

CHALMERS



Welding Simulation of a Gear Wheel Using FEM

Master's Thesis in Applied Mechanics

ANDREAS ROBERTSSON

JERK SVEDMAN

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2013
Master's thesis 2013:50

MASTER'S THESIS IN APPLIED MECHANICS

Welding Simulation of a Gear Wheel Using FEM

ANDREAS ROBERTSSON

JERK SVEDMAN

Department of Applied Mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2013

Welding Simulation of a Gear Wheel Using FEM
ANDREAS ROBERTSSON
JERK SVEDMAN

© ANDREAS ROBERTSSON AND JERK SVEDMAN, 2013

Master's Thesis 2013:50
ISSN 1652-8557
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Chalmers Reproservice
Göteborg, Sweden 2013

Welding Simulation of a Gear Wheel Using FEM

Master's Thesis in Applied Mechanics

ANDREAS ROBERTSSON

JERK SVEDMAN

Department of Applied Mechanics

Division of Material and Computational Mechanics

Chalmers University of Technology

ABSTRACT

Welding is a complex manufacturing process and it is important to predict the effects of welding early in the product development. Today, many imperfections caused by welding are solved by doing corrective procedures, which is costly. The ability to predict a priori the properties and shape of the product after welding will therefore save cost and time and increase the possibility of getting the product right at first time. The objective of this project was to develop a simulation tool in Abaqus where a weld process with addition of filler material could be simulated and residual stresses and deformations could be determined. The tool was made to cope with different geometries, materials, weld processes and weld paths. The addition of filler material was handled by the use of so-called quiet elements where material not yet added is still present in the analysis but given very low material properties. The analysis was conducted as a three-dimensional sequentially coupled thermomechanical problem. Comparison was made with other weld simulations where real test data were available. The Goldak heat source was used and it has been shown to work well on different kinds of geometries and weld paths. The thermal results compare well with other simulations, however while the mechanical analysis looks reasonable further study is required.

Keywords: Finite element method, welding, Abaqus, residual stress, Goldak heat source

Contents

ABSTRACT	I
CONTENTS	1
PREFACE	3
NOTATIONS	5
1 INTRODUCTION	7
1.1 Background	7
1.2 Objectives	7
1.3 Limitations	7
2 THEORY	9
2.1 Weld modelling	9
2.2 Thermal analysis	10
2.3 Mechanical analysis	13
3 METHOD	15
3.1 Pre-processing	15
3.2 Thermal analysis	15
3.3 Mechanical analysis	16
3.4 Verification process	17
4 RESULTS	22
4.1 Thermal analysis	22
4.1.1 The UET Taxila plate	22
4.1.2 The IIW plate	24
4.2 Mechanical analysis	26
4.2.1 The UET Taxila plate	27
4.2.2 The IIW plate	28
4.3 The weld path	29
5 DISCUSSION	31
5.1 The thermal analysis	31
5.1.1 The UET Taxila plate	31
5.1.2 The IIW plate	31
5.2 The mechanical analysis	31
5.2.1 The UET Taxila plate	32
5.2.2 The IIW plate	32
5.3 The weld path	32

5.4	Gear wheel	33
5.5	Future work	33
6	CONCLUSIONS	34
	REFERENCES	35

Preface

This thesis concludes a five-year engineering program at Chalmers University of Technology in Mechanical Engineering and leads to a degree in Master of Science. The thesis was carried out at the office of Vicura AB in Trollhättan, Sweden, during the spring of 2013.

We want to thank Vicura AB for providing the resources to carry out this project. Special thanks to our supervisor at Vicura AB, Henrik Tersing, who has helped us a lot during this time. We also want to thank our examiner Lennart Josefson, Chalmers University of Technology, and Per Lindström, Det Norske Veritas AS, for their expertise in the area.

Göteborg June 2013

AndreasRobertsson
Jerk Svedman

|

Notations

Common abbreviations

2D	Two dimensional
3D	Three dimensional
FE	Finite element
FZ	Fusion zone
HAZ	Heat-affected zone

Parameters – Upper case roman

A	Area	$[m^2]$
E	Young's modulus	$[Pa]$
I	Electrical current	$[A]$
Q	Heat input rate	$[Js^{-1}]$
T	Temperature	$[^{\circ}C]/[K]$
U	Voltage	$[V]$

Parameters – Lower case roman

a, b, c_1, c_2	Heat source geometry parameters	$[m]$
c_p	Specific heat capacity	$[Js^{-1}^{\circ}C^{-1}]$
f_f	Fraction of heat in front of torch	$[-]$
h	Thermal convection coefficient	$[Js^{-1}m^{-2}K^{-1}]$
k	Thermal conductivity coefficient	$[Js^{-1}m^{-1}^{\circ}C^{-1}]$
l	Element side length	$[m]$
q	Spatial heat input rate	$[Js^{-1}m^{-3}]$
t	Time	$[s]$

Parameters – Greek letters

α	Thermal expansion coefficient	$[^{\circ}C^{-1}]$
ε	Radiation emissivity	$[-]$
ϵ_{th}	Thermal strain	$[-]$
ρ	Density	$[kg\ m^{-3}]$
σ	Stefan-Boltzmann constant	$[Js^{-1}m^{-2}K^{-4}]$

1 Introduction

This chapter describes the background to the problem and the motivation to perform this thesis work. It contains background, objectives and limitations of the problem. The thesis work was carried out in cooperation with Vicura AB, which is an engineering consultant company in Trollhättan within the transmission area, and Chalmers University of Technology.

1.1 Background

Welding is a complex manufacturing process and it is important to predict the effects of welding early in the product development. Such effects can be residual stresses and deformation due to heating of the component and the influence of the added filler material. Today, many of these problems are solved by doing corrective procedures, which is costly. The ability to predict a priori the properties of a product after welding will therefore save cost and time and will increase the possibility of getting the product right the first time. A large number of welding simulations carried out over the years have been performed for specific cases and generally in 2-dimensions and the need for a general simulation tool is substantial.

1.2 Objectives

The main objective of this project was to develop a simulation tool which is capable of carrying out three-dimensional welding simulations. The tool should be able to handle different geometries, materials and welding processes. The tool should consider methods for applying heat from the weld torch as well as modelling of filler material and high temperature behaviour. The user of the tool should be able to freely define the weld path on any geometry, as well as the number of weld passes. The results were then compared with other simulations in order to verify the simulation tool. With this simulation tool it should be possible to carry out welding simulations of a gear wheel.

1.3 Limitations

The project was done with ABAQUS/Standard and comparisons with other software were not conducted. The high temperature behaviour and changes in the microstructure due to rapid heating and cooling were simplified due to limitations in time. The heat source was modelled as a Goldak heat source [1], which is a Gaussian double elliptic distribution described later, and no other heat distributions were considered.

The simulation tool conducts a sequentially coupled thermomechanical analysis instead of a fully coupled analysis in order to save computational time. The tool only simulates weld processes where filler material is added. The activation of filler material was done with the so-called quiet element method. Quiet elements are elements with reduced values for the material properties which becomes active once a certain criterion is fulfilled.

The main testing and verification of the simulation tool were made on butt-welded plates, however some simulations were made on other geometries such as a gear wheel welded onto a shaft to demonstrate that the weld path could be freely defined.

2 Theory

Welding is a complex joining method to simulate with accuracy since it is a fully coupled thermo-mechanical-metallurgical problem. As stated above, the analysis made in this thesis is sequentially coupled and therefore the thermal and mechanical analysis will be described separately in this report.

2.1 Weld modelling

A sequentially coupled analysis assumes that the mechanical behaviour does not affect the thermal properties. In reality this is not the case since deformations and temperature changes can cause the microstructure of the material to change which affects the material properties. The couplings between the different analyses and the microstructural change are illustrated in Figure 2.1 and explained in Table 2.1.

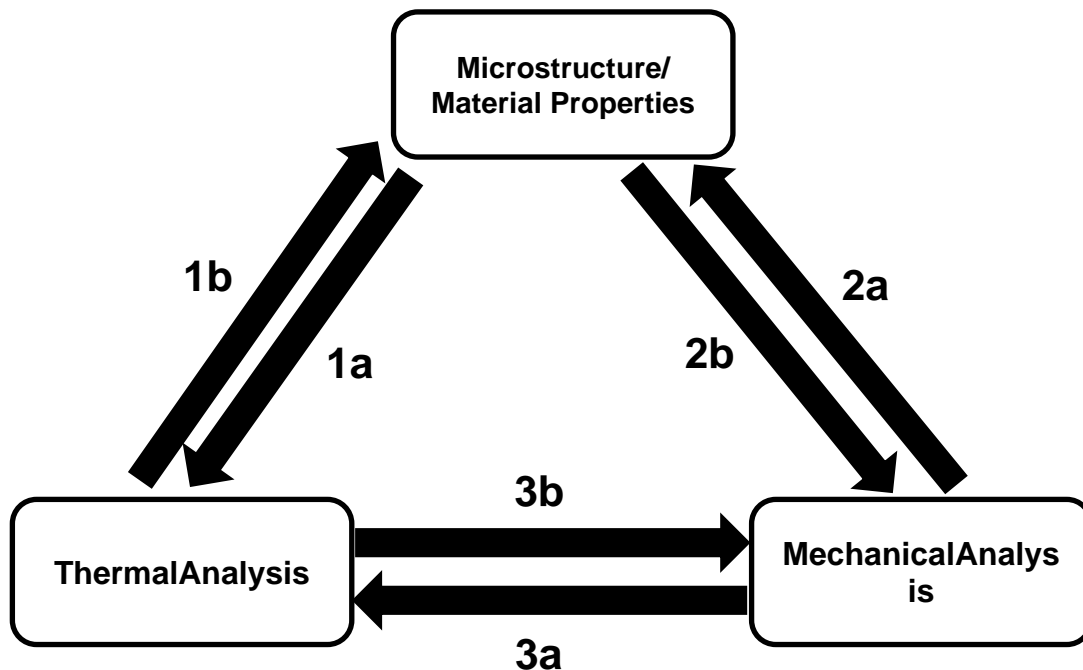


Figure 2.1 – Couplings in a welding simulation.

Table 2.1 – Thermo-mechanical-metallurgical couplings.

1a.	Latent heat due to phase changes depends on microstructure and temperature
1b.	Temperature changes the microstructure
2a.	Deformations affect the microstructure
2b.	Mechanical properties depend on microstructure and temperature
3a.	Heat is generated due to deformation
3b.	Thermal expansion drives the deformation

Changes in heat capacity, specific heat and latent heat will occur due to microstructural changes (coupling 1a). Deformations will generate small amounts of heat due to plastic dissipation (coupling 3a) and also affect the microstructure of the material (coupling 2a). However, these two couplings are considered negligible and the contribution to the thermal analysis can be ignored [2]. The microstructure and in turn the mechanical properties are dependent on the temperature and rate of temperature change (coupling 1b and 2b). If these changes can be predicted before the analysis, the coupling between the thermal and mechanical analysis is weak and a sequentially coupled analysis is motivated where the temperature history is imported in the mechanical analysis (coupling 3b) [2].

