



NTNU – Trondheim
Norwegian University of
Science and Technology

Finite Element Modelling of Weldments

Gaute Thorson Kolstad

Master of Science in Product Design and Manufacturing

Submission date: June 2012

Supervisor: Christian Thaulow, IPM

Norwegian University of Science and Technology
Department of Engineering Design and Materials

THE NORWEGIAN UNIVERSITY
OF SCIENCE AND TECHNOLOGY
DEPARTMENT OF ENGINEERING DESIGN
AND MATERIALS

**MASTER THESIS SPRING 2012
FOR
STUD.TECHN. GAUTE THORSON KOLSTAD**

**FINIT ELEMENT MODELING OF WELDMENTS
Finit element modellering av sveiste forbindelser**

A demanding challenge for steel structures in arctic regions is the risk of brittle fracture. Qualifying materials for this conditions is a complex process that demands testing of both the base material and the weldments. The most common way to test materials against brittle fracture is by crack tip opening displacement (CTOD) testing. This test procedure works fine for the base material and the melted material in the weld, but for the heat affected zone around the weld, the results show large scatter. It has therefore been proposed that testing of the heat affected zone could be done by testing of weld simulated specimens. This will remove most of the scatter, and the number of expensive test specimen is thus reduced.

This testing procedure needs to be studied and qualified, before commercial use can take place. This shall be done by comparing weld simulated specimens, with welds by doing simulations with the finite element method (FEM). The results can give a qualitative indication of how conservative a weld simulated specimen is, compared with a real weld.

Comparison of the results shall be done with the Weibull stress as a fracture criterion. This criterion takes both statistical effects and the stress level into account.

Several variables can be studied, to get an overview of the distinctions between a weld simulated specimen and a real weld.

The thesis work will be divided into three parts:

Part 1: Development of a model to transfer CTOD values measured from weld thermal simulated specimens to real welds using FE MBL modeling and the Weibull stress. The model should include the following parameters: Yield strength mismatch of HAZ, yield strength mismatch of the weld metal, and width of the HAZ. It is foreseen that a base material with yield stress 450 MPa being used. Two versions of the base material should be used, one with continuous hardening and one with Lüders strain (3% length). Mismatch levels of the HAZ and weld metal should range from -10 to +30%. Widths of the HAZ should be in the

range 0.2 to 0.5 mm. Two defect positions should be considered: At the fusion line and in the middle of the HAZ. The model should also consider the effect of geometry constraint.

Part 2: A sensitivity study on selected cases should be carried out. The sensitivity study should address the effects of varying defect positions and hardening mismatch.

Part 3: Weld thermal simulated specimens will have to be smaller than typical plate thicknesses. In order to assess the importance absolute size of the specimens FE modeling of SENB05 specimens with different dimensions should be carried out. The Weibull stress should be used to establish a relation for how to modify the CTOD values obtained for smaller specimens to larger plate thicknesses. It is proposed that the study should consider dimensions in the range 10-100 mm.

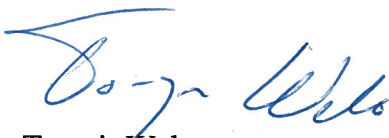
The thesis should include the signed problem text, and be written as a research report with summary both in English and Norwegian, conclusion, literature references, table of contents, etc. During preparation of the text, the candidate should make efforts to create a well arranged and well written report. To ease the evaluation of the thesis, it is important to cross-reference text, tables and figures. For evaluation of the work a thorough discussion of results is appreciated.

Three weeks after start of the thesis work, an A3 sheet illustrating the work is to be handed in. A template for this presentation is available on the IPM's web site under the menu "Undervisning". This sheet should be updated when the Master's thesis is submitted.

The thesis shall be submitted electronically via DAIM, NTNU's system for Digital Archiving and Submission of Master's thesis.

Contact person

Erling Østby, SINTEF



Torgeir Welo
Head of Division



Christian Thaulow
Professor/Supervisor



NTNU
Norges teknisk-
naturvitenskapelige universitet

Institutt for produktutvikling
og materialer

Preface

This master study has been done at the department of engineering design and materials at the Norwegian University of Science and Technology (NTNU). This thesis complete the course *TMM4911 - Materials, Master Thesis*. Responsible supervisor at NTNU has been Professor Christian Thaulow. Co-supervisors have been from SINTEF Senior Researcher Erling Østby and Professor Odd Magne Akselsen and from Statoil Professor Mons Hauge. The work has been done in cooperation with SINTEF and the Arctic Materials project.

The thesis was performed in the period from January to June 2012. Parts of this thesis is based on a pre study in the course *TMM4511 - Materials, Specialization Project* and a summer job at SINTEF in 2011.

I would like to use this opportunity to thank my supervisors, Professor Christian Thaulow, Senior Researcher Erling Østby, Professor Mons Hauge and Professor Odd Magne Akselsen for support, contributions and discussions. Eventually I would like to thank Hedvig Andersen, Inga Ringdalen Vatne and Christer Honoré Erslund for help regarding practical problems and proof reading.

Gaute Thorson Kolstad

Abstract

Brittle fracture is a large problem for steel structures in the arctic region. It is thus important to qualify materials and welds so they do not behave in a brittle manner. Since fracture testing of the heat affected zone (HAZ) around a weld gives a lot of scatter, doing weld simulated testing is proposed as an alternative method. In this thesis cracks in weld simulated HAZ specimens are compared to cracks in real welds, by use of finite element simulations.

A weld simulated specimen is usually more brittle than a real weld. The goal of this thesis is thus to find a general rule for how much more brittle a weld simulated test specimen is, compared to a real weld on a structure. It would then be possible to establish how brittle a real weld is based on the result from the weld simulated testing.

As a fracture criterion the Weibull stress is used, which is a statistical criterion. Crack tip opening displacement (CTOD) is used as a measure on how brittle a specimen is.

To compare weld simulated specimens with real welds, two 2D modified boundary layer (MBL) models are used. One homogeneous model to represent weld simulated specimens, and one with three different materials to represent a real weld. The three materials in the weld model are base material, welded material and heat affected zone. On the two models a large parameter study is performed. The variables investigated are:

- Position of the crack relative to the HAZ.
- Size of the HAZ.
- Geometry constraint.
- Mismatch in WM.
- Mismatch in HAZ.
- Hardening.
- The Weibull exponent m .

There have also been made 3D models to investigate the size effect on the weld simulated specimen. This is because a weld simulated specimen is limited to a cross-section of 10x10 mm.

The parameter study concludes that it is mainly the size of the HAZ, the yield stress mismatch and the geometry constraint, that make weld simulated specimens more brittle than welds. The 3D simulations are however concluding that the geometry constraint effect can not be included, due to the size of the small test specimen.

Based on these results a general relationship is proposed between the critical CTOD for a weld simulated specimen (δ_{ws}), and the critical CTOD for a real weld (δ_w):

$$\begin{aligned} \delta_w &= \delta_{ws} & \delta_{ws} &\in [0, 0.05) \\ \delta_w &= 20\delta_{ws}^2 & \delta_{ws} &\in [0.05, 0.1) \end{aligned}$$

There are three requirements for this relationship to be valid:

- At least 10% overmatch in HAZ compared to base material.
- No more than 10% undermatch in HAZ compared to welded material.
- Maximal brittle HAZ thickness of 0.5 mm.

Sammendrag

Sprøbrudd er et stort problem for stålkonstruksjoner i arktiske strøk. Det er derfor viktig å kvalifisere materialer og sveiser så de ikke oppfører seg sprøtt. Sprøbruddtesting av det varmepåvirkede området rundt en sveis, gir ved bruk av ekte sveiser stor spredning i resultatene. Et forslag til en alternativ metode, er å gjøre sprøbruddtester på sveisesimulerte prøver. I denne masteroppgaven blir sprekker i sveisesimulerte prøver sammenlignet med sprekker i sveiste prøver med elementmetoden.

En sveisesimulert bruddprøve er vanligvis sprøere enn en ekte sveis vil være. Målet i denne masteren er derfor å finne en generell regel for hvor mye sprøere en sveisesimulert prøve er, sammenlignet med en ekte sveis på en konstruksjon. Med en slik regel vill det bli mulig å finne ut hvor sprø en ekte sveis er, på bakgrunn av sveisesimulerte prøver.

Som sprøbruddparameter brukes Weibullspenningen, som er et statistisk sprøbruddkriterium. CTOD brukes til å si noe om hvor sprø en prøve er.

For å sammenligne sveisesimulerte prøver med ekte sveiser, er det brukt to 2D MBL modeller. Den ene er homogen og skal etterligne en sveisesimulert prøve, mens den andre inneholder tre materialer og etterligner en ekte sveis. De tre materialene er grunnmateriale, sveist metall og varmepåvirket område. På disse to modellene er det gjort et stort parameterstudie. Variablene som er studert er:

- Posisjon på sprekken i forhold til varmepåvirket sone.
- Størrelse på varmepåvirket sone.
- Geometri constraint.
- Flytespenning mismatch i sveist metall.
- Flytespenning mismatch i varmepåvirket sone.
- Fastning.
- Weibulleksponenten m .

Det har også vært laget 3D modeller for å undersøke om det er noen størrelseseffekt fra den sveisesimulerte prøven. Dette gjøres siden den sveisesimulerte prøven kun kan ha et tverrsnitt på 10x10 mm.

Parameterstudiet konkluderer at det er i hovedsak størrelsen på den varmepåvirkede sonen, flytespenning-mismatchen og geometriconstrainten som gjør sveisesimulerte prøver sprøere. 3D analysene viser på den annen side at geometrieffekten forsvinner på grunn av størrelsen på den sveisesimulerte prøven.

På bakgrunn av disse resultatene er det lagt fram et forslag til en sammenheng mellom kritisk CTOD for en sveisesimulert prøve (δ_{ws}) og kritisk CTOD for en ekte sveis (δ_w):

$$\begin{array}{ll} \delta_w = \delta_{ws} & \delta_{ws} \in [0, 0.05) \\ \delta_w = 20\delta_{ws}^2 & \delta_{ws} \in [0.05, 0.1) \end{array}$$

Det er tre krav som stilles til sveisen for at denne sammenhengen skal være gyldig:

- Minst 10% overmatch i varmepåvirket sone sammenlignet med grunnmateriale.
- Maksimalt 10% undermatch i varmepåvirket sone sammenlignet med sveistmetall.
- Maksimal utstrekning på sprø varmepåvirket sone på 0.5 mm.

Contents

Preface	iii
Abstract	v
Sammendrag	vii
Nomenclature	xii
1 Introduction	1
2 Background	3
2.1 Arctic Materials	3
2.2 Steel weldments	3
2.2.1 CTOD testing of welds	4
2.2.2 CTOD testing of weld simulated specimens	4
2.3 Previous work	5
2.3.1 Experimental testing	5
2.3.2 Pre study	6
2.3.3 Ranestad	7
3 Fracture mechanics	9
3.1 Linear elastic fracture mechanics	9
3.2 Elastic plastic fracture mechanics	10
3.2.1 CTOD	10
3.3 Geometry constraint	10
3.4 Local failure criterion	11
3.4.1 Derivation of the Weibull stress	11
3.4.2 The m parameter	13
3.4.3 Weibull as a fracture parameter	14
3.5 Thickness requirements for CTOD testing	14
3.6 Out of plane sampling	15
4 Heat affected zone	17
4.1 Mismatch	18
4.1.1 The mismatch effect	18
5 FE modeling of cracks	19
6 Method description and implementation	21
6.1 MBL model	21
6.1.1 Scripting	23
6.1.2 Validity	23
Validity tests	23
Validity area	24
6.1.3 Parameters	25
6.2 3D simulations	26

6.2.1	SENB model	27
6.2.2	SENT model	29
6.2.3	Validity	29
6.3	Materials	31
6.4	Post processing	32
6.4.1	Plots	33
7	Results of parameter study	35
7.1	Real material curves	35
7.2	Placement of the crack	35
7.3	Effect of size	36
7.4	T -stress at MBL model	37
7.5	Mismatch in yield stress	38
7.6	Mismatch in hardening	40
7.7	General hardening	40
7.8	The effect of m	41
8	Discussion	43
8.1	Out of plane sampling	43
8.2	Constraint and specimen size	43
8.2.1	SENB comparison	43
8.2.2	SENT comparison	44
8.3	Mismatch and size effect	46
8.4	Discussion	48
9	Conclusion	51
10	Suggestions for future work	53
11	Bibliography	55
Appendix A	Curves	57
Appendix B	First Python code	86
Appendix C	Second Python code	96
Appendix D	Calculate Weibull	103
Appendix E	Materials	107

Nomenclature

E	Elasticity modulus [MPa]
ν	Poisson ratio
μ	Shear modulus
σ_{ys}	Yield stress [MPa]
σ_{xx}	Normal stress in x-direction [MPa]
σ_n	Stress normal to crack plane [MPa]
$\sigma_{0.2}$	Stress at 0.2% plastic strain [MPa]
σ_1	Maximal principle stress [MPa]
r	Distance from crack tip [mm]
θ	Angle from crack plane
K	Stress intensity factor [MPa $\sqrt{\text{m}}$]
K_I	Stress intensity factor, mode I [MPa $\sqrt{\text{m}}$]
K_{II}	Stress intensity factor, mode II [MPa $\sqrt{\text{m}}$]
K_{III}	Stress intensity factor, mode III [MPa $\sqrt{\text{m}}$]
K_{Ic}	Critical stress intensity factor [MPa $\sqrt{\text{m}}$]
T	T -stress [MPa]
δ	Crack tip opening displacement (CTOD) [mm]
δ_c	Critical CTOD [mm]
δ_w	CTOD for welded specimens [mm]
δ_{ws}	CTOD for welded simulated specimens [mm]
J	The J -contour integral [MPa m]
a	Crack depth [mm]
a_0	Crack depth [mm]
a_c	Critical microcrack size [mm]
b_0	Ligament length [mm]
R	Radius of crack tip [mm]
W	Specimen thickness [mm]

B	Specimen thickness out of plane [mm]
L	Specimen length [mm]
m	Scalar dependent on stress state
σ_W	Weibull stress [MPa]
m	Weibull modulus
V	Volume of plastic zone [mm ³]
V_0	Characteristic volume [mm ³]
γ	Effective surface energy [J/mm ²]
b	Microcrack density distribution
α	Numerical constant related to the choice of V_0
β	Constants assumed independent of V_0
C	Numerical constant dependent on crack geometry
σ	Stress normal to microcrack plane [MPa]
σ_u	Weibull parameter
k	Weibull parameter
p	Probability of fracture in volume V_0
P	Probability of fracture
N	Number of volumes V_0
n	Number of elements in the plastic zone
C_1	Constant dependent of m
C_2	Constant dependent of m
$\Delta t_{8/5}$	Time used to cool down from 800°C to 500°C [s]

1 Introduction

Brittle fracture is a large problem for steel structures in the arctic region. Qualifying materials for these conditions is a complex process that requires testing of both base material (BM) and the weldments. Today the most common way to test welded materials against brittle fracture is by crack tip opening displacement (CTOD) testing. Both welded metal (WM) and heat affected zone (HAZ) need to be tested. This is performed on welded test specimens. When doing CTOD tests in the HAZ, a large problem is that there is a lot of scatter. This complicates the testing, and it is thus desirable to find a better test procedure. Testing of homogeneous weld simulated specimens could be a solution to this problem. This will remove most of the scatter, and the number of expensive test specimens is thus reduced. Testing of weld simulated specimens is quite new and needs to be studied and qualified, before commercial use.

This thesis compares weld simulated specimens with welds, by doing simulations with the finite element method (FEM). The results can give an indication of how much more brittle a weld simulated specimen is, compared to a real weld.

Two types of 2D modified boundary layer (MBL) models are made to compare weld simulated specimens with real welds. One homogeneous model to represent a weld simulated specimen, and one model with three materials to simulate a real weld. The three materials in the weld model are BM, HAZ and WM. With these models, a large parameter study is performed in order to explore qualitative effects of different variables. The variables studied are:

- Position of the crack relative to the HAZ.
- Size of the HAZ.
- Geometry constraint.
- Mismatch in WM.
- Mismatch in HAZ.
- Hardening.
- The Weibull exponent m .

3D models of CTOD test specimens are also made, to investigate if there are any size effects, compared to a real structure.

The comparison of different CTODs is performed with Weibull stress as a fracture criterion. This criterion takes both statistical effects and stress level into account.

The goal of the thesis is to make a first proposal for a general relation between the CTOD of weld simulated specimens and the CTOD of a real weld on a structure.

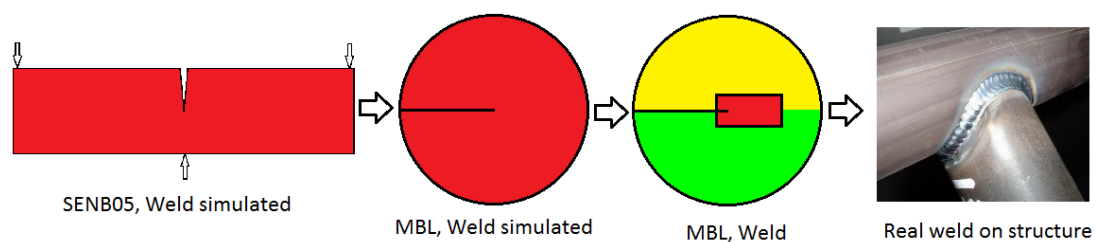


Figure 1.1: The different models, for CTOD comparison.

2 Background

2.1 Arctic Materials

SINTEF is carrying out a large research project called Arctic Materials. Major industry companies like Aker Solutions, Statoil, Total, ENI, DNV, Nippon Steel, JFE, Scana and Bruck are participating. The main goal of the project is to establish criteria and solutions for safe and cost-effective applications of materials for hydrocarbon exploration and production in arctic regions. Sub goals are:

- Extend the application range for structural steels by characterization of brittle fracture resistance in welded steel structures. Including the effects of geometry constraint and local variation of material properties.
- Define criteria for low temperature application for steel and weldments including safety factors (design temperatures down to -60°C).
- Develop specifications for qualification testing of materials to be applied down to -60°C .
- Define material criteria for safe application under large deformation (strains $> 5\%$) of pipelines and other structures, including safety factors.
- Develop requirements for polymer coatings to be used down to -60°C , and under large temperature variations.
- Develop basis for applications of light weight structures using materials like high strength steel, composites, and hybrid solutions.

The project is divided into four main work packages [1]:

- WP 1 - Steel fabrication and mechanical characterization
- WP 2 - Strength and toughness criteria for safe material utilization
- WP 3 - Polymer coatings
- WP 4 - New concepts and materials solutions

One of the most challenging problems for steel structures in arctic conditions is brittle fracture. With a decrease in temperature, the steel changes from ductile to brittle. It is therefore important to develop and qualify steels that are ductile even at low temperatures. One of the main topics of this research project is thus to develop guidelines, procedures and criteria for qualifying steels to use under arctic conditions.

2.2 Steel weldments

Welds are critical areas of a steel structure, especially concerning brittle fracture. It is thus important to establish efficient and stable test procedures for the welds to qualify a material.

To test a material with respect to brittle fracture, CTOD testing is the most common test procedure. Today most of material testing considering brittle fracture in weldments, are done with CTOD testing of cracks using welded specimens. Another way of testing is to use CTOD testing on a weld simulated specimen. Both these test procedures have their benefits and faults.

In the research project Arctic materials a requirement has been proposed that a structure has to withstand a CTOD of 0.1 mm to avoid brittle fracture [2]. Two demands to this requirement are that the global stress is no more than 80% of the yield stress, and that no crack has a depth larger 5 mm. These requirements are for the global structure, it is thus important to be able to link the CTOD from the test results, to a CTOD for the global structure.

2.2.1 CTOD testing of welds

CTOD testing of real welds is today the most common way of testing a material and its welds for brittle fracture. Both WM and HAZ is then tested. A common test procedure is single edge notched bending (SENB). This test uses three point bending, see figure 2.1. The crack is first machined out, and then made sharp by fatigue. The size of the test can vary, but it is common to have the relationship $a/w = 0.5$, where a is crack depth and w the thickness of the specimen.

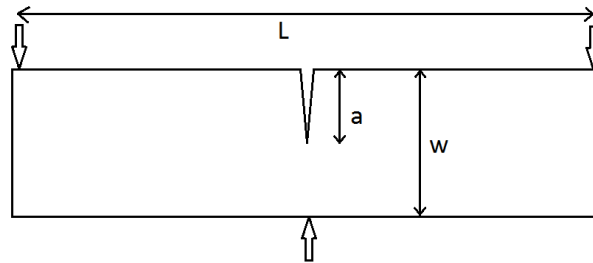


Figure 2.1: A SENB test specimen.

One of the main problems by this test method is that there is a lot of scatter in the results for the HAZ. This is due to large toughness gradients in a small area, and that the brittle areas may be of limited size. The crack fronts may therefore have large variations in toughness. This causes a lot more scatter in the weld testing, than the real brittle microstructure actually has [3]. It is thus desirable to find better test procedures.

Using this type of high constraint specimen is an efficient but conservative way of testing. This is because a real structure will always have a lower constraint level than a SENB05 (SENB with $a/w = 0.5$). To link the critical CTOD measured in the testing to the real structure, there should be a factor taking the constraint effect into account [2].

2.2.2 CTOD testing of weld simulated specimens

Weld simulated specimens are specimens that are given a certain heat cycle in an oven to get the same microstructure as an area in the HAZ. The specimens are thus homogeneous and the crack will hit the same microstructure every time. This reduce the scatter compared with testing of real welds. The specimen can today only have a 10x10 mm cross section and still be homogeneous, because of the oven. CTOD testing of weld simulated specimens are usually done with SENB05 [4].

Compared to real welds, there are four major factors making the probability of brittle fracture in weld simulated specimens higher:

- The probability of brittle fracture is depending on the size of the brittle area. The brittle area is much larger in a weld simulation specimen than in a real weld and the probability for fracture is thus higher.
- The brittle area where crack propagation often occurs has a high yield stress. In a weld the surrounding materials will lower the effective yield stress because of the mismatch effect. This will reduce the probability for brittle fracture in a weld compared to a weld simulated specimen.
- Since the level of constraint is high in a SENB05 specimen and low in a real structure, brittle fracture should occur at a lower deformation for a weld simulated specimen.
- There are also an effect from the crack sampling out-of-plane when testing several specimens.

These factors should be taken into account when comparing the critical CTOD from the testing, with the requirement for the global structure. This master thesis is looking into these effects, especially the three first.

When testing weld simulated specimens several tests have to be performed to get a statistical foundation. For weld simulated specimens a test series of ten samples is proposed, [2]. When testing welds, this amount need to be larger since the scatter is high.

2.3 Previous work

2.3.1 Experimental testing

In the Arctic Material project material testing have been performed comparing the two mentioned test methods, welds and weld simulated specimens [5]. The testing has been done at different temperature levels, and when comparing the behaviour on the different temperature levels, an interesting trend appears. For low temperatures, when the weld simulated specimen shows very brittle behaviour the welded specimen shows the same brittle behaviour. The two test procedures then have an approximately 1 : 1 relationship, see figure 2b. For higher temperature the weld simulation specimens show a slightly more ductile behaviour. This leads to a large increase in ductility for the welded specimens, see figure 2a. This indicates that the toughness of the weld increase rapidly if the toughness of the weld simulated specimen exceeds a threshold value [5].

The points in figure 2.2 where the CTOD is very large arise because the crack missed the fusion line (FL) [5]. This shows how large the scatter is for testing of welds.

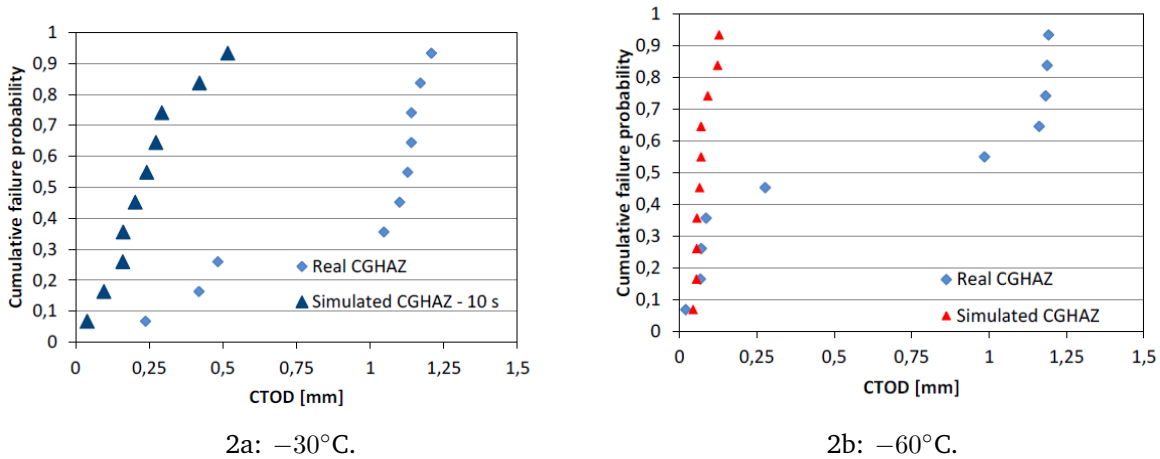
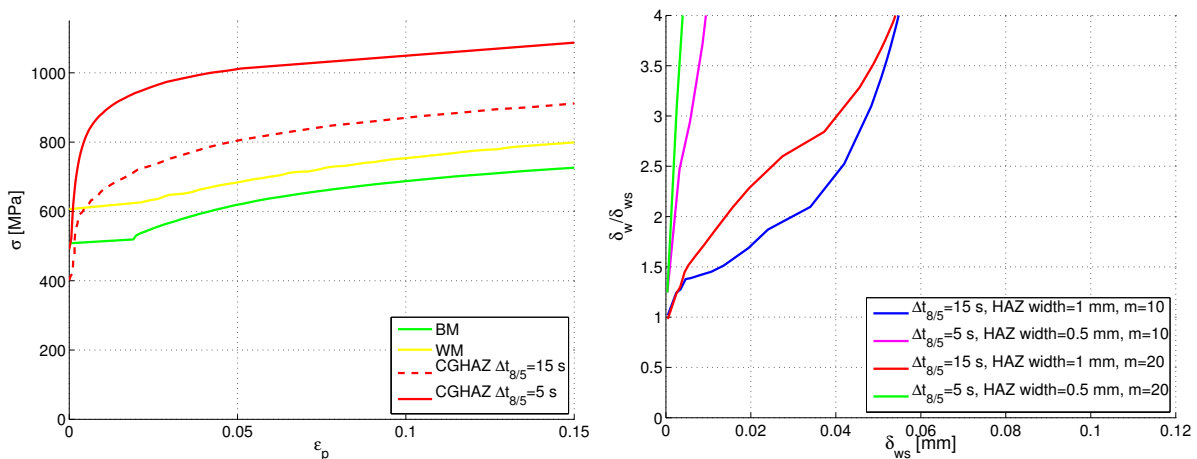


Figure 2.2: CTOD plotted against cumulative failure probability for experimental results [5].

2.3.2 Pre study

A pre study has been done to this master thesis [6], with a parameter study using the same 2D model as used here. For the pre study real material data was used for simulations. The material was a rolled plate with yield strength of 420 MPa. Coarse grained HAZ was used with two different cooling rates, $\Delta t_{8/5} = 15$ s and $\Delta t_{8/5} = 5$ s. The material curves are plotted in figure 3a.

When comparing a simulation of a real weld and a simulation of a weld simulated specimen, the weld simulated specimen is much more brittle, see figure 3b. This figure is explained later, see figure 6.13. The HAZ width used on $\Delta t_{8/5} = 15$ s is 1 mm, and 0.5 mm for $\Delta t_{8/5} = 5$ s. The crack is placed at FL and $T = 0$. This result is later compared with the more general parameter study done in this thesis.



3a: True stress plotted against true plastic strain at -60°C . 3b: Comparison of simulations of real welds and weld simulated specimens. Crack is placed at FL and $T = 0$.

Figure 2.3: Some material and simulation data from the pre study [6].

The pre study concludes, as mentioned above [5], that there is a nonlinear rela-

tionship in fracture toughness between welds and weld simulated specimens. It also concludes that this relationship is dependent on several factors. The two factors which contribute the most, are level of mismatch between the HAZ and the other materials, and the size of the HAZ. The position of the crack also has some influence.

An interesting conclusion of the pre study is that the results are quite independent of level of T -stress and the Weibull parameter m . This assertion is discussed in more detail later in this thesis, see chapter 7.

