position A, B and C [8].*

There are two basic changes which contribute to the development of the residual stresses. Macroscopic volumetric changes are caused by contraction and expansion or by varying cooling rate between top and bottom welding surfaces. Further, microscopic volumetric changes are caused during cooling due to metal transformation, mainly from austenite to martensitic. [7].
2.6 Correlation between the varying temperature and stresses

Varying cooling and heating rate close to the weld have big impact on development of the residual stresses. They are called thermal stresses. Temperature acts on a volumetric change and strength of the tested material. At the beginning of that process, the temperature is going up while heat source comes closer to the exact point. Yield strength of the material is inversely proportional to the temperature and for that reason strength decreases with thermal expansion at the same time. Furthermore, rising temperature leads to an increase of compressive residual stress. When heat source comes nearly to the point of interest compressive stress immediately decrease and may be even omitted. Finally, while crossing the point by the heat source temperature of base metal gradually reduces which leads to shrinkage of the heat affected zone (HAZ) [12]. It is worth to note that the greatest tensile residual stress formations occur along the weld metal zone. Figure posted below shows described process.

![Figure 3: Distribution of weld residual stresses [25].](image)

2.7 Cooling rate

During welding, top and bottom surfaces of welding joint are exposed on relatively high cooling rate. On the reverse side, there are middle parts of weld and heat affected zone (HAZ), where cooling rate is lower (figure 4). Generally, these differences cause thermal expansion and contraction through the welded plate which may even finally lead to the improvement of compressive (at the surface) and tensile (in the middle part of weld) residual stress [9].
Heat input does influence the temperature distribution, but there is no linear effect on residual stress. Refers to research led by A. M. Paradowska and J. W. H. Price, better to use smaller heat input to obtain smaller magnitude of residual stress. Compressive residual stress exists near to the weld center line when low heat input is applied to the components [20]. When higher heat input is applied a tensile residual stress will occur. To obtain better quality and beneficial residual stress, there are some important things which should be taken into account by welder needs to know before start welding: material specification – base and filler material, welding parameter – heat input especially, welding process and welding sequence.

2.9 Fatigue

First, fatigue features were investigated by Wilhelm Albert. In 1837 he detected the failure of metal chain and it was applicable in mining industry. Next person whose developed science of fatigue was August Wohler. He measured the service loads on rail road axles and several years later published the results of fatigue testing. Wohler stated that fatigue life is dependent on the pick stress and the stress amplitude which is base of fatigue analysis [10].

Metal fatigue was defined by the American Society for Testing of Materials (ASTM) as: “The process of progressive, localized, permanent, structural change occurring in a material subjected to conditions that produce fluctuation stresses and strains that may culminate in cracks or complete fracture after sufficient number of fluctuations” [11]. It is commonly known that localized stresses and strains have significant impact on damage and it occurs in the material at atomic level.
The figure 5 shows that fatigue process can be divided into two stages. Stage 1 is when the crack moves along the maximum shear stress direction since shear stress is moving along the slip direction. Stage 2 begins when crack has grown through few grains. The crack growth direction will be perpendicular to the applied stress direction [12].

![Figure 5: Stages of fatigue crack process [24].](image)

Fatigue life of weld seams significantly differ from the fatigue of non-welded materials. It is hard to perform in industrial area detailed characterization for various structures. To lead well-balanced fatigue research it is necessary to lead the investigations in laboratory. The best way to obtain satisfying results depends on reasonable assumptions where each of them is justified. Additionally, extreme practical conditions should be maintained. Fatigue resistance analysis should assume that cracks which may occur on welded joins will appear after the welding process. Danger of that fatigue cracks is insignificants for all procedure. Besides that, initial cracks propagation may be used as a criterion for very high cycle fatigue. During that cycle, crack growth analysis can be used for low and high cycle fatigue. [13] [14].
**Figure 6:** Stress versus number of cycles (S-N) curve with crack growth diagram [17].

**Figure 7:** Typical S-N curve with three crack areas [17].

*Figure 6* shows that non-welded metals with good surface condition are marked by smaller crack growth probability. According to the *figure 7*, the most time-consuming stage is a crack nucleation and it directly effects on a lifetime of that material.
2.10 FE-modeling and simulation.

FE-modeling is a dominant discretization technique in structural and mechanical engineering. The fundamental concept in the physical interpretation of the FEM is the subsection of the investigated model into disjoint components in simple geometry which is named as finite elements. In terms of a finite number the response of each element can be expressed in the form of degrees of freedom. Each of them is characterized as an unknown function value or functions, at a set of nodal points.

In case of weld joints metal plates can be modeled by using normally available commercial finite element software. Composition of semi-infinite plates, weld-groove angle should lead to creating a mesh consisting element nodes with degrees of freedom. Plates should be fixed at the ends and owing to heat convection heat will be escaped at the top surface of plates [15]. To facilitate all welding simulation process it is needed to evaluate if the separation mechanical and thermal analyses could be computationally efficient.

At the beginning is important to express the evaluation of the temperature distribution at welding process and later subcooling is ended. Then, temperature field is applied and used in the mechanical model to execution the residual stress analysis. Additionally, heat input during welding can be modeled in software by the equivalent heat input which includes heat flux along the body.
2.11 Modeling process.

Modeling process consist three basic parts which effect on each other. Figure 8 presents interactions between the most common factors such as microstructural models, thermal heat-flow models and mechanical models in weld modeling.

![Figure 8: Block diagram showing the most common dependences in weld modeling [24].](image)

Interactions description:

1. Heating and cooling rates coming from phase transformations influence the microstructural changes.

2. Latent heat located in heat sinks absorbs energy and giving off this energy on cooling processes.

3. The volume change caused by rearrangement in the atoms results in a mechanical strain change.

4. Changes in phase change can also be a result of mechanical deformations.

5. Temperature changes effect on the expansion and compression of the materials which can result mechanical distortion.

6. Mechanical deformation has also impact on the temperature changes by heat generation caused by distortions.
Procedure chart in welding analysis was shown in figure 9. At the beginning of the process with the determination of the heat input parameters it is known that this parameter has an influence in microstructural changes. Further, the microstructural changes results on mechanical properties of the material. Additionally, the temperature changes related to the phase transformation should be also considered due to impact on mechanical strains development. After that, with the adjusted mechanical and thermal properties the residual stresses can be computed.