During welding, melted filler material will be added and merged together with the material being welded, denoted as parent material further on. When modelling the filler material in a FE analysis, two different methods are commonly used [3]. One method is the use of inactive elements where the elements ahead of the heat source are inactive until the heat source has passed. This has been shown to cause problems since the inactive elements do not follow the displacements of the neighbouring elements which can result in distorted elements upon activation [3]. The other method is to use so-called quiet elements where the material ahead of the weld torch is isolated until a certain condition is fulfilled. The isolation in the thermal analysis is done by reducing the conductivity and in the mechanical analysis it is by reducing Young's modulus, the yield strength and the thermal expansion coefficient. Reducing these properties too much has been shown to cause numerical problems and a reduction of two orders of magnitude has been shown to be sufficient. The activation should also be made smoothly since abrupt changes in material properties can also cause numerical issues [4].

Computational welding simulations are very resource demanding and in order to keep the simulation times down it is important to have a sufficient simulation platform [5]. The first things to consider are the purpose of the analysis, the accuracy required and the resources available. From that the complexity of the model can be decided [4]. Simplifications can then be made based on that decision. One simplification that is commonly made is to simulate the process in 2D. This can be motivated if the conductivity is considerably lower in the weld direction compared to the transverse direction [4]. If this is not sufficiently accurate, a 3D model can be used with substructuring. In that case, parts of the model are assumed to behave linearly and are then replaced with boundary conditions [2]. The most computationally expensive and also the most accurate analysis is a full 3D model. Adaptive meshing can then be used to save time by concentrating the elements in regions where large gradients are expected and allowing the mesh to become coarser further away from the concentrated region. If the geometry is symmetric about a plane, only one side of the symmetry plane needs to be modelled along with corresponding boundary conditions.

2.2 Thermal analysis

The first step in calculating residual stresses in a sequentially coupled welding simulation is to perform a thermal analysis where the temperature field over time is calculated. The heat source can either be modelled as a static heat source with prescribed temperature, a static heat source with volumetric flux or as a moving heat source with volumetric flux. The latter is used in this project and it is the most complex type of analysis and also the most realistic since it can predict stresses in

longitudinal direction [4]. When performing a full 3D analysis with a moving heat source the heat input from the weld torch needs to be modelled as a distributed heat flux on both the filler and parent material. The heat input rate, Q , that is generated from the weld can be calculated as follows:

$$Q = UI\eta \quad (2.1)$$

where U is the voltage, I is the current and η is the weld efficiency. In this project, a double ellipsoid heat distribution was used. This model was suggested by Goldak et al. [1] in 1983 and is commonly accepted by the community. The Goldak heat source model consists of two Gaussian distributions, one in front of the weld torch and one behind it. Figure 2.2 is a visual general representation of the heat input rate distribution in the xz -plane ($y = 0$) in the local coordinate system. Equation (2.2) shows the mathematical representation of the Goldak heat source model for a moving frame of reference.

$$\begin{aligned} q_f(x, y, z) &= f_f \frac{6\sqrt{3}Q}{abc_1\pi\sqrt{\pi}} e^{-3\left(\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{z^2}{c_1^2}\right)} \\ q_r(x, y, z) &= (2 - f_f) \frac{6\sqrt{3}Q}{abc_2\pi\sqrt{\pi}} e^{-3\left(\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{z^2}{c_2^2}\right)} \\ q(x, y, z) &= \begin{cases} q_f & \forall z \in [0, \infty) \\ q_r & \forall z \in (-\infty, 0) \end{cases} \end{aligned} \quad (2.2)$$

It is important to know that $q(x, y, z)$ is the distribution of the heat input rate $2Q$, but since the heat source normally is applied on or relatively near the surface, only one Q is distributed. Table 2.2 shows the terms of equation (2.2).

Table 2.2 – The parameters in equation (2.2).

Term	Meaning	Dimension
$q(x, y, z)$	Spatial heat input rate	$ET^{-1}L^{-3}$
Q	Heat input rate	ET^{-1}
a, b, c_1, c_2	Geometry parameters	L
f_f	Fraction of heat in front of torch	—
x	Transverse direction of weld	L
y	Depth direction of weld	L
z	Welding direction	L

The geometry parameters can be hard to predict since they are dependent on the geometry of the product, of the welding parameters and material properties. It can be shown that a, b, c_1 and c_2 are the diameters of the distribution when $q(x, y, z) = 0.05q(0,0,0)$. f_f is the fraction of the heat for the front distribution. If f_f has a value of 2, all heat is distributed to the front ellipsoid. It is also worth noting that if

$$c_2 = \frac{c_1(2 - f_f)}{f_f}$$

the distribution is continuous at $z = 0$.

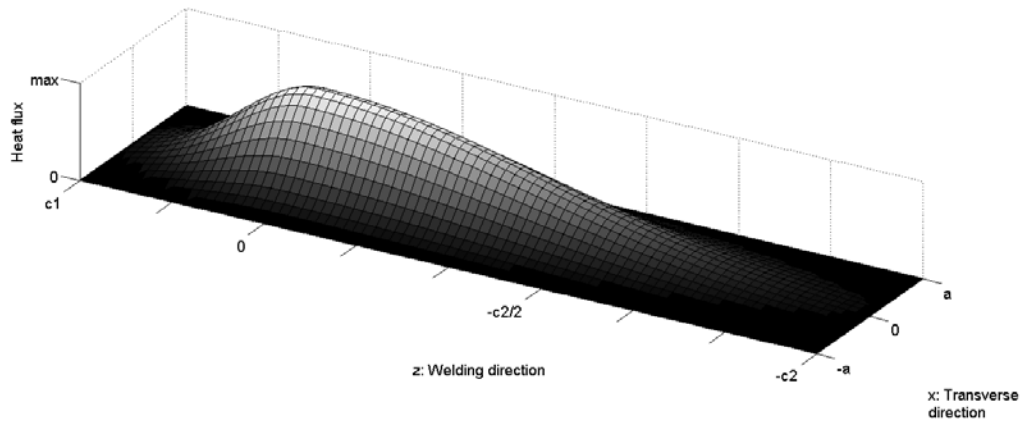


Figure 2.2 – Illustration of the heat source in the xz-plane.

The thermal analysis in a weld simulation consist of boundary conditions, initial conditions, interpass temperature, radiative heat transfer and convective heat losses. The initial conditions set the initial temperature of the material as well as the ambient temperature. If boundaries are set at the interpass temperature, it specifies the temperature allowed on the material before another weld bead is applied. The radiative heat transfer and convective heat losses are applied on all the surfaces exposed to the ambient surroundings. During the deposition of the weld bead, the radiative heat transfer may be ignored since the heat losses are already accounted for in the weld efficiency [4]. The convective heat transfer near the source can also be suppressed since the air around the source is heated from the torch and is therefore higher than the ambient temperature. The convective heat losses and radiative heat transfer may be combined into a single convective condition dependent on temperature according to the following equations [6]:

$$\begin{aligned} q_{loss} &= h_{total} \cdot A(T - T_{amb}) \\ h_{total} &= h + \varepsilon\sigma(T + T_{amb})(T^2 + T_{amb}^2) \end{aligned} \quad (2.3)$$

where q_{loss} is the heat loss, A is the surface area, h is the thermal convection coefficient, ε is the radiation emissivity, σ the Stefan-Boltzmann constant, T is the current temperature and T_{amb} is the ambient temperature.

The material properties needed in the thermal analysis are the density, thermal conductivity, specific heat capacity, latent heat capacity, and solidus and liquidus

temperatures. The density is usually kept constant while the conductivity and specific heat is temperature dependent. In order to account for the phase transformations, latent heat may be prescribed between the solidus and liquidus temperature. Latent heat due to solid state phase transformations was ignored. To compensate for convective stirring effects in the fusion zone, the conductivity is usually increased above the melting temperature [7]. If material properties are unavailable at high temperatures, extrapolations can be made up to the melting temperature and kept constant since the residual stress calculations are less sensitive to changes in the thermal properties compared to the structural properties [4, 8]. If extrapolations have been made, the heat flux may need to be adjusted to obtain realistic fusion and heat-affected zones [8].

In order to achieve a stable solution, some guidelines on the time increments and element lengths may be considered. To avoid errors due to oscillations, the time increments used (Δt) are defined as:

$$\Delta t > \frac{\rho c_p}{6k} \Delta l^2 \quad (2.4)$$

where ρ is the density, c_p is the heat capacity, k the thermal conductivity and Δl is the element length [4]. If the element length in the weld direction is too large, the heat flux may be discontinuous between elements and causing an uneven temperature distribution. A guideline on the sufficient element length in the welding direction is at most 75% of the diameter of the heat source in the same direction [4].

The area around the weld can be categorized into different zones. The fusion zone (FZ) is closest to the weld and is where all material has peak temperatures above melting temperature and consists of both filler and parent material. The heat-affected zone (HAZ) consists of parent material only and does not melt during the analysis but still reach temperatures high enough to change the microstructure and material properties. Material outside the HAZ does not change in microstructure but is still subjected to temperature changes in the area closest to the weld torch. The size of the different zones depends on the parameters in the Goldak heat source as well as the welding parameters and material properties. The heat source parameters should be modified so that all of the filler material as well as some of the parent material reaches melting temperature during the thermal analysis in order to get a realistic FZ.

2.3 Mechanical analysis

When calculating the residual stresses and deformations in the mechanical analysis, the temperature field from the thermal analysis is imported as input. When an unrestrained material is heated or cooled it will expand or contract according to the following equation:

$$\epsilon_{th} = \alpha(T - T_{ref}) \quad (2.5)$$

where ϵ_{th} is the thermal strain, α is the coefficient of thermal expansion T is the current temperature and T_{ref} is a reference temperature. The material close to the weld will experience rapid heating and will therefore try to expand. Material that is not heated at the same rate will prevent this expansion and therefore cause compressive stresses to occur. If these stresses reach above the yield limit, the material will start to deform plastically. When the weld process is complete, the material will be cooled to the ambient temperature which causes the previously

expanded material to contract again. If plastic deformations have occurred, the material will be smaller than its initial state resulting in residual tensile stresses in the contracted material and residual compressive stresses in the surrounding material. The filler material will be added in melted form and should therefore be strain and stress free when added and with a reference temperature on the thermal expansion equal to the melting temperature [4].

The mechanical boundary conditions should represent how the product is fixed during the welding process and prevent rigid body motion. The influence on different support types and positions can be investigated by varying the boundary conditions. If symmetry conditions are used, the symmetry plane also needs to be mechanically constrained accordingly.