2.3.3 Ranestad

A great deal of research has been done on the effect of mismatch in welds. The effect of mismatch has been studied with FEM in a PhD thesis by Ø. Ranestad [7]. In his thesis he uses a MBL model to simulate a three material system in a weld, as shown in figure 1b. The materials are welded metal, heat affected zone and base metal. For the three materials he assumes that elastic properties and plastic hardening are equal, but with different yield stresses. For the base material the yield stress is set equal to 500 MPa, while the heat affected zone have a yield stress at 600 MPa. The yield stress in the welded metal is then changed to vary the local mismatch. The simulations are run with a yield stress of 700 MPa, 650 MPa, 600 MPa and 550 MPa. He also performed a reference simulation where the model is homogeneous with a yield stress of 600 MPa.

From the simulations a normalized plot with the maximal principle stress at the ligament is shown. In figure 2.4 the constraint effect of the local mismatch is illustrated. The reference simulations with no mismatch have the highest level of stress, since there is no soft material to lower the stresses. In the other simulations the stress level in the ligament decreases with decreasing yield stress in the welded metal.

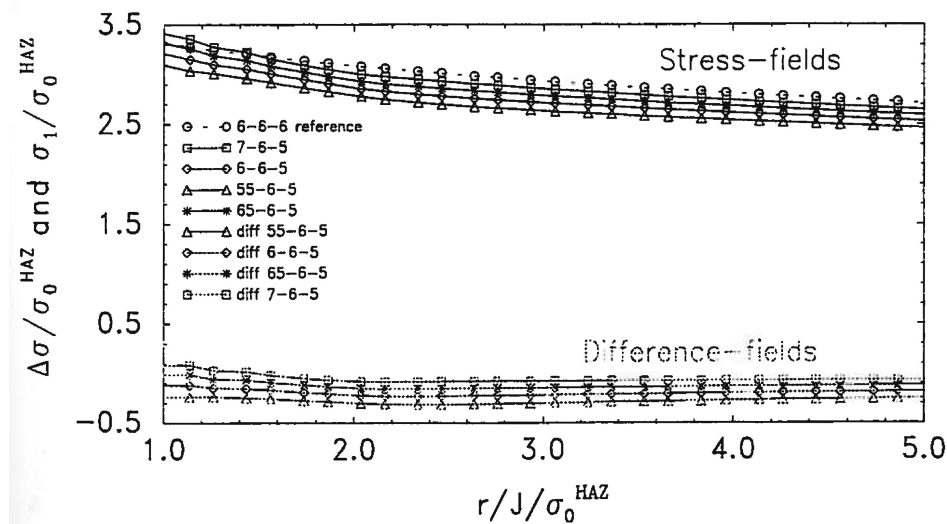


Figure 2.4: The normalized stress against normalized length, over the ligament [7, p.1-7].

3 Fracture mechanics

Fracture is a complex mechanism which scientists have tried to understand for generations. The models that exist are all based on assumptions. In this report, crack tip opening displacement (CTOD, δ) and the Weibull stress (σ_W) are the main parameters used. Therefore this theory focuses on these, but there are also other parameters mentioned.

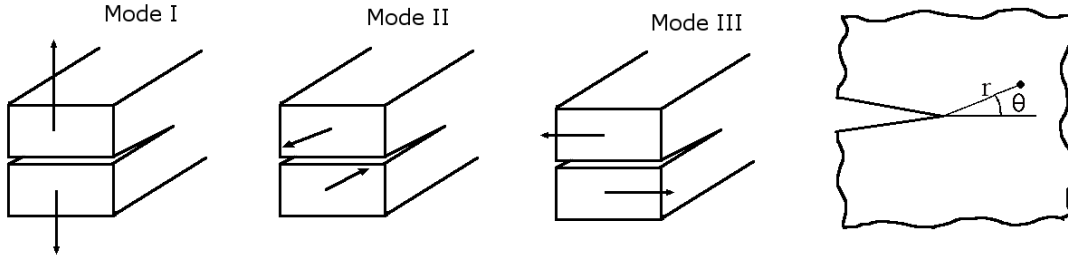


Figure 3.1: The different modes of loading to the left, and the definition of r and θ to the right [8].

It is common to say that a crack have three main types of loading: mode I, II and III. Mode I is loading normal to the crack plane, mode II is shear loading in the crack direction and mode III is shear loading out of plane, see figure 3.1. The stress around a crack tip is often described in terms of r , the distance from the crack tip, and θ , the angle with respect to the crack plane, see right side of figure 3.1.

3.1 Linear elastic fracture mechanics

The stress intensity factor K is a widely used parameter to describe the loading of a crack. This parameter is based on an isotropic linear elastic material, and can be calculated from the global stress field and the crack geometry. K is depending on the loading mode and it is thus usually divided into K_I , K_{II} and K_{III} , for the three modes of loading. The geometry of the crack is also important for K , for instance type of crack and size. An important characteristic of the stress intensity factor is that it uniquely describes the stress field around the crack tip, with a $1/\sqrt{r}$ singular field. The equations for the stress field are based on a series expansion where at least the terms of order \sqrt{r} and higher are neglected.

In this project only K_I will be used since only mode I will be considered. The displacement field around the crack tip for mode I loading is described by these expressions [9]:

$$u_x = \frac{K_I}{2\mu} \sqrt{\frac{r}{2\pi}} \cos\left(\frac{\theta}{2}\right) \left[\kappa - 1 + 2 \sin^2\left(\frac{\theta}{2}\right) \right] + T \frac{(1 - \nu^2)}{E} r \cos(\theta) \quad (1)$$

$$u_y = \frac{K_I}{2\mu} \sqrt{\frac{r}{2\pi}} \sin\left(\frac{\theta}{2}\right) \left[\kappa + 1 - 2 \cos^2\left(\frac{\theta}{2}\right) \right] - T \frac{\nu(1 + \nu)}{E} r \sin(\theta) \quad (2)$$

where ν is the Poisson rate, μ the shear modulus, T the T -stress which is described in chapter 3.3, $\kappa = 3 - 4\nu$ for plane strain and $\kappa = \frac{3-\nu}{1+\nu}$ for plane stress and E is the elastic

modulus. These equations also consist of a term which depend on K_{II} and \sqrt{r} , but it is not included here.

The stress intensity factor can also be used as a fracture criterion. Since linear elasticity is assumed, this criterion only holds for very brittle materials where there is little plastic deformation prior to failure. The critical stress intensity K_{Ic} then works as a material constant, and failure occur when K_I reaches K_{Ic} .

3.2 Elastic plastic fracture mechanics

To have a fracture criterion that is valid for more plastic deformation, elastic-plastic fracture mechanics has to be used. This introduces two new fracture parameters, CTOD and J . The J contour integral is not used in this report.

3.2.1 CTOD

CTOD is one of the most commonly used fracture parameters. Under loading a sharp crack will experience plastic deformations and blunt, see figure 3.2. The CTOD can then be defined as the distance across the crack where the blunting stops, but this is no clear definition. In FE modeling of cracks, the CTOD is usually defined as the displacement normal to the crack plane for the first two nodes away from the crack tip.

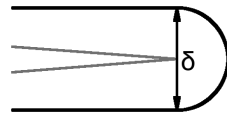


Figure 3.2: CTOD (δ) [8].

It is possible to link K_I to CTOD for small plastic zones and linear elastic behavior [9]:

$$\delta = \frac{K_I^2}{m\sigma_{ys}E'} \quad (3)$$

where σ_{ys} is the yield stress and m is a dimensionless constant that is approximately 1 for plane stress and 2 for plane strain. E' is equal E for plane stress and equal $\frac{E}{1-\nu^2}$ for plane strain.

CTOD can also be used as a fracture criterion where fracture occurs when $\delta = \delta_c$. CTOD is then a material constant, but usually depends on both temperature and constraint level.

3.3 Geometry constraint

In equations (1) and (2) there are terms which are scaled by the T -stress (T). T -stress is defined as the global stress in the crack direction (σ_{xx}). The T -stress comes from the second, constant term in the stress field series expansion.

T -stress is often used as a parameter to describe the level of geometry constraint on the crack. The level of constraint in a model has a large influence on the plastic zone

around the crack tip. This is due to the formation of a more hydrostatic stress field, if the constraint level is high (high T -stress). According to the Mises yield criterion it is only the deviatoric stress level that defines when a material yield [10]. This implies that a highly constrained crack can have very high stresses around the crack tip without a big plastic zone. A crack with low constrain will have more stress relaxation due to plastic deformation. This may have a big influence on the crack behavior and how much global load it can stand.

Typical level of T -stress for a single edge notch bending (SENB) specimen with crack depth thickness ratio $\frac{a}{w} = 0.5$ is $T = 0$. For a single edge notch tension (SENT) specimen with $\frac{a}{w} = 0.5$ the constraint level is typically $T = -0.5\sigma_{0.2}$. Here $\sigma_{0.2}$ is the stress at 0.2% plastic strain.

3.4 Local failure criterion

The classic fracture criteria using K , J and CTOD have several weaknesses when it comes to predict fracture in real structures. When the geometry becomes complex, the deformation high or the temperature is changing, none of them are valid any more. Another problem is that they can not predict the scatter in fracture testing, especially for brittle fracture. One of the approaches to overcome these problems has been to connect micromechanical fracture criterion, with the global fracture behaviour. These methods use as a basis that fracture occurs from a defect in the material, and the critical step is the propagation from this. The most promising of the local failure criteria is the Local Approach. This was developed by a French research group, Beremin [11]. They had two main assumptions in the method [12]:

- The failure probability inside a small uniformly stressed volume can be expressed by the stress level and the distribution of microcracks in this volume.
- The total failure probability of the whole component follows a *weakest link* mechanism, and failure in one part of the component results in final failure of the whole component.

The method is based on dividing a volume V into small volumes V_0 , and calculate the probability of fracture in each of these volumes. Then use them to calculate the fracture probability of the volume V . The volume V is set as the plastic zone because it is shown that there has to be some plastic deformation for fracture to occur in steel [13].

3.4.1 Derivation of the Weibull stress

To get a local fracture criterion for each volume part, a relation between stress and the critical microcrack size is used. Beremin assumed that this could be described by a Griffith relation [12]:

$$a_c = \frac{2E\gamma}{C\sigma^2} \quad (4)$$

where γ is the effective surface energy, a_c the critical microcrack size, σ the tension stress normal to the microcrack plane and C a numeric constant dependent on the crack geometry. To get a statistical approach they needed a statistical distribution of

microcracks, which could be changed for different materials. To get this they proposed that the microcrack density distribution $b(a)$ inside a volume V_0 could be expressed as [12]:

$$b(a) = \frac{\alpha}{a^\beta} \quad (5)$$

where α is a numerical constant related to the choice of V_0 , and β is a constant assumed independent of V_0 . The probability to find a microcrack larger than a_c is then:

$$p(a) = \int_{a_c}^{\infty} b(a) da = \int_{a_c}^{\infty} \frac{\alpha}{a^\beta} da \quad (6)$$

Introducing $k = \frac{2E\gamma}{C}$ and inserting equation (4) into (6) the probability of fracture for a given stress can be written as:

$$p(\sigma) = \int_0^\sigma \frac{2\alpha}{k^{\beta-1}} \sigma^{2\beta-3} d\sigma \quad (7)$$

By introducing $m = 2\beta - 2$ the integral becomes:

$$p(\sigma) = \int_0^\sigma \frac{2\alpha}{k^{\frac{m}{2}}} \sigma^{m-1} d\sigma = \frac{2\alpha}{mk^{\frac{m}{2}}} \sigma^m \quad (8)$$

Into this expression a new variable is introduced: $\sigma_u = \sqrt{k} \left[\frac{m}{2\alpha} \right]^{\frac{1}{m}}$ which leads to:

$$p(\sigma) = \left(\frac{\sigma}{\sigma_u} \right)^m \quad (9)$$

In this equation σ_u and m are assumed material constants that are independent of temperature. They also link the local fracture behavior to the continuum mechanics stress field. Whether σ_u really is temperature independent, can be discussed. In equation (5) α is assumed dependent of V_0 . Then from the definition of σ_u , the independent parameter should be $V_0 \sigma_u^m$.

To link the local fracture probability (9) to the global fracture probability, a weakest link approach is used. Then the probability of failure in a volume V_0 is assumed statistically independent, and failure in one volume is critical. The total failure probability is then:

$$P_f = 1 - \prod_{i=1}^N (1 - p(\sigma_i)) \quad (10)$$

Where N is the number of volumes, and $p(\sigma_i)$ is the failure probability in volume i . This can be rewritten if $p(\sigma_i)$ is small for a uniformly stressed volume:

$$\ln(1 - P_f) = \ln \prod_{i=1}^N (1 - p(\sigma_i)) = \sum_{i=1}^N \ln(1 - p(\sigma_i)) \approx - \sum_{i=1}^N p(\sigma_i) \quad (11)$$

$$\ln(1 - P_f) \approx \sum_{i=1}^N \left(\frac{\sigma_i}{\sigma_u} \right)^m = - \frac{V}{V_0} \left(\frac{\sigma}{\sigma_u} \right)^m \quad (12)$$

$$P_f \approx 1 - \exp \left(- \frac{V}{V_0} \left(\frac{\sigma}{\sigma_u} \right)^m \right) \quad (13)$$

This equation is valid only for a uniformly stressed volume. To get a model that can handle inhomogeneous stresses and are suited for finite element calculations, $\sum_{i=1}^N p(\sigma_i)$ is replaced with $\sum_{j=1}^n \left(\frac{\sigma_1^j}{\sigma_u}\right)^m \frac{V_j}{V_0}$ in equation (11). Where n is the number of volumes V_j , or elements in a FEM calculation and σ_1^j the maximal principle stress of the element j . $\frac{V_j}{V_0}$ is just a scaling based on the assumption that the probability scales with the volume. This gives:

$$P_f = 1 - \exp\left(-\sum_{j=1}^n \left(\left(\frac{\sigma_1^j}{\sigma_u}\right)^m \frac{V_j}{V_0}\right)\right) \quad (14)$$

Beremin introduced the Weibull stress (σ_W) defined as [11]:

$$\sigma_W = \sqrt[m]{\sum_{j=1}^n (\sigma_1^j)^m \frac{V_j}{V_0}} \quad (15)$$

By inserting this in equation (14) the probability of failure becomes:

$$P_f = 1 - \exp\left(-\left(\frac{\sigma_W}{\sigma_u}\right)^m\right) \quad (16)$$

3.4.2 The m parameter

m is one of the two constants in the Weibull stress, see equation (15). Later in this thesis this parameter show some interesting effects. It is therefore studied in more detail here.

m is meant to be a material constant and is defined by:

$$m = 2\beta - 2 \quad (17)$$

where β is defined in equation (5). Equation (5) represents a Poisson distribution over the microcrack density. β is then a parameter saying something about the width of this distribution. Since the length of the microcrack (a) is raised to the power of β , the distribution gets very sensitive to m .

The width of the Poisson distribution is important for how a material behaves, and which factors that are most critical for fracture to occur. A narrow distribution, high β and m , means a lot of microcracks of a certain size. This implicate that it is the height of the stress peak that decides when fracture occurs, since this stress peak will hit one of the many equally sized microcracks. For a wide distribution, low β and m , there will be microcracks in many sizes, but fewer of each type. This will implicate that it is the size of the plastic zone that is important for when fracture occur. This is because a large plastic zone has an increased probability of hitting a large microcrack. The height of the stress peak is no longer very important since the probability for a large crack in this area is small.

Ideally m should be found without material testing, and established just by metallurgical methods. However what m a material has, is not only dependent on the crack density. For instance how the material arrests cracks is also important. Because of this m has become more of a fitting parameter than the physical parameter it was meant to be. There has thus been developed other solutions for estimating m .

Finding m was first done by testing of notch tensile specimens. The results from these tests were however not correct when a crack was present. Therefore fracture mechanics

testing for estimating m was proposed. For fracture testing with small scale yielding it was shown that σ_u and m are non-unique, because of this the test procedure has to be more complex, and involve testing of specimens with different constraint [14]. The testing procedure then becomes an iteration process to get the correct m . For a normal steel m is usually in the range 7 – 20.

3.4.3 Weibull as a fracture parameter

The validity of the Weibull stress as a fracture parameter has been discussed a lot after it was introduced by Beremin [11]. The method has several flaws, like the assumption that the parameter m is insensitive to temperature and constraint and that microcrack distribution does not vary under plastic deformation [15]. Because of this there has been published several improvements. The problem with these improvements is that all of them are very complex, and therefore not useful from an engineering point of view.

A lot of testing has been performed of the Weibull stress, with different conclusions. Some find very good relationship between the Weibull stress and the fracture testing [16]. Others find good relationships by introducing some extra equations [15], [17]. Since brittle fracture is such a complex phenomena and dependent on several parameters, there is probably no ideal model applicable to all microstructures. In this thesis the general Weibull stress is used, despite its limitations. It is thus important to know its restrictions.

One approach to compare different geometries and constraints, is to use toughness scaling [18]. By toughness scaling different models are compared with each other, based on a fracture parameter. It is assumed that the probability for fracture is set by this fracture parameter, regardless of geometry and constraint. The load, or deformation level, of different models can then be compared, to see the effect of geometry or constraint, on this fracture parameter, see figure 6.13.

For the Weibull stress this implies that the two variables V_0 and σ_u can be disregarded when using toughness scaling.

3.5 Thickness requirements for CTOD testing

When performing fracture mechanics testing there are some requirements for the size of the test specimens. This is to avoid influence of the boundaries and the free surfaces. In BS7448 [19] and ASTM E-1820 [20] there is developed a standard which is based on K from linear elastic fracture mechanics:

$$B, a_0, b_0 \leq 2.5 \left(\frac{K_Q}{\sigma_{ys}} \right)^2 \quad (18)$$

K_Q is the largest K used or the critical, B is the out of plane thickness, a_0 crack depth and b_0 the length of the ligament ($b = w - a$). The requirement needs to be valid for B , a_0 and b_0 . By inserting the relationship between δ and K from equation (3) the requirement can be rewritten as:

$$B, a_0, b_0 \leq 2.5 \frac{m E' \delta_Q}{\sigma_{ys}} \quad (19)$$

m is not the Weibull exponent but a dimensionless constant. This requirement is very stringent and is based on materials which behave in an almost perfectly brittle manner.

Another requirement from ASTM E-1921 [21] is based on J as the fracture parameter, and include thus small scale yielding:

$$K_{jc} \leq \sqrt{\frac{Eb_0\sigma_{ys}}{30(1-\nu^2)}} \quad (20)$$

Substitute K with δ from equation (3) leads to:

$$\delta_{jc} \leq \frac{b_0}{30m} \quad (21)$$

3.6 Out of plane sampling

When using statistical fracture mechanic it is not only the in-plane sampling effect, that needs to be counted for. There is also a sampling effect out-of-plane, longer cracks leads to higher probability for hitting a critical microcrack. This effect is important when comparing different sizes of test specimens or a real crack with a finite length. To be able to compare different crack lengths a reference length needs to be made, the different crack length should be corrected to this reference length [22].

For the Weibull stress the out-of-plane correction can be calculated by replacing V_j with $A_j L_j$ in equation 15. L_j can then be substituted by \tilde{L}/L_j to calculate $\tilde{\sigma}_W$, which is σ_W for reference length \tilde{L} :

$$\sigma_W = \sqrt[m]{\sum_{j=1}^n (\sigma_1^j)^m \frac{A_j L_j}{V_0}} \quad (22)$$

$$\sigma_W = L_j^{\frac{1}{m}} \sqrt[m]{\sum_{j=1}^n (\sigma_1^j)^m \frac{A_j}{V_0}} \quad (23)$$

$$\tilde{\sigma}_W = \left(\frac{\tilde{L}}{L_j}\right)^{\frac{1}{m}} \sqrt[m]{\sum_{j=1}^n (\sigma_1^j)^m \frac{A_j}{V_0}} \quad (24)$$

4 Heat affected zone

A weld is made by melting two materials together. Next to the melted metal there is a zone where the temperature has been high enough for the base material to change material properties. This area is called the heat affected zone (HAZ). In steel this area reaches from the fusion line (FL) and down to approximately 600°C. Throughout this zone the maximal temperature decreases and the material properties are therefore changing continuously. In steels, the HAZ is usually divided into four zones. This is because of the different changes in microstructure steel undergoes during heating. The zones are the tempered zone, the partly recrystallized zone, the recrystallized zone and the grain growth zone, see figure 4.1 .

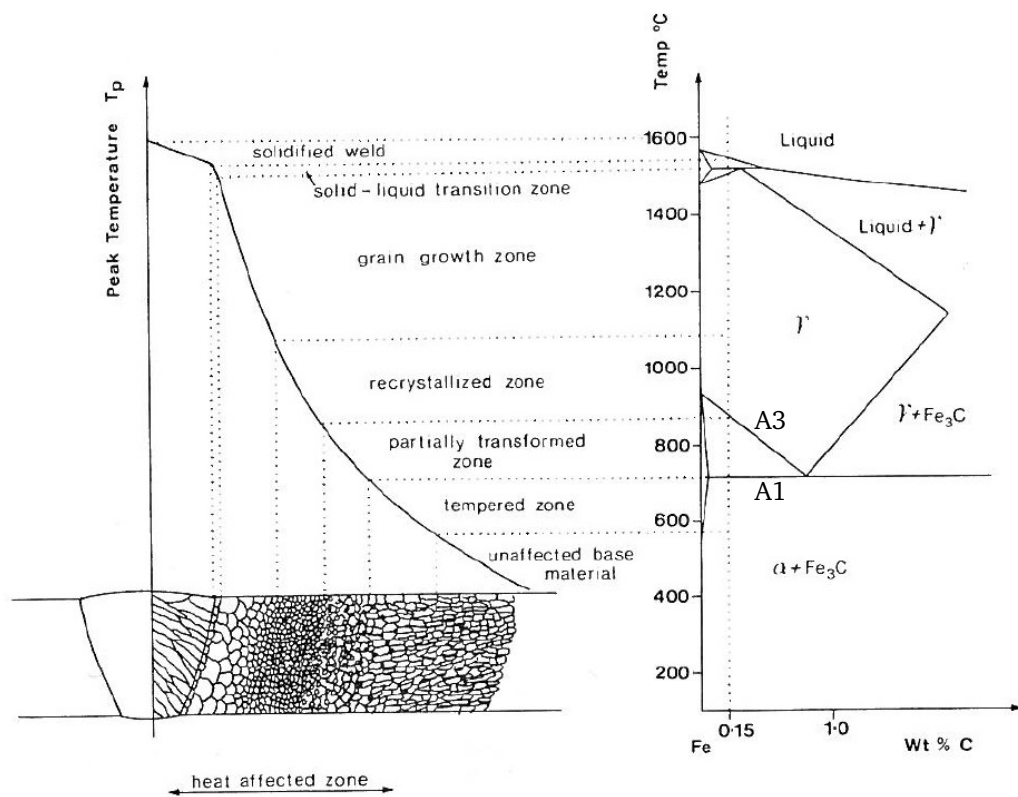


Figure 4.1: Overview of the different zones in a weld, compared with a phase diagram for steel [23].

The tempered zone is where the material has been influenced by the heat, but the material experiences no phase transformation. This zone goes up to the $A1$ line at approximately 730°C, where the transition to austenite (γ -iron) starts. This transformation continues up to the $A3$ line, where all ferrite (α -iron) is transformed to austenite. At what temperature this partly recrystallized zone ends, is depending on the alloying elements, as shown in figure 4.1. The next zone is the recrystallized zone. For low temperatures this zone consists of small austenite grains, but when the maximal temperature increases, the grains start to grow. The grain growth zone is where considerable grain growth has occurred and it is often called the coarse grained heat affected zone

(CGHAZ). This is often one of the most brittle microstructures in a weld, and the one which is simulated in this thesis.

4.1 Mismatch

When there are different material properties in two connected materials, this will cause a stress and strain effect called mismatch. Since there are big differences in material properties for the material zones in a weld, this effect will be important to the stress field.

In a weld two forms of mismatch are defined: global and local. The global mismatch is between WM and BM, and the local mismatch is between the HAZ and either the WM or BM. There is also local mismatch between the different zones in the HAZ but this is not considered in this thesis. Materials can have mismatch in different material properties, but only mismatch in yield stress and hardening is considered here. These are also considered the most important mismatch properties. Young's modulus and Poissons ratio are assumed equal in the different materials.

4.1.1 The mismatch effect

On the interface between two materials certain laws from the continuum mechanics have to be satisfied. One of them is that the strain on the interface plane needs to be continuous over the interface. This law causes a mismatch effect when yielding occurs, if the two materials have different yield stresses. The soft material will yield first and the strain will increase, since plastic flow is incompressible. Since the strain needs to be continuous over the interface, the hard material will stop the strain increase in the soft material. This implies an increase of stresses in the interface plane in the opposite to strain direction for the soft material, and stresses in the strain direction for the hard material. The stress in the opposite direction of the strain from yielding will cause a lower deviatoric stress state in the soft material. A less deviatoric stress state will reduce further yielding since it is assumed that yielding is independent of the hydrostatic stress. In the hard material there will be a higher deviatoric stress state and yielding will occur earlier than if it was alone.

The mismatch effect induces, for the soft material, an increase of the stresses at the interface along with higher mismatch. And for the hard material a decrease of stresses along with higher mismatch.

Hardening mismatch will cause some of the same effect, but then later than at the yielding point.

5 FE modeling of cracks

Modeling of welds using the finite element method (FEM) has been performed for a long time. There are several factors to consider when doing an analysis of this kind.

There are two ways to model a crack tip: either it can start sharp where the crack tip is defined as one node, or it can start as a half circle. The sharp crack is probably most realistic but can cause convergence problems, since all deformation around the crack tip will be taken by one element. A circular crack can take larger deformations, but is not valid until there has been some deformation. A rule of thumb is that the result is not valid until the CTOD (δ) is five times the initial diameter, [9, p.576].

The mesh around the crack tip should be very fine, which implies a large scaling of the mesh. There are two main methods to scale the mesh. The easiest one is to define an element size at the crack tip, and let the finite element program make a scaling of the mesh. The other, and more work demanding way is to define the mesh scaling yourself. This is easy for a circular crack tip and circular model, but can get quite complex. The best way to have a huge scaling is a circular mesh, see figure 3a. It has been discussed a lot whether the irregular mesh that the program generates makes more numerical errors. For few and big elements this might be a non-neglectable argument, but it is probably no problem for the number of elements that are used today. However it is usually favourable to have a regular mesh if you are going to present the model to other people, since a nice mesh appears more valid.

For modelling cracks, it is common to use an MBL approach. This is a method using a displacement controlled boundary. On this boundary there can be displacement fields which are dependent on the load. The distance from the crack tip to the boundary should be so large that the plastic field does not reach the boundary. This is to avoid that the boundary influences the stress field.

If the stress field close to the crack tip is going to be examined, or a large deformation is applied, it is important to do a geometric nonlinear analysis. In this way the stiffness matrices, and therefore the geometry, is updated each time step [24].

For 2D simulations the element type should be plane strain elements, as long as the model is not a thin sheet. For the 3D models, regular 3D elements will make the stress field plane-stress-like at the surface, and plane-strain-like in the middle, as long as the model is of a certain size.

6 Method description and implementation

To investigate the effect HAZ has on the stress field, FEM is used. The FEM program that is used is ABAQUS 6.10 [25].

In this master thesis two types of models have been used, 2D MBL model and 3D models of different test specimens. The MBL model have been used to run a large parameter study to qualitatively investigate the effect of different variables. It was also used quantitatively to make a proposal for a design line to compare weld simulated specimens with real structures. The 3D models were mainly used to validate the MBL model.

6.1 MBL model

A good way to control the loading parameters around a crack in FEM is to use an MBL model. The displacement field around the boundary is controlled by the equations (1) and (2), where K_I and T can be controlled, see figure 6.1. The crack emerges from the left and ends in the middle of the model. The model is built by two Python scripts, see Appendix B and Appendix C.

Two types of MBL of MBL models have been used. The weld simulated model is a homogeneous model where all the material is HAZ, see figure 1b. This is often called reference simulation. The other model is a simulations of a weld, where the model consist of three different materials, see figure 1a.