![Diagram](image)

*Figure 9: Procedure chart – welding analysis [24].*
3. Method

The consideration in this thesis will be done by using a numerical method. The results will consist of simulation process with further analysis of obtained results. To handle with established analysis which will be performed based on finite element calculation as an efficient tool for welding simulation with the help of FEA software such as ANSA Pre-Processor, META Post-Processor and by using welding GUI implemented in LS-PrePost.

The fillet and butt weld models will be prepared during this thesis. Using preprocessor software proposed by Volvo CE, ANSA Pre-Processor, solids with meshing and type of material will be created. Furthermore, can be possible to export models to the LS-PrePost to carry out the welding simulation. During that process welding sequence, structure and thermal boundary conditions, weld pool geometry and welding parameters such as weld speed, weld power with efficiency will be determined. Finally, by using LS-Dyna solver I will get a set of results which contribute to show the temperature distribution or position dependence as a function of time. In that software, further investigations will be continued: tensile and compressive residual stress deformation, heat input dependence to properties of the models and parametric studies such as weld order or direction.

3.1 Finite element method

The finite element method (FEM) is a computational method used to receive approximate solutions of boundary value problems. The finite element method is a way to obtaining a numerical answer to a complex engineering problem. For a better understanding of FEM concept, that process consists in the cutting of a structure into few, relevant elements which could describe the behavior of each part in a simple way.

In simulation of welding processes, the choice of finite element is justified. That method has a proper impact on accuracy and computational costs. The most preferable are the linear hexahedron and the quadratic tetrahedron [16]. The figures below show linear hexahedron set.

![Figure 10: Set of linear hexahedron with node numbering [16].](image-url)
Mesh refinement is usually divided into p-refinement, h-refinement or other combination. For instance, in p-refinement the polynomial degree of shape function is preserved, however, the element density is changed for h-refinement and vice versa [2].

In order to introduce the concept of isoparametric finite elements, the eight-node isoparametric quadrilateral element consideration was posted below. The region in the parent domain is defined by four corner points with four other points on the lines between corners. Figure 12 shows the mapping region into the arbitrary quadrilateral shown in the global xy-plane. Each of the eight points in the parent domain can be associated with an element shape function. Summary for eight nodes isoparametric quadrilateral element was shown below on figure 11 [3].

\[
\begin{align*}
N_1^e &= -\frac{1}{4}(1 - \xi)(1 - \eta)(1 + \xi + \eta); & N_5^e &= \frac{1}{2}(1 - \xi^2)(1 - \eta) \\
N_2^e &= -\frac{1}{4}(1 + \xi)(1 - \eta)(1 - \xi + \eta); & N_6^e &= \frac{1}{2}(1 + \xi)(1 - \eta^2) \\
N_3^e &= -\frac{1}{4}(1 + \xi)(1 + \eta)(1 - \xi - \eta); & N_7^e &= \frac{1}{2}(1 - \xi^2)(1 + \eta) \\
N_4^e &= -\frac{1}{4}(1 - \xi)(1 + \eta)(1 + \xi - \eta); & N_8^e &= \frac{1}{2}(1 - \xi)(1 - \eta^2)
\end{align*}
\]

*Figure 11: Shape functions for eight-node isoparametric quadrilateral element [3].*

Domains are independently integrated by Gaussian quadrature scheme for deviation strains. The variable number of nodes and the interelement compatibility make the grade element extremely efficient in mesh grading algorithms. The mesh grading algorithm developed automatically identifies mid-edge and mid-face locations and adds the necessary node to the definition of the graded element [3].

The two-dimensional four- and eight-node isoparametric elements can be easily extend. Three-dimensional element is similar to 8-node, but constrains associated with the mesh grinding are settled in the shape functions to ensure compatibility. According to *figure 10*, nodes 1-8 are vertex whereas nodes 9-20 and 21-26 are optional mid-edge and mid-face nodes. Shape functions for graded elements must be calculated, remembering about if a node is absent the matching shape function will be zeroed.
Depending on the number of the nodes and their location, the element is subdivided into number of linear subdomains (1-8). The boundaries will be created by the graded element and the adjacent linear elements. Each subdomain will be integrated by a 2x2x2 Gaussian quadrature scheme for the deviatoric strains. The mesh grinding developed identifies the mid-edge and mid-face positions and adds the important node to the graded elements.

3.2 Meshing

Mesh quality is important to perform well designed model. To increase model accuracy is recommended to use finer mesh. Nevertheless, mesh cannot be too fine to avoid ill-conditioned system. On the other hand, finer mesh takes more time to solve. It is recommended to prepare coarser mesh and then repeat the analysis with a finer mesh until the change in the solution is sufficiently small. At each stage one can also visualize the element error that shows where the meshing improvement would provide the largest gain in accuracy. In most situations, first coarser mesh is almost sufficiently accurate over significant regions and the mesh only needs to be refined in local regions to reach greater accuracy.

Manufacturers provide fully or semi-automatic meshing tools which are proprietary meshing algorithms. Those solutions are fast, robust and still are undergoing continuous development to improve the mesh flow and quality.

A picture posted below present example of T – joint weld meshing. This model consists areas of different meshing densities. Application of more accurate meshing on the load exposed areas has a justification. Mostly from reason, that on locations will occur the greatest temperature and stress changes (HAZ – heat affected zone).

![Figure 13: Example of T–joint weld model meshing.](image)
3.3 LS-PrePost

Nowadays, on the modern engineering market there are a lot of techniques which have been applied in industry. In that case LS-PrePost have been used to express advanced computational method for fatigue assessment of welding joints and thermo physical material designing.

LS-PrePost is an explicit FE-solver which makes each time step fast. All the same time small time steps are required. On the basis of input data with material properties can be possible to obtain lifetime assessment of welded structures with some variations.

In this thesis LS-PrePost is used for welding analysis. Welding GUI in LS-PrePost software enables performing parametric study of different variables.

a) Welding sequence

The user can freely change weld sequence settings. The bookmark for the Sequence is placed on the bookmarks bar at the top of Welding Simulation window. Then, it is possible to determine weld order at its discretion. Furthermore, to each weld, corresponding structural boundary condition in a form of a clamping and air segment (thermal boundary condition) can be chosen. If the icon is highlighted in green, this option is possible to perform. Described settings are shown on the figure 14.