The temperature dependent material properties needed for the mechanical analysis were the density, Young's modulus, Poisson's ratio, yield strength, hardening modulus, anneal temperature and thermal expansion coefficient. The residual stresses are mostly influenced by the tensile material properties, i.e. the yield strength, hardening modulus, anneal temperature and the hardening model used. In order to get reliable results these properties should be thoroughly considered and validated [4]. The yield strength at room temperature affects the residual stresses the most and reliable properties up to $0.7T_m$ are desirable for accurate results.

There are different kinds of strain hardening models to consider and depending on available data, accuracy required and type of welding procedure, a choice of hardening model can be made. The models to be considered are isotropic hardening, kinematic hardening and mixed isotropic-kinematic hardening and each have variations of how they can be implemented. When the material reaches high temperatures and changes phase, the plastic strains and work hardening accumulated at lower temperatures are usually eliminated. This is called annealing and the way it is implemented differs depending on what strain hardening model is used. The kinematic hardening history is removed and the back stress is reset to zero when the material becomes elastic-perfectly plastic. To remove the isotropic hardening history, a two-stage annealing function can be implemented where the material ceases to harden above a lower annealing temperature but still retains its hardening history, and above an higher annealing temperature the hardening history is removed [9]. This is to account for the effect of high temperature softening behaviour.

When a material is exposed to high temperatures it will experience phase changes. During a weld process the rate of cooling in the HAZ and FZ is very high, resulting in the solid phase change occurring at low temperatures which can result in creation of martensitic microstructures [10]. These changes will in turn introduce a change of volume. Depending on the cooling rate, material composition and peak temperature reached, the magnitude of the volume change can be determined from dilatation-temperature diagrams [11].

3 Method

The methods used to pre-process, perform the two different analyses and verify the results are presented in this chapter. The software packages mainly used in this project were ABAQUS/Standard 6.11-1 by Dassault Systèmes Simulia Corp. and ANSA 14.0.0 by BETA CAE Systems S.A. Abaqus user subroutines were created in the programming language Fortran since Abaqus/Standard did not support, for example, the Goldak heat source.

3.1 Pre-processing

The meshes for the models were created using the pre-processor ANSA 14.0. It was important to use relatively small elements in the vicinity of the heat source to accurately predict the temperature and stress distribution. It was also important to coarsen the mesh further away from the heat source to save computational time. For convenience the same mesh was used for both the thermal and mechanical analyses. Element type DC3D8 was used for thermal analysis and C3D8 was used for mechanical analysis. DC3D8 can be used for heat transfer analysis and has only one temperature degree of freedom at each node, C3D8 is purposed for stress/displacement analysis with 3 translational degrees of freedom at each node. Both element types are 8-node linear brick elements with 8 integration points.

Sets were also created in ANSA defining which elements that belonged to the weld geometry, which surfaces the boundary layer conditions would be active on, which nodes the initial conditions were active on and which nodes that belonged to the symmetry plane if symmetry conditions were used. These sets were later used in the Abaqus input file to apply corresponding loads and boundary conditions. The Abaqus input file was first created in ANSA which then could be altered manually to suit the purpose without large difficulties.

A pre-program was written in Fortran to create the input files needed for the Abaqus subroutines. The program calculated the local coordinate system for the weld nodes to be used later in the DFLUX subroutine. The program also calculated which elements that should be activated and at which time depending on the position of the weld. If a multi-pass weld was simulated, the program also computed which surfaces that the boundary layer condition should be active on and at which time since another weld bead applied on a free surface would remove the need for a boundary layer condition. The required input for the pre-program were the weld nodes, surfaces related to the weld nodes, node definitions, boundary layer surfaces, weld element definitions and weld parameters.

3.2 Thermal analysis

In the thermal analysis an Abaqus input file was modified from the one generated from ANSA to fit the purpose. Several features had to be implemented, such as the user subroutines DFLUX, USDFLD, UEXTERNALDB and temperature and field variable dependent material properties. Initial conditions were set on the entire geometry along with boundary conditions, if needed. The analysis was performed in several different steps depending on number of weld beads. In the case where several beads are simulated, only the beads that have been already applied or are being

applied are present in the model while the others are removed for the time being. In the step where welding is performed, the heat flux is applied by specifying a non-uniform distributed body flux on an element set, resulting in Abaqus accessing the user subroutine DFLUX which specifies where and how much spatial heat flux is applied. In the weld step there are also boundary layer conditions acting on all free surfaces. Between each weld bead there is also a step for cooling to a given interpass temperature and one for activating the next bead being simulated. In both the weld step and the cooling step a heat transfer analysis is performed where step time and minimum time increment is specified. When introducing a bead to the model it have the temperature as given in the initial condition given from start and it will not receive any heat from the underlying bead or surrounding material since it is still thermally isolated until the heat source have passed. The material parameters are specified depending on temperature and field variables. The activation field variable varies smoothly as the heat source approaches, activating the conductivity gradually. Output variables sought for in the analysis are also specified in the input file.

The Abaqus routine DFLUX was modified so that the heat input rate was distributed according to Goldak et al.[1] by calculating the relative position for each integration point at any given time. To achieve this, the transformation matrix between the global coordinate system and the local, welding, coordinate system had to be calculated for each time increment and integration point. The local coordinate systems for each node along the weld path were generated by the pre-program. To calculate the weld direction was straight forward since all weld node positions were known. In order to calculate the normal directions the program uses a mean value of the normal direction of the connecting surface elements. Linear interpolation was used to find the position between chosen weld nodes and proximal interpolation was used to find the local coordinate system of the heat source. With the algorithm described above, any arbitrary weld path could be chosen by the user.

Another subroutine used was UEXTERNALDB which had the function of reading input files for each weld step. Since the boundary layer condition and some thermal properties are a function of the weld position history, the subroutine USDFLD had to be created where field variables were defined depending on weld position.

3.3 Mechanical analysis

The mechanical analysis follows a similar procedure as the thermal. As mentioned before, the same mesh is used but with different element types. The same initial temperature is used. Mechanical boundary conditions were added to prevent rigid body motion and to represent the actual fixation. The same steps are also used for removing and introducing weld beads as in the thermal analysis. When a bead is introduced in the mechanical analysis it is strain free even though it may have been distorted from its original shape. The weld and cooling steps are different in the way that a temperature field is imported from the thermal analysis result file and imported at corresponding time in the static analysis causing thermal strains to occur. Since a mechanical analysis is more complex, a smaller limit on the minimum allowed time increment had to be set in order to get converging results. The mechanical properties are also dependent on temperature and on the field variables in the same way as in the thermal analysis with a smooth activation as the weld approaches. Since the same activation condition is used and no heat flux is applied, only UEXTERNALDB and USDFLD need to be used and work in the same way as in the thermal analysis.

Due to extensive computational times for the mechanical analysis of the IIW plate, a 2D analysis was performed in order to acquire results within the time frame of the project. In the 2D analysis, the element type used was CPE4, which is a 4-node bilinear plane strain element with 3 translational degrees of freedom in each node. The temperatures were imported from a cross section in the 3D thermal analysis in the middle of the plate.

3.4 Verification process

In the beginning of the thesis work it was important to run simulations with relative short CPU times to verify that the user subroutines were working properly before continuing to models with finer meshes. A paper [6] was found to meet these requirements including a complete mesh specification. The paper also specified the values of the Goldak geometrical parameters and weld process parameters, which further made the results comparable. The paper did only present thermal results from their analysis and left the mechanical analysis for future investigations. It is worth noting that [6] used ANSYS 10.0. In this report, this model will be referred to as UET Taxila plate and can be seen in Figure 3.1. The UET Taxila plate had two tack welds at each end of the weld geometry and consisted of 12464 nodes and 9000 elements. The mesh at the region near the weld at a cross section in the middle of the plate can be seen in Figure 3.2. It was fixed in all translational directions near the beginning of the weld and in the thickness direction near the end. Since it is a half model, a symmetry condition was also introduced.

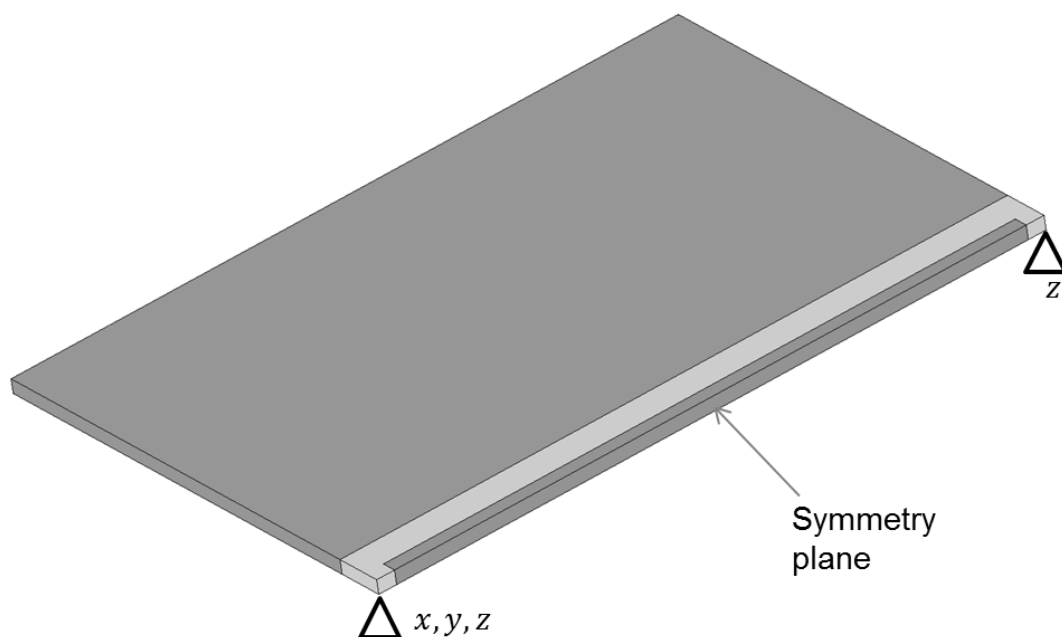


Figure 3.1 – Half model of the 150 · 200 · 3mm UET Taxila plate.

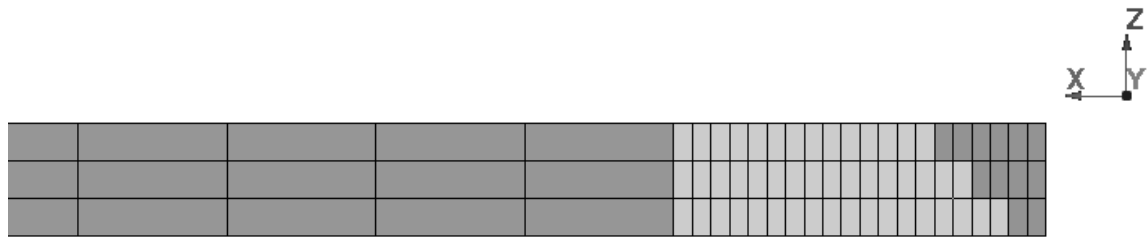


Figure 3.2 – The mesh at the region near the plate for the UET Taxila plate.