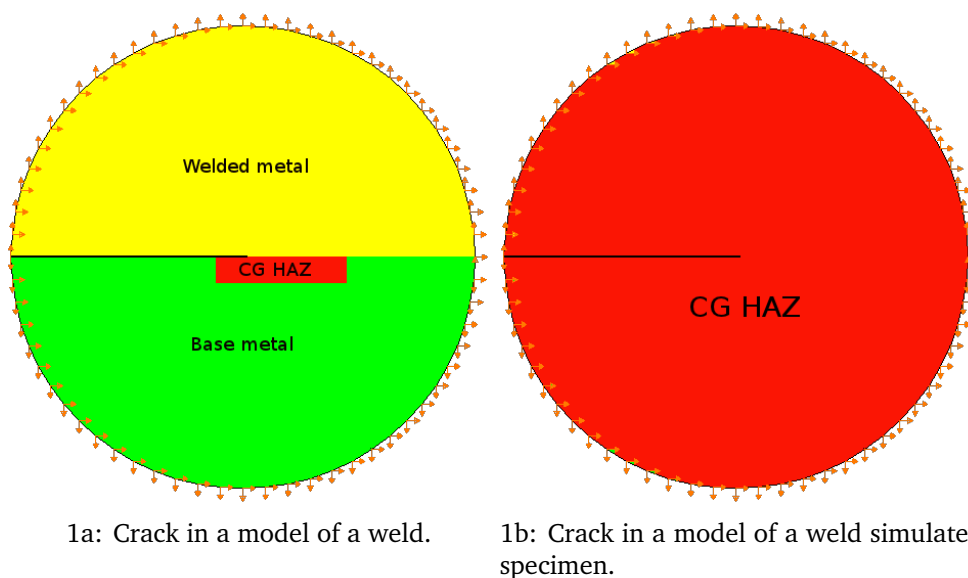
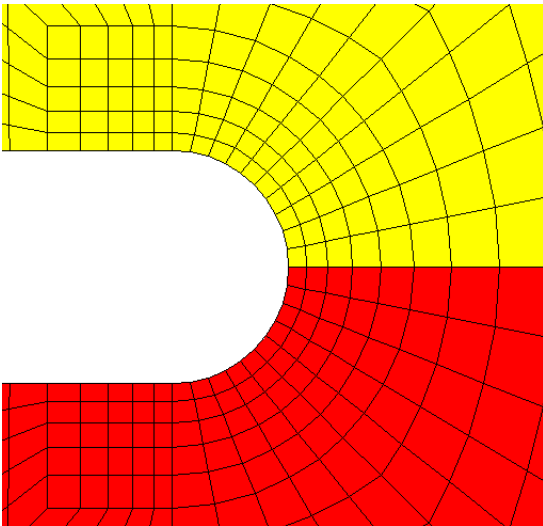


Figure 6.1: Schematic drawing of the MBL model with the displacement field.

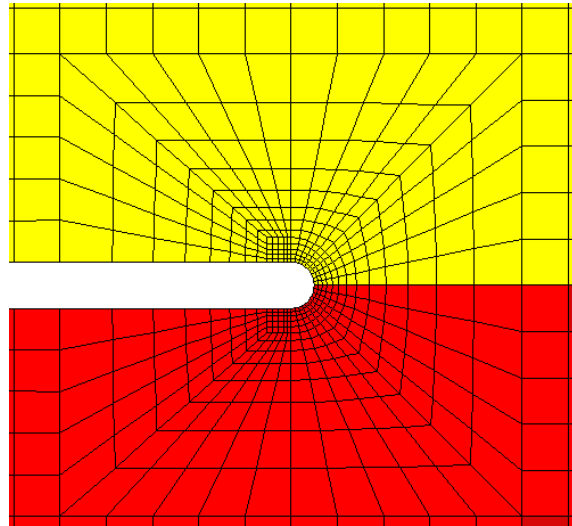
To avoid the boundary influencing the stress field around the crack, the model has a radius of 5000 mm, with the crack in the middle.

To achieve convergence for larger deformation the crack tip is modelled circular. Two different sizes of the crack tip radius have been used, $R = 0.005$ mm and 0.01 mm. This is to test the validity and convergence of the model.

In order to simulate a weld with three different material zones, have a circular crack tip and a large boundary, certain restrictions have to be applied to the mesh. To have a model with a traditional structural mesh, the mesh is divided into five zones. Zone one is the innermost zone around the crack tip. It has to have a circular mesh on the right side of the crack tip. On the left side there is a square mesh to scale up the element size, see figure 2a.



2a: The mesh around the crack tip. Mesh zone 1.



2b: The transition to square mesh. Mesh zone 2.

Figure 6.2: The inner mesh.

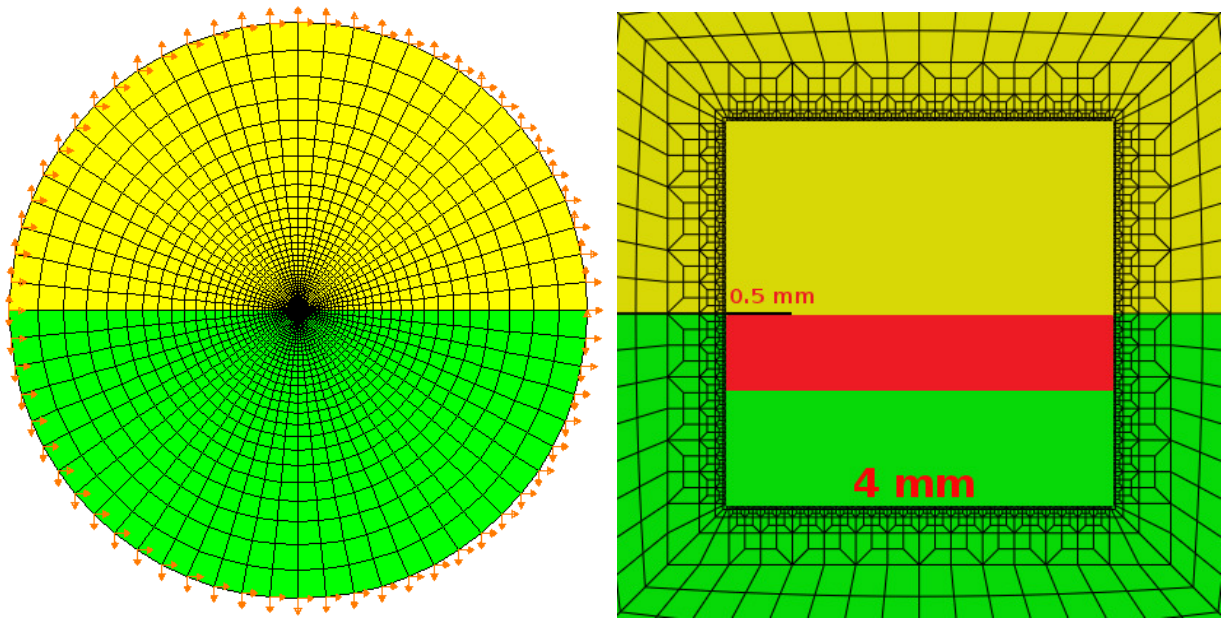
In the area where the HAZ can move a square mesh is needed to get a HAZ with a smooth surface. Because of this a transition zone between the mesh in zone 1 and zone 3, which is the square mesh, is necessary. This transition zone is zone 2, see figure 2b.

Zone 3 consists of elements which are 0.01×0.01 mm. The zone is 4×4 mm, and the crack reaches 0.5 mm into it. Inside this area it is possible to change the thickness and position of the HAZ, see figure 3b.

Between the fine square mesh and the boundary a large scaling of the mesh is necessary. The best way to have this scaling of element size is by a circular mesh, see figure 3a. This circular mesh consists of 64 elements in each circle and is mesh zone 5. Between the fine square mesh and the circular mesh there needs to be a transition zone, to get 64 elements around the square. This is mesh zone 4, see figure 3b.

The mesh consists of four nodes, linear, plane strain, continuum elements. In ABAQUS this is called a CPE4 element [25]. Since the mesh is so fine, a simple element like CPE4 is sufficient to gain enough accuracy. The meshing algorithm is implemented in the code in Appendix B.

The simulations are calculated with ABAQUS standard, and nonlinear geometry is used.



3a: The whole model with mesh.

3b: The square region around the cracktip.

Figure 6.3: The outer mesh.

6.1.1 Scripting

The MBL model has been made by scripting in Python instead of using a GUI, like ABAQUS CAE. This way of generating a model usually takes more time, and requires more of the user. This method was still chosen due to several factors. The main factor was that a structural mesh in the transition zones requires every element to be drawn. The user would then have to draw several thousand lines by hand. By scripting this is done with some code loops. Another important benefit from scripting is that the model can be parametrized, and changes are easily applied. For a more complex model than the MBL, scripting would be harder. An other advantage of scripting is the controllability of the model. If the model is made by the use of a GUI it is hard to control what have been done. Since the script is saved it is easy to change, reuse and study by others.

6.1.2 Validity

Validity tests

For FEM calculations it is important to validate the model and the results. To validate the Weibull stress calculated in the model there is a connection between K_I and the σ_W that can be used. It can be shown that for a self-similar stress field around a crack a relationship between σ_W and J exists [26]:

$$\sigma_W = C_1(m)J^{\frac{2}{m}} \quad (25)$$

Here C_1 is a constant dependent on m . Since $J \propto K_I^2$ equation (25) can be rewritten to:

$$\sigma_W = C_2(m)K_I^{\frac{4}{m}} \quad (26)$$

Also here C_2 is a constant dependent on m . This is just valid for a self similar-stress field, and thus only for the homogeneous reference simulations. By approximating the relation between the numerical Weibull stress and applied K_I with a power function, it is possible, from equation (26), to estimate what m is used. This estimated value of m can then be compared with the m used in the calculation. If they are equal it is a validation of the model and the post processing. This calculation has been done for the two different crack radii and for two values of m , [6].

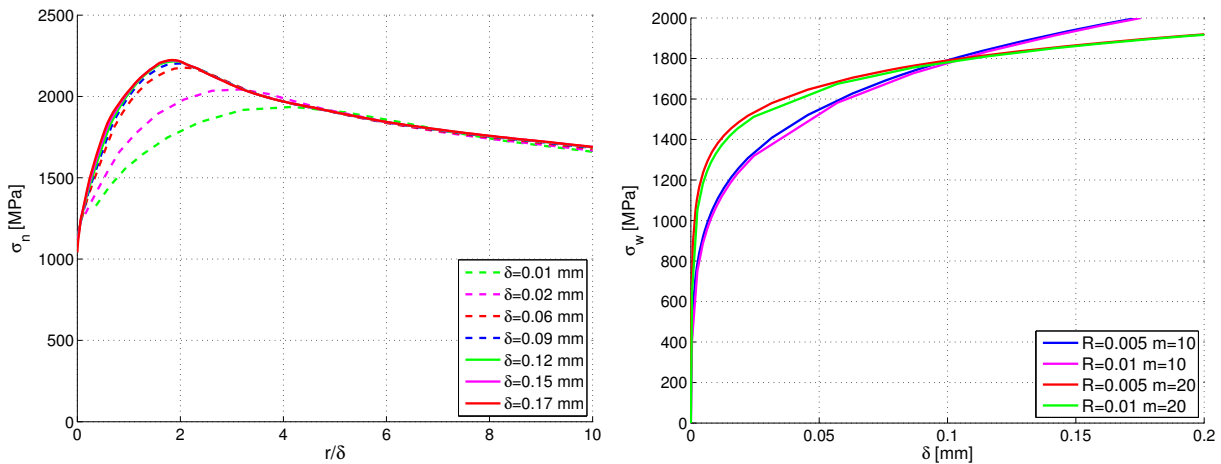
R [mm]	m	Numeric m	m	Numeric m
0.005	10	10.002	20	19.904
0.01	10	10.011	20	19.998

This result show that the model is accurate enough.

Validity area

Since the crack tip starts with a finite radius the simulations are not valid until there has been some deformation.

One way to control if the stress field has converged, is by plotting stress field in the ligament. The normal to crack stress (σ_n) is then plotted against normalized distance r/δ , for different values of CTOD. These curves should be equal if the stress field is self-similar. Figure 4a shows these curves for different levels of CTOD for the reference simulation at -60°C , $T = 0$ and $R = 0.01$ mm. The stress field becomes self-similar when $\delta \approx 0.1$ mm, and have thus converged. This is in accordance with Anderson [9, p.576], see chapter 5.



4a: σ_n plotted against normalized distance form crack tip for $R = 0.01$ mm. 4b: σ_W plotted against CTOD for $R = 0.01$ mm.

Figure 6.4: These figures gives an indication of the validity area to the models with a finite start radius at the crack tip.

For the Weibull stress however, the effect of the wrong stress field is not that large. Figure 4b shows the σ_W plotted against CTOD for a simulation with $R = 0.01$ mm compared to a simulation where the crack tip radius is halved to $R = 0.005$ mm. Here it is clear that there is not any large effect from R on σ_W . This is because the calculation

of σ_W is integrated over a larger area. The effect of some elements around the crack tip having slightly lower stress level is thus negligible. This effect should be highly dependent on m , but even for $m = 20$ the Weibull stress is just slightly lower for the large crack. Since it is only the Weibull stress that is used in this thesis, the results are considered valid for smaller deformations as well.

6.1.3 Parameters

With the MBL model one large parameter study and two smaller series have been performed. These are described in the following sections.

The large parameter study

In the large parameter study the variables have been: material type, position of the crack, size of the crack, mismatch in HAZ, mismatch in WM and T -stress, see figure 6.5.

Material	Crack position	Width of HAZ	Mismatch in HAZ	Mismatch in WM	T-stress	
Theoretic material curve with $\sigma_{0.2} = 450$ MPa	At fusion line	1 mm	1.0	1.0	0	
			1.1	1.1	-0.25 $\sigma_{0.2}$	
			1.2	1.2	-0.5 $\sigma_{0.2}$	
		0.5 mm	1.0	1.0	1.0	0
				1.1	1.1	-0.25 $\sigma_{0.2}$
			1.1	1.2	-0.5 $\sigma_{0.2}$	
Material curve from the rolled steel with yield limit 420 MPa, tested at -60°C	In middle of HAZ	0.25 mm	1.2	1.3	0	
			1.1	1.4	0	
			1.2	1.2	0	
		0.25 mm	1.0	1.0	0	
			1.1	1.1	-0.25 $\sigma_{0.2}$	
			1.2	1.2	-0.5 $\sigma_{0.2}$	

Figure 6.5: Overview of the parameter study.

The parameter study contains of 340 weld simulations, and 8 reference (weld simulated) simulations. The parameter matrix has been increased on basis of the results obtained from the start matrix.

Two different material types are used, they are explained in detail in chapter 6.3.

There have been performed simulations with two different positions of the crack. One where the crack is at FL between WM and HAZ, and one where the crack is in the middle of the HAZ.

Mismatch level is chosen based on empirical experience. For a wide HAZ the yield stress in the HAZ will not be that high as in a small HAZ, [2]. This is due to changes in cooling time, and thus different microstructures. The pre study takes a closer look at

this [6]. The HAZ with a width of 0.5 mm have the largest study, since this turned out to be the most interesting case.

Three different values of T -stresses have been used:

- $T = 0$
- $T = -0.25\sigma_{0.2}$
- $T = -0.5\sigma_{0.2}$

T -stress is calculated relative to $\sigma_{0.2}$ of BM. All the simulations are compared with a reference simulation where $T = 0$.

Side studies

One of the minor parameter studies was done to investigate the effect of undermatch in yield stress for HAZ, as long as there was a vigorous overmatch in hardening. A higher hardening in HAZ than in the two other materials are quite common. Yield stress and hardening in WM and BM were kept constant at $\sigma_{0.2} = 450$ and $n = 0.07$ (black line in figure 12a). Four simulations with different different yield stress and hardening in HAZ were performed:

- $\sigma_{0.2} = 400$ MPa, $n = 0.012$
- $\sigma_{0.2} = 400$ MPa, $n = 0.015$
- $\sigma_{0.2} = 350$ MPa, $n = 0.012$
- $\sigma_{0.2} = 350$ MPa, $n = 0.015$

Four reference simulations were also made for comparison.

In the second minor parameter study the effect of general hardening is investigated. BM still has $\sigma_{0.2} = 450$ MPa, but the hardening is changed between three levels:

- $n = 0.07$
- $n = 0.12$
- $n = 0.15$

The hardening is set equal for the three different materials, but with three different mismatch conditions:

- Even match.
- 10% overmatch in HAZ and 20% overmatch in WM.
- 20% overmatch in HAZ and 30% overmatch in WM.

6.2 3D simulations

Common CTOD specimens of welds can be made in many sizes, but weld simulated specimens can only practically be made with a cross section of 10x10 mm. Because of this some simulations should be made to investigate the effect of specimen size on the Weibull stress. It is the in plane area effect that is interesting as well as any 3D effect from thickness.

To investigate the size effect on the test specimen, four different SENB models were made, all with $a/w = 0.5$ to keep the constraint level constant. Based on the results from

these simulations, it was decided that some simulations should be made to compare the 10x10 mm specimen with SENT specimens. This was to validate the constraint effect on the small SENB specimen.

The middle of the specimen will have the highest stresses, because of a plane strain like condition there. The Weibull stress is thus only calculated for a 1 mm thick element layer in the middle of the specimen, see blue elements in figure 8a. This will make the calculation conservative, and the Weibull stress will be comparable with the MBL result without any correction.

A theoretic material with $\sigma_{0.2} = 450$ and $n = 0.07$ is used in all the 3D simulations, see chapter 6.3. The models are homogeneous and made using ABAQUS cae.

6.2.1 SENB model

The sizes simulated for SENB were:

- $w = 10$ mm
- $w = 25$ mm
- $w = 50$ mm
- $w = 100$ mm

w is the thickness of the quadratic cross section. The length of the specimens is set equal $4w$. The specimens are loaded with three point bending, see figure 2.1, with $a/w = 0.5$. The symmetry of the model is utilised, so only one forth of the model is simulated, see red symmetry planes in figure 8b. The crack tip is 0.01 mm in radius to get convergence in a larger strain area, a validity simulation is run with a radius of 0.005 mm. The mesh is circular around the crack tip with ten elements at the tip, see figure 6a. The mesh transitions into a circular pattern until it ends in a quadratic mesh for the rest of the model, see figure 6b and 6.7. The model with $w = 10$ has an element size of 0.5 mm in the general area, the rest of the simulations have an general element size of 1 mm.

In the out of plane direction there are large elements, see figure 8a, which is done to keep the simulation time down. Closest to the symmetry plane there is an element set who is 1 mm thick. It is from these elements that the Weibull stress is calculated, see blue elements in figure 8a. After this element set, there are ten more planes with elements, with a bias such that the mesh is densest close to the free surface. This is because the stress gradient in this direction, is largest close to the free surface due to a change of stress field from plane stain to plane stress.

The force is applied by keeping the nodes along the bottom of the ligament fixed in y direction, while pushing the nodes in the upper end of the model down. The elements which these two node sets are attached to are elastic, see orange elements in figure 6.7 and 8a. This is to avoid large deformations in the loading areas. These elastic elements are excluded in the Weibull calculation. The loading is displacement controlled.

The elements used are a quadratic eight node element type without reduced integration, in ABAQUS called C3D8. The mesh is generated structured. The simulations are calculated with ABAQUS standard, and nonlinear geometry is used.

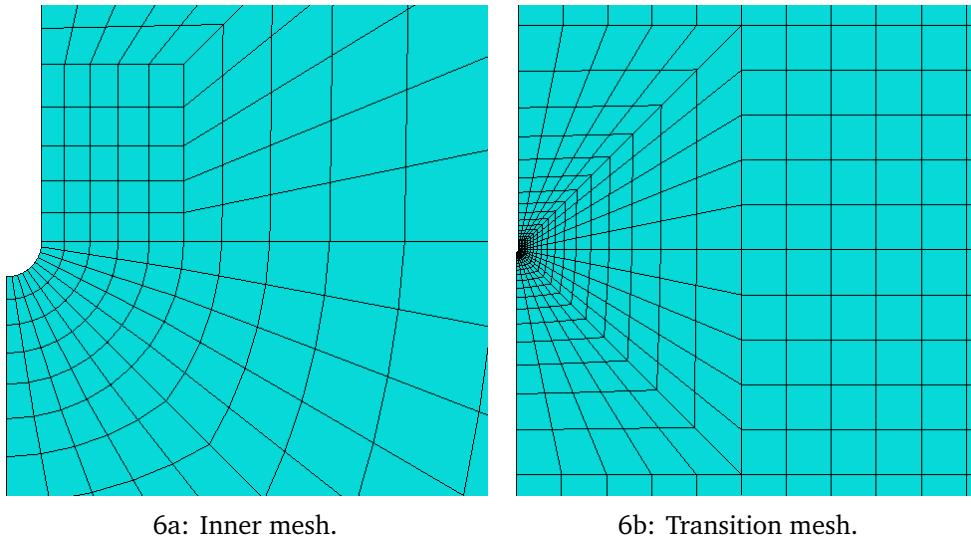


Figure 6.6: Mesh.

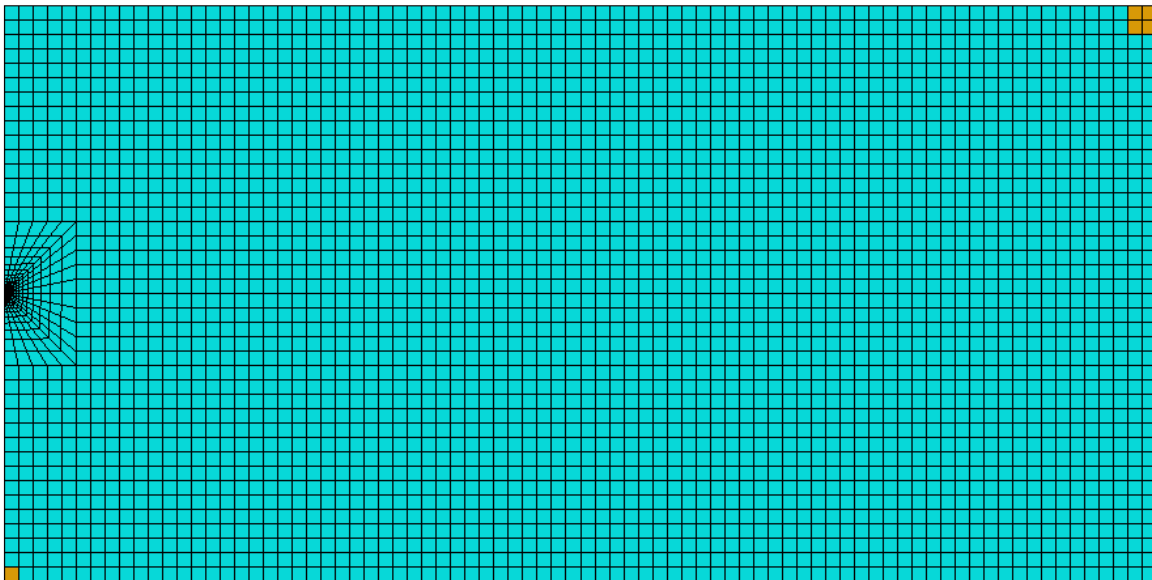
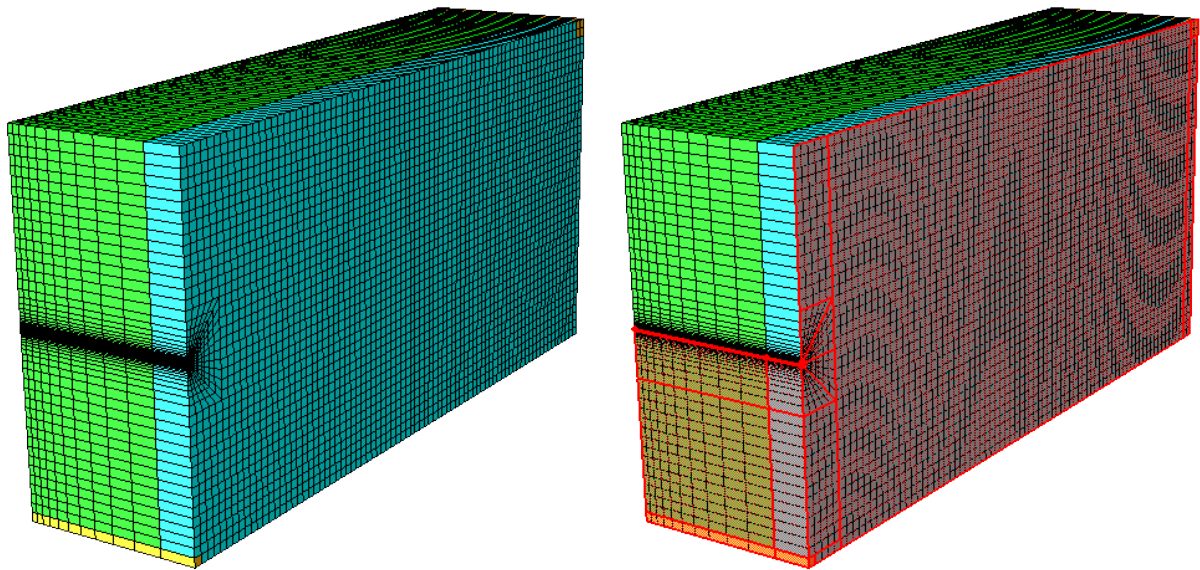


Figure 6.7: The mesh for the model with $w = 10$ mm. Elastic elements are orange.



8a: Elastic elements are orange and Weibull elements are blue.

8b: Symmetry planes are marked red.

Figure 6.8: 3D model with $w = 10$ mm.

6.2.2 SENT model

Simulations with SENT are made to simulate the constraint level of a crack in a real plate. The plate thickness is set to:

- $w = 20$ mm
- $w = 50$ mm
- $w = 100$ mm

Crack depth is set to 5 mm since this is the proposed maximal allowed crack depth [2].

The SENT models have the same meshing algorithm, elements and calculation procedure as the SENB model. The only differences are the loading and where the crack is placed, see figure 6.9. The loading is in the normal to crack plane direction, and all the elements in the end of the specimen are used as boundary. There is no need of any elastic elements since the load is applied over a such large area. The length of the SENT specimens is $10w$. The long length is to avoid boundary effects from the displacement controlled boundary.

6.2.3 Validity

The validity of the mesh is tested by refining the mesh for the model with $w = 10$ mm, since this will have the coarsest mesh. The elements in the general region is then set to be 0.25 mm. One simulation with 0.005 mm radius on the crack tip is also run, to see the influence of this. This simulation also has the fine mesh. In the plot for the Weibull stress the mesh size shows a small effect, see figure 10b. There is no differences for the two simulations with fine mesh and different crack radius.