![Welding Simulation](image)

*Figure 14: Welding simulation in LS-PrePost software – Sequence bookmark.*
b) Weld parameters

In this bookmark the user should match *the welding path*. It means that trajectory with corresponding reference path need to be determined. Later, *weld pool geometry* with relevant factors is required. Shape of heat input is expressed by Goldak double ellipsoidal heat source (figure 15).

Weld source moves with prescribed motion and weld path with corresponding path are treated as beam element.

![Figure 15: Welding simulation in LS-PrePost software – Welds bookmark.](image)
c) Structural and thermal boundary conditions

Last bookmarks let the user to determine boundary conditions. From structural point of view is necessary to pick the proper coordinates to fix the clamps. That condition let to avoid unexpected movements of plates during the welding (figure 16).

![Welding simulation in LS-PrePost software – Structural Boundary Condition bookmark.](image)

*Figure 16: Welding simulation in LS-PrePost software – Structural Boundary Condition bookmark.*

*Thermal boundary condition* was described more specifically in theoretical chapter. Initial temperature of metal plates, convection and radiation factors should be taken into consideration (figure 17).

![Welding simulation in LS-PrePost software – Thermal Boundary Condition bookmark.](image)

*Figure 17: Welding simulation in LS-PrePost software – Thermal Boundary Condition bookmark.*
3.4 ANSA Pre-Processor

ANSA is pre-processor software produced by BETA CAE Systems. It is an advanced pre-processing CAE tool which provides the most valuable functionality from CAD data to already prepared ready-to-run solver input file. ANSA fulfills CAE pre-processing requirements such as shell and volume elements generation, batch meshing, assembly functions, modeling of welding processes and pre-processing analyses setup for major FEA solvers. ANSA owes its success due to innovative concept of GUI adaptation [4].

The figures below present two types of welding models which were proposed by Volvo to model and carried out analysis in this thesis:

a) T-joint weld model [24]

Dimensions of the T-joint weld model used in this thesis:

- 300mm x 300mm x 150mm x 10mm (length, width, height and thickness respectively),

- 300mm x 600mm x 50mm x 10mm,

- 300mm x 300 mm x 50 mm x 10mm.

Figure 18: Fillet weld (T-joint) model with weld trajectory and reference paths in ANSA software.
Weld seams diameters used in investigations:

- 5,65mm x 5,65mm x 300mm (cathetus, cathetus, length)
- 8mm x 8 mm x 300mm
- 8mm x 8 mm x 600mm

To perform a welding process it is necessary to determine trajectory and reference line for the weld seam. Trajectory line should be placed at the middle line along the weld seam. Well-constructed reference line has to be perpendicular to and located in relevant distance above a trajectory line. On the figure 20 are shown example of trajectory and reference lines.
b) Butt weld model

Dimensions of the butt weld model: 200mm x 100mm x 6mm (length, width and thickness respectively), angle between surfaces 60˚, gap between baseplates 2mm (figure 21) [23].

Figure 20: Trajectory and reference lines in ANSA software.

Figure 21: Butt weld model in ANSA software.
4. Results

In this section the results will be presented and analyzed. In section 4.1 T-joint weld model investigation will be performed and in section 4.2 Butt weld model simulation respectively.

4.1 T-joint weld model

4.1.1. Meshing density impact on the temperature and stress distribution.

In this section, results of the investigation in relation to example welding parameters data will be presented. In T-joint weld model simulations was used material database based on Steel S355. Those studies have been conducted to confirm correctness of the cooperation between output file used in LS-PrePost based on and prepared in ANSA Pre-Processor. Additionally, meshing impact on total computing time will be performed.

4.1.1.1. Dimensions of the model.

First investigation has been carried out according to the following dimensions of the T-joint model (figure 22). Heat affected zone (HAZ) and weld seam dimensions are shown on the figure 23. Accurate dimensions of the meshing are shown on the figure 24.

Figure 22: General dimensions of the T-joint model.
Figure 23: Heat affected zone (HAZ) dimension.

Figure 24: Dimension of the meshing – case 1.
4.1.1.2. Welding application settings in LS-PrePost.

This model has been fixed by using clamp located on the side of start first welding order – yellow color (figure 25). This clamp was fixed during all welding process (figure 26).

![Figure 25: Clamping condition.](image)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>== Welds ==</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18 Weld_1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19 Weld_2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Springback</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>== Struc. B.C. ==</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5 Clamp_1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6 Clamp_2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7 Clamp_3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8 Clamp_4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>== Therm. B.C. ==</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Air segm.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Figure 26: Welding orders with corresponding clamp.](image)
The size parameters used in the double ellipsoid model which were explained in section 2.3 “Thermal analysis and heat source” (figure 27). Additionally, in this simulation were used other parameters, such as velocity of the weld seam, weld power, efficiency factor, number of step per element, NCYC and cool down time. Those example settings were set according to the previous tests at Volvo CE.

For both welding orders used exactly the same values of weld stroke. Clamp_1 was applied during the first weld. The cooling time is equal to 5 seconds. Then, after first cooling down time second weld order was applied. At the end of the process, next time same cooling down time was applied. Finally, spring back went back to the starting point and all the simulation has been finished after 55 seconds.

![Figure 27: Weld pool geometry and welds stroke parameters.](image)

To determine thermal boundary condition initial temperature of the baseplate has set as a 20°C. Figure 28 attached below shows the thermal settings.

![Figure 28: Thermal boundary settings.](image)
4.1.1.3. Temperature distribution.

To investigate the temperature distribution during the welding process node 48024 was used. This node is located on the middle line path across the weld seam and has a direct contact with the edge of the weld seam (figure 29). The results of the temperature distribution are following:

![Temperature distribution history at node 48024 during a welding process.](image)

Figure 29: Temperature distribution history at node 48024 during a welding process.
Referring to the chart, the highest temperature has been obtained within second welding order as the heat flux along the weld line and it was 281°C. It could be concluded that cooling down time 5 seconds is not sufficiently to obtain initial temperature of the baseplate. At the end of the process temperature of the baseplate at chosen node was 145°C which is sharply too high comparing to the initial temperature.
4.1.1.4. Stress distribution.