The main verification in this project was made against the thermal and mechanical results from the Swedish and Norwegian delegation in the International Institute of Welding's (IIW) RSDP Round Robin, Phase 2 [12]. Results from other participants along with material data is found in [13]. In this report, this model will be referred to as the IIW plate. As mentioned before, no 3D mechanical analysis was performed on the IIW plate due to extensive computational time. The IIW plate had a two pass TIG butt-weld and is shown in Figure 3.3 and the geometry of the weld grooves in Figure 3.4. It consisted of 107856 nodes and 94135 elements. The mesh from a cross section in the middle is shown in Figure 3.5. The boundary conditions used in the 2D mechanical analysis were a symmetry condition and the mesh is also seen in Figure 3.5.

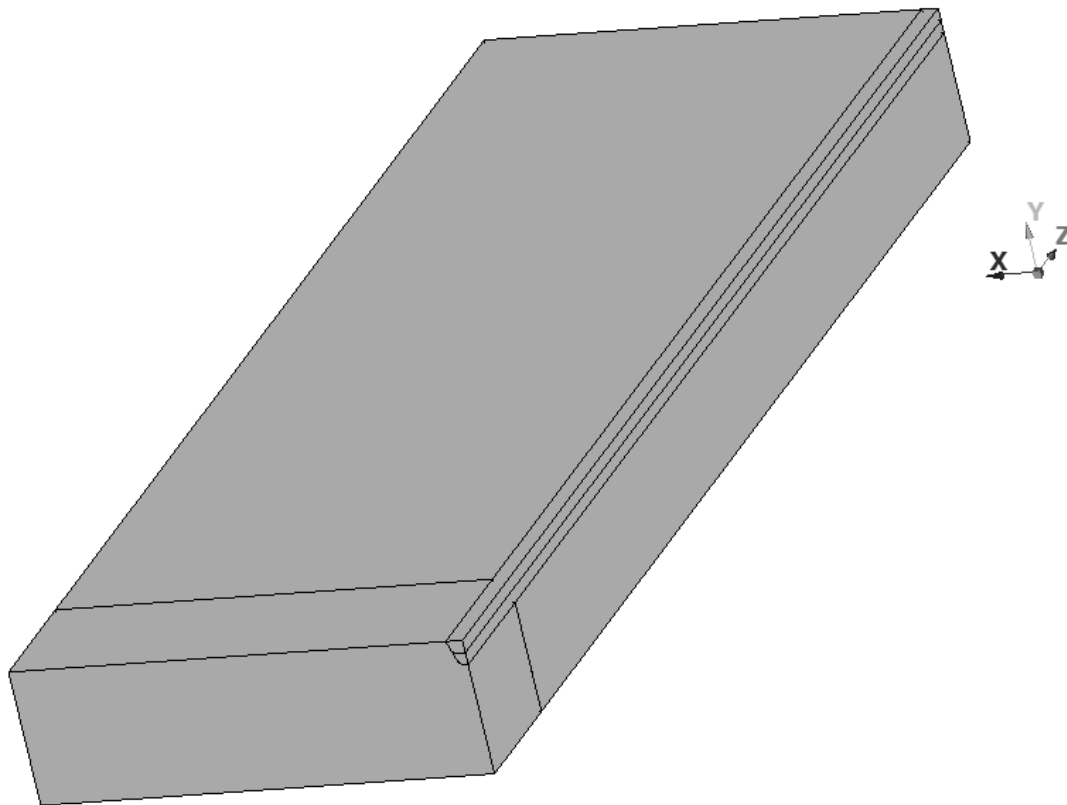


Figure 3.3 – Half model of the 270·200·30mm IIW plate with a 30mm long run-on plate.

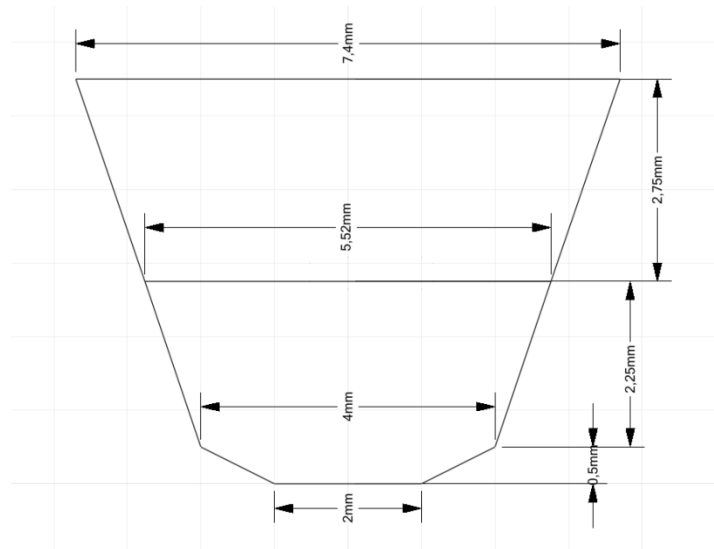


Figure 3.4 – The dimensions of a cross section of the two weld grooves for the IIW plate.

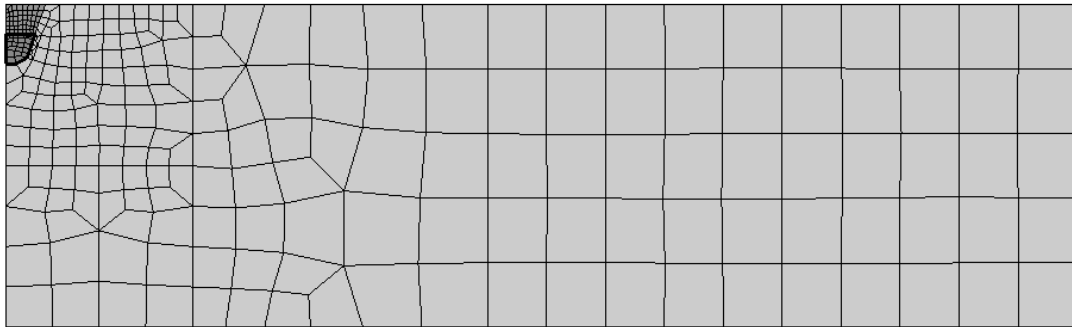


Figure 3.5 – The mesh from a cross section of the IIW plate.

One important part of the project was to design the simulation tool in such a way that the weld path could be used on models with complex geometries. This was verified by conducting a thermal analysis on two different 3D models: The Console and the Gear wheel.

The Console was a model to test that the simulation tool could handle directional changes properly. The first change of direction consisted of two 45° bends and the second change was a 90° bend as seen in figure 3.6.

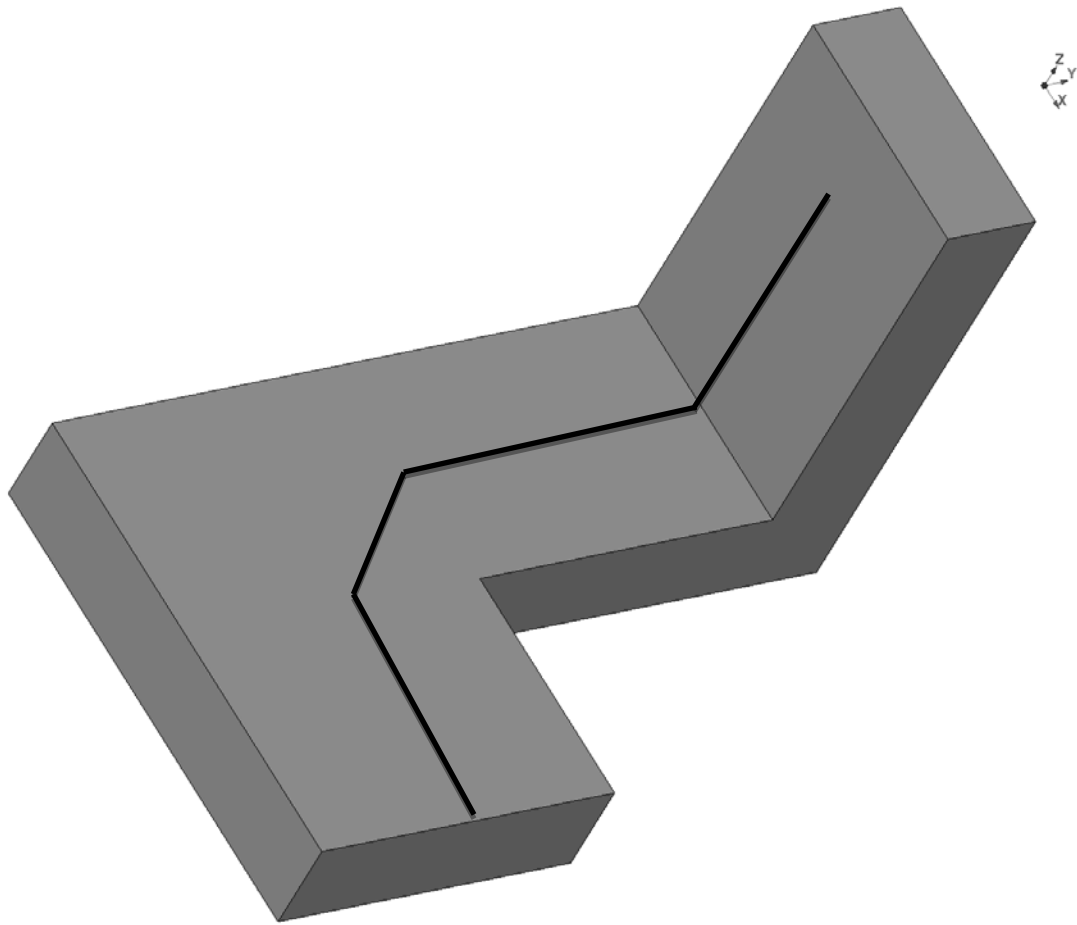


Figure 3.6. Illustration of the Console.

The gear wheel model was a simplified model of a gear wheel welded onto a shaft. The gear wheel was simplified as a disc with larger radius than the shaft. The weld path followed the perimeter of the shaft where the shaft met the gear wheel as seen in figure 3.7.