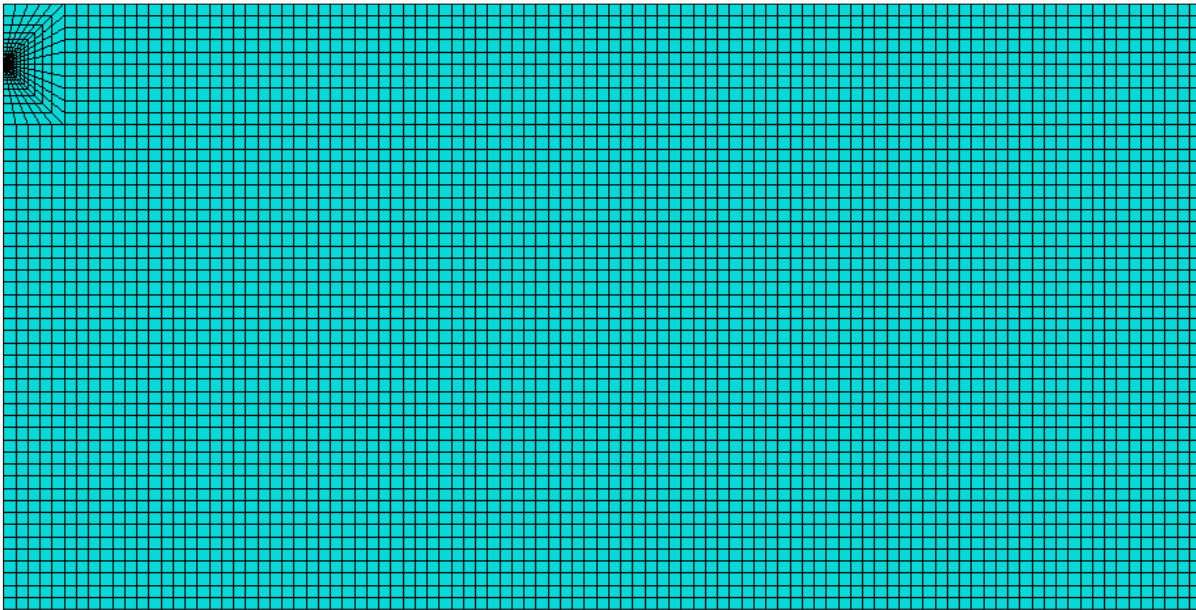
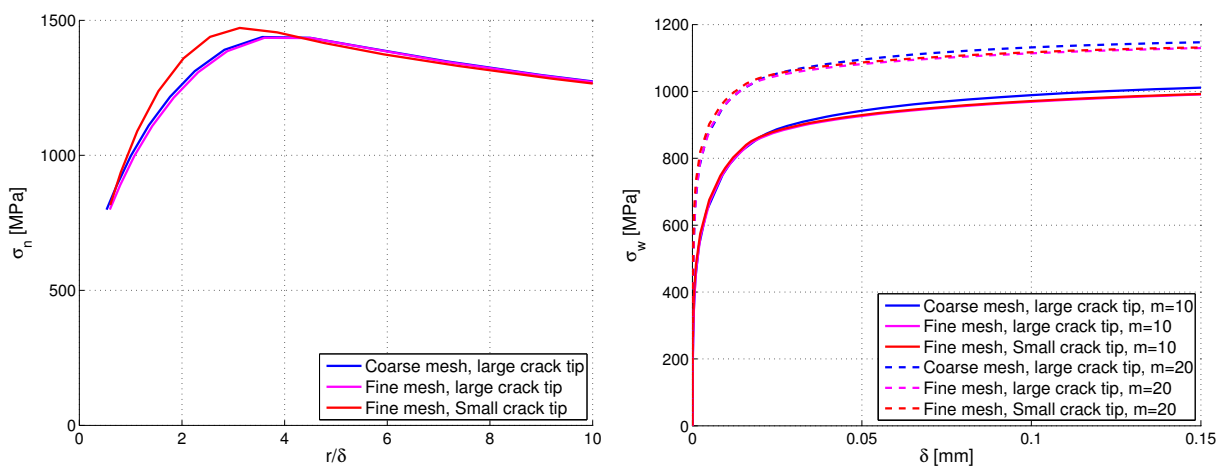


Figure 6.9: The mesh for the sent model with $w = 50$ mm, $a = 5$ mm.

The stress field in front of the crack tip vary some for different crack tip radii, see figure 10a. This is for a CTOD of 0.02 mm. The model with $R = 0.05$ have a higher stress close to the crack tip, since the stress field for the models with $R = 0.01$ have not converged yet. This effect is neglected since it is only the Weibull stress that is used, and the effect here is small, since it integrates over a large area.



10a: Stress field in front of crack with normalized length, for CTOD on 0.02 mm. 10b: Weibull stress plotted against CTOD. Blue and magenta line are on top of each other.

Figure 6.10: Comparison of the specimen with $w = 10$ mm and crack tip radius 0.01 mm (blue line), with a simulations where the mesh is finer (magenta), and a simulation with fine mesh and crack radius of 0.005 mm (red).

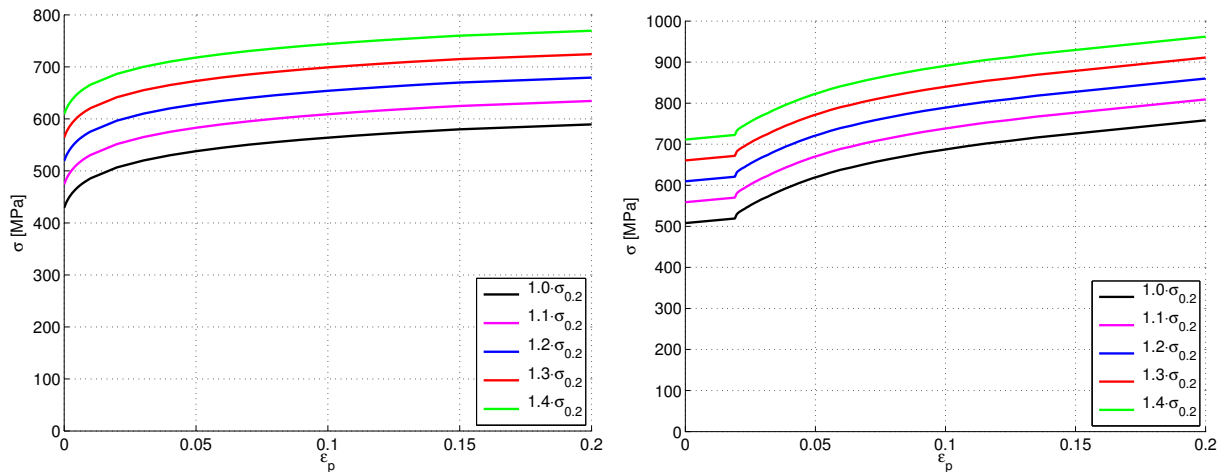
6.3 Materials

In this master thesis two different types of BM have been used, one real and one theoretic. The real material is a rolled steel with yield stress at 420 MPa, tested at -60°C . This material then has a yield stress at 519 MPa. There is a clear Lüders band in this material see figure 11b. The theoretic material is constructed with a power law:

$$\sigma = \sigma_y \left(1 + \frac{\varepsilon_p E}{\sigma_y} \right)^n \quad (27)$$

n is the hardening modulus and σ_y is the yield stress. The materials used in this thesis are defined from $\sigma_{0.2}$ instead of σ_y . An iteration is thus done to find σ_y for the different materials, the code for this is included in Appendix E.

For the large parameter study performed, the materials from figure 6.11 are used. For these simulations the hardening is constant for a material type. BM is always one of the black lines in the figure. For the real BM $\sigma_{0.2}$ is equal 509 MPa and $\sigma_y = 507.8$ MPa. The theoretic BM has $\sigma_{0.2} = 450$ MPa and $\sigma_y = 429.1$ MPa.



11a: Theoretic materials with different yield stress, $n = 0.07$.

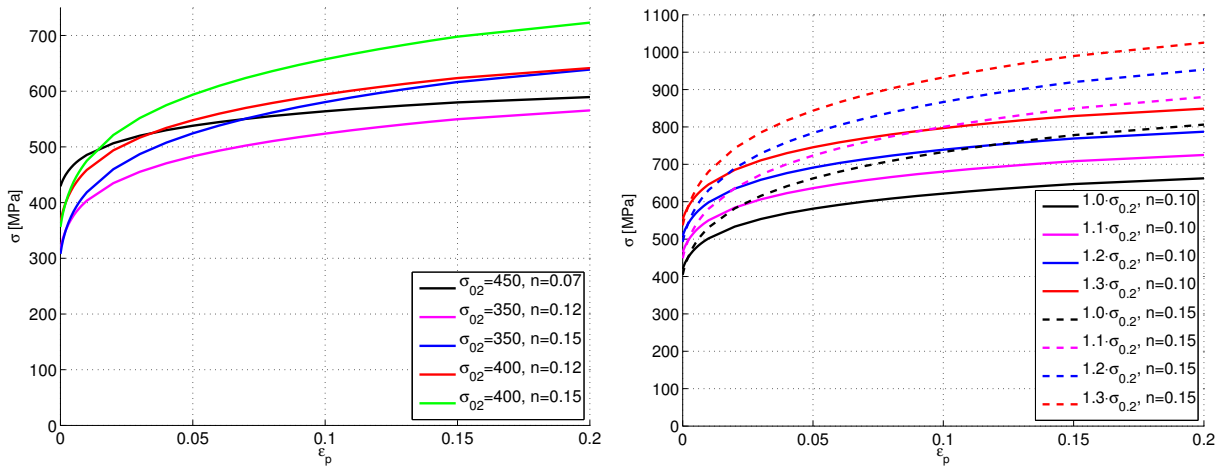
11b: Real materials with different yield stress.

Figure 6.11: Stress-strain curves for different materials.

In the simulations different levels of $\sigma_{0.2}$ mismatch is used, see chapter 6.1.3. The different levels of mismatch are indicated in figure 6.11 and 12b with a factor times $\sigma_{0.2}$ for the BM. Materials with 10%, 20%, 30% and 40% $\sigma_{0.2}$ mismatch are made.

There has also been run some simulations with both hardening and yield stress mismatch. These materials have $n = 0.12$ and 0.15 and $\sigma_{0.2} = 350$ MPa and 400 MPa, see chapter 6.1.3 and figure 12a.

The simulations where the effect of general hardening is studied, the hardening used are $n = 0.07$, $n = 0.10$ and $n = 0.15$. There has also been different levels of mismatch in WM and HAZ, see chapter 6.1.3 and figure 12b.



12a: Materials used to investigate effect of hardening mismatch.

12b: Materials used to investigate effect of general hardening.

Figure 6.12: Stress-strain curves for materials with different yield stress and hardening.

6.4 Post processing

One of the main parameters used in this project is the Weibull stress. The Weibull stress is calculated by equation (15). ABAQUS does not calculate this stress, so it is necessary to do by post processing. The post processing is done by Matlab and a shell script. During simulation ABAQUS saves data for each increment in a text file (filename.dat). This file is then divided into one file for each increment by a shell script. These files are possible to read with Matlab and then calculate and save the desired data to a new file for each simulation, see Appendix D. These files can be used for plotting and study.

To calculate the Weibull stress three variables need to be saved from each element at each time increment. From equation (15) it is clear that the maximal principle stress and the volume for each element must be saved. Since the calculation shall be done only over the plastic area, equivalent plastic strain is also saved to know if an element is in the plastic area. For the SENB simulations pressure is also saved, since elements that yield due to pressure should be excluded. The Matlab script loops through all the elements at each time increment and calculate the Weibull stress for this increment with equation (15). This calculation is done with four different levels of m ($m = 7.5$, $m = 10$, $m = 15$ and $m = 20$). Since the Weibull stress is only used to compare different simulations, the volume V_0 is irrelevant and set equal to 1. For both the MBL model and for the 2D models, the Weibull stress is calculated for a 1 mm thick element layer.

For the 2D models, only half of the model is used to the Weibull calculation, because of the symmetry. From equation (15) the Weibull stress thus needs to be multiplied by $\sqrt[m]{2}$ to double the integration volume.

In addition to the Weibull stress, the CTOD and the stress normal to the crack plane at the ligament is saved. The CTOD is the displacement normal to the crack plane, of the first nodes where the crack tip ends. The stress normal to the crack plane is saved for the centroidal of the element on one side of the ligament, as well as position of the centroid. This data is also rewritten to new files by the Matlab script.

6.4.1 Plots

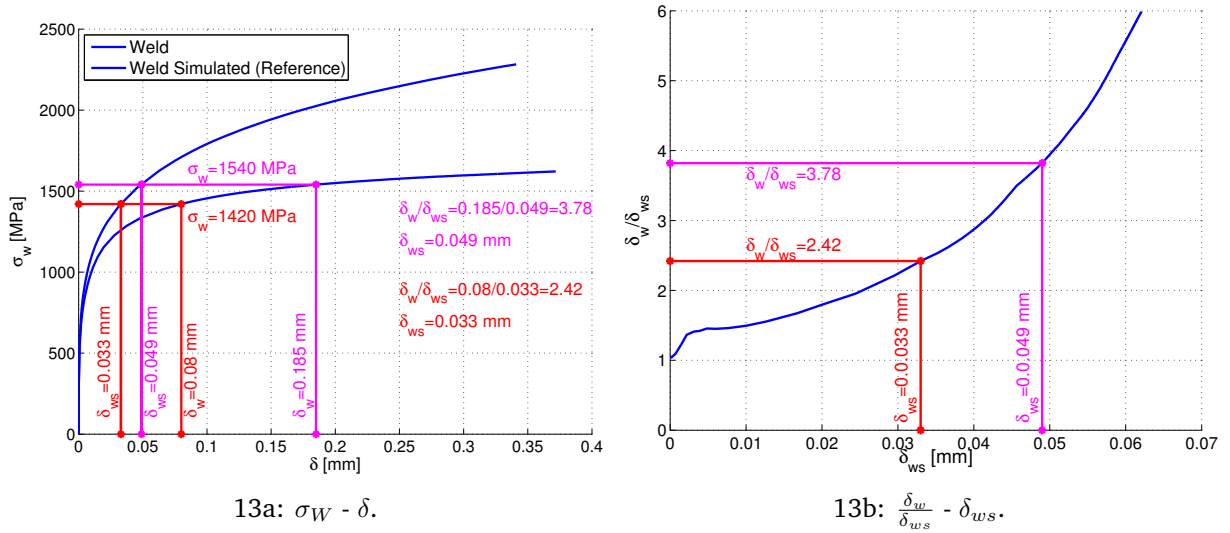


Figure 6.13: Illustration of how the $\delta_w/\delta_{ws} - \delta_{ws}$ plot is made.

To compare the results from the three material, weld, simulation with the reference weld simulated simulations, a special plot is used. This plot assumes Weibull stress as a fracture criterion and uses toughness scaling. The plot assumes that it is equal possibility of fracture in a weld simulated specimen and a weld, when the Weibull stress is equal. To compare how critical different load levels are for the two types of simulations, the CTOD for different Weibull stresses are compared. This is done by dividing the CTOD for a given Weibull stress in the weld, with the CTOD at the same Weibull stress for the reference simulation. The CTOD ratio between the two simulations is then plotted against the CTOD in the reference simulation, see figure 6.13. This plot shows how many times more deformation the weld can take if a certain deformation level is achieved with a weld simulated specimen.

Due to the use of toughness scaling the parameter V_0 and σ_u can be disregarded. The Weibull exponent m is however still important, this is commented on later. Most of the plots are for both $m = 10$ and $m = 20$.

Some of these plots are added in the results and discussion chapters, for the rest see Appendix A.

7 Results of parameter study

In this chapter the different parameters and their effects on the CTOD ratio, between CTOD in a weld simulated specimen and CTOD in a weld, are studied and discussed in detail.

In the plots used there are some special indexes for level of $\sigma_{0.2}$ mismatch in the legends. For instance h20 means 20% overmatch in HAZ compared to BM, and w00 means evenmatch in WM compared to BM.

7.1 Material type

For the simulations with real material all the materials have Lüder bands. This is not realistic since Lüder band do not occur in HAZ. The simulations should therefore have been performed with Lüder band in WM and BM, but with a theoretical material curve for HAZ. Without Lüder band in HAZ the graphs with real material in figure 7.1 would probably have raised faster.

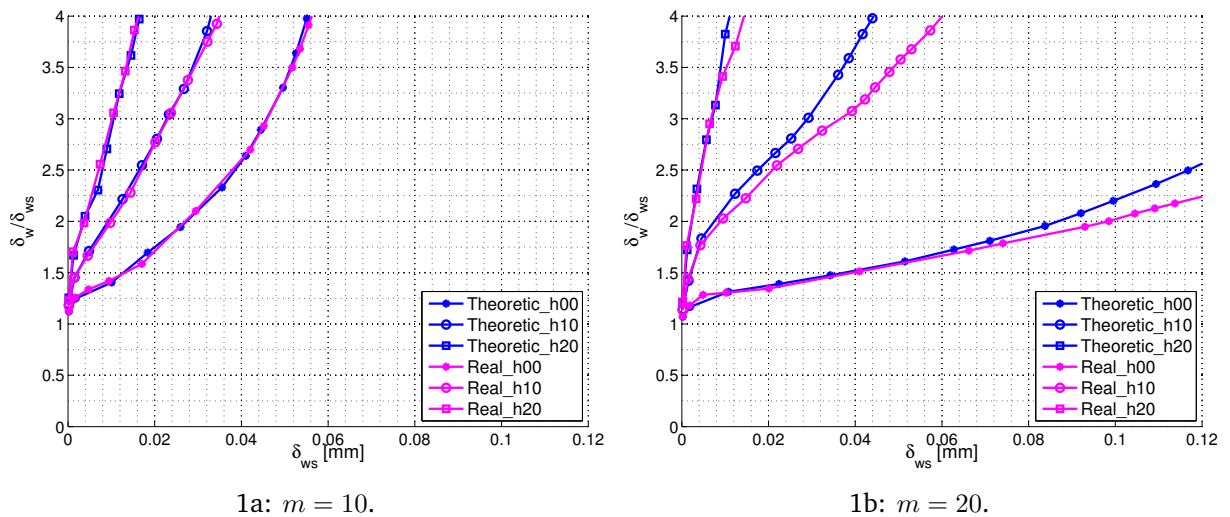


Figure 7.1: Comparing effects of material type for three levels of mismatch in HAZ. Blue line is theoretic material and magenta is real material. $T = 0$, crack at FL, HAZ thickness 0.5 mm and evenmatch between WM and BM.

One conclusion can however be drawn from these simulations: The general yield stress does not contribute significantly to the results. The difference between the results for the two material types is quite small, see figure 7.1. This is even though the yield stress in the real material is higher, $\sigma_y = 507.1$ MPa, than for the theoretic material, $\sigma_y = 429.1$ MPa. The hardening is almost equal, but with a Lüder band in the real material, see figure 6.11

7.2 Placement of the crack

There have been performed simulations with the crack at FL and with the crack in the middle of the HAZ. Figure 7.2 is comparing the position of the crack for different

mismatch levels. From the curves there is a clear trend that to position the crack in the middle of the HAZ is worse than at FL. This is perfectly natural since it is only the stress field of the HAZ that is considered. Placing the crack in the middle of the HAZ will give more high stresses in the HAZ, and thus a higher Weibull stress.

For a real crack this effect will probably be the opposite [5]. This is because the material is usually most brittle and contains most inclusions close to FL. The simulations with the crack on FL will thus be used mostly in the rest of this thesis, since this is the most critical case in a real weld.

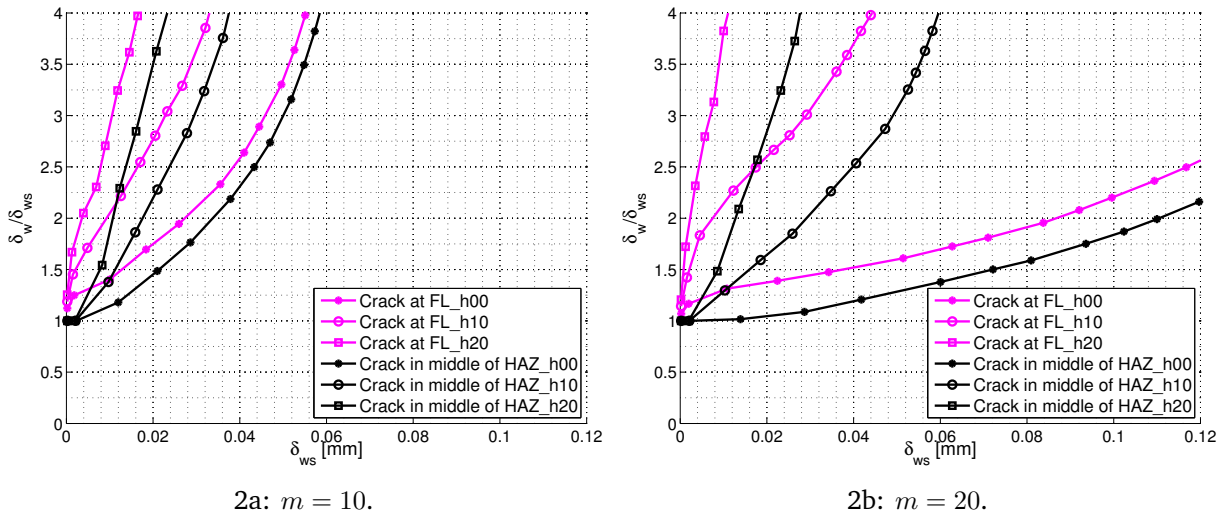


Figure 7.2: Comparing effect of crack position for three levels of mismatch in HAZ. Black line is crack at FL and magenta is crack in middle of HAZ. $T = 0$, theoretic material, HAZ thickness 0.5 mm and evenmatch between WM and BM.

When comparing figure 2a and 2b the black and purple lines are further apart for $m = 20$ than for $m = 10$. This can be explained by how m is influencing the Weibull stress, see chapter 3.4.2. For $m = 20$ the high stresses are important and the position of the crack has a larger effect. $m = 10$ makes the area of the HAZ more important and therefore less difference since this is kept the same.

There is also a large difference in the behaviour of the general curves dependent on m . This is discussed later, see chapter 7.8.

7.3 Effect of size

One of the main parameters that makes weld simulations more brittle than welds, is that the brittle area of the HAZ is much larger in a weld simulated specimen. Since brittle fracture is a statistical phenomena the size of the brittle zone is important. From figure 7.3 this appears clearly. The CTOD ratio between the two models is increasing with smaller HAZ size. The lines with dots on are for simulations with a homogeneous material, these lines shows therefore only the size effect.

A similar study was performed in the pre study, but with mismatch in the materials and thus without the effect of size only.

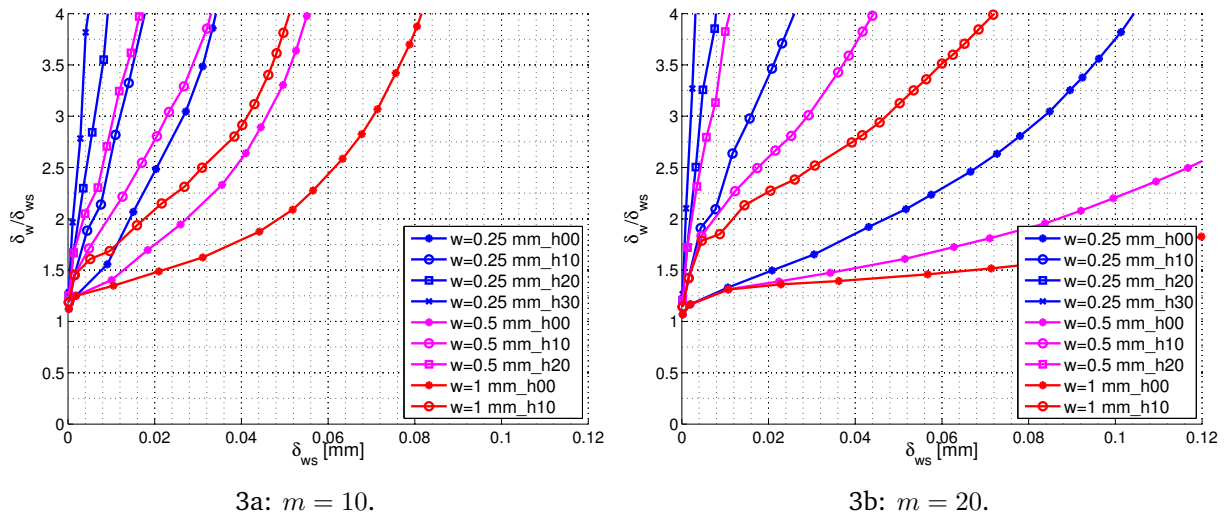


Figure 7.3: Comparing effect of HAZ size for three different levels of HAZ mismatch. Blue line is HAZ width 0.25 mm, purple 0.5 mm and red 1 mm. $T = 0$, theoretic material, crack at FL and evenmatch between WM and BM.

For real welds the fracture rarely appears far away from FL [2], a 1 mm brittle HAZ is thus quite unrealistic. A HAZ width of 0.5 mm will thus mainly be used further in this thesis. The 0.5 mm HAZ will always be conservative compared with the 0.25 mm HAZ for a realistic CTOD. The HAZ size can also be estimated with Rosenthal equation [27], this was performed in the pre study [6].

7.4 T -stress at MBL model

In the pre study [6] a parameter study of the T -stress effect on the Weibull stress was performed. The Weibull stress for these simulations has been compared with reference simulations with the same T -stress level. Since the weld simulated specimens are tested with SENB05 the T -stress is equal zero. The different weld specimen should thus be compared to a reference simulation with $T = 0$.

This comparison is done in the parameter study of this thesis. The result is very different from the result obtained in the pre study. In the pre study it was concluded that the T -stress had almost no effect. When comparing reference simulations with $T = 0$ the effect is large, see figure 7.4. This indicates that it might be possible to make an addition in critical CTOD for a weld simulated specimen, compared to a real weld, based on the geometry effect. This is further discussed in chapter 8.2.

The large effect can be explained by a difference in stress state in front of the crack tip, between the welded specimens and the reference simulation. The reference simulation will have a much higher level of hydrostatic stress, and thus obtain higher principal stress before yielding. This will imply a higher Weibull stress since it is calculated from the principal stress. The effect will be largest for $m = 20$ since this is extra sensitive to the maximal principal stress.

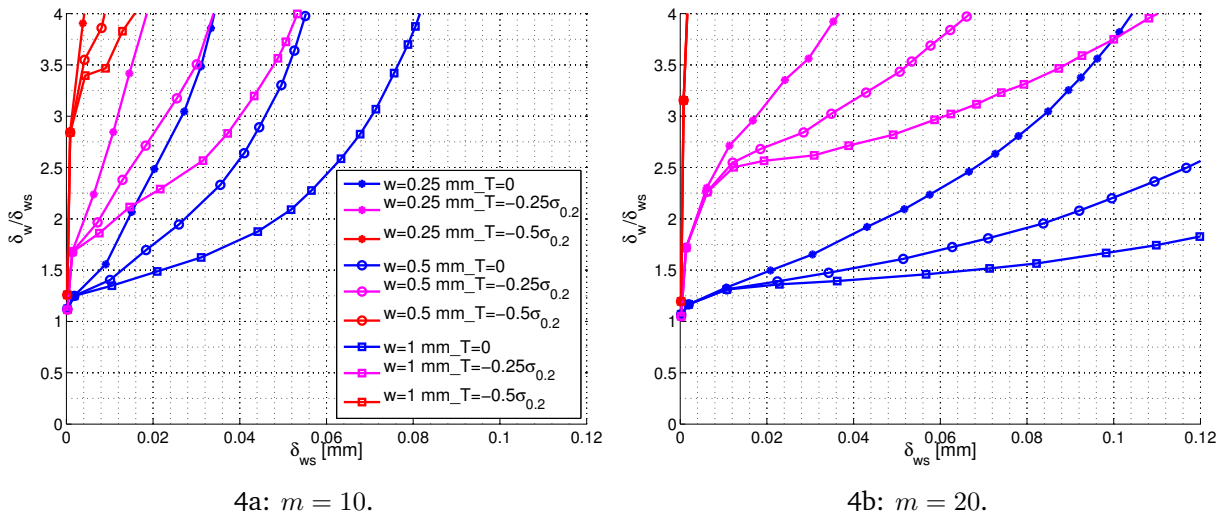


Figure 7.4: Comparing the effect of T -stress for three different HAZ thicknesses. Blue line is $T = 0$ MPa, purple $T = -0.25\sigma_{0.2}$ and red $-0.5\sigma_{0.2}$. Theoretic material, crack at FL and evenmatch between WM, BM and HAZ.

7.5 Mismatch in yield stress

One of the key variables in this thesis is the level of mismatch between the different materials. The CTOD ratio is highly dependent of the mismatch properties, see figure 7.5. In this figure both WM and HAZ have different levels of mismatch compared with BM. The blue line with dots is a homogeneous simulation where only the sampling size effect occurs.