To investigate the stress distribution during the welding process line across the weld line has been proposed to show the most accurate stresses precisely on the middle path line through the model (figure 31 and figure 32). The results of the stress distribution are following (figure 33):

![Stress distribution diagram](image)

*Figure 31: Stress distribution with proposed middle path line in vertical projection.*
Figure 32: Longitudinal stress distribution with proposed middle path line in vertical projection.
Referring to the chart posted above, the highest longitudinal stresses have been obtained close to edge of the stiffener (two peaks). In longitudinal stress distribution has been observed stress gap precisely in the middle of the baseplate and this is due to insufficient parameters of the weld power. Then, stresses rapidly decreased as a result of applied welding order. After that, stresses have increased once again. That varying tendency is caused due to applied clamp and that is why stress lines are not symmetric. However, in transverse residual stresses distribution from the beginning of the baseplate could be observed smooth compressive tendency until to the contact with the weld seam. Then, a short increase of the stresses at the beginning the heat affected zone and after that, rapid drop through the stiffener.
4.1.1.5. Parametric study.

Presented above methodology of the comparison was conducted for few other models differing to each other dimensions of the elements in the heat affected zone. Figure 34 shows size of elements for second model (case 2) and figure 35 size of elements for third model (case 3).

![Figure 34: Dimensions of the meshing – case 2.]

![Figure 35: Dimensions of the meshing – case 3.]

Maciej Maczugowski
<table>
<thead>
<tr>
<th>Case</th>
<th>Analysis</th>
<th>Number of elements</th>
<th>Number of nodes</th>
<th>Element size (HAZ)-length [mm]</th>
<th>Element type</th>
<th>Step/element</th>
<th>CPU [m]</th>
<th>Memory [m]</th>
<th>Total Solution Time</th>
<th>Cooling down time [s]</th>
<th>Weld Geometry [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Thermal and Mechanical</td>
<td>44475</td>
<td>53060</td>
<td></td>
<td>0,5</td>
<td>1</td>
<td>500</td>
<td></td>
<td>00:11:51</td>
<td>5</td>
<td>3 3 3 6</td>
</tr>
<tr>
<td>2</td>
<td>Thermal and Mechanical</td>
<td>140100</td>
<td>156420</td>
<td></td>
<td>0,5</td>
<td>1</td>
<td>500</td>
<td></td>
<td>00:37:38</td>
<td>5</td>
<td>3 3 3 6</td>
</tr>
<tr>
<td>3</td>
<td>Thermal and Mechanical</td>
<td>560400</td>
<td>619470</td>
<td></td>
<td>0,5</td>
<td>1</td>
<td>1000</td>
<td></td>
<td>04:07:15</td>
<td>5</td>
<td>3 3 3 6</td>
</tr>
</tbody>
</table>
Figure 36: Collective summary of the temperature distribution at node N48024 placed on the middle path line across the baseplate.
Figure 37: Collective summary of longitudinal stresses along the weld seam.
Figure 38: Collective summary of transverse stresses along the weld seam.
Referring to the charts mentioned above could be concluded general dependency size of elements on temperature and stress distributions. For a model consists 560400 elements the highest measured temperature was 409°C (case 3), for case 2 temperature was 350°C and for first case 281°C respectively. The greater number of elements the higher temperature values. The general tendency for each curve varies mostly in the heat peaks. Rest of the temperature distribution stages proceed similar and their values as well.

According to the stress distribution plots could be deduced that density of meshing has relatively big impact on the reached values of the extremes. Tensile residual stress extremes occur close to the heat peaks and there are directly dependent to the temperatures. The higher temperature the higher stress. Observed stress gap between 125 mm and 175 mm along the length of the baseplate was caused by the heat input parameters. Values of the weld speed and weld power are probably not sufficient to obtain real stress distributions curves. Compressive stress differences are also noticeable and for the model with the densest meshing attain lower value of those stresses.
4.1.2. Influence of the type of the clamping and weld pool geometry on the temperature and stress distribution.

In this section, results of the investigation will be presented in relation to the weld input power and type of the clamping. Weld input power factor will be studied to model a correct heat input to the structure. Type of the clamping will be investigated to improve the knowledge how the stress distribution will differ with the different types of the clamping. Additionally, the weld pool geometry has been improved.

4.1.2.1. Dimensions of the model.

Second set of simulations have been carried out according to the following dimensions of the T-joint model (figure 39). Heat affected zone (HAZ) and weld seam dimensions are shown on the figure 40. Accurate dimensions of the meshing are shown on the figure 41.

![Figure 39: General dimensions of the T-joint model.](image-url)
Figure 40: Heat affected zone (HAZ) dimensions.

Figure 41: Dimension of the meshing.
4.1.2.2. Welding application settings in LS-PrePost.

Models have been fixed by using different clamps. In case 1 fixing are placed in two corners from side of the beginning first welding order (figure 42). Case 2 consists one, longitudinal clamp along the weld seam and is placed precisely in the middle of the left side of the baseplate (figure 43).

Figure 42: Clamping conditions – case 1.

Figure 43: Clamping conditions – case 2.
For case1 and case2 were used exactly the same parameters of the weld stroke. Models differ in weld pool geometry and type of clamping. That set of clamping was used to increase rigidity of a model to avoid unexpected movements. Moreover, cooling down time has changed and increased to 60 seconds in relation to section 4.1.1 to attain initial temperature of a baseplate. For case1 both clamps were applied within both orders of welding and for case2 one clamp respectively. The simulation has been finished after 300 seconds. Described settings are shown below (case1 - figure 44 and case2 - figure 45).

<table>
<thead>
<tr>
<th>Weld pool geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ellipsoid</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>a</td>
</tr>
<tr>
<td>b</td>
</tr>
<tr>
<td>cf</td>
</tr>
<tr>
<td>Cr</td>
</tr>
<tr>
<td>Ff</td>
</tr>
<tr>
<td>Fr</td>
</tr>
<tr>
<td>n</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Weld stroke</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>Weld Power</td>
</tr>
<tr>
<td>Eff. factor</td>
</tr>
<tr>
<td>Step/ele</td>
</tr>
<tr>
<td>NCYC</td>
</tr>
<tr>
<td>Cooldown Time</td>
</tr>
</tbody>
</table>

Figure 44: Weld pool geometry and welds stroke parameter – case 1.
To determine thermal boundary condition initial temperature of the baseplate has set as 20°C. *Figure 46* attached below shows thermal settings.