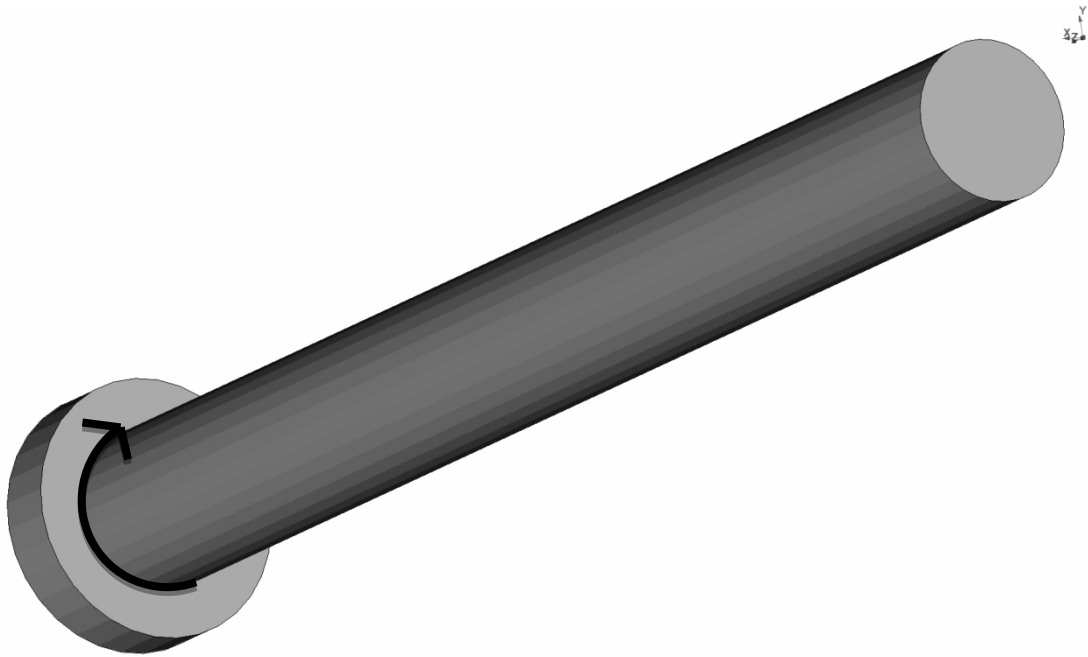


Figure 3.7. Illustration of the simplified gear wheel on a shaft.

4 Results

In this chapter the results from the performed analyses are presented together with the specification of the weld process.

4.1 Thermal analysis

The results from the thermal analysis shows temperatures at different times and positions where thermocouples were placed in the real tests performed in [6, 13]. The weld process parameters and the heat source parameters are also presented for each analysis.

4.1.1 The UET Taxila plate

The thermal results from the analysis of the UET Taxila plate are presented and compared in this section. The results were obtained using the same weld process parameters and heat source parameters as [6]. The parameters used are presented in table 4.1. The latent heat due to fusion was 2.1 GJm^{-3} between the solidus and liquidus temperatures, the rest of the material properties used in the simulation can be found in [6].

Table 4.1 – Parameters used in the UET Taxila plate analysis.

Heat source parameters					Welding process parameters			
a	b	c_1	c_2	f_f	U	I	η	Welding speed
10mm	2.5mm	8mm	20mm	0.6	175V	12.5A	90%	3mm/s

Figures 4.1 and 4.2 show the temperatures over time for five nodes along the transversal line located on the upper surface, 10mm from the weld start in the welding direction, for both the analysis carried out by [6] and by the authors of this report.

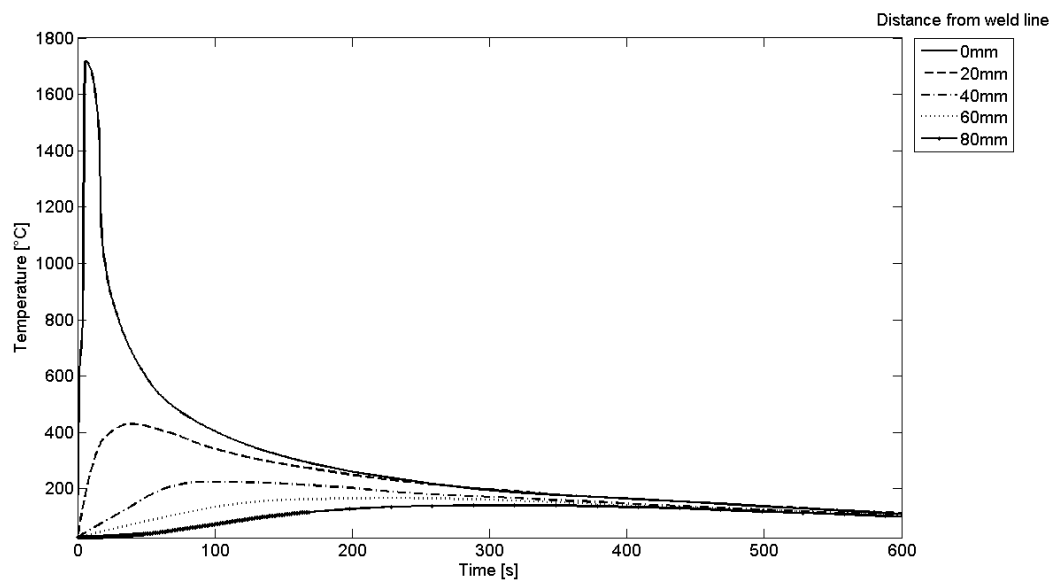


Figure 4.1 – Thermal results on upper surface transverse to weld line as presented by [6].

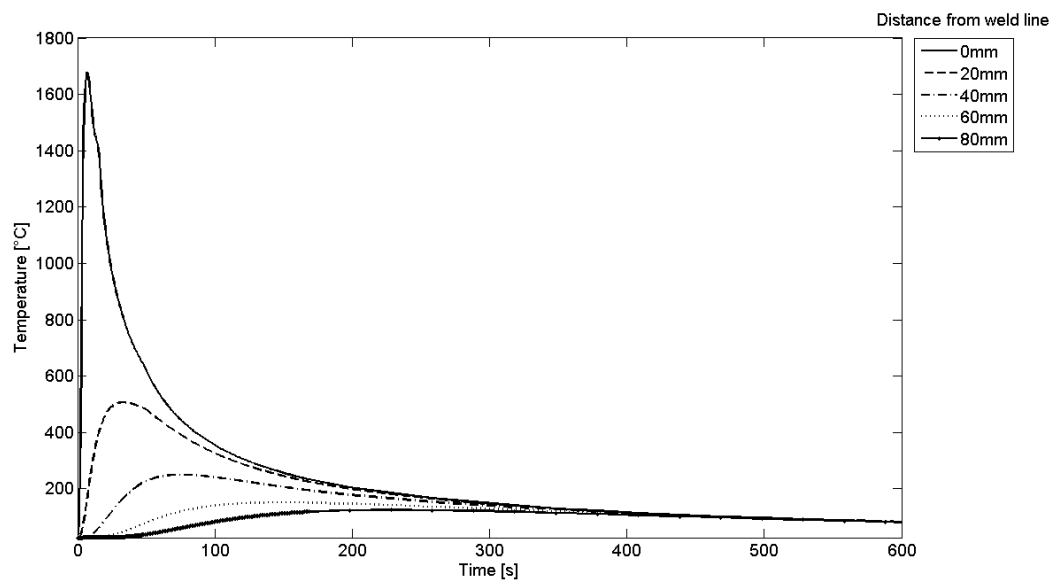


Figure 4.2 – Thermal results on upper surface transverse to weld line for the UET Taxila plate.

4.1.2 The IIW plate

The heat source parameters and welding process parameters used for each weld deposited are shown in Table 4.2. Since the heat source parameters and the boundary layer conditions were not specified, the weld efficiency has been modified compared to [12] in order to get realistic FZ. The material properties used can be found in [13] and athermal convection coefficient of $8 \text{ Js}^{-1}\text{m}^{-2}\text{K}^{-1}$ and a radiation emissivity of 0.1 were used.

Table 4.2 – Parameters used in the IIW plate analysis.

Weld	Heat source parameters					Welding process parameters			
	a	b	c_1	c_2	f_f	U	I	η	Welding speed
First deposit	2.357mm	2.357mm	1mm	3mm	0.5	9V	155A	72%	0.683mm/s
Second deposit	3.3657mm	3.3657mm	1mm	3mm	0.5	9V	155A	75%	0.683mm/s

The positions of the thermocouples where the temperatures were read can be seen in Figure 4.3. The thermocouples T1-T3 and T5 were located 90mm from end of the plate in the weld direction. T1-T3 were located on the upper surface at distances of 3mm, 5mm and 7mm, respectively, from the edge of the groove. T5 was located on the bottom surface directly under the groove. Thermocouple T4 was not used. Figures 4.4-4.8 show the temperature results from the thermal analysis and are shown in comparisons with the highest and lowest temperature curves from the participants, denoted P1-P7. The results from the physical test are also shown in the same figures.

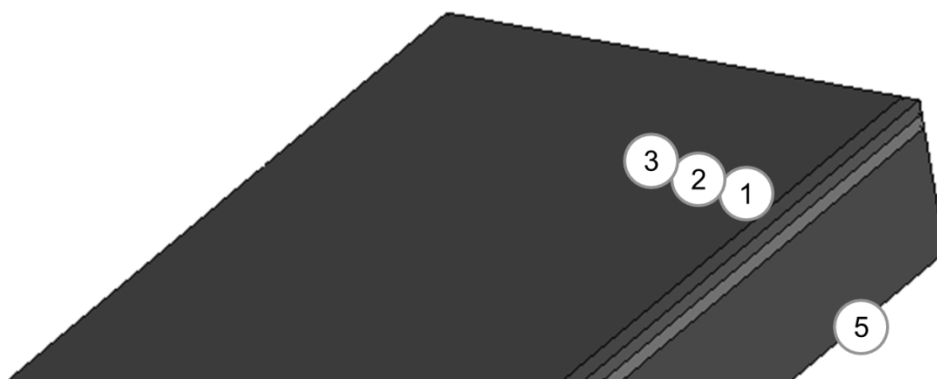


Figure 4.3 – Positions of the thermocouples.

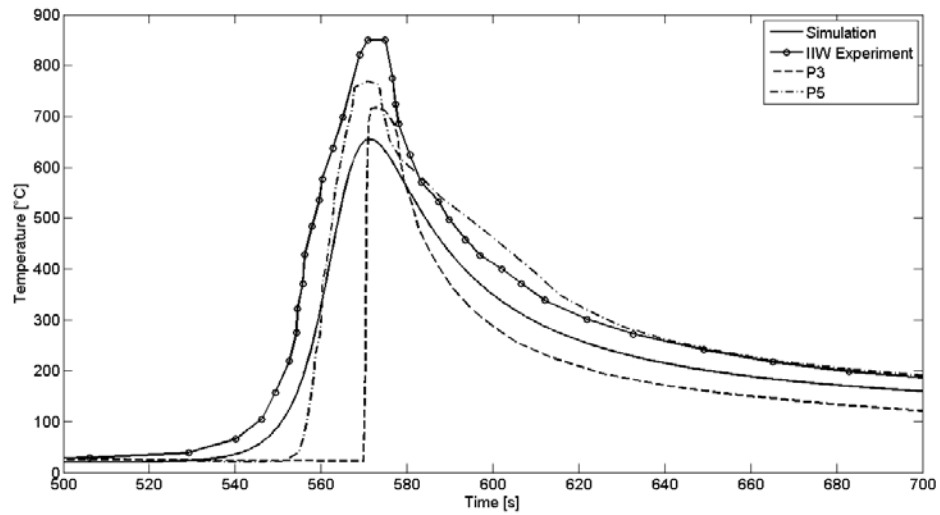


Figure 4.4 – Thermal results for T1, first weld pass.