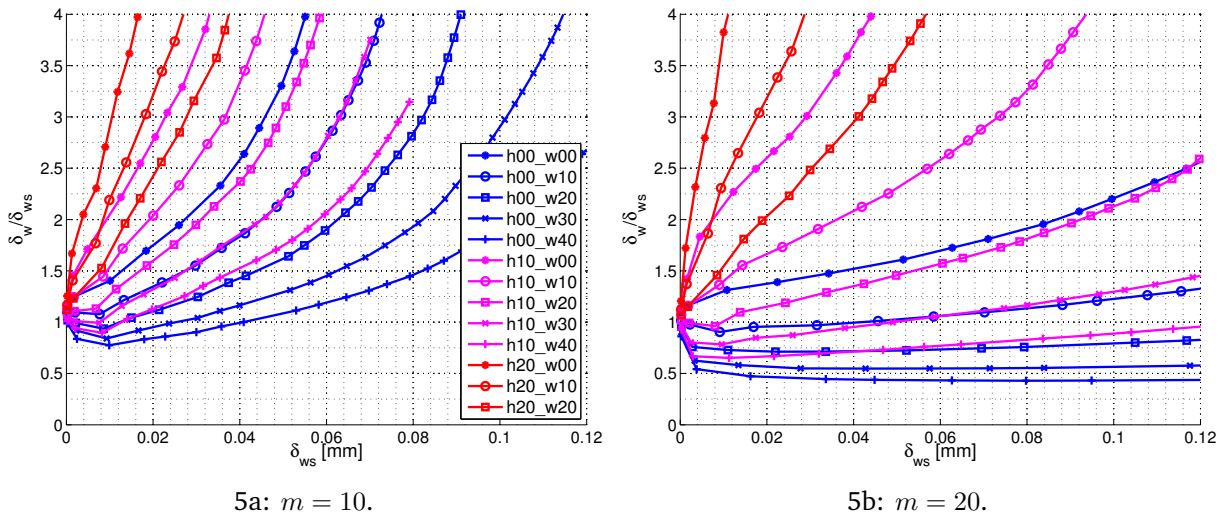
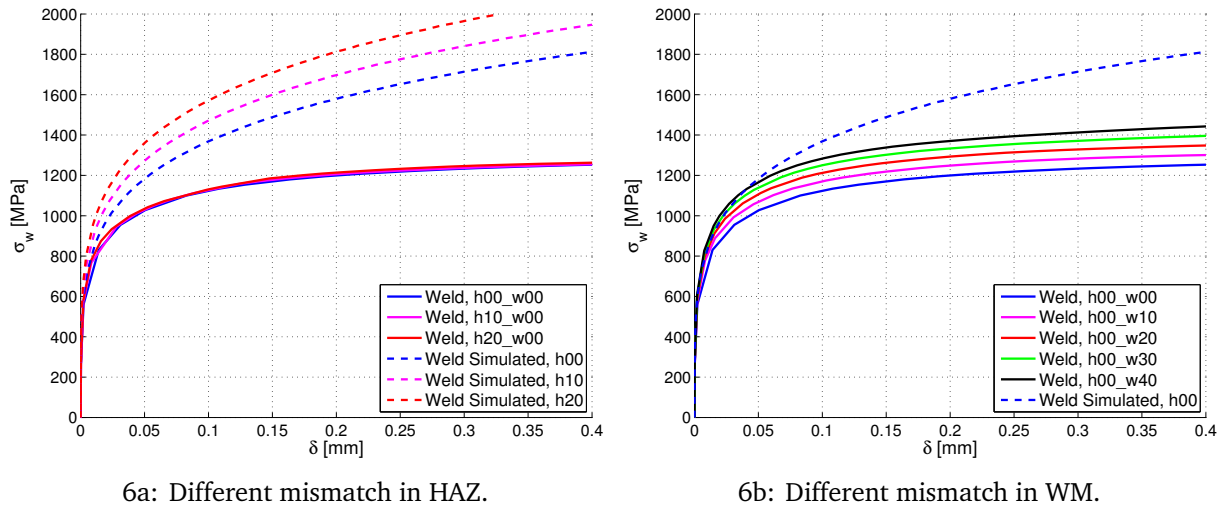


Figure 7.5: Different mismatch properties. Colour indicate level of mismatch in HAZ and marker indicate level of mismatch in WM. Mismatch is relatively to BM. These simulations are with the theoretic material, crack at FL, crack width 0.5 mm and $T = 0$.

The trends of figure 7.5 are quite clear. High mismatch in HAZ make the ratio between CTODs high, while high mismatch in WM does the opposite. This is as expected

from chapter 4.1, and in accordance with the results of Ø. Ranestad [7]. For a HAZ with high overmatch to both BM and WM, the two soft materials will make the HAZ softer, since they will yield first. For the reference simulation without this soft material, yielding will occur later, and the stress level will be higher. This will cause a higher Weibull stress in the homogeneous material, and larger probability for fracture, see figure 7.6.



6a: Different mismatch in HAZ.

6b: Different mismatch in WM.

Figure 7.6: Comparison of Weibull stress in weld and weld simulated specimens. These simulations are with the theoretic material, crack at FL, crack width 0.5 mm, $T = 0$ and $m = 10$.

In figure 6a different mismatch levels in the HAZ are compared for the weld and the weld simulated analyses. An interesting effect is that the Weibull stress in the welded specimens is almost independent of the yield stress in the HAZ. This is because the softer materials, WM and BM, lower the stress field so much that the yield stress in the HAZ almost has no influence. It is thus just the Weibull stress in the reference simulation that makes the difference for the graphs in figure 7.5. This means that a real weld will break at almost the same CTOD for different yield stresses in the HAZ. A weld simulated specimen with high yield stress will, on the other hand, break at much lower CTOD than one with lower yield stress.

For different levels of mismatch in WM, see figure 6b, it is the Weibull stress in the welded specimens that changes. This simulations uses the same reference simulation (dashed line), since the yield stress in the HAZ is constant. The high yield stress in WM will here increase the stress level, and prevent yielding. The yield stress in WM is therefore an important factor regarding brittle fracture, since it actually changes the critical CTOD for the weld.

There are some simulations where the CTOD ratio is below 1, see figure 7.5. These simulations either have undermatch between HAZ and WM, and no overmatch between HAZ and BM, or a larger overmatch than 10% between HAZ and WM. A CTOD ratio below 1 means that the weld has higher probability of fracture than the weld simulated one. This even though the brittle area is smaller.

7.6 Mismatch in hardening

Usually the hardening is higher in HAZ than in WM and BM. A small parameter study was therefore performed where there was undermatch in yield stress for the HAZ, but higher hardening. BM and WM had even match, in both yield stress and hardening, $\sigma_{0.2} = 450$ and $n = 0.07$.

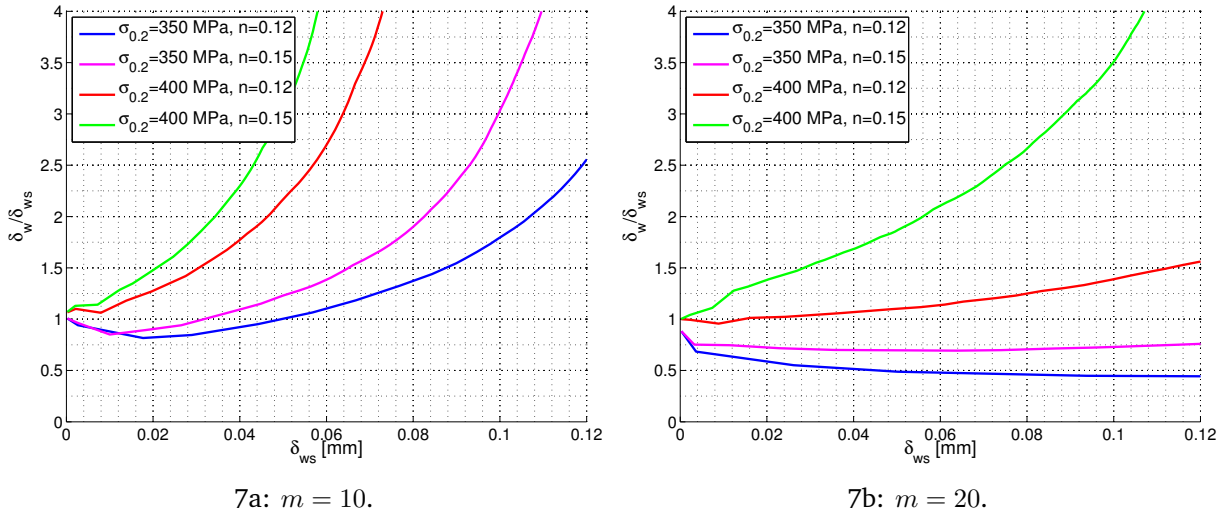


Figure 7.7: Different hardening and yield stresses for HAZ, even match for WM and BM. These simulations are with the theoretic material, crack at FL, crack width 0.5 mm and $T = 0$.

From figure 7.7 it is clear that both yield stress and hardening are important factors when comparing welds and weld simulated specimens. The yield stress is however the dominating factor for relatively small CTODs. It is thus only the simulation with $n = 0.15$ and 50 MPa undermatch which gives good result for both m values. These simulations show that it is very important to not have yield stress undermatch in the HAZ. Hardening overmatch in HAZ with even match or better in yield stress, would most likely give a very positive effect. This is not studied here since evenmatch in hardening is most critical, and thus conservative.

7.7 General hardening

Since hardening may have a large influence on the CTOD ratio between weld simulated specimens and welds, a series of simulations with different hardening properties was performed. All the simulations have evenmatch in hardening, but different mismatch properties for WM and HAZ.

Figure 7.8 shows the effect of different hardening in the materials. From the figure it appears clearly that a high general hardening lowers the CTOD ratio. This effect may arise from a change in the stress field when the hardening increases. For a higher hardening the stress peak will be higher and closer to the crack tip, see figure 7.9. Higher stress level will lead to higher Weibull stress, in both weld and weld simulated specimen, which may cause a change in the CTOD ratio.

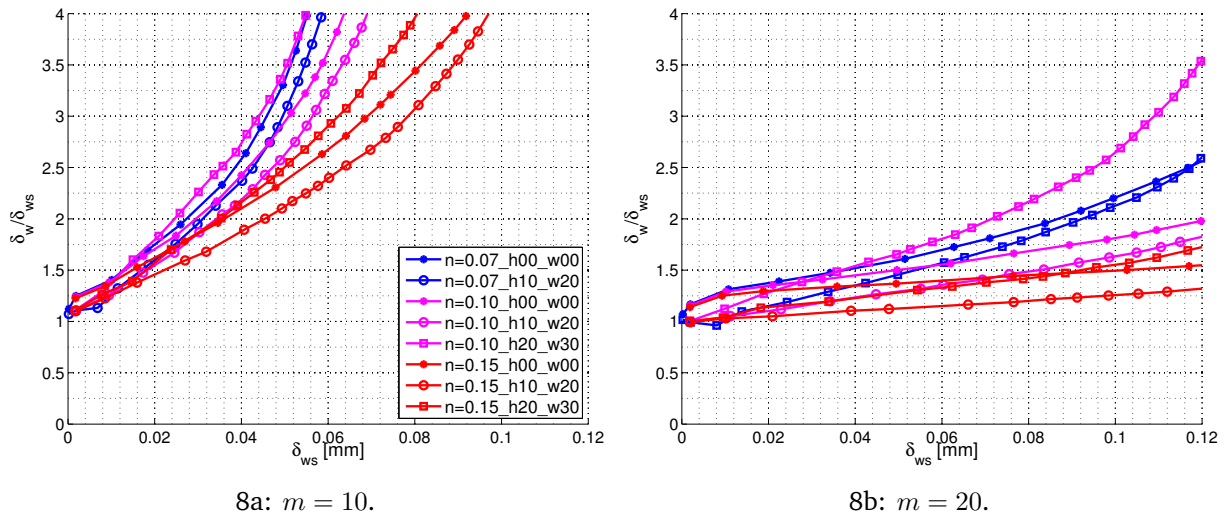


Figure 7.8: Different yield stress mismatch for WM and HAZ, equal hardening for all materials. These simulations are with the theoretic material, crack at FL, crack width 0.5 mm and $T = 0$.

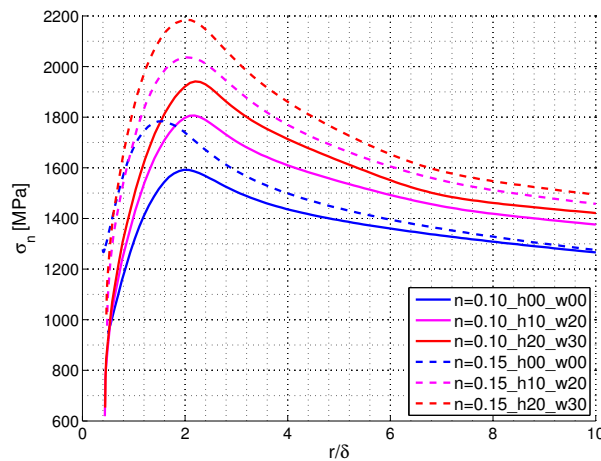


Figure 7.9: Stress normal to crack plane in front of crack with normalized length, $\delta = 0.1$ mm. $T = 0$, crack at FL and theoretic material.

A hardening of $n = 0.15$ for WM and BM is unlikely to occur [2], and these simulations was only performed to see the qualitative effect.

7.8 The effect of m

From the pre study [6] it was concluded that the Weibull parameter m had almost no effect. From the plots made here this conclusion is clearly wrong. The plots from the pre study are however not wrong. In the pre study all the simulations were done with high overmatch in the HAZ, which implies a fast growing CTOD ratio. When studying figure 7.3 and 7.5 it is clear that as long as the CTOD ratio is growing fast, the result is almost independent of m . For a slowly growing ratio, m has larger influence.

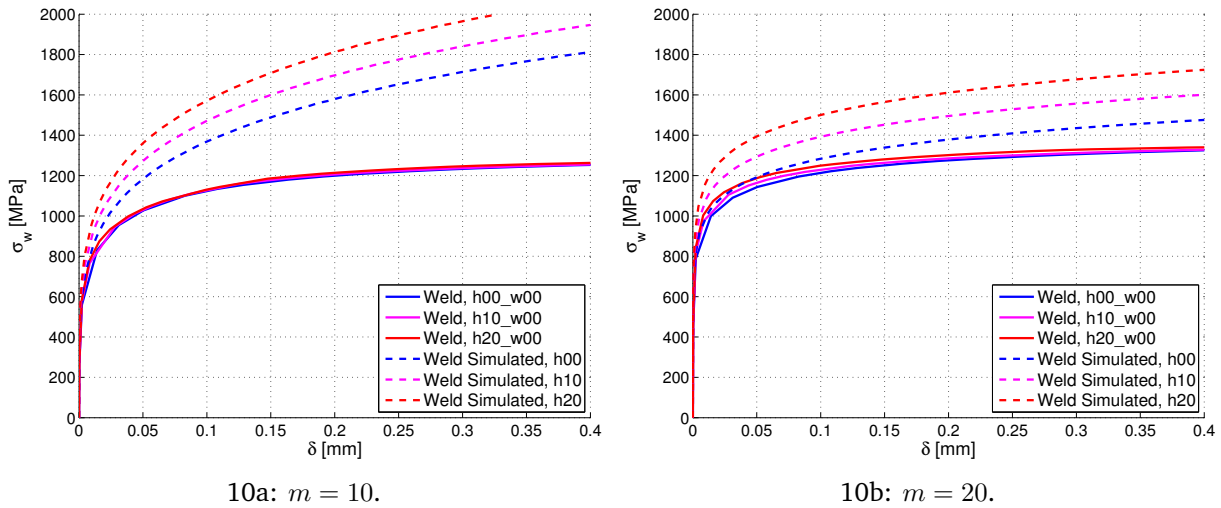


Figure 7.10: Comparing the effect of m on the Weibull stress, for different mismatch level in the HAZ. These simulations are with the theoretic material, crack at FL, crack width 0.5 mm and $T = 0$.

This effect can be explained by the graphs in figure 7.10. This graphs are for the same simulations as the purple lines in figure 7.2. In figure 7.2 the purple line with dots are dependent of m , but the two other purple lines are almost independent. In figure 7.10, however, m has a large influence on all the weld simulated simulations (dashed lines). The reason why just one of the purple lines is affected by m is due to the shape of the Weibull stress curve. All the Weibull stress graphs are raising fast until a turning point where they flatten. The weld specimens are then converging against a threshold Weibull stress. The shape and the CTOD before the turning point is almost independent of m , and m only affects the Weibull stress at the turning and threshold point. This implies that as long as the weld simulated turning point is higher than the threshold point of the weld, the CTOD ratio is almost independent of m , since how much higher the turning point is, is irrelevant.

The Weibull stress in the weld simulated specimens are often higher for $m = 10$ than for $m = 20$. The simulations with $m = 20$ will thus reach the point where the threshold of the welded specimen is higher than the turning point of the weld simulated specimen, earlier. This will then make a large difference in plots like figure 7.2.

m is therefore important for the Weibull stress curve, and the probability for fracture. For comparison of MBL models $m = 20$ is the most critical, and should be used as design reference to avoid estimating m for the material.

8 Discussion

In chapter 2.2.2 four different effects that may make CTOD testing of weld simulated specimens more brittle than real welds were mentioned. In this chapter it is discussed if it is possible to make a general relationship between weld simulated CTOD and CTOD for a structure, and take these effects into account.

8.1 Out of plane sampling

For weld simulated specimens the whole crack front will be in the brittle microstructure. Statistically this implies, from equation (24), that ten specimens with $w = 10$ mm are the same as a test with a 100 mm long crack front. Practically this will however not be exact. A test specimen has free surfaces on both sides, and will therefore not have plane strain all the way through. The area without plan strain, will have a lower probability for fracture and can thus not be used in the calculation. How large the area without plane strain is, is not calculated here, but this area should be subtracted. Testing of several specimens give however an increased probability for fracture. In real structures a 100 mm long crack where the the whole crack front is in a brittle microstructure will rarely appear. Often only a small part of a crack will hit the brittle microstructure, testing of ten specimens will thus make the testing conservative.

8.2 Constraint and specimen size

Chapter 7.4 concludes that there is a large positive effect from geometry constraint in a real weld compared to a weld simulated specimen. Before these results are used, there should be investigated if the small size of the weld simulated specimen have any affect. A weld simulated specimen can only have a cross section of 10x10 mm, due to the oven. This implies certain restrictions to how large deformation the sample can take, and still be valid.

In chapter 3.5 there are listed two requirements to the size of a fracture mechanics test specimen. The first requirement, equation (19), is based on linear elastic behaviour and is very stringent. This equation demands, depending on m and σ_{ys} , that a specimen needs to be in the size area of 50 – 400 mm, for $\delta = 0.1$ mm to be valid. This requirement is too stringent since there will be plasticity around the crack tip, and a plastic requirement should be used, [9, p.311]

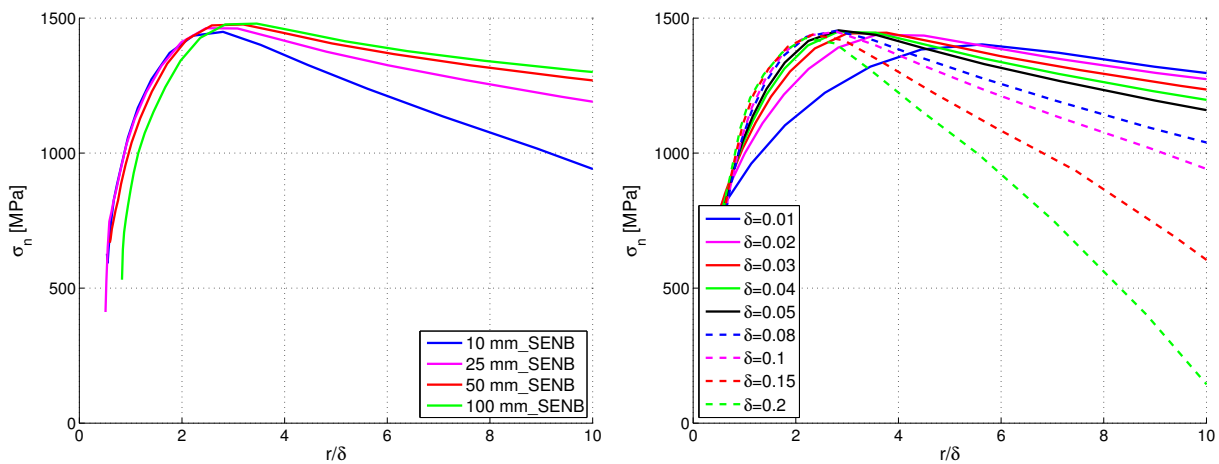
The second requirement, equation (21), is based on the J -integral, and takes some plasticity into account. This requirement treats the size of the ligament, and is only dependent on δ and the constant m . The weld simulated test specimens, with $w = 10$ mm and $a/w = 0.5$, should from this equation be valid until $\delta = 0.08$ mm for $m = 2$ and $\delta = 0.11$ mm for $m = 1.5$.

8.2.1 SENB comparison

From the second requirement there should not be any large size effect in the weld simulated specimens, as long as the deformation is small. To control this there has been run a simulation series of SENB05 specimens with different heights, see figure 8.2

and 1a. From these figures it is clear that the small specimens deviate from the larger samples quite early. The Weibull stress for $m = 10$ deviates already at $\delta = 0.02$ mm. At the stress field plot for $\delta = 0.1$ mm, there is a clear drop in stress already at $r/\delta = 4$, for the specimen with $w = 10$ mm. Figure 1b shows the stress in the ligament of the simulation with $w = 10$ mm, for different CTODs. The lines should ideally be equal, for $r/\delta > 3$, if the ligament size was large enough. The figure shows that the stress field is dropping already for $\delta = 0.02$ mm. This concludes that the size of the ligament is influencing the stress field for $\delta > 0.02$ mm, and that the requirement in equation (21) is not strict enough.

On the left side of figure 1b the lines are deviating for small CTODs. This is an effect of the large crack tip radius.



1a: Different w for SENB specimens.

1b: SENB with $w = 10$ and different sizes on the SENT specimens.

Figure 8.1: Stress field in front of crack with normalized length, $\delta = 0.1$ mm.

The stress field for the simulation with $w = 10$ mm has almost the same stress peak as the other specimens, see figure, 1a. The stress is however dropping more rapidly for increasing r . This influence of the ligament size is thus affecting the Weibull stress with $m = 10$ the most, since the size of the stress field is important. For $m = 20$ the height of the stress peak is most important, and the effect is therefore smaller even though it appears at the same CTOD, see figure 2a and 2b.

8.2.2 SENT comparison

Since the validity area was so small for the SENB05 specimen with $w = 10$, some simulations were performed to compare this simulation with lower constrained specimens. SENT specimens with a crack depth of 5 mm was made in different sizes. Figure 8.4 and 8.3 compare the small SENB05 specimen with different SENT specimens.

Before studying figure 3a a Weibull artefact should be explained. The green line for the SENT 100 mm specimen behaves quite different than the others. For a certain level of CTOD the Weibull stress starts increasing to a higher stress than expected. This is due to a phenomena that the Weibull stress cannot handle. Since the crack

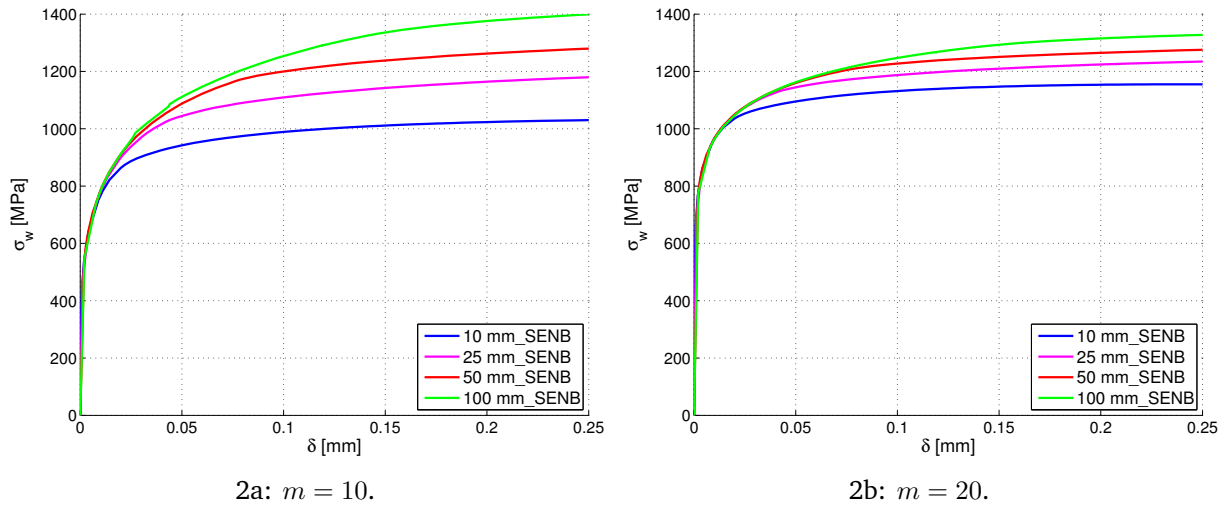


Figure 8.2: Weibull stress plotted against CTOD for different w on the 3D SENB specimens.

depth of this simulation is so small compared to the ligament, a lot of force need to be applied to reach a large CTOD. Because of this the specimen reach global yielding before a large CTOD occurs. The global stress level is just above the yield limit, but since the integration area for the Weibull stress becomes so large, these low stresses get an influence. This only happens for $m = 10$, since area has so little influence for larger m . Also for the specimen with $w = 50$ this effect occurs, but not in the same scale.

This effect is not realistic and should not be taken into account. The Weibull stress for $w = 100$ mm and $w = 50$ mm is thus only valid until $\delta \approx 0.04$ mm, for $m = 10$.

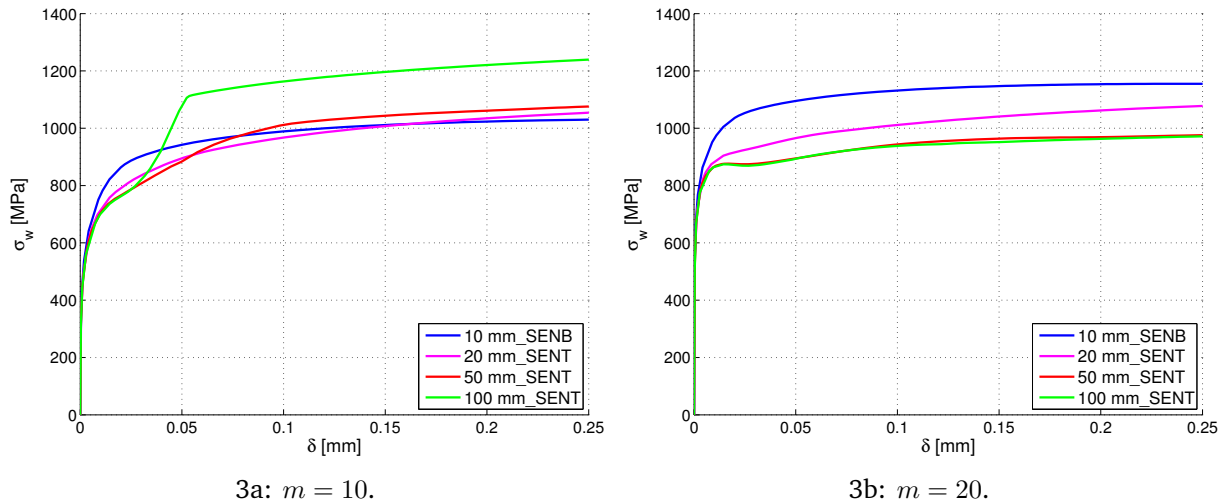


Figure 8.3: Weibull stress plotted against CTOD for different w on the 3D SENT specimens.

When comparing SENB05 with SENT specimens, the Weibull stress in the SENB05 specimen should be higher, due to higher constraint level. For the SENB05 with $w = 10$ mm this effect does however not occur, see figure 3a. This is because the small ligament influences the stress field when $w = 10$ mm, as mentioned above. The ligament size only affects the lower region of the stress field and the stress peak is still higher in

the SENB specimen, see figure 8.4. This implies that the ligament effect only occurs for $m = 10$. The constraint effect is thus normal for $m = 20$, see figure 3b.

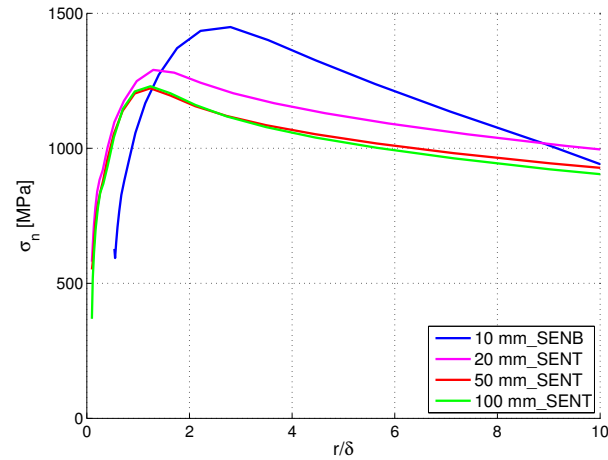
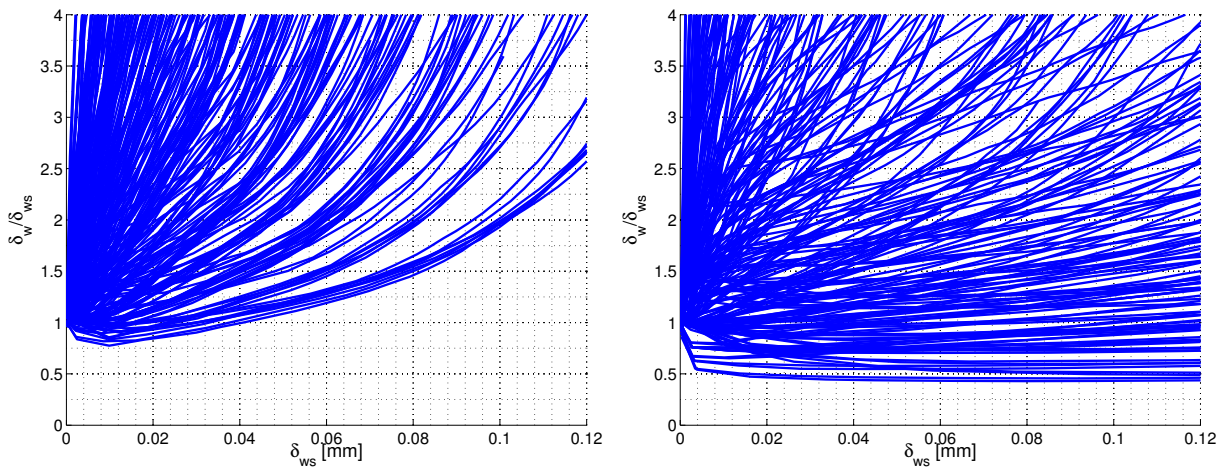


Figure 8.4: Stress field in front of crack with normalized length, $\delta = 0.1$ mm.