*Figure 45: Weld pool geometry and welds stroke parameter – case 2.*

*Figure 46: Thermal boundary settings.*
4.1.2.3. Temperature distribution.

To investigate the temperature distribution during the welding process node 83043 was used. This node is located on the middle line path across the weld seam and has a direct contact with the edge of the weld seam (figure 47).

![Temperature distribution history at node 83043 during a welding process.](image)

4.1.2.4. Stress distribution.

To investigate the stress distribution during the welding process line across the weld line has been proposed to show the most accurate stresses precisely one the middle path line through the model (figure 48).
4.1.2.5. Parametric study.

Presented above methodology of the comparison was conducted for two models. Table 2 concerns described differences between models. Figure 49 presents consolidated chart of temperature distribution for chosen node N83043, figure 50 performs differences of longitudinal stresses and figure 51 differences of transverse residual stresses.
<table>
<thead>
<tr>
<th>Case</th>
<th>Analysis</th>
<th>Number of elements</th>
<th>Number of nodes</th>
<th>Element size (HAZ)-length [mm]</th>
<th>Element type</th>
<th>Step/element</th>
<th>CPU [m]</th>
<th>Memory [m]</th>
<th>Total Solution Time</th>
<th>Cooling down time [s]</th>
<th>Weld Geometry [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Thermal</td>
<td>45000</td>
<td>52969</td>
<td>0,5 1</td>
<td>quads</td>
<td>0,5</td>
<td>1</td>
<td>500</td>
<td>00:20:0</td>
<td>60</td>
<td>3 3 7 7</td>
</tr>
<tr>
<td>2</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>0,5 1</td>
<td>quads</td>
<td>0,5</td>
<td>1</td>
<td>500</td>
<td>02:08:3</td>
<td>60</td>
<td>5 5 8 9</td>
</tr>
<tr>
<td>2</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>0,5 1</td>
<td>quads</td>
<td>0,5</td>
<td>1</td>
<td>500</td>
<td>02:05:2</td>
<td>60</td>
<td>3 3 7 7</td>
</tr>
</tbody>
</table>
Referring to figure 49 could be concluded influence of improved weld pool geometry on achieved temperature. Using smaller dimensions, it is possible to reach 1253°C, but when geometry of a heat source will increase the highest temperature might be 1356°C.

Figure 49: Collective summary of the temperature distribution at node N83043 placed on the middle path line across the baseplate.
Regarding to figure 50 there are significant difference in stress distribution in relation to types of the clamping. For case1 where two corner clamps were applied longitudinal stress curve looks smoother than for case2. Nevertheless, peak of the longitudinal stresses looks more relevant due to improved weld pool geometry.
Figure 51: Collective summary of transverse residual stresses along the weld seam at the bottom of the baseplate.

According to figure 51 could be observed similar tendency in transverse stress distribution. Basing on prominent stress gap between 60mm and 80mm of a distance it is easy to conclude where fixing from case2 has been located. As well as on longitudinal stress plot for case 2 lower extremes close to the heat affected zone have been observed.
4.1.3. Influence of the step per element parameter on the temperature and stress distribution.

In this section, results of the investigation will be presented in relation to step per element factor to expand knowledge how the temperature and stress distributions will change and how strongly effect on computing time.

4.1.3.1. Dimensions of the model.

Second set of simulations have been carried out according to the following dimensions of the T-joint model (figure 52). Heat affected zone (HAZ) and weld seam dimensions are shown on the figure 53. Accurate dimensions of the meshing are shown on the figure 54 [24].

![Figure 52: General dimensions of the T-joint model.](image)
Figure 53: Heat affected zone (HAZ) dimensions.

Figure 54: Dimension of the meshing.
4.1.3.2. Welding application settings in LS-PrePost.

This model has different type of clamping than previous. For first welding order was applied one clamp from the same side as a welding order – yellow color and during second weld order clamp was changed on second fixing respectively – blue color. That set of clamping was used to increase rigidity of model to avoid unexpected movement and was presented on the figure 55. The clamps were modeled to have the same dimensions as the vise holding the sample during welding.

![Clamping condition](image)

*Figure 55: Clamping condition.*

For both welding orders used exactly the same values of weld pool geometry and weld stroke except of step/element factor. That factor increase or decrease computing data time by division each element. The cooling down time has been set as a 500 seconds for both welding orders. All simulation has been finished after 1360 seconds. That description is shown on the figure 58.
To determine thermal boundary condition initial temperature of the baseplate has been set as a 20°C. Figure 57 attached below shows thermal settings.
4.1.3.3. Parametric study.

To investigate the temperature distribution during the welding process node N69050 was used. This node is located on the middle line path across the weld seam and has a direct contact with the edge of the weld seam (figure 58). Summary results of the temperature distribution for all variations are presented on the figure 60.

Figure 58: Temperature distribution history at node 69050 during a welding process.
To carry out the stress distribution analysis, middle line path across the weld seam has been proposed (figure 59). The results of the stress distribution are shown on the figure 61 and figure 62):