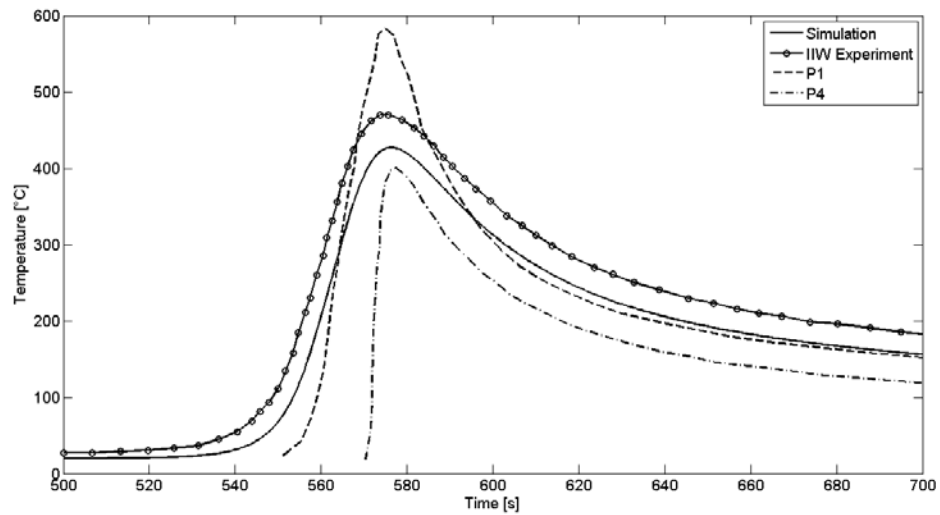


Figure 4.5 – Thermal results for T3, first weld pass.

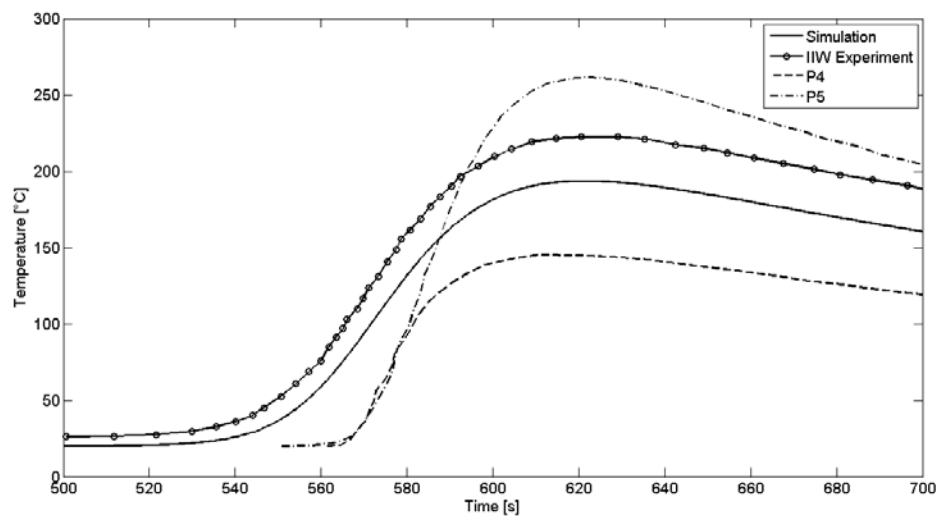


Figure 4.6 – Thermal results for T5, first weld pass.

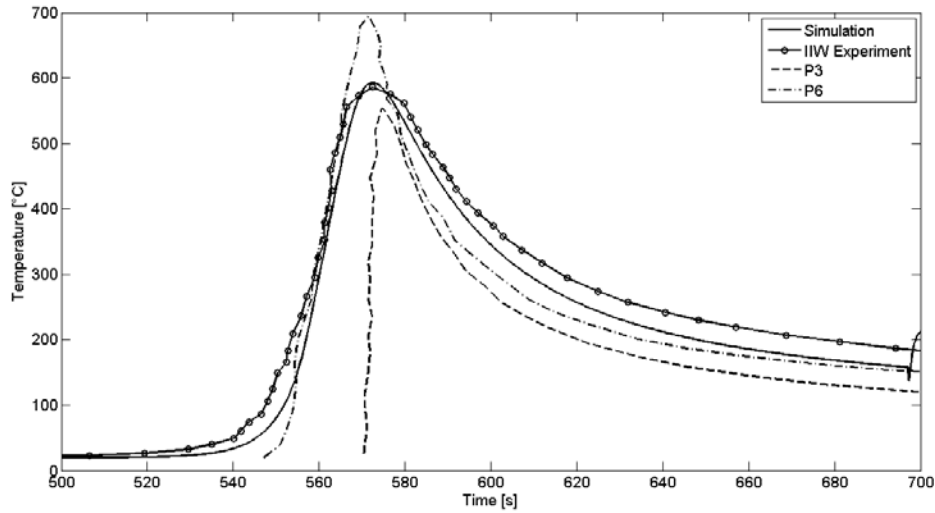


Figure 4.7 – Thermal results for T2, second weld pass.

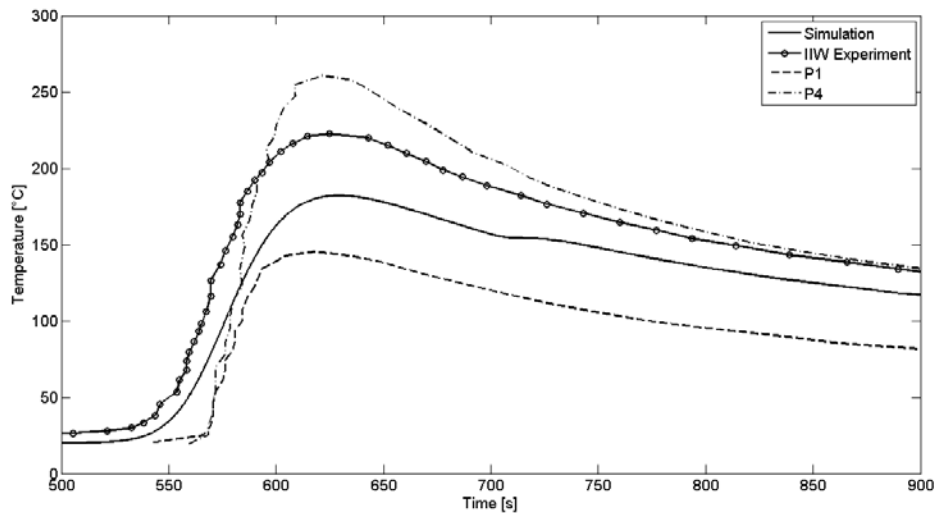


Figure 4.8 – Thermal results for T5, second weld pass.

4.2 Mechanical analysis

The results from the mechanical analysis show residual stresses in the principal directions after the plate has been cooled down to room temperature. The displacements at different locations are also shown for the UET Taxila plate.

The mechanical material properties for both the UET Taxila plate and the IIW plate can be found in [12, 13]. The material used for the weld was Avesta 316L and for the parent material 316L-N-SPH was used. The thermal expansion coefficient was reduced to zero above melting temperature and for the inactive material.

4.2.1 The UET Taxila plate

The mechanical results from the UET Taxila plate can be seen in figures 4.9 and 4.10 at the transversal line in the longitudinal middle of the plate, 1mm under the top surface.

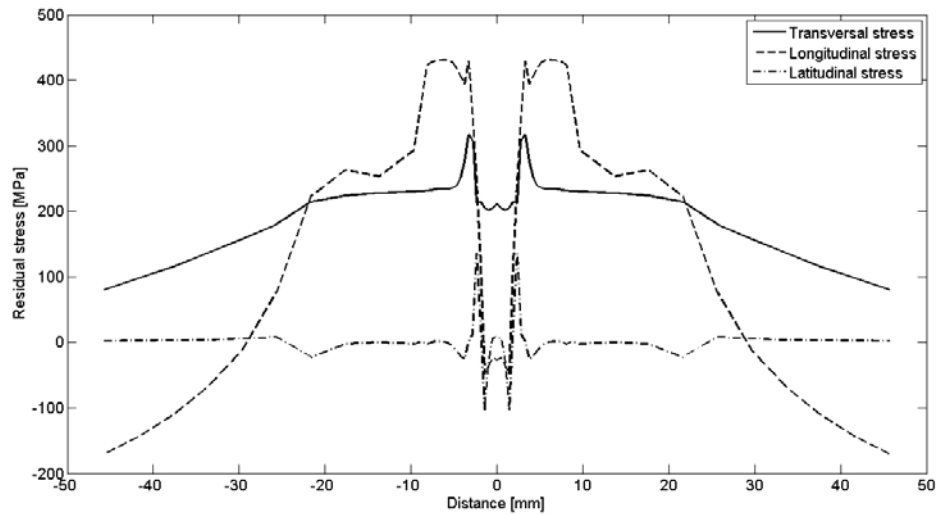


Figure 4.9 – Residual stresses from the mechanical analysis of the UET Taxila plate.

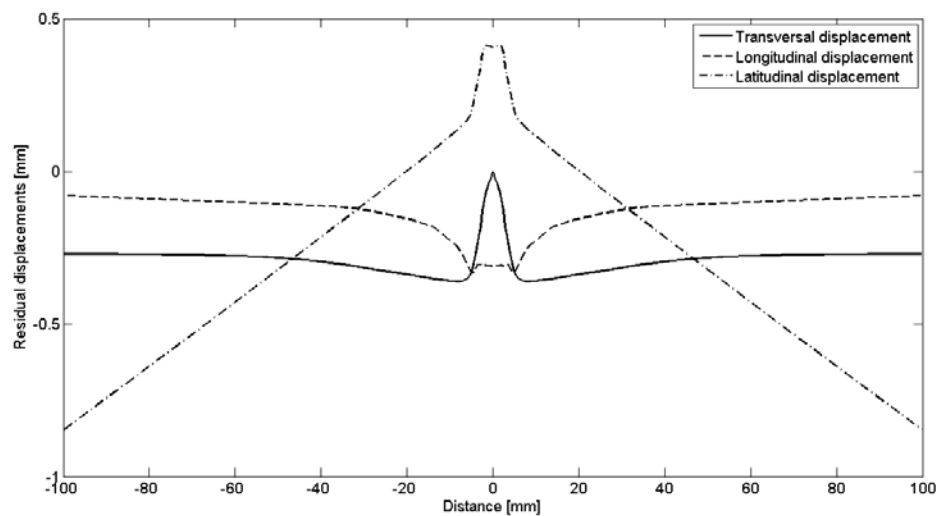


Figure 4.10 – Residual displacements from the mechanical analysis of the UET Taxila plate.