This implies that the CTOD from the weld simulated specimens gets no positive effect because of constraint. There is a positive effect for $m = 20$, but this cannot be counted for, since an general constraint benefit should be valid for both $m = 10$ and $m = 20$.

8.3 Mismatch and size effect

From the results of the parameter study it is clear that the size of the the HAZ and mismatch have large influence on the CTOD ratio between a weld simulated specimen and a real weld. These are the two main factors that make weld simulated specimens more brittle than welds. Figure 8.5 shows the effect of size and mismatch by plotting all the simulations in the parameter study.



5a: $m = 10$.

5b: $m = 20$.

Figure 8.5: All the simulations in the large parameter study.

From figure 8.5 it is clear that there are a lot of simulations where the weld simulated specimens are much more brittle than the weld. It is however also some simulations where the difference is smaller, and even some where the weld is more brittle. To be able to make a general relationship between the CTOD for a weld and the CTOD for a weld simulated specimen, it is thus important to establish some realistic requirements to the weld:

- At least 10% overmatch in HAZ compared to BM.
- Not more than 10% undermatch in HAZ compared to WM.
- Maximal brittle HAZ thickness of 0.5 mm.

With these three requirements a much more consistent trend occurs, see figure 8.6. It is now possible to propose a first suggestion to a lower design line, for size and mismatch effects. This line establishes a conservative relationship between a weld and a weld simulated specimen, to take the effect of HAZ size and mismatch into account. If given a certain critical CTOD for a weld simulated specimen, it is possible to use this line to find the critical CTOD for a weld, as long as the requirements are fulfilled. The line is valid for both $m = 10$ and $m = 20$ and will thus be valid for a large spectrum of microstructures.

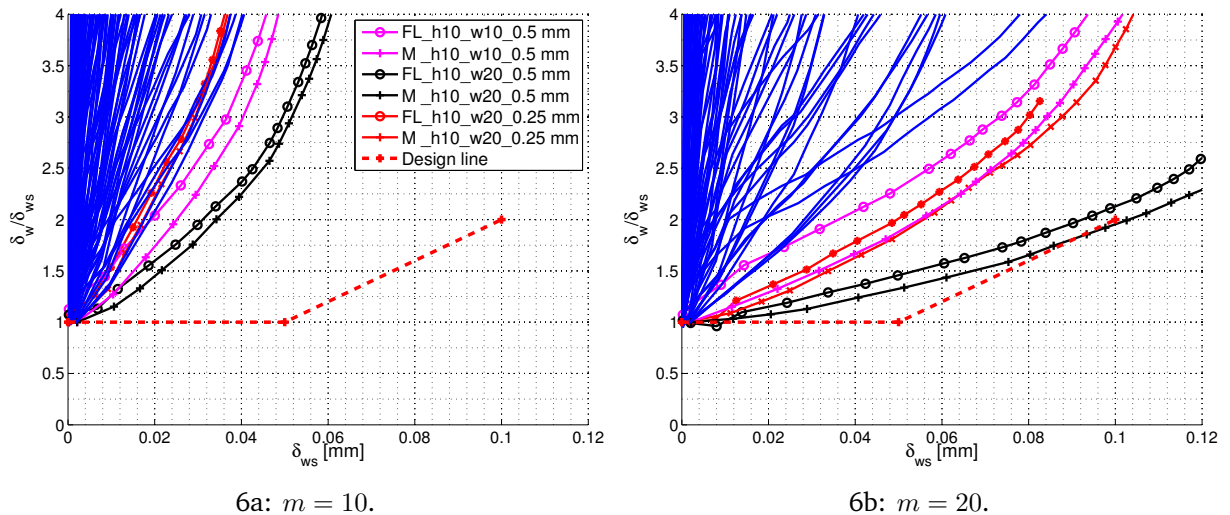


Figure 8.6: Simulations with at least 10% overmatch in HAZ compared with BM and not more than 10% undermatch between HAZ and WM. Without the simulations with 1 mm HAZ.

The design line says that there is no positive effect for a weld simulated specimen as long as the critical CTOD is less than 0.05 mm. For such low CTOD values the positive effect from size and mismatch in the weld have not yet occurred. This is due to that the high stresses and the plastic zone needs to reach the boundary of the HAZ first. From a $\delta = 0.05$ mm to $\delta = 0.1$ mm, the design line grows linearly to the point where the CTOD of a weld simulated specimen may be multiplied by 2. This means that if the critical CTOD is 0.1 mm in a weld simulated specimen, it is 0.2 mm in a real weld.

The relationship between a weld simulated specimen and a real weld is now possible to describe mathematically. δ_{ws} is tested CTOD of weld simulated specimen and δ_w is

equivalent CTOD of a weld.

$$\delta_w = \delta_{ws} \quad \delta_{ws} \in [0, 0.05) \quad (28)$$

$$\delta_w = 20\delta_{ws}^2 \quad \delta_{ws} \in [0.05, 0.1) \quad (29)$$

This non linear relationship between CTOD in weld simulated specimens and CTOD in real welds is qualitatively in accordance with the real testing mentioned in chapter 2.3.1. For small CTODs the relation is approximately 1 : 1, but for larger CTODs weld simulated specimens are more brittle than real welds.

A factor that is not included in this model is general hardening. From chapter 7.7 it is concluded that an increased hardening in all the materials will have a negative effect. From figure 7.8 it can be shown that the relationship above is not valid even for $n = 0.10$. How large this effect is and how it can be taken into account needs further study.

There are however some positive effects which are not included either. One is that there is usually higher hardening in HAZ, than in the two other materials. In chapter 7.6 this is discussed and might give an indication of how large this effect is. Also the Lüder band that often occurs in BM and WM will give a positive effect. Higher hardening in HAZ and Lüder in BM and WM might together give a large positive effect, but this also needs further study.

8.4 Discussion

This proposed design line is a first sketch and there are several factors that need to be studied before it can be used by the industry. It was designed to be as general as possible, and it might be too general, and more requirements need to be applied. It is also possible to make several design lines, where higher lines have more requirements, and demands more testing.

This parameter study has been performed with theoretical materials, and the qualitative effect of mismatch is studied. In the pre study, real materials were used in the simulations. These simulations show very different results, see figure 2.3. In this figure weld simulation is very brittle compared to real welds. This is due to higher mismatch conditions and Lüders band. It is therefore probably possible to establish a more favourable relationship, where welds are even more ductile compared to weld simulated specimens, if a real material is tested, and several design lines are made. Even to move the threshold CTOD for when the weld simulation gets a benefit, should be possible.

From chapter 8.2 it is concluded that there is, at this point, not possible to give a positive effect from high constraint in a weld simulated test specimen. This is based on the lack of constraint effect for $m = 10$, for $m = 20$ it is however a large constraint effect. In the design line based on size and mismatch, $m = 20$ is the lower bound. For only $m = 10$ the design line would be higher and with fewer requirements. This means that the worst effect of both $m = 10$ and $m = 20$ is taken into account. If a material has $m = 10$, there should be no constraint effect, but the effect of size and mismatch should be higher. For a material with $m = 20$ the effect of size and mismatch is correct, but there should be an effect of constraint. This indicates that the design line is very conservative. To utilise the effect of m better and take it into account, more simulations need to be performed.

When comparing the design equation with the requirement proposed in the Artic Material research project, a minimum critical CTOD in the weld simulation specimen is possible to establish. A minimum CTOD of 0.1 mm in the structure will from equation (29) be equivalent to at least a critical CTOD of 0.07 mm in the weld simulated specimen.

9 Conclusion

The parameter study investigated several variables, and what effect they cause on the ratio between the CTOD on a weld simulated specimen and the CTOD on a real weld. The different variables are first studied one by one:

- From the study of different material types, there was not much effect on the CTOD ratio. The main conclusion from this comparison was that general yield stress level, is of minor importance.
- The position of the crack has some influence. For numerical simulations a crack placed in the middle of HAZ is the worst case. For real welds, the crack is however most critical if it is positioned at FL.
- The effect of HAZ size is quite important. The probability of fracture in a weld reduces with smaller HAZ area, and the CTOD ratio between the two models is thus increasing.
- Mismatch in yield stress compared to BM has large influence on the CTOD ratio. It is most sensitive to mismatch in HAZ, but also mismatch in WM is important. Higher overmatch in HAZ makes the weld simulated specimen more brittle compared to a real weld. Overmatch in WM is however making the weld more brittle while the weld simulated specimen is the same. With overmatch in WM compared to both BM and HAZ the weld may even behave more brittle than a weld simulated specimen.
- Overmatch in hardening for HAZ, makes the weld simulated specimen more brittle compared to a weld. The effect is however not so strong that undermatch in HAZ can be accepted if the hardening is high.
- Increase in general hardening makes the weld simulated specimen less brittle compared to a weld. This effect might be important and should be studied further.
- The Weibull exponent m shows large influence on the result from the MBL model, as long as the difference between weld and weld simulation is not too large. It is then $m = 20$ which is the most critical case.
- Geometry constraint should make weld simulated testing more brittle than testing of real welds. The MBL simulations with different level of T -stress confirm this. The effect does however disappear for the 3D model of a weld simulated specimen, due to its small size. The ligament on the test specimen is too short and influences therefore the stress field, even though it is valid from the requirement in ASTM E-1921 [21]. This only occur for $m = 10$.

When all the different variables are studied, there are only the size and the mismatch effects which contribute significantly, to make a weld simulated specimen more brittle than a weld. To avoid that a weld is more brittle than a weld simulated specimen, three requirements are suggested:

- At least 10% overmatch in HAZ compared to BM.
- Not more than 10% undermatch in HAZ compared to WM.
- Maximal brittle HAZ thickness of 0.5 mm.

With these requirements fulfilled a conservative and general relationship between CTOD in a weld and CTOD in a weld simulated specimen can be proposed:

$$\begin{aligned}\delta_w &= \delta_{ws} & \delta_{ws} &\in [0, 0.05) \\ \delta_w &= 20\delta_{ws}^2 & \delta_{ws} &\in [0.05, 0.1)\end{aligned}$$

This design line is a first proposal and has large potential for improvements. Some effects that are not included are, general hardening and Lüder band in WM and BM. It is also important to remember that it is based on the Weibull stress as an fracture parameter, and the limitations this implies.

The design line is taking the worst effects from both $m = 10$ and $m = 20$ into account. $m = 20$ is most critical regarding size and mismatch, while $m = 10$ is most critical for geometry constraint. Combining these effects may give a less strict design line.

In the Arctic Materials research project it has been proposed that a structure needs to withstand a CTOD of 0.1 mm. This implies, from the design line, that the minimum allowed CTOD in a weld simulated specimen is 0.07 mm.

10 Suggestions for future work

This master thesis has several limitations that is possible to study further:

- The HAZ can be made more complex. This is done by including several of the microstructures in the HAZ. In the model used in this thesis this is easy to implement. The calculation of the Weibull stress becomes however more complex. This is because the Weibull parameter σ_u needs to be calculated for the different microstructures, to be able to compare them. Material curves for all the different microstructures need to be established as well.
- General hardening reduces the CTOD ratio between the two models. This effect is not studied thoroughly in this thesis, and could be investigated in more detail.
- The parameter study performed here is very general, real materials and welds would probably have a larger difference in critical CTOD. For instance the Lüder band might have big influence. The design line proposed here might thus be too conservative.
- The design line made is very general and applicable for most cases. Some special lines might be established as well, but with more strict requirements.
- The worst sides of both $m = 10$ and $m = 20$ is taken into account in the design line, this makes the line very conservative. It might be possible to establish a less strict design line if this effect was studied in more detail.

11 Bibliography

- [1] Artic Materials. www.sintef.no/projectweb/arctic-materials/task-descriptions/, 10/11 2011.
- [2] E. Østby. Personal conversation, 03 2012.
- [3] O.M. Akselsen E. Østby and C. Thaulow. Low temperature toughness in SA welding of 420 MPa steel. 2011. The 21st ISOPE Conf., Maui, USA, June 19-24.
- [4] E. Østby C. Thaulow and O.M. Akselsen. Fracture toughness scatter and effect of constraint in weld thermal simulated HAZ microstructures at -60°C . 2011. The 21st ISOPE Conf., Maui, USA, June 19-24.
- [5] E. Østby G.T Kolstad O.M. Akselsen C. Thaulow and M. Hauge. Comparison of fracture toughness in real weld and thermally simulated CGHAZ of a 420 MPa rolled plate. 2012.
- [6] G.T Kolstad. *FEM modeling of weldments*, 2011. NTNU.
- [7] Ø. Ranestad. *The effect of strength mismatch and local geometry on the brittle behaviour of steel weldments*. PhD thesis, NTNU, 1998.
- [8] E. Zimmermann. *FE modelling of local stress level i second phase particles in steel*. Bachelor's Thesis, 2011.
- [9] T. L. Anderson. *Fracture mechanics*. CRC Press Taylor & Francis Group, 3rd edition, 2005.
- [10] R. C. Hibbeler. *Mechanics of materials*. Prentice Hall College Div, 4th edition, 1999.
- [11] F. M. Beremin. *A local criterion for cleavage fracture of nuclear pressure vessel of steel*, volume 14 A. Met. Trans. A, 1983.
- [12] E. Østby. State of the art of models and treatment of brittle fracture in steels. Technical report, SINTEF, 2008.
- [13] J. R. Low. volume 46A. Trans. AMS, 1945. (Taken from Østby [1998]).
- [14] Procedure to measure and calculate material parameters for the local approach to fracture using notched tensile specimens. *ESIS P6-94*, 1998.
- [15] G. Bernauer W. Brocks and W. Schmitt. Modification of the beremin model for cleavage fracture in the transition region of a ferritic steel. *Engineering Fracture Mechanics*, 64:305–325, 1999.
- [16] C. Sainte-Catherine M. Di Fant B. Duquaire and P. Balladon. Prediction of toughness scatter in heat affected zones with local approach. *Engineering Fracture Mechanics*, 46:115–125, 1993.

- [17] C. Ruggieri. An engineering methodology to assess effect of weld strength mismatch on cleavage fracture toughness using the Weibull stress approach. *International Journal of Fracture*, 164:231–252, 2010.
- [18] C. Ruggieri and R. H. Dodds jr. A transferability model for brittle fracture including constraint and ductile tearing effects: a probabilistic approach. *International Journal of Fracture*, 79:309–340, 1996.
- [19] British Standard. Fracture mechanics toughness tests, 2005. BS 7448.
- [20] American Society for Testing and Materials. Standard test method for measurement of fracture toughness, 2005. ASTM E-1820, Philadelphia.
- [21] American Society for Testing and Materials. Standard test method for determination of reference temperature, T_0 , for ferritic steels in the transition range, 2008. ASTM E-1921, Philadelphia.
- [22] K. Wallin. Master curve analysis of the "Euro" fracture toughness dataset. *Engineering Fracture Mechanics*, 69, 2002.
- [23] K. Easterling. *Introduction to the physical metallurgy of welding*. Butterworths, 1983.
- [24] R.D.Cook D.S.Malkus M.E.Plesha and R.J.Witt. *Concepts and Applications of Finite Element Analysis*. John Wiley, 4th edition, 2002.
- [25] Abaqus documentation. <https://www.sharcnet.ca/software/abaqus610/documentation/docs/v6.10/index.html>, 20/9 2011.
- [26] E. Østby. Prediction of brittle fracture in steel based on local fracture criteria. Technical report, 1998.
- [27] D. Rosenthal and R. Schmerber. *Thermal study of arc welding*, volume 17. Welding J, 1938.

Appendix A Curves

All the simulations in the parameter study are plotted in this appendix, for both $m = 10$ and $m = 20$. The Figures have equal T -stress, position of crack, size of HAZ and mismatch level in WM. Type of material and mismatch level in HAZ vary in the figures. The labels indicate mismatch level in HAZ, and whether the material type is real or theoretic. All the figures have the same scale at the axes, this causes some strange plots.

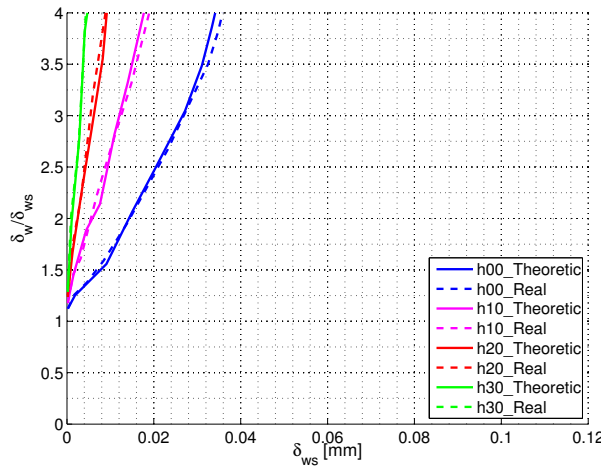
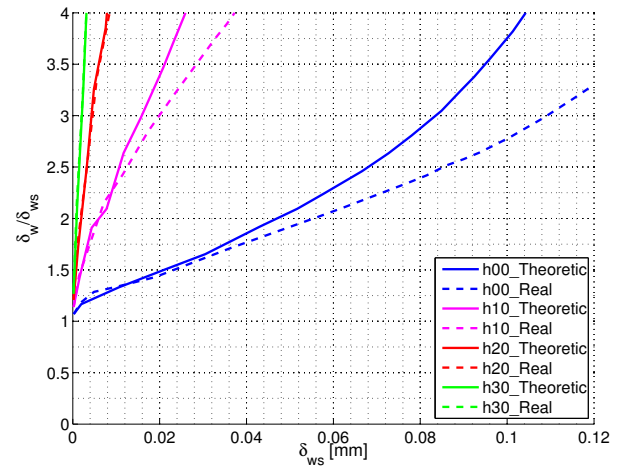
A.1a: $m = 10$.A.1b: $m = 20$.

Figure A.1: Crack at FL, HAZ width 0.25 mm, w00, $T=0$.

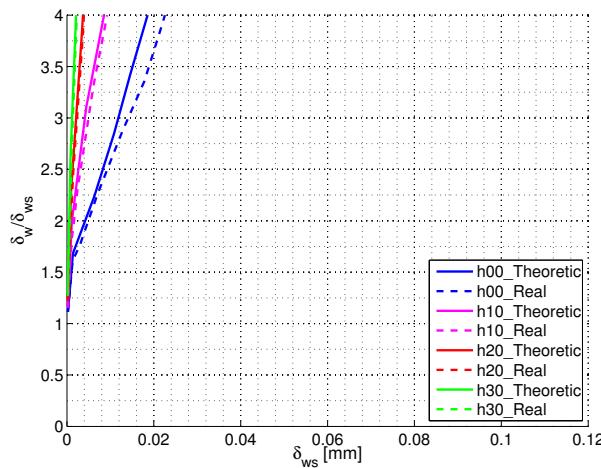
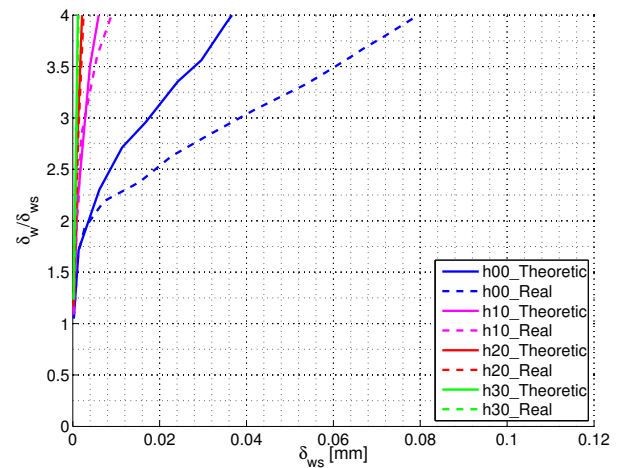
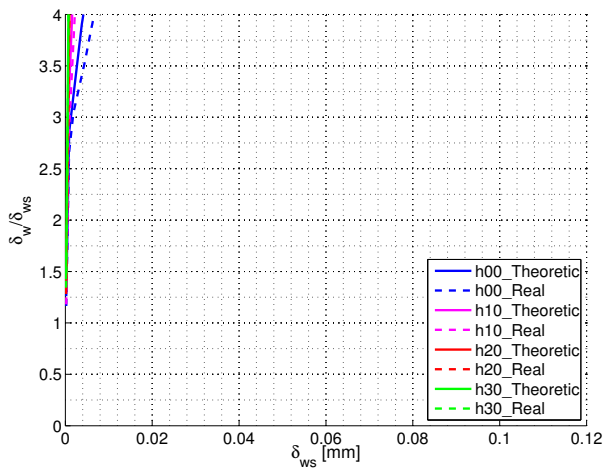
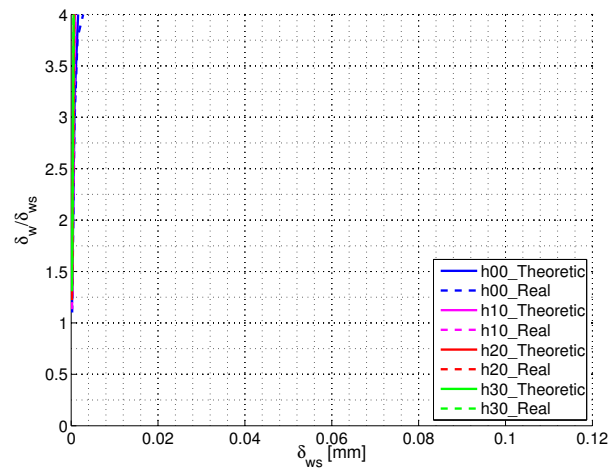
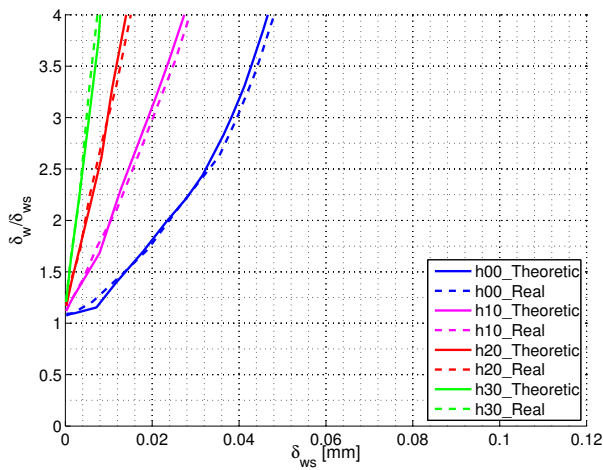
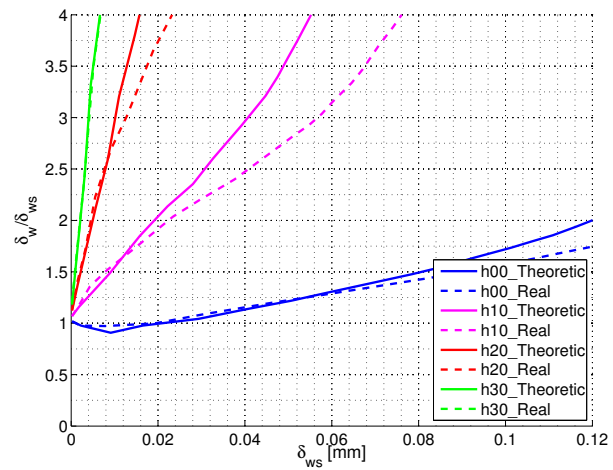
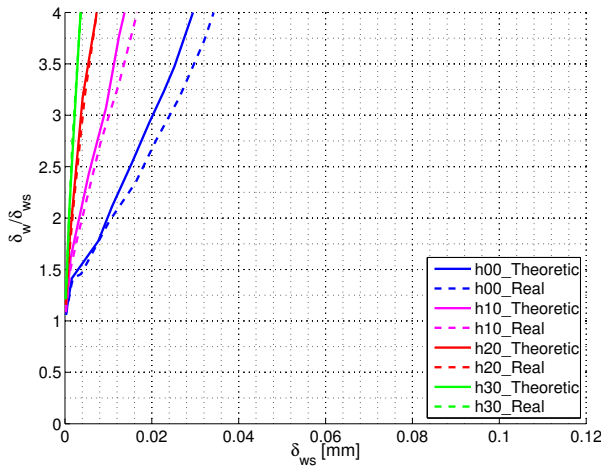
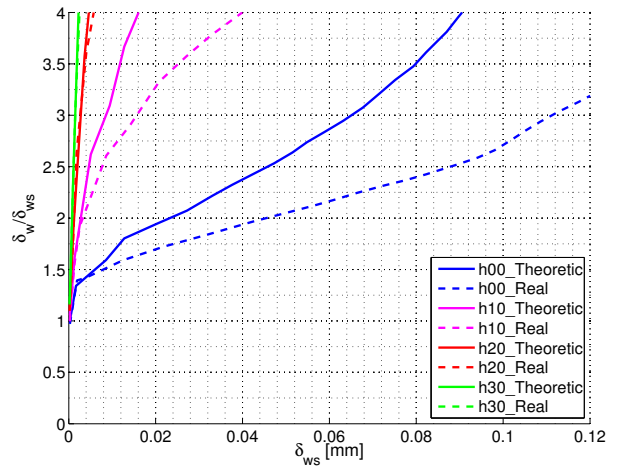
A.2a: $m = 10$.A.2b: $m = 20$.

Figure A.2: Crack at FL, HAZ width 0.25 mm, w00, $T=-0.25\sigma_{0.2}$.

A.3a: $m = 10$.A.3b: $m = 20$.Figure A.3: Crack at FL, HAZ width 0.25 mm, w00, $T = -0.5\sigma_{0.2}$.A.4a: $m = 10$.A.4b: $m = 20$.Figure A.4: Crack at FL, HAZ width 0.25 mm, w10, $T = 0$.

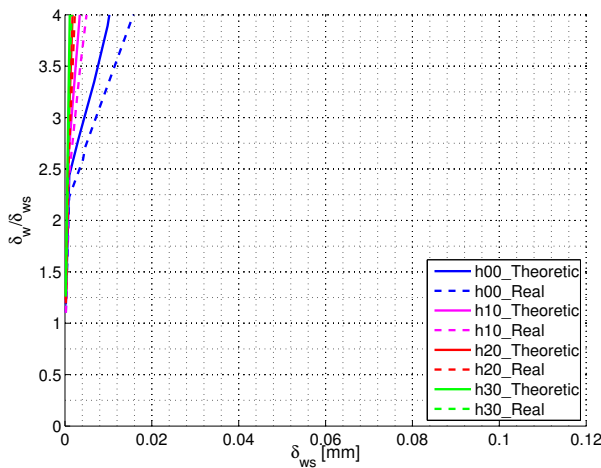


A.5a: $m = 10$.

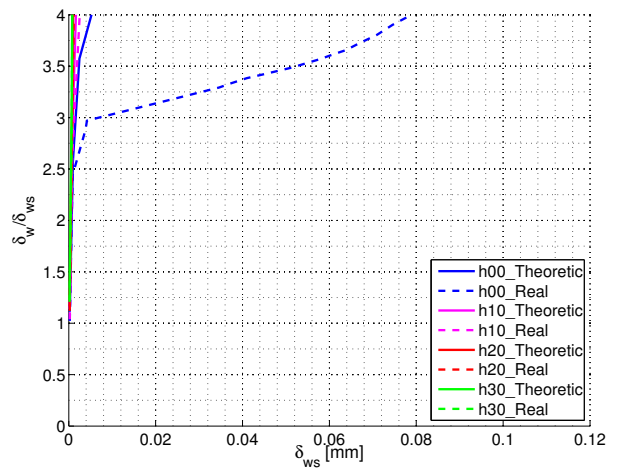


A.5b: $m = 20$.