*Figure 59: Longitudinal stress distribution with proposed middle path line in vertical projection.*
<table>
<thead>
<tr>
<th>Case</th>
<th>Analysis</th>
<th>Number of elements</th>
<th>Number of nodes</th>
<th>Element size (HAZ)-length [mm]</th>
<th>Element type</th>
<th>Step/element</th>
<th>CPU</th>
<th>Memory [m]</th>
<th>Total Solution Time</th>
<th>Cooling down time [s]</th>
<th>Weld Geometry [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>1 2 5</td>
<td>quads</td>
<td>1,5</td>
<td>1</td>
<td>500</td>
<td>16:29:45</td>
<td>500</td>
<td>6 12 6 12</td>
</tr>
<tr>
<td>2</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>1 2 5</td>
<td>quads</td>
<td>2</td>
<td>1</td>
<td>500</td>
<td>20:21:36</td>
<td>500</td>
<td>6 12 6 12</td>
</tr>
<tr>
<td>3</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>1 2 5</td>
<td>quads</td>
<td>3</td>
<td>1</td>
<td>500</td>
<td>21:08:41</td>
<td>500</td>
<td>6 12 6 12</td>
</tr>
<tr>
<td>4</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>1 2 5</td>
<td>quads</td>
<td>4</td>
<td>1</td>
<td>500</td>
<td>27:51:57</td>
<td>500</td>
<td>6 12 6 12</td>
</tr>
<tr>
<td>5</td>
<td>Thermal and Mechanical</td>
<td>45000</td>
<td>52969</td>
<td>1 2 5</td>
<td>quads</td>
<td>6</td>
<td>1</td>
<td>500</td>
<td>36:54:51</td>
<td>500</td>
<td>6 12 6 12</td>
</tr>
</tbody>
</table>
Figure 60: Collective summary of the temperature distribution at node N69050 placed on the middle path line across the baseplate.
Figure 61: Collective summary of longitudinal stresses along the weld seam.
Figure 62: Collective summary of transverse stresses along the weld seam.
Referring to the figure 60 mentioned above could be concluded that general tendency for each curve varies mostly in the heat peaks and there are not any significant values between each of them. The highest temperature has been detected in second model consisting step/element 2 and it was 783°C. On the other hand, the lowest value has been measured in first model (step/element 1.5) and it was 754°C.

Regarding the longitudinal residual stresses (figure 61), no reasonable pattern can be established since the time both have increased and decreased regardless at the step per element factor.

According to the figure 62 which presents transverse residual stresses could be concluded fact that for the models with lower step per element factor lower variations in compressive stresses can be observed. Situation is opposite for the model with higher step per element factor where lower variations in tensile stresses are exist.

Figure 63 presents step per element factor dependence on residual stresses. In general conclusion, the most significant impact during last consideration has computing time. Calculation takes quite a long time and taking into account this fact, could be concluded that the most effective model is that one with the lowest step per element factor.
4.1.4. Correlation with published measurements. [24].

In this section, results of the investigation will be presented in relation to the document “Progress toward a model based approach to the robust design of welded structures” which contribute to the development of fatigue life and stress distribution knowledge.

4.1.4.1. Problem description.

This simulation has been carried out according to the same dimensions as on the figure 52. A study of the model was made in three locations along the weld on a sample shown below (figure 64). The first cross – section location was made 200 mm from the top of the sample. The second cross – section was made at 300 mm from the end of the sample and a third one 400 mm from the end of the weld.

For each location multiple residual stresses were measured. The stresses were made on the vertical plate 7 mm from the top of the baseplate and last measurement 15 mm respectively. The locations are shown in figure 65.

![Figure 64: Sketch showing the three investigated cross- sections.](image-url)
4.1.4.2. Comparison of measurements and FEM simulation of longitudinal stresses $S_x$, transverse stresses $S_y$ and transverse through the thickness $S_z$, 7mm above the baseplate.

Clamps were defined in the Ansa Software as prescribed nodes. Figure 66 shows the clamps location and geometry (case1). Clamp_1 has been prescribed for welding_2 and clamp_2 for welding_1 respectively. For more accurate mapping the welding and cooling procedure two different sets of structural boundary conditions were applied to constrain ridged body motion. The first clamp (nf1) has been applied between welding_1 and welding_2 to avoid model movements. The second set of clamp (nf2) has been used during the cooling process after fully completed welding_2. The pair of clamps, clamp_1 with clamp_2 and nf1 with nf2 were designed to provide the same dimensions as the vice versa.
Weld pool geometry and weld stroke parameters in that consideration have been expressed on figure 67. Thermal boundary conditions were the same as on the figure 57.

Figure 66: Clamping set up for case1 in horizontal projection.

Figure 67: Weld pool geometry and weld stroke parameters.
Figure 68 shows predicted residual stresses from the American paper (Paper-FEM-VrWeld), two measurements from the paper (Paper – measurement 24-2 and 23-2) and the result of simulation made in LS-PrePost. X-label was determined as a cross – section of the stiffener (distance in mm). Results of the FEM simulation compared to the paper measurements and VrWeld simulation 7mm above the baseplate are following:

![Diagram](image)

Figure 68: Comparison between the simulated and measured residual stresses (from paper) with simulations in LS-PrePost in the z-direction.

Referring to the figure above could be concluded that LS-PrePost simulations is similar to the measurements [24]. Magnitude of the stresses on the right side of the stiffener is a bit higher than during the experiments. The most important fact is such that thesis tests have been made with much better accuracy than in VrWeld simulation. The error bars for the measurement stresses are +/- 23 MPa.
Figure 69: Comparison between the simulated and measured residual stresses (from paper) with simulations in LS-PrePost in the y-direction.
Figure 70: Comparison between the simulated and measured residual stresses (from paper) with simulations in LS-PrePost in the x-direction.

Figure 69 shows comparison of the stresses in y-direction and figure 70 comparison of the stresses in x-direction respectively. In comparing to the results in z-direction can be observed smaller stress differences between predicted and simulated values from the paper with simulations in LS-PrePost. Those differences are constant and vary between 10-30 MPa at the whole distance. Last experimental point should be neglected due to uncertainty of the measuring equipment. The error bars for the measurement stresses in y-direction are +/- 34 MPa and +/- 25 MPa in x-direction.
4.1.4.3. Parametric study – direction of the weld order influence on the longitudinal residual stress distribution.

In this section two different cases will be compared. *Case 1* has been already described in section 4.1.4.2 (*figure 66*). *Case 4* was presented in *figure 71* and differs comparing to the case1 direction of the welding_2. Start location of the second welding sequence has been constructed from the opposite side of the baseplate. Weld pool geometry, weld stroke parameters and thermal boundary conditions in that consideration have been expressed on *figure 67* and *figure 57*.

*Figure 71: Clamping set up for case 4 in horizontal projection.*
Figure 72: Comparison between the simulated and measured residual stresses (from paper) with simulations in LS-PrePost in the z-direction – parametric study – weld order influence.

Figure 72 shows comparison of the case1 and case4 in z-direction. Referring to the figure above could be concluded that direction of the weld sequence has not any significant influence on the longitudinal residual stress distribution. Only one slight difference can be observed on the left side of the stiffener and value of the stresses for case4 is 6 MPa higher than for case1. Additionally, same as on the previous section, the thesis simulations have been made with much better accuracy than in VrWeld simulation. The error bars for the measurement stresses are +/- 23 MPa.
4.1.4.3. Parametric study – cooling boundary conditions influence on the longitudinal residual stress distribution.