4.2.2 The IIW plate

The 2D mechanical results from the IIW plate are shown in Figure 4.11 and the results from [12] are shown in Figure 4.12. The figures show residual stresses versus distance from the weld centre line 3mm below the top surface after both weld beads have been applied.

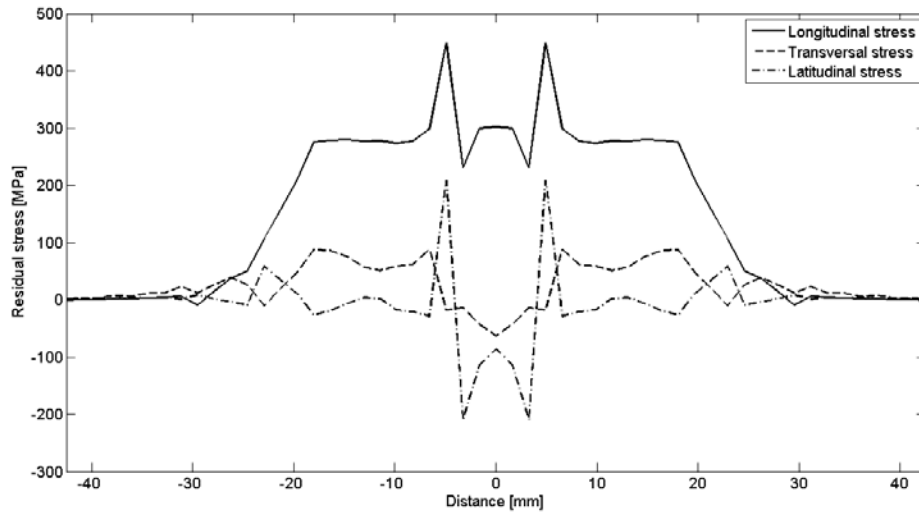


Figure 4.11 – Residual stresses from the mechanical analysis of the IIW plate.

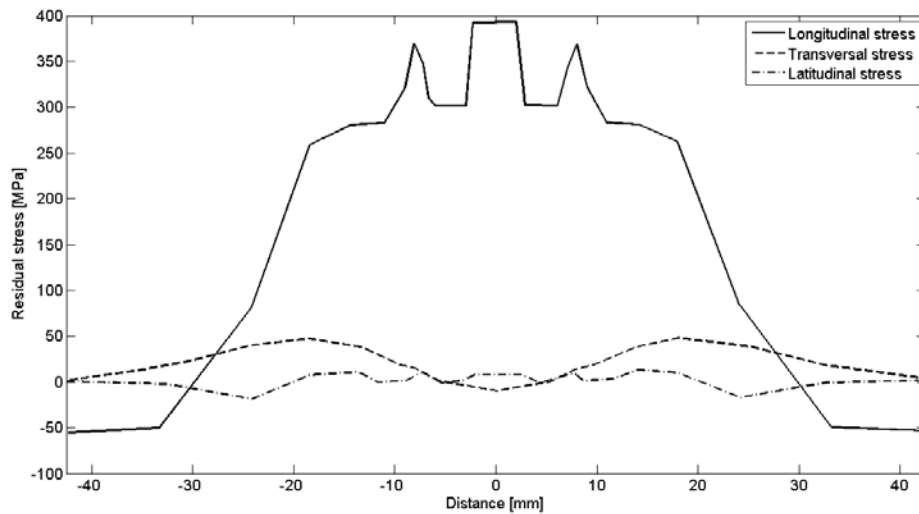


Figure 4.12 – Residual stresses from the 2D analysis with same materials in [12].

4.3 The weld path

Figure 4.13 shows the temperature field in the console analysis at a time that demonstrates the weld path curvature. The conductivity for this analysis was very low and no film condition was set so that the weld path history clearly could be seen. Figure 4.14 focuses on the second directional change, note that the temperature scale is changed due to that higher temperatures are present at this point.

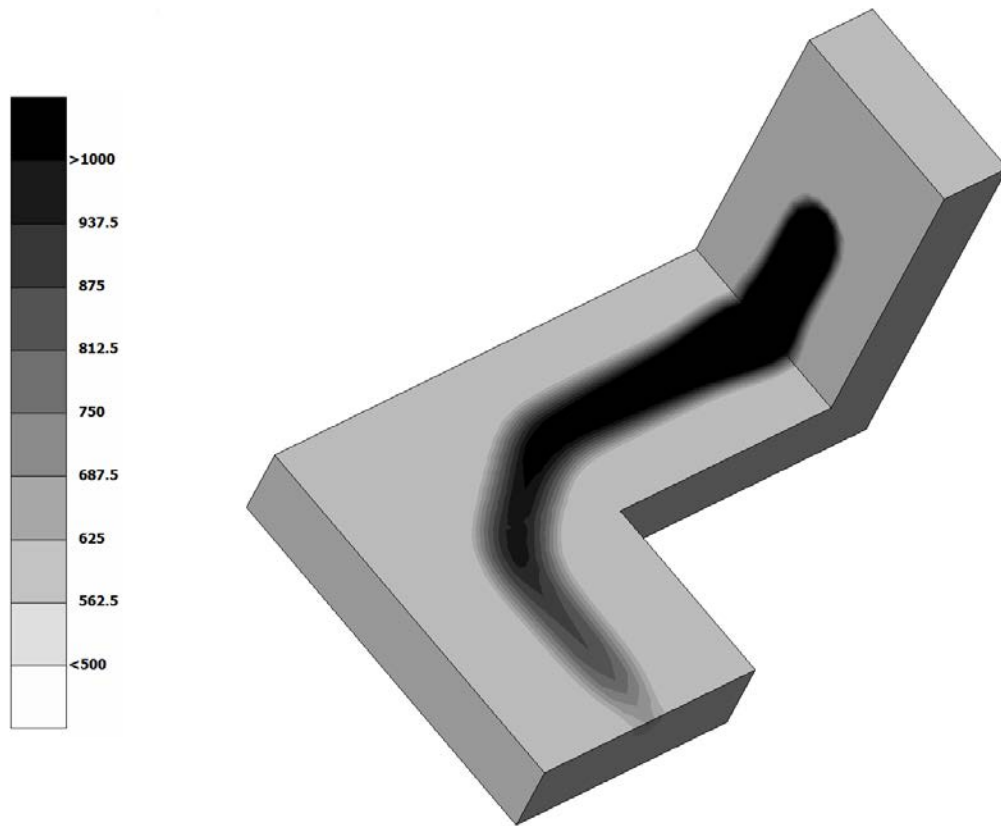


Figure 4.13 – Thermal results of the Console analysis.

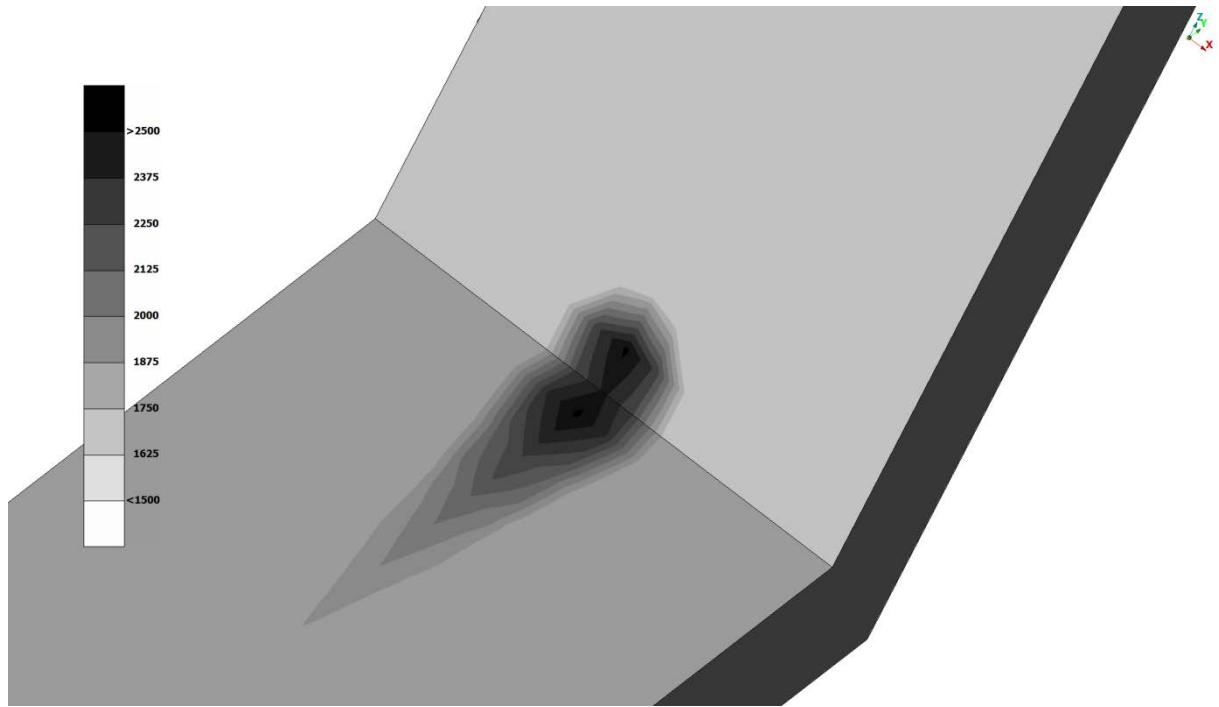


Figure 4.14 – Thermal results of the Console analysis, focused on the second directional change.

Figure 4.15 shows the results from simulation of the Gear wheel. The temperature for four different nodes located equally spaced around the weld path during a full revolution weld is presented. The temperature scale ranges from T_{min} to T_{max} rather than displaying the actual temperature results due to that the purpose of the analysis was to demonstrate the weld path abilities of the simulation tool.