Figure A.5: Crack at FL, HAZ width 0.25 mm, w_{10} , $T = -0.25\sigma_{0.2}$.

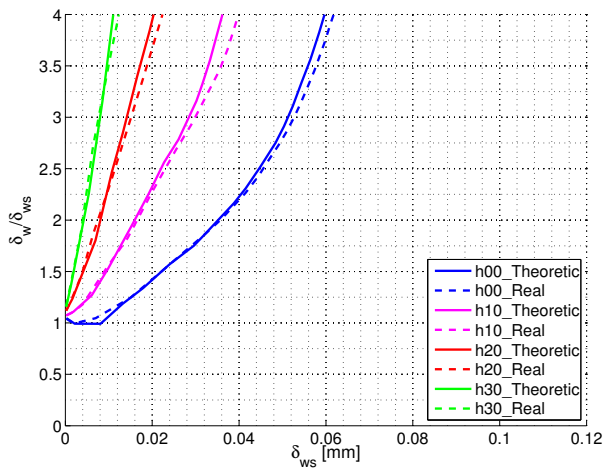
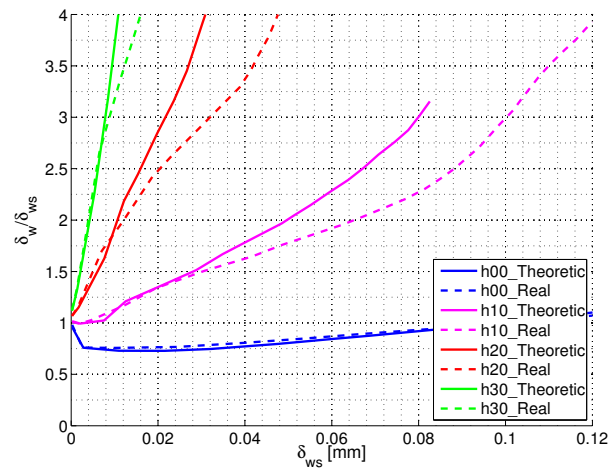
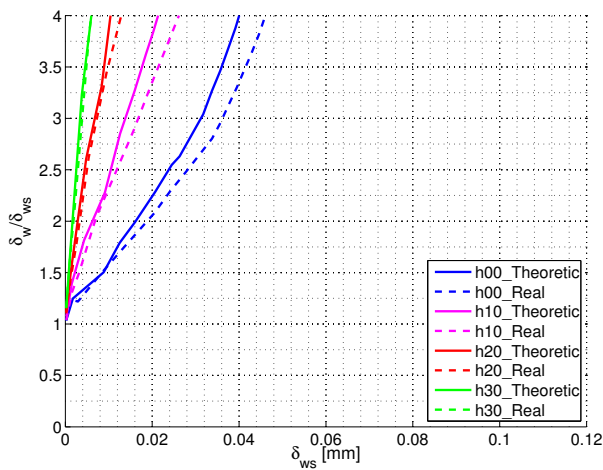
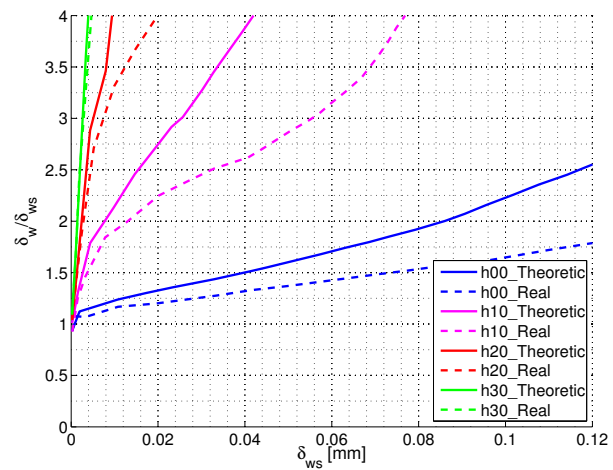


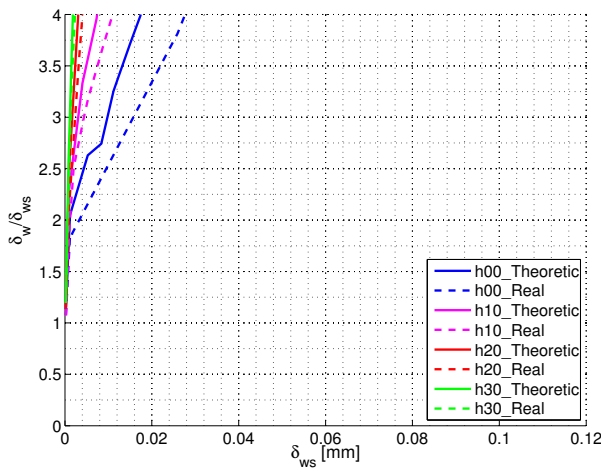
A.6a: $m = 10$.



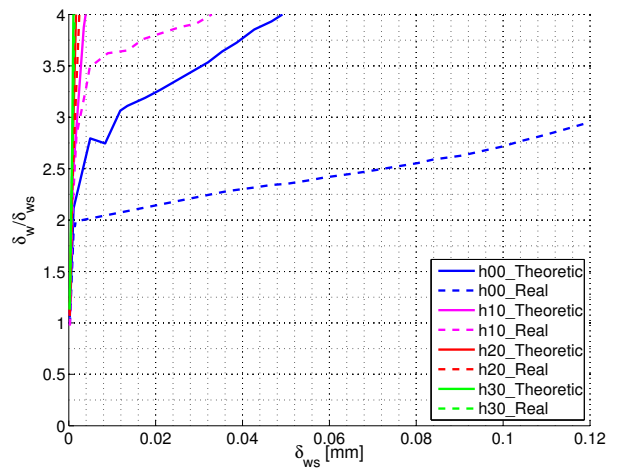
A.6b: $m = 20$.

Figure A.6: Crack at FL, HAZ width 0.25 mm, w_{10} , $T = -0.5\sigma_{0.2}$.

A.7a: $m = 10$.A.7b: $m = 20$.Figure A.7: Crack at FL, HAZ width 0.25 mm, w20, $T=0$.A.8a: $m = 10$.A.8b: $m = 20$.Figure A.8: Crack at FL, HAZ width 0.25 mm, w20, $T = -0.25\sigma_{0.2}$.

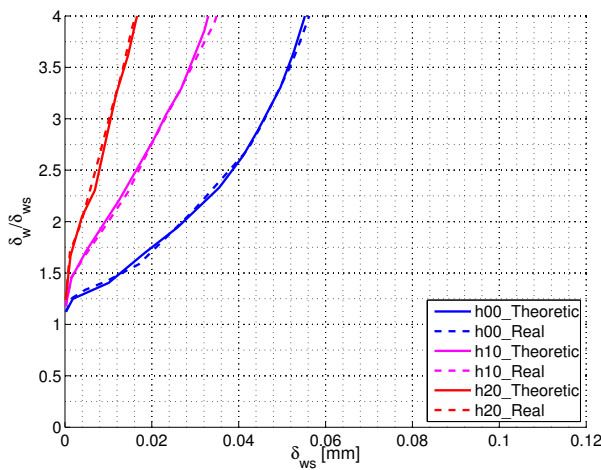


A.9a: $m = 10$.

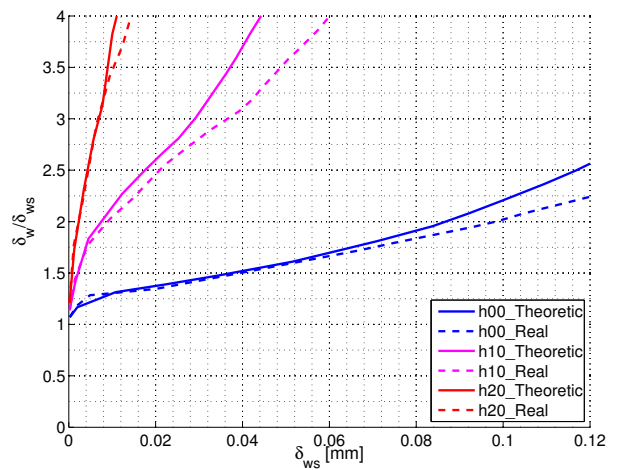


A.9b: $m = 20$.

Figure A.9: Crack at FL, HAZ width 0.25 mm, w20, $T = -0.5\sigma_{0.2}$.

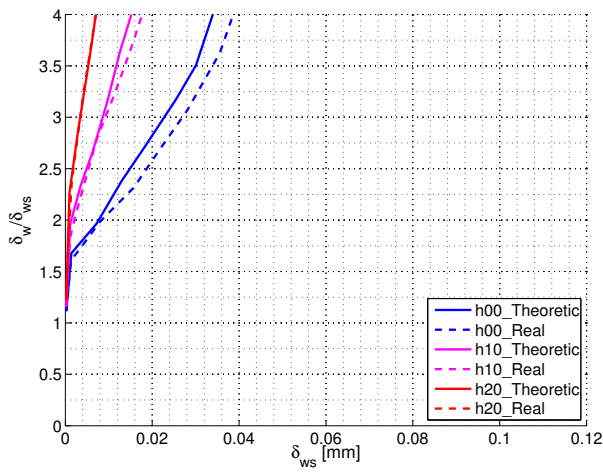
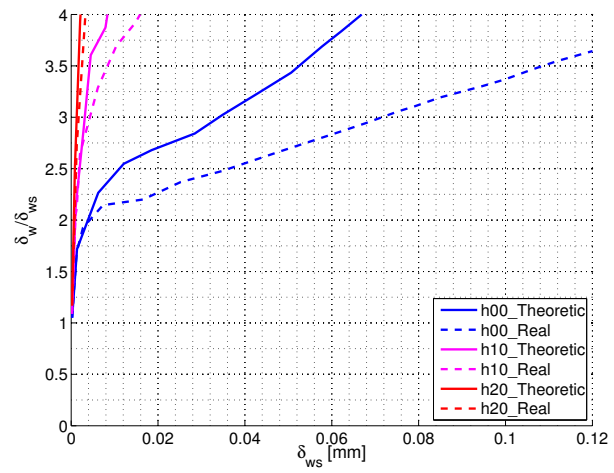
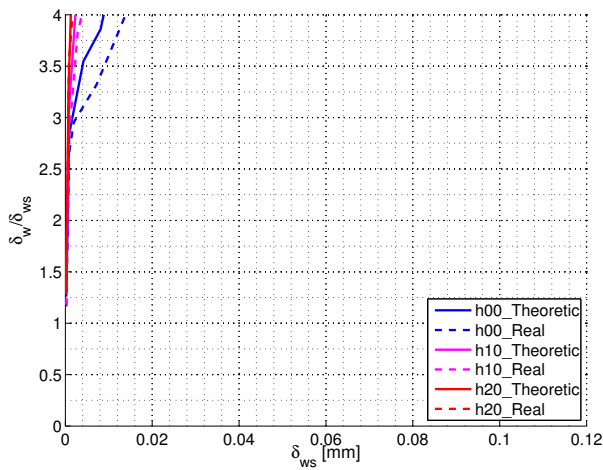
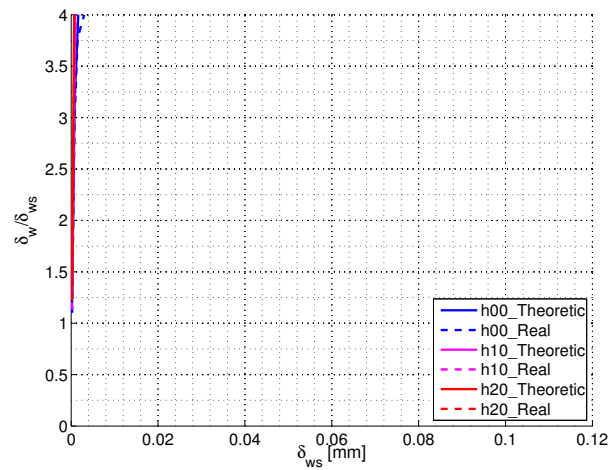


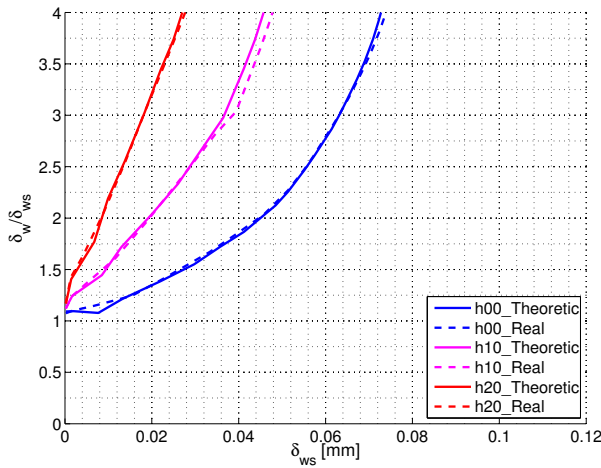
A.10a: $m = 10$.



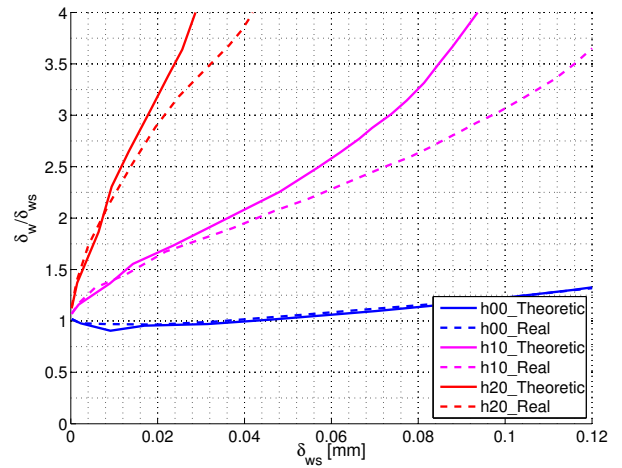
A.10b: $m = 20$.

Figure A.10: Crack at FL, HAZ width 0.5 mm, w00, $T = 0$.

A.11a: $m = 10$.A.11b: $m = 20$.Figure A.11: Crack at FL, HAZ width 0.5 mm, w00, $T = -0.25\sigma_{0.2}$.A.12a: $m = 10$.A.12b: $m = 20$.Figure A.12: Crack at FL, HAZ width 0.5 mm, w00, $T = -0.5\sigma_{0.2}$.

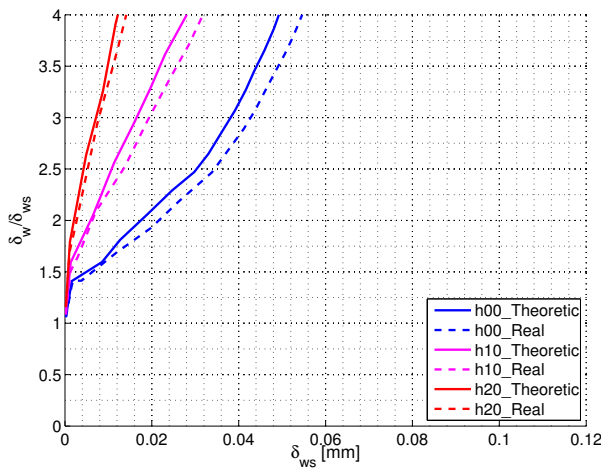


A.13a: $m = 10$.

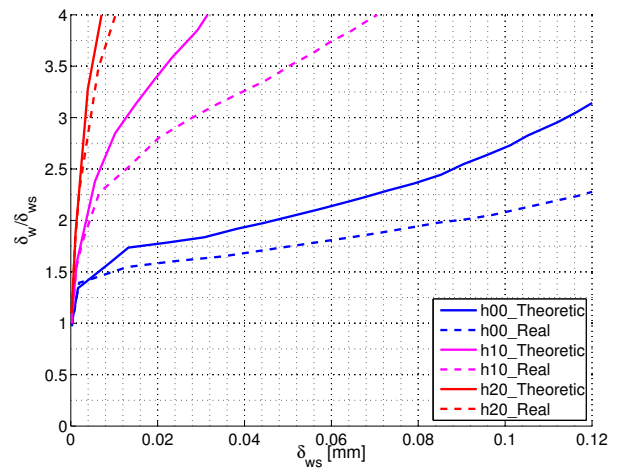


A.13b: $m = 20$.

Figure A.13: Crack at FL, HAZ width 0.5 mm, w10, $T=0$.

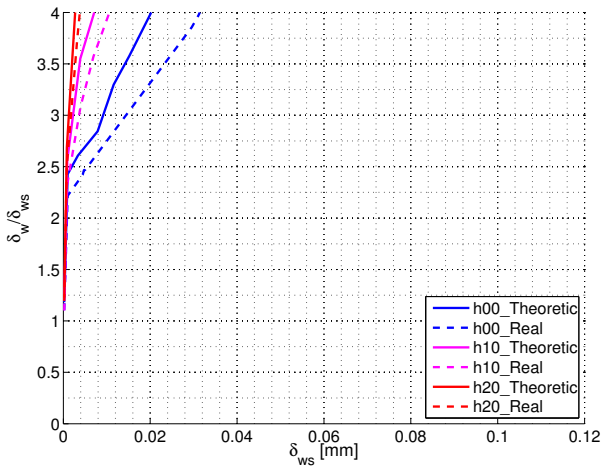


A.14a: $m = 10$.

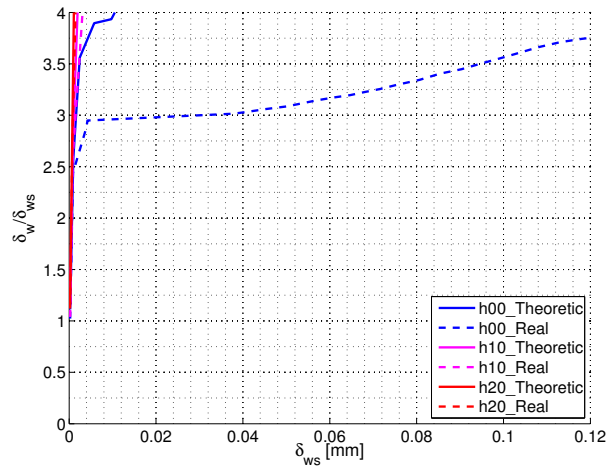


A.14b: $m = 20$.

Figure A.14: Crack at FL, HAZ width 0.5 mm, w10, $T = -0.25\sigma_{0.2}$.

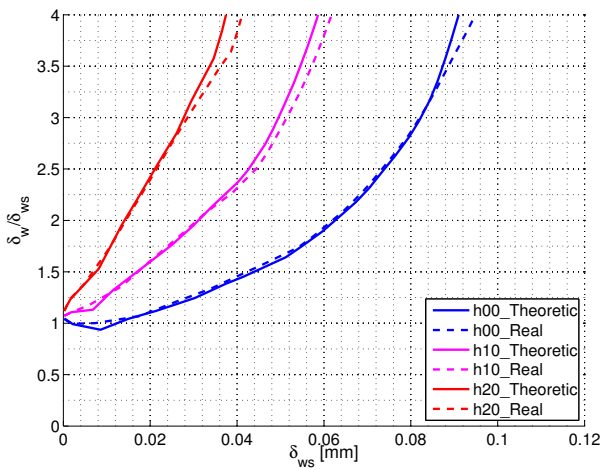


A.15a: $m = 10$.

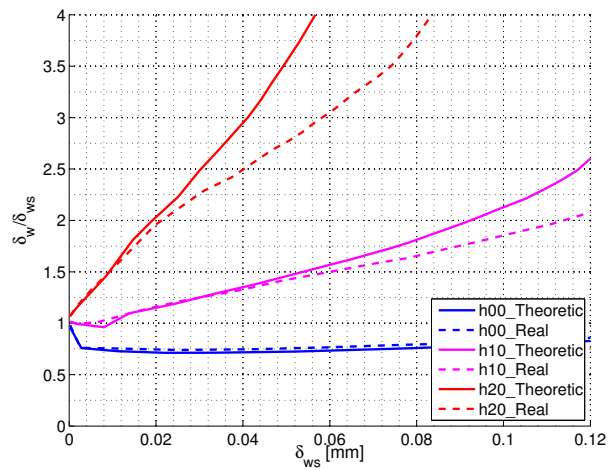


A.15b: $m = 20$.

Figure A.15: Crack at FL, HAZ width 0.5 mm, w10, $T = -0.5\sigma_{0.2}$.

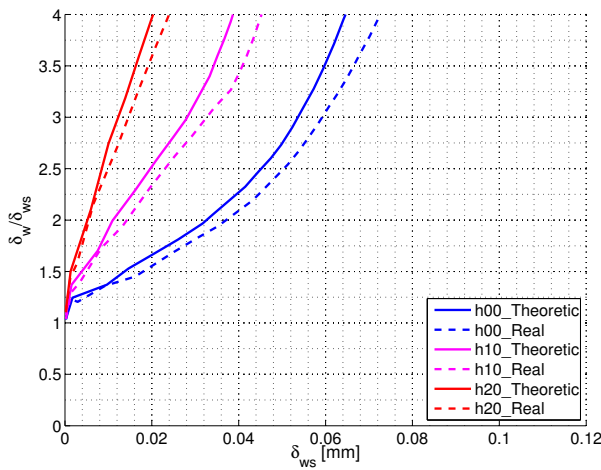


A.16a: $m = 10$.

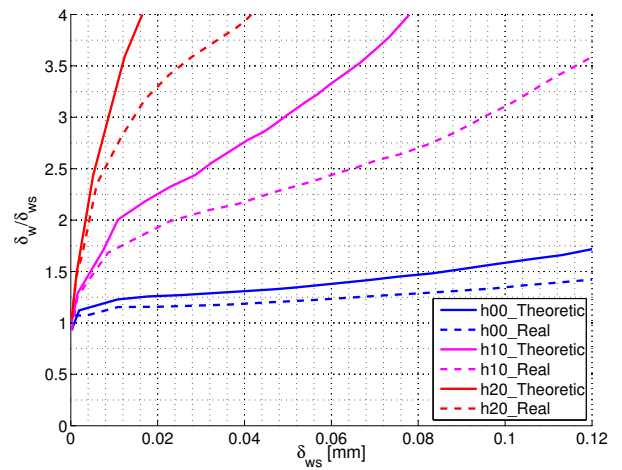


A.16b: $m = 20$.

Figure A.16: Crack at FL, HAZ width 0.5 mm, w20, $T = 0$.

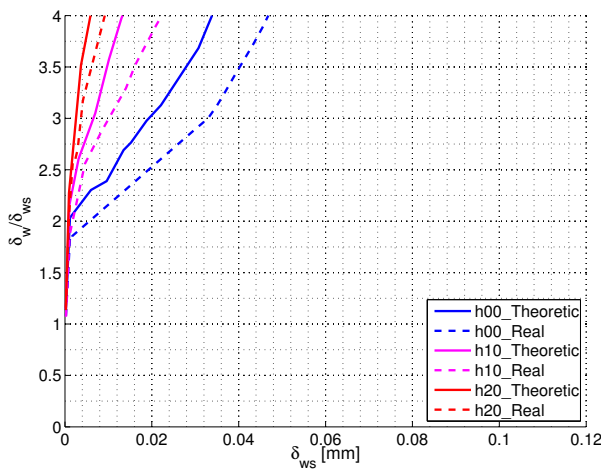


A.17a: $m = 10$.

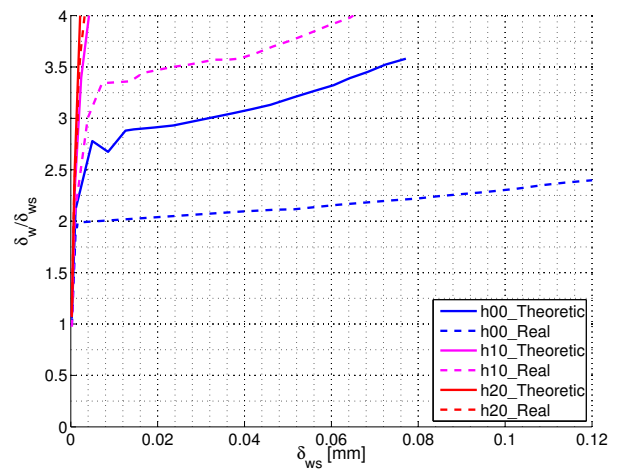


A.17b: $m = 20$.

Figure A.17: Crack at FL, HAZ width 0.5 mm, w20, $T = -0.25\sigma_{0.2}$.

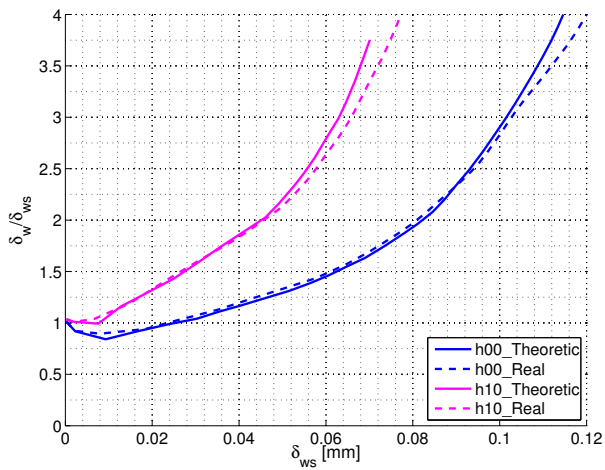
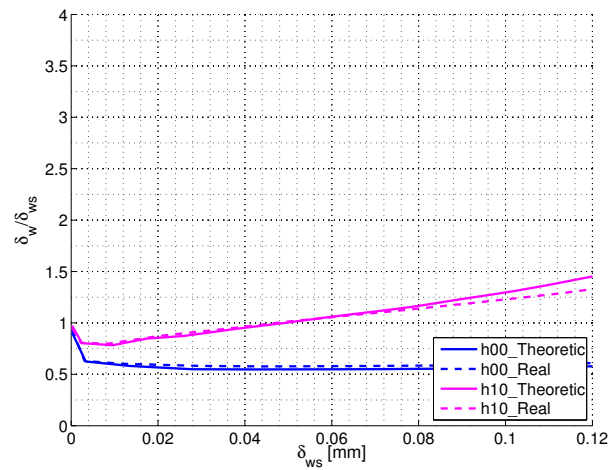
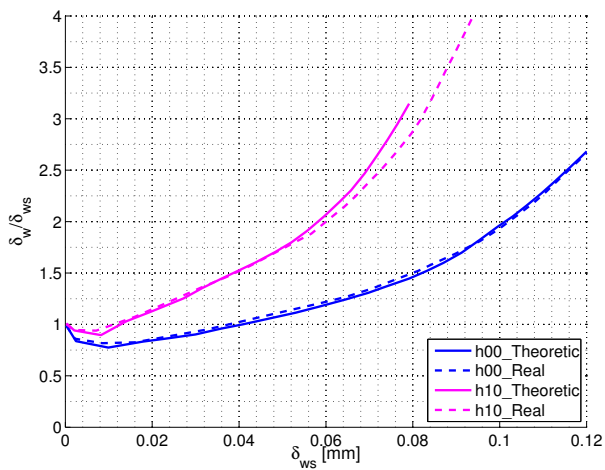
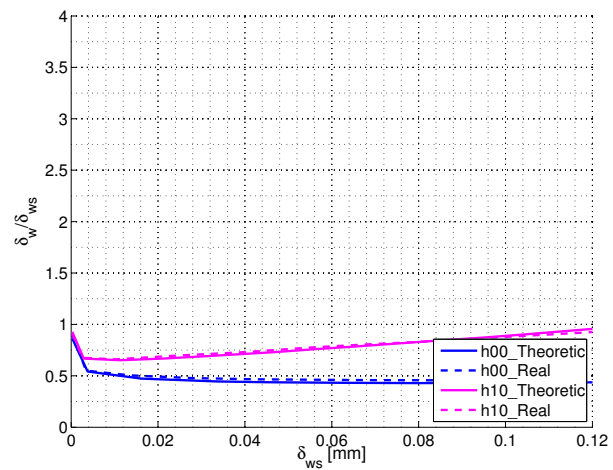


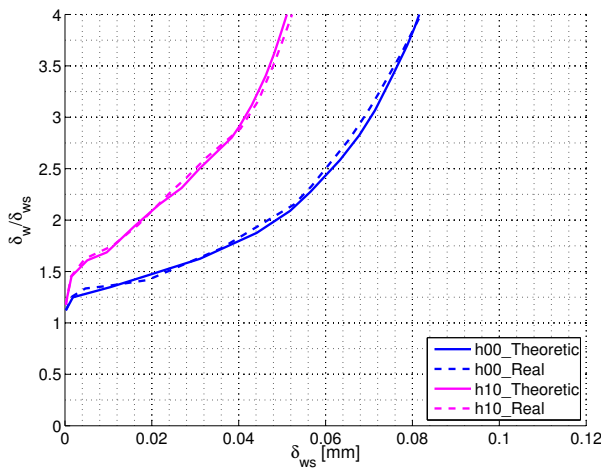
A.18a: $m = 10$.