In this section three different cases will be compared to each other. Case1 has been already described in section 4.1.4.2 (figure 66). Figure 73 shows other the clamp location (case2). As in a previous case clamp_1 and clamp_2 have been used in welding procedure, but during the rotation of the sample and the cooling process clamp named as a stiffix has been applied. That clamp was located precisely in the middle and at the top of the stiffener.

![Figure 73: Clamping set up for case2 in horizontal projection.](image)
Figure 74 presents third case of the clamps. To simplify the welding and cooling procedure directly in LS-PrePost software, clamp_1 has been used for welding_2 and additionally for the cooling process.

*Figure 74: Clamping set up for case3 in horizontal projection.*

Weld pool geometry, weld stroke parameters and thermal boundary conditions in that consideration have been expressed on figure 67 and figure 57.
Figure 75: Comparison between the simulated and measured residual stresses (from paper) with simulations in LS-PrePost in the z-direction – parametric study – cooling boundary conditions influence.

Figure 75 shows comparison of the case1, case2 and case3 in z-direction. Referring to the figure above could be concluded that neither fixing at the stiffener nor simplified settings of the clamping have not considerable influence on the longitudinal residual stress distribution. From the left side of the stiffener and values of the stresses differ to each other from 10-20 MPa. Additionally, in the middle of the vertical plate can be seen 20 MPa difference between simplified case3 and case1 with case2. It might cause by the reduction of the supports during cooling process. Same as on the previous section, the thesis simulations have been made with much better accuracy than in VrWeld simulation. The error bars for the measurement stresses are +/- 23 MPa.
4.1.4.4. Parametric study – application of the tack welds and his influence on the longitudinal residual stress distribution.

In this section two different cases will be compared to each other. Case2 has been already described in section 4.1.4.3 (figure 73). Second case has been defined directly in the ANSA Pre-Processor and each six tack welds were determined as a solid body, so type of the contact between tack weld, stiffener and baseplate have not been prescribed. Figure 76 express described above the tack weld model. Weld pool geometry, weld stroke parameters and thermal boundary conditions in that consideration have been expressed on figure 67 and figure 57.

Figure 76: Tack weld model specification.
Figure 77 shows comparison between simulations with tack weld application and ordinary weld without that improvement. Referring to the figure above could be concluded that level of stresses for both cases are similar, but not the same. Stress deviation is maximum 5 MPa higher for simulations without tack application. That observation leads to conclusion that if analyze takes into account only stress distribution cannot be seen any big deviation between compared simulation. Otherwise, if consideration concerns fatigue life, it will might turn out that tack welds could more significantly effect on improvement of the welding process.
4.1.4.5. Parametric study – torch angle influence on the longitudinal residual stress distribution.

In this section three different settings will be compared to each other. Case3 has been already described in section 4.1.4.3 (figure 74). First set up has been described for 90-degree angle between weld trajectory and surface of a weld seam (figure 78). Second set up expresses 75-degree angle (figure 79) and last one 105-degree angle respectively (figure 80).

Other parameters such as weld pool geometry, weld stroke parameters and thermal boundary conditions in that consideration have been expressed on figure 67 and figure 57.

![Figure 78: Torch angle 90-degree between trajectory and weld seam.](image-url)
Figure 79: Torch angle 75-degree between trajectory and weld seam.

Figure 80: Torch angle 105-degree between trajectory and weld seam.
Figure 81 shows comparison of three different settings concerning angle between weld trajectory and surface of a weld seam in z-direction. Referring to the figure above could be concluded that for 90 and 105 degrees value of stresses are almost the same. Difference between those settings is maximum 5 MPa. Only result for 75-degree angle could be astonishing. From the beginning of the left side of a stiffener tensile residual stress is 40 MPa higher than for 105-degree angle and 45 MPa for 90-degree angle. Moreover, along the 7-mm line on the cross-section of a stiffener residual stress increase much slower than for rest two settings and finally on the right side of a vertical plate values are comparable.
4.1.4.6. Parametric study – shape of the stiffener in contact with the weld seam and his influence on the longitudinal residual stress distribution.

In this section, two different settings will be compared to each other. Case3 has been already described in section 4.1.4.3 (figure 74) and it is treated as a first setup. Second model has different dimensions of the stiffener. Surface which has a direct contact with a surface of the baseplate has 6 mm. Heat affected zone (HAZ) and weld seam dimensions are shown on the figure 82. Accurate dimensions of the meshing are shown on the figure 83. Other parameters such as weld pool geometry, weld stroke parameters and thermal boundary conditions in that consideration have been expressed on figure 67 and figure 57.

Figure 82: Heat affected zone (HAZ) dimensions – T-joint model with a gap.
Figure 83: Dimension of the meshing – T-joint model with a gap.
Referring to the figure 84 could be concluded that narrow down have not any big influence on the stress distribution 7 mm above the baseplate. Higher stress concentration has been observed closer to the surface of the baseplate, but not on the tested line.
The residual stress distribution simulated in LS-PrePost software seems match to experimental results taken from the American paper [24]. In general, LS-Dyna simulations fit much better than paper simulations and values of the stresses are almost same as experimental. Some anomalies have been observed closer to the right edge of the stiffener. It might be caused by the differences in the material. Authors of the document have used material ASTM A572 grade 50 high strength low alloy steel (HSLA) [24]. In thesis simulations were used comparable material structural steel S355. For that reason, anomalies on the right side of the stiffener might have been observed. To decrease differences between paper experimental data with LS-Dyna simulations error bars for the stress in x-direction is +/- 25 MPa, in y-direction +/- 34 MPa and +/- 23MPa for the z-direction.
4.2. Butt weld model

4.2.1. Weld stroke parameters impact on the temperature and stress distribution.

In this section, results of the investigation in relation to welding parameters based on given measurements [23] will be presented.

In Butt weld simulations was used other material database which is based on Low Carbon Steel (ASTM A36). Those studies have been conducted to confirm correlation with measured data and additionally cooperation between output file used in LS-PrePost based on and prepared in ANSA Pre-Processor.