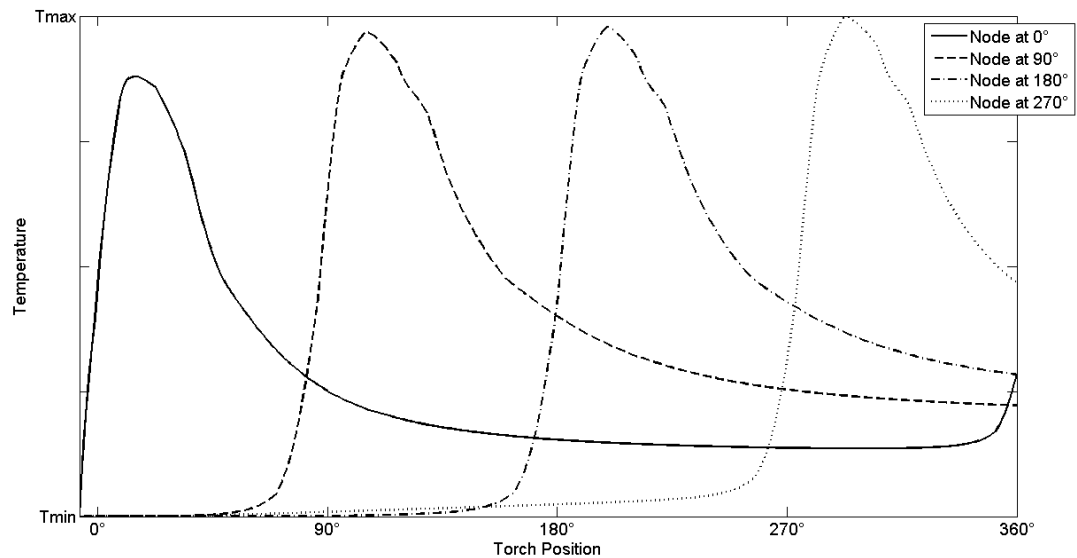


Figure 4.15 – Temperature result at different nodes along the weld line for the gear wheel analysis.

5 Discussion

This chapter mainly discusses the three basis parts of the simulation tool; the thermal analysis, the mechanical analysis and the weld path. Suggestions for future work are also included in this chapter.

5.1 The thermal analysis

The results from the thermal analyses show that the simulation tool provides good predictions of the temperature history when compared to results from analyses conducted by others. However, even if the results look promising, there are some concerns that need to be addressed. The material data, especially the conductivity, has to be well defined in the temperature range experienced by the material. Small changes in the material data proved to change the size of the FZ considerably. The implementation of the increased conductivity in the weld pool also showed to affect the FZ and peak temperatures considerably. Depending on how much it is increased and during what temperature range the increment was made, the temperature field could show large variations.

5.1.1 The UET Taxila plate

The largest temperature difference at the investigation points between the results from the analysis and [6] was found to be 77°C at the second point from the weld line. This difference can be partly explained by different solvers. In [6] an explicit solver was used while the results in this report were acquired with an implicit solver. The contribution from the latent heat implementation is unknown since it is not mentioned in [6] but still should be accounted for. Other than that, Figure 4.1 and Figure 4.2 show that the results correspond well.

5.1.2 The IIW plate

In 4.1.2., it can be seen that the temperature results from the thermal analysis were in the range given by both the participants in the IIW benchmark and the corresponding physical experiment. This shows that the simulation tool gives a good indication of the temperature history, which is an important input for calculating residual stresses and displacements in the mechanical analysis.

5.2 The mechanical analysis

The results from the mechanical analyses are not sufficient enough to verify the simulation tool's ability to predict residual stresses and displacements. As mentioned in Chapter 2.3, the temperature dependent material properties have to be sufficiently accurate. This is hard to acquire since material testing at high temperatures are expensive and rarely conducted. The way this high temperature material behaviour is implemented could affect the results significantly. The implementation of the annealing function may vary in different software packages and the effects on the result have not been studied within this project. Some difficulties with the quiet element technique were encountered in the mechanical analysis where material not yet

activated experienced stresses before activation had occurred. This resulted in premature plastic deformation in material that should be added stress and strain free. This is not only physically untrue but also caused convergence issues. This was solved by activating the quiet elements smoothly and closer to the torch. Another source of error could be the use of elements of the same order in the mechanical and thermal analyses since it may cause incompatible strain problems.

The dip in the residual longitudinal stresses, seen in Figure 4.9 and Figure 4.11, in the near proximity of the weld line is unphysical since no solid state phase transformations are present in this analysis. This may be caused by the implementation of the thermal strain for the filler metal in Abaqus in combination with the approach chosen for the incorporation of the weld metal. Further investigations are needed to resolve this discrepancy.

5.2.1 The UET Taxila plate

No real conclusion can be made from the mechanical results presented in Figure 4.9 and Figure 4.10 even though a trend can be seen when comparing with Figure 4.12 in terms of shape i.e. the longitudinal stresses are the highest followed by the transversal stress and the peak values occur in the vicinity of the weld groove.

5.2.2 The IIW plate

During the project it became clear that the mechanical analysis would take too long time to complete within the time frame of this project due to the size of the model and the low weld speed. As mentioned before a 2D model was used for the welding simulation in order to get some comparable results. The results and [12] were not similar enough to verify the simulation tool. However, when observing the stress results in [13], it can be seen that results from other participants show great variation which implies that the results are very dependent on how the tensile properties are implemented since these properties were not given.

5.3 The weld path

As seen in Figure 4.13 and Figure 4.15 the simulation tool created the weld path as intended. Both the gear wheel and the console analysis showed that the simulation tool was capable of changing its local coordinate system as it progressed. However, in the console case the temperature increased considerably at the last 90° bend as seen in Figure 4.14. This is because more heat flux than Q is distributed since more material was exposed by the distribution compared to when the heat source moves on a plane surface as illustrated in Figure 5.1.

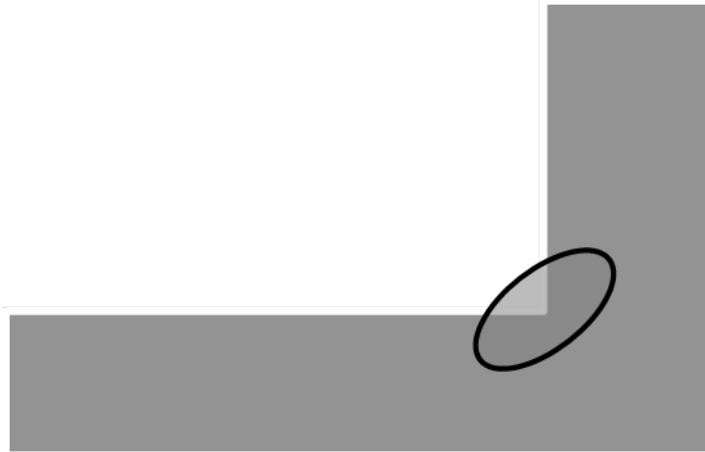


Figure 5.1 – Illustration of weld path problem near geometrical changes.

One solution for this problem could be to not define the nodes closest to the sharp bend as weld nodes. The heat source would then “take a short cut” and therefore not as much heat flux would be applied. This solution was however never investigated. This problem also occurs in the gear wheel case, but this can be solved by decreasing the total heat flux to two thirds since six out of eight octants constantly are exposed.

5.4 Gear wheel

When the three parts; freely defined weld path, thermal analysis and mechanical analysis is properly combined, this simulation tool can be used on complex geometries like a gear wheel and predict residual stresses and deformations a priori.

5.5 Future work

The thermal analysis part of the simulation tool needs to be further verified, both with other analyses and preferably with more physical experiments. However, it is important that the physical tests are well documented.

As for the time of this project, the simulation tool could not properly treat the progression of the heat source near sharp bends. A solution for this problem needs to be implemented before using the simulation tool on geometries with this particular property. The solution discussed in 5.3 seems to be a practical solution which does not result in any change of the simulation tool itself, but rather how the user defines the weld path.

Since no actual verification was done for a 3D mechanical analysis this also has to be done before applying the simulation tool in the industry. No thermal or mechanical analyses were made on other geometries than butt-welded plates and therefore further analyses needs to be done on more complex cases.

6 Conclusions

The objective of this report was to create a simulation tool that can be used in welding simulations on different geometries and materials together with a freely defined weld path in order to get residual stresses and deformations. The weld path was proven to be able to handle different paths, even though some problems occurred near geometrical changes. The thermal analyses done in this project proved to give good results and were well suited as input data to the mechanical analyses. The mechanical analyses to find the residual stresses and deformations need to be properly verified.

The simulation tool can be a good starting point for future 3D welding simulations if given further investigations and development.

References

- [1] Goldak, J. A., Chakravarti, A., Bibby, M. (1983): A New Finite Element Model for Welding Heat Sources. *Metallurgical and Materials Transactions B* **15**(1984), pp. 299-305.
- [2] Lindgren, L.-E. (2004): Numerical modeling of welding. *Computer Methods in Applied Mechanics and Engineering* **195** (2006), pp. 6710-6736.
- [3] Lundbäck, A. (2010): *Modelling and Simulation of Welding and Metal Deposition*. Doctoral Thesis, Luleå University of Technology, Luleå, Sweden.
- [4] British Energy Generation Ltd (2009): *Assessment of the integrity of structures containing defects*. R6-Revision 4, Amendment 7.
- [5] Lindström, P. R. M., Faraji, M. (2004): *Review and selection of a Finite Element Simulation Platform for Academic and Industrial Analyses of In-Service Welding Operations*, ASME 23rd International Conference on Offshore Mechanics and Arctic Engineering, OMAE2004-51246, Vancouver, British Columbia, Canada.
- [6] Afzaal Malik, M., Ejaz Qureshi, M., Ullah Dar, N. (2007): Numerical Simulation of Arc Welding Investigation of various Process and Heat Source Parameters. *UET Taxila Journal of Failure of Engineering Materials & Structures* **30**(2007), pp. 127-142.
- [7] Andersson, B. A. B. (1978): Thermal Stresses in a Submerged-Arc Welded Joint Considering Phase Transformations. *ASME Journal of Engineering Materials and Technology* **100** (1978), pp. 356-362.
- [8] Yaghi, A. H., Tanner, D. W. J., Hyde, T. H., Becker, A. A., Sun, W. (2011): *Abaqus Thermal Analysis of the Fusion Welding of a P92 Steel Pipe*. 2011 SIMULIA Customer Conference, Barcelona, Spain, pp. 622-638.
- [9] Smith, M. C., Bouchard, P. J., Turski, M., Edwards, L., Dennis, R. J. (2011): Accurate prediction of residual stress in stainless steel welds. *Computational Materials Science* **54** (2012), pp. 312-328.
- [10] Karlsson, R. I., Josefson, B. L. (1989): Three-Dimensional Finite Element Analysis of Temperatures and Stresses in a Single-Pass Butt-Welded Pipe. *ASME Journal of Pressure Vessel Technology* **112** (1990), pp. 76-84.
- [11] Alberg, H. (2005): *Simulation of Welding and Heat Treatment, Modelling and Validation*. Doctoral Thesis, 2005:33, Luleå University of Technology, Luleå, Sweden.
- [12] Lindström, P. R. M., Josefson, B. L. (2012): *2D, Axisymmetric and 3D Finite Element Analysis Assessment of the IIW RSDP Round Robin Initiative, Phase 1 and Phase 2*. IIW 65th Annual Assembly 2012, Denver, Colorado, USA.
- [13] Wohlfahrt, H. (2004): *IIW "Round Robin" Update-Results for Residual Stress and Distortion Prediction*, IIW- X/XII/XV- RSDP-97-04, International Institute of Welding.