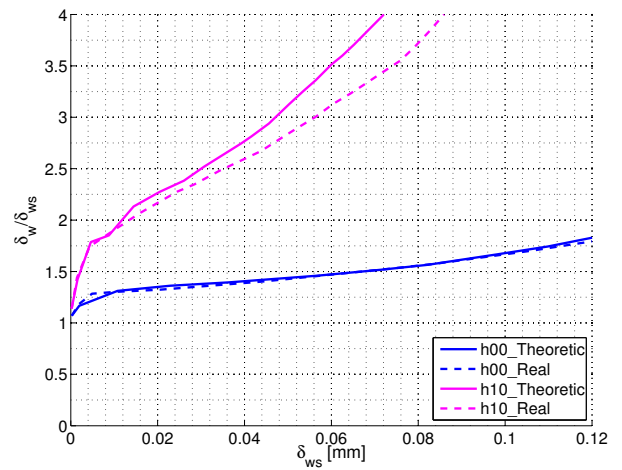
A.18b: $m = 20$.

Figure A.18: Crack at FL, HAZ width 0.5 mm, w20, $T = -0.5\sigma_{0.2}$.

A.19a: $m = 10$.A.19b: $m = 20$.Figure A.19: Crack at FL, HAZ width 0.5 mm, w30, $T=0$.A.20a: $m = 10$.A.20b: $m = 20$.Figure A.20: Crack at FL, HAZ width 0.5 mm, w40, $T=0$.

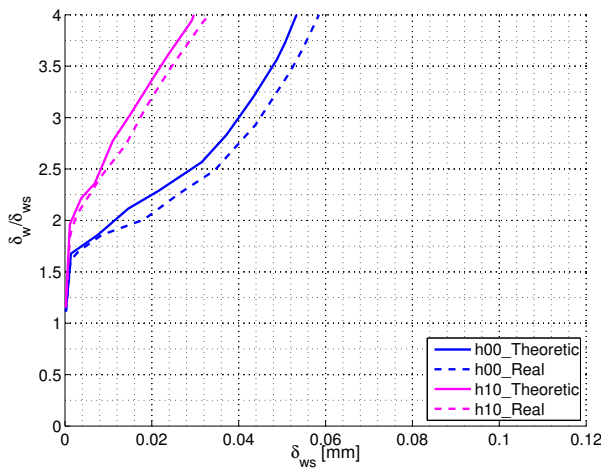


A.21a: $m = 10$.

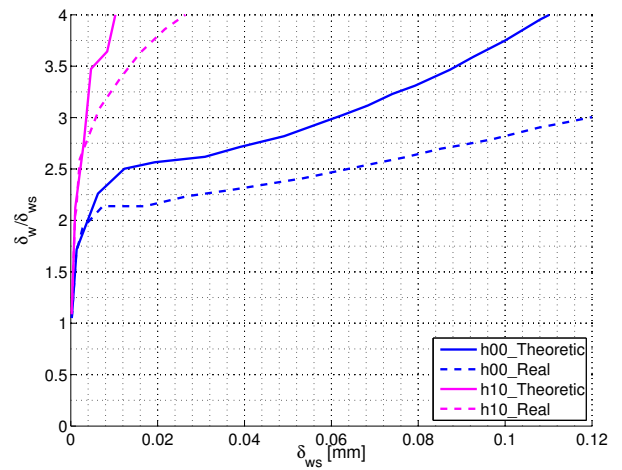


A.21b: $m = 20$.

Figure A.21: Crack at FL, HAZ width 1 mm, w_{00} , $T=0$.

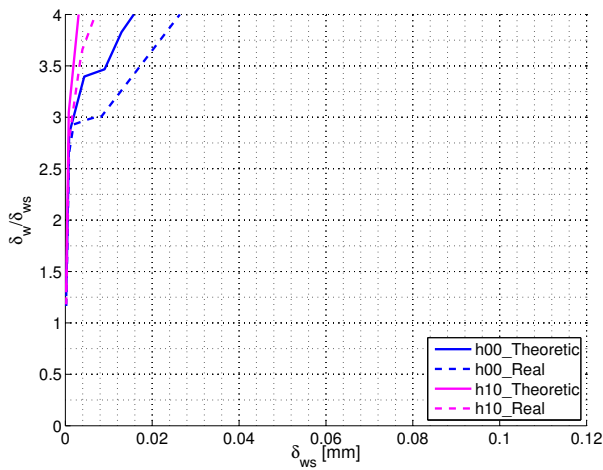
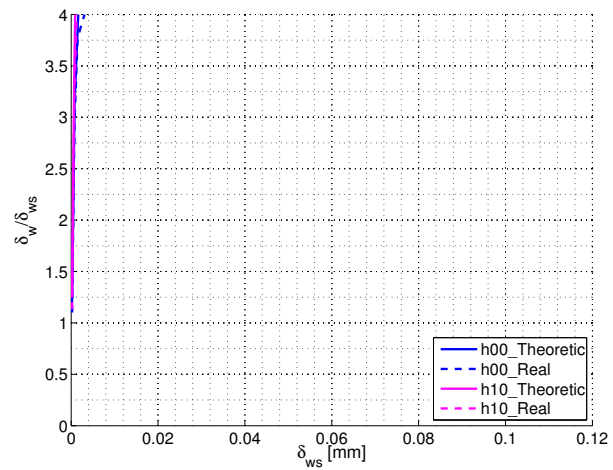
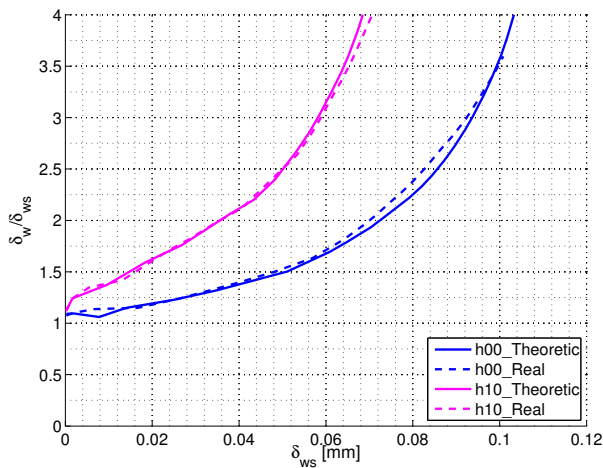
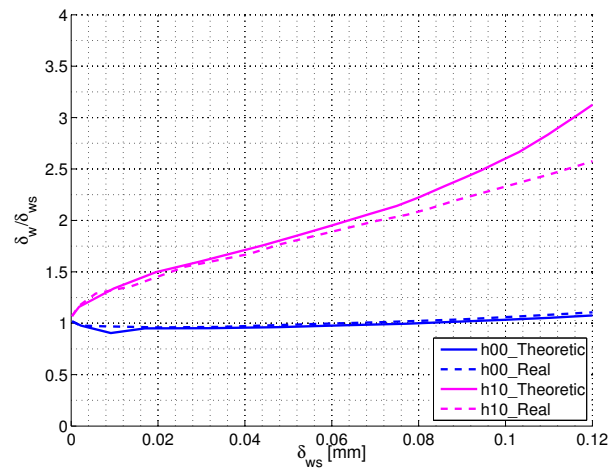


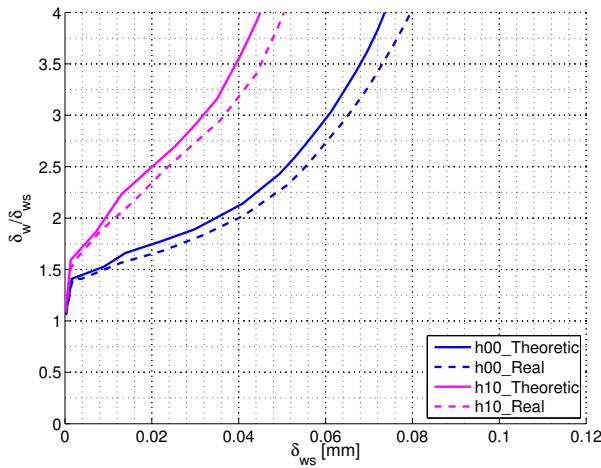
A.22a: $m = 10$.



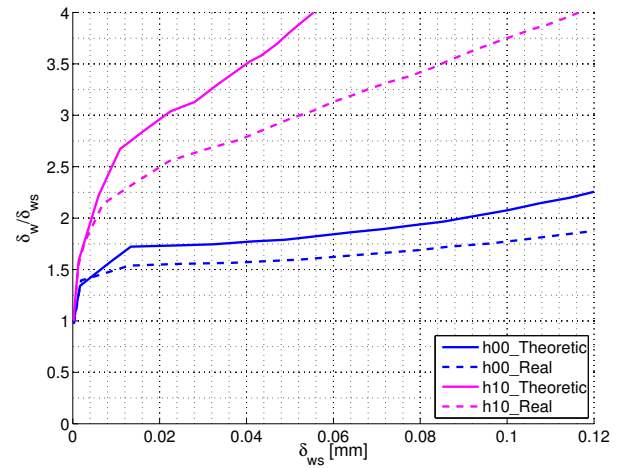
A.22b: $m = 20$.

Figure A.22: Crack at FL, HAZ width 1 mm, w_{00} , $T=-0.25\sigma_{0.2}$.

A.23a: $m = 10$.A.23b: $m = 20$.Figure A.23: Crack at FL, HAZ width 1 mm, w00, $T = -0.5\sigma_{0.2}$.A.24a: $m = 10$.A.24b: $m = 20$.Figure A.24: Crack at FL, HAZ width 1 mm, w10, $T = 0$.

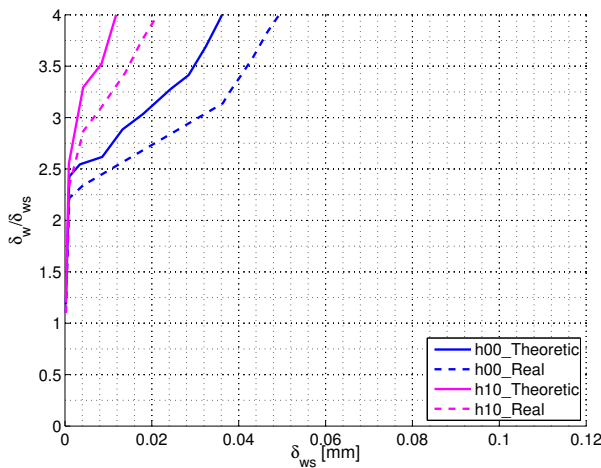


A.25a: $m = 10$.

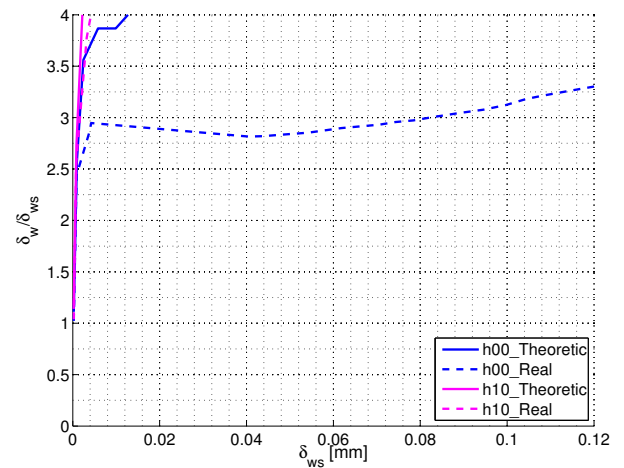


A.25b: $m = 20$.

Figure A.25: Crack at FL, HAZ width 1 mm, w10, $T = -0.25\sigma_{0.2}$.

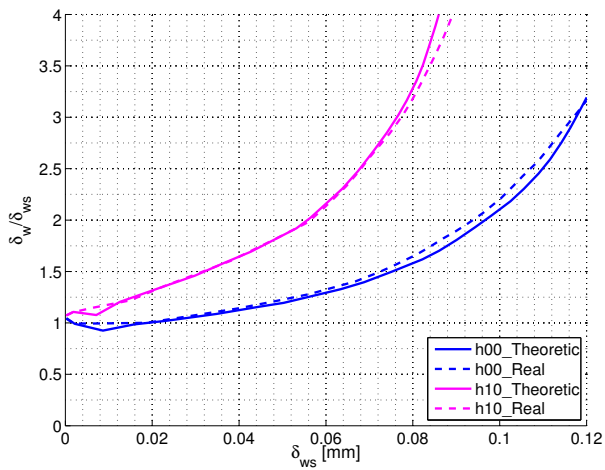
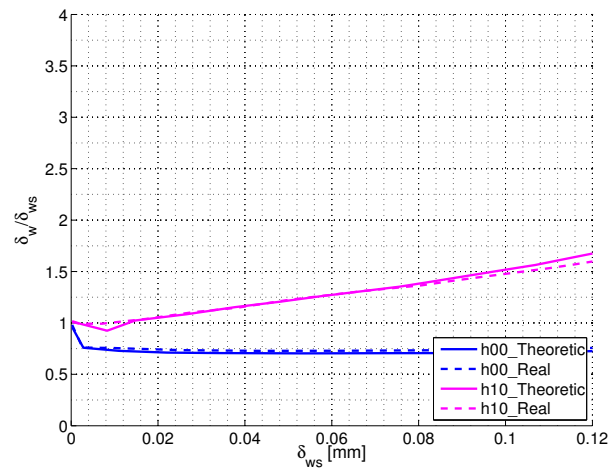
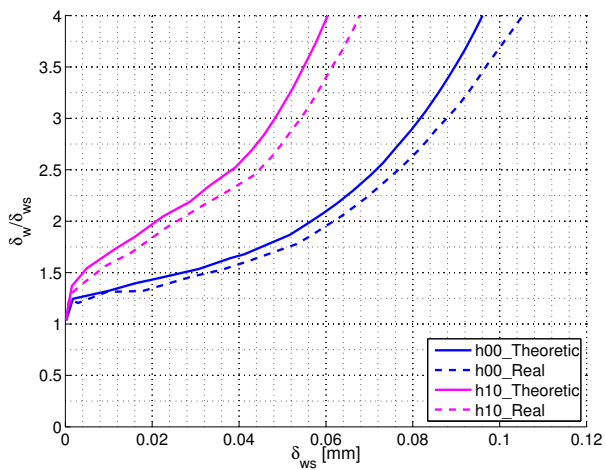
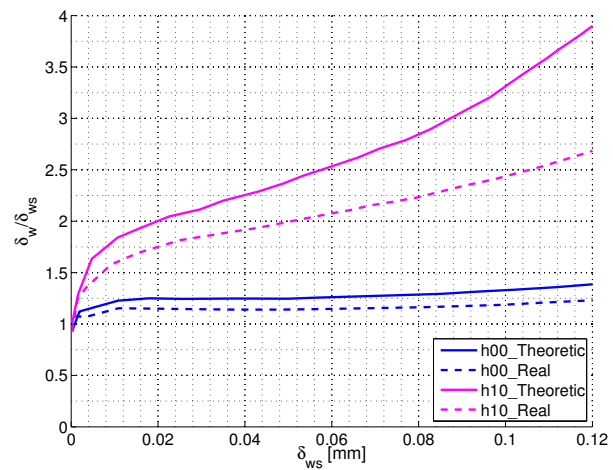


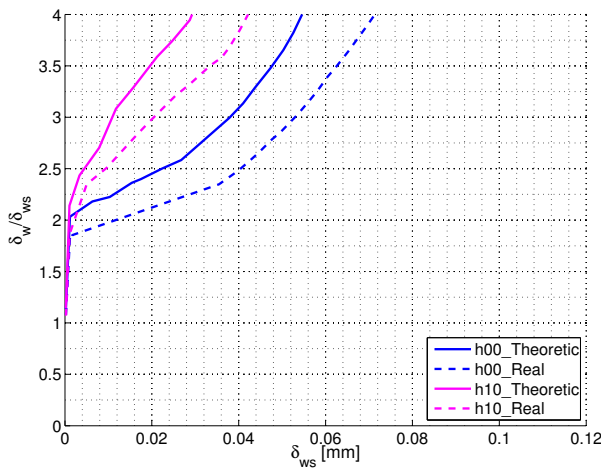
A.26a: $m = 10$.



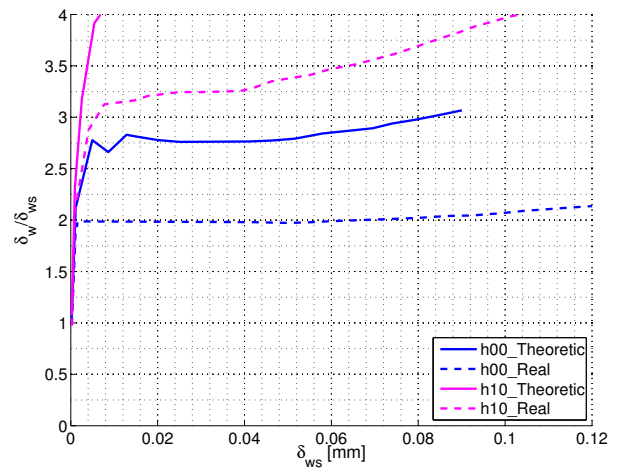
A.26b: $m = 20$.

Figure A.26: Crack at FL, HAZ width 1 mm, w10, $T = -0.5\sigma_{0.2}$.

A.27a: $m = 10$.A.27b: $m = 20$.Figure A.27: Crack at FL, HAZ width 1 mm, w_{20} , $T = 0$.A.28a: $m = 10$.A.28b: $m = 20$.Figure A.28: Crack at FL, HAZ width 1 mm, w_{20} , $T = -0.25\sigma_{0.2}$.

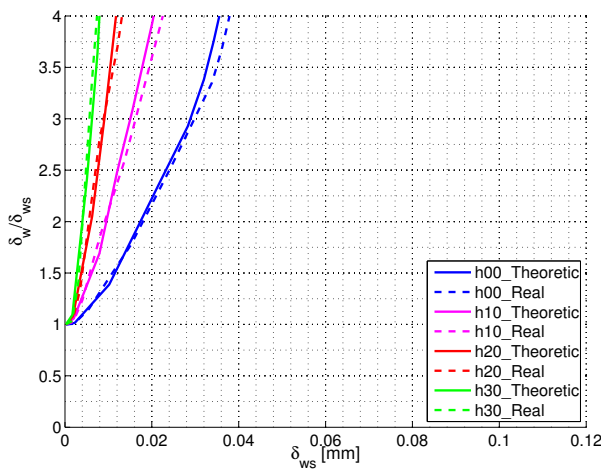


A.29a: $m = 10$.

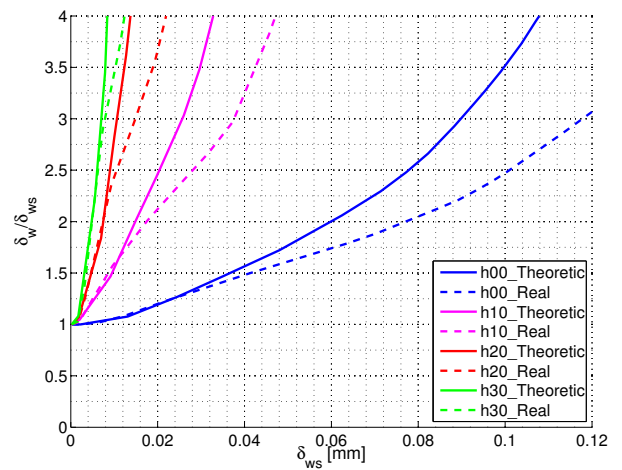


A.29b: $m = 20$.

Figure A.29: Crack at FL, HAZ width 1 mm, $w20$, $T = -0.5\sigma_{0.2}$.

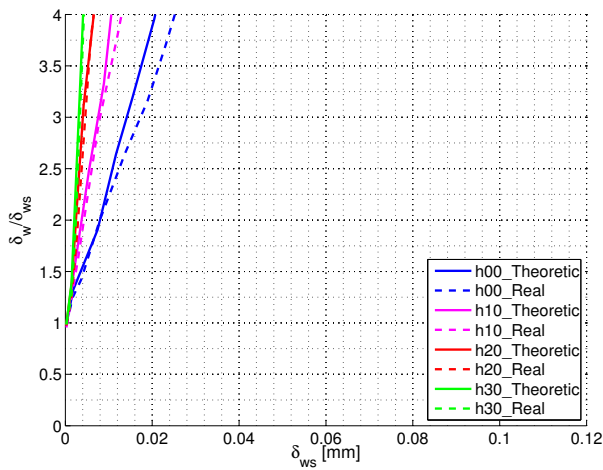
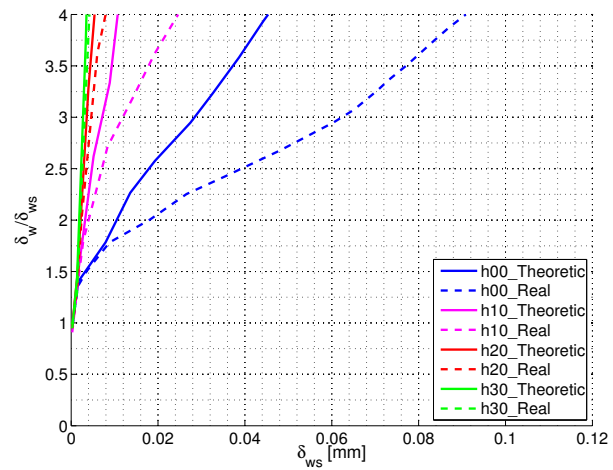
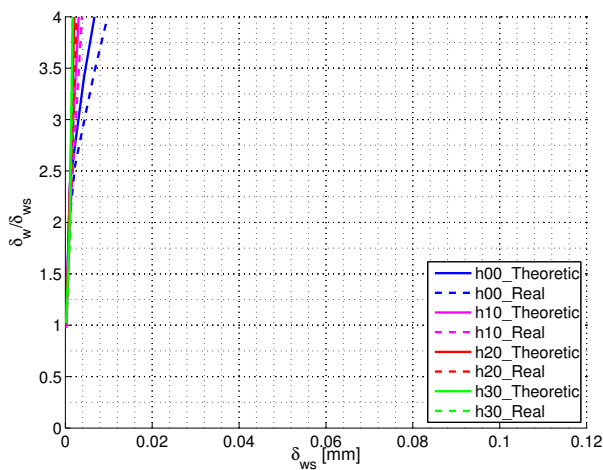
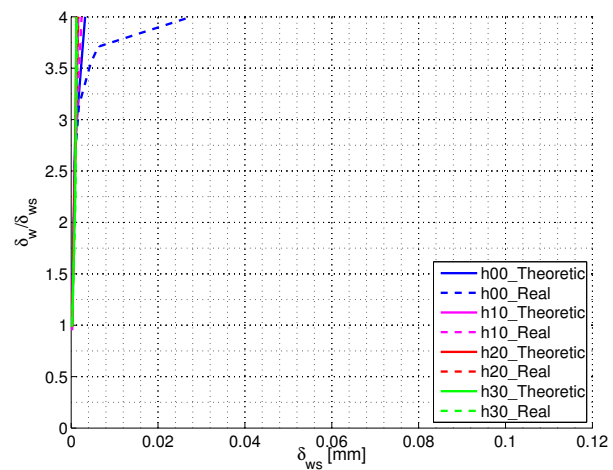


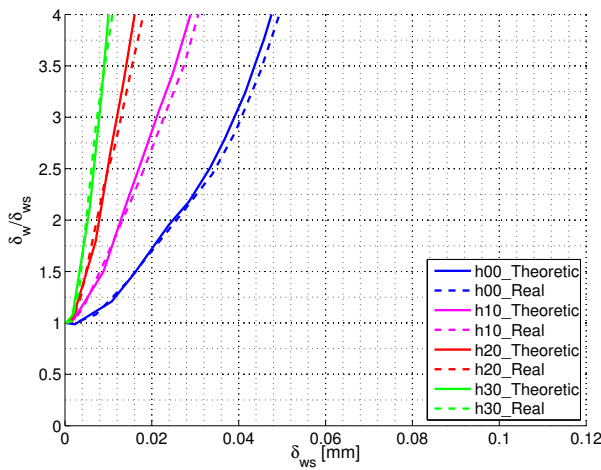
A.30a: $m = 10$.



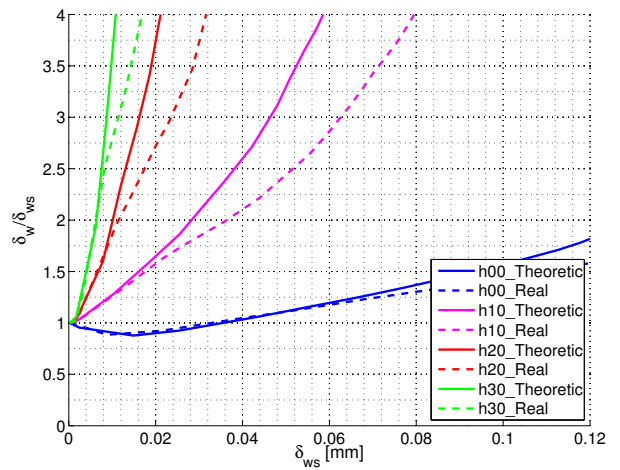
A.30b: $m = 20$.

Figure A.30: Crack in middle of HAZ, HAZ width 0.25 mm, $w00$, $T=0$.

A.31a: $m = 10$.A.31b: $m = 20$.Figure A.31: Crack in middle of HAZ, HAZ width 0.25 mm, w_{00} , $T = -0.25\sigma_{0.2}$.A.32a: $m = 10$.A.32b: $m = 20$.Figure A.32: Crack in middle of HAZ, HAZ width 0.25 mm, w_{00} , $T = -0.5\sigma_{0.2}$.

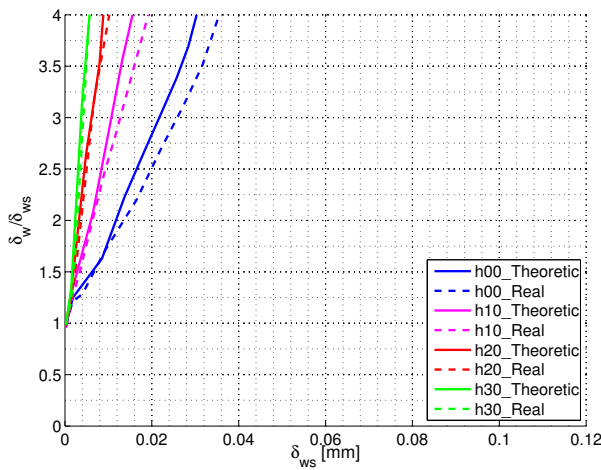


A.33a: $m = 10$.

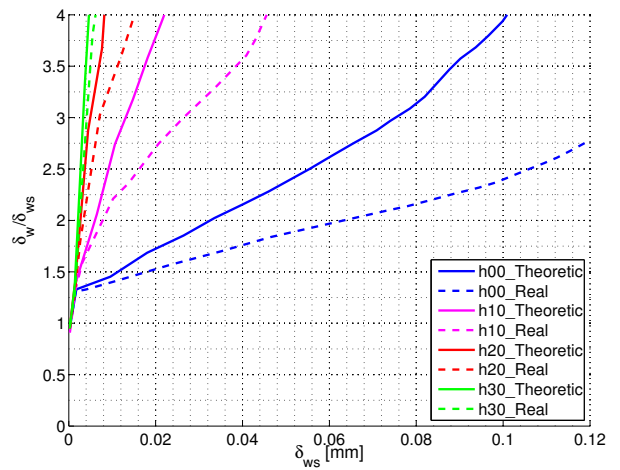


A