4.2.1.1. Dimensions of the model.

This simulation has been carried out according to the following dimensions of the Butt weld model (figure 85). Heat affected zone (HAZ) is shown on the figure 86. Accurate dimensions of the meshing are shown on the figure 87.

Figure 85: General dimensions of the Butt weld model.
Figure 86: Heat affected zone (HAZ) dimensions.

Figure 87: Dimension of the meshing.
4.2.1.2. Welding application settings in LS-PrePost.

This model has been fixed by using two clamps located as on the figure below (figure 88) to ensure rigidity during the welding process. In case1 during welding process clamp middle1 and middle2 have been applied and during the cooling process middle2 has been removed and only middle1 was fixed. In case2 for both processes – welding and cooling procedure – both clamps were applied. Described set up are shown on the figure 89.

![Figure 88: Clamping conditions in Butt weld model.](image)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>== Welds ==</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20 Weld_1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Springback</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>== Struc. B.C. ==</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1 loong1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 loong2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5 middle1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6 middle2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7 corner1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8 corner2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9 corner3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10 corner4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>== Therm. B.C. ==</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Air segm.</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Figure 89: Welding order with corresponding clamps.](image)
For butt welding order used relevant parameters of the weld pool geometry, step per element and NCYC variables. Rest of the weld stroke factors were used due to reference document [23] to recreate the same conditions as in that document:

- Weld speed $v$ - 2 mm/sec,
- Efficiency $\eta$ – 75%,
- Voltage $V$ - 25V,
- Current $I$ - 90A,
- Cooling down time – 50 seconds;

All simulation process has been finished after 200 seconds.

![Weld pool geometry and welds stroke parameters](image)

Figure 90: Weld pool geometry and welds stroke parameters.

To determine thermal boundary condition initial temperature of the baseplate has set according to reference document [23] as a $27^\circ$C. Figure 91 attached below shows thermal settings.
4.2.1.3. Temperature distribution.

To investigate the temperature variation with distance along the transverse direction on the baseplate line across the weld seam has been used (figure 92)
Figure 92 shows the temperature variation with distance along the transverse direction on the baseplates away from the heat affected zone (HAZ). It can be observed that the temperature is maximum around the line of the welding and gradually reduces towards the edge of the plate. The maximum temperature obtained in thesis simulations (LSDYNA) was 1830°C while that at the end of the plate was maintained at around 27°C. In comparison, value of the highest temperature derived from Nigerian simulation was 1832°C.

4.2.1.4. Stress distribution.

To investigate the longitudinal and transverse stress distribution during the welding process two lines have been proposed: along the weld line (longitudinal stress) and across the weld line (transverse stress). To indicate relevant stresses those lines were placed precisely in the middle of the model. Plots attached below present comparison between X-Ray measurements and FEM simulation from the Nigerian paper with LS-PrePost thesis simulations.

Figure 93: Comparison of the X-ray diffraction and FEM simulation of the longitudinal residual stress (Sx) along the weld line in longitudinal direction.
Figure 94: Comparison of the X-ray diffraction and FEM simulation of the transverse residual stress (Sy) along the weld line in longitudinal direction.
Figure 95: Comparison of the X-ray diffraction and FEM simulation of the *longitudinal* residual stress (Sx) across the weld line in longitudinal direction.
Figure 96: Comparison of the X-Ray measurements and FEM simulation (Nigerian paper) with LS-PrePost thesis simulation of longitudinal residual stress ($\sigma_l$) along the weld line.

Comparison of the X-ray diffraction and FEM simulation of the transverse residual stress ($S_y$) across the weld line in longitudinal direction.
Regarding to longitudinal and transverse residual stress plots $\sigma_x$ and $\sigma_y$ (figure 93 and 94) could be concluded that values of the stresses are relevant in comparing to reference [23]. Residual stress distribution for simulation case1 looks more similar to real X-ray measurements which prove correctness of that fixing instead of case2 which differs more slightly.

According to the figure 95 tendencies of thesis simulations in comparing to the Nigerian simulation and measurement are mostly similar. Extreme values of tensile stress (400 MPa) and compressive stresses (200 MPa) are also at the same level. One difference is about transition from tensile to the compressive stress and it comes between 10mm and 20mm from the weld center line.

Referring to the figure 96, transverse stresses ($\sigma_y$) obtained in thesis simulations have approximately 40 MPa lower values in first half of the baseplate than reference values of Nigerian paper. It might be caused by uncertainties of material database.

Differences between peaks of the temperature might be caused by slight differences in material databases between reference [23] and thesis simulations. It also could has influence on transverse residual stresses in x-direction and stress transition from tension to compression at the 10 mm from the beginning of the baseplate. Additionally, might cause lower values of the transverse residual stresses in y-direction.
5. Conclusions

From the obtained results of this thesis can be concluded, that computational time is the most important factor in welding analysis. Investigated models have uncomplicated construction, but should be taken into account that Volvo CE develops much more complex structures which consist hundreds of thousands of elements. During this thesis, each next model contained smaller number of elements. That was supposed to simplify construction by reduce the mesh size to limit computing time.

Moreover, another factor which has significant influence on operating time is time step that represents by step per element factor. The decreased time steps, increases the simulation time. For this reason, steps must be well matched to the needs. At the same time, small time steps and accuracy are required. In section 4.1.3.3 has been shown relation between step per element factor and residual stresses. The simulations with step per element 2 up to step per element 6 exhibiting similar level of the highest stresses. Can be concluded that it is not necessary to decrease step per element factor at the expense of accuracy. It only has an influence on a computing time.

As described in section 4.1.2.5 clamping size effects on residual stress distribution. If the clamp is larger, the residual stresses will be lower. Improved weld pool geometry causes higher temperature peak so those parameters should be selected carefully to avoid too high stresses.

Referring to the section 4.1.4.5 It might be supposed that change of a torch angle significantly change the residual stress distribution, but in that case weld seam might has unrealistic shape in the industry.

According to the butt weld results the stresses are similar in comparing to reference document [23]. That conclusion proves the correctness of the simulations. The most significant influence on some temperature and stress deviations between LS-PrePost simulations and reference document have uncertainties of the material database. Most of the values are known, but certain factors such as stress-strain curves have been assumed according to the material behavior for different temperature.
References


Faculty of Technology
351 95 Växjö, Sweden
Telephone: +46 772-28 80 00, fax +46 470-832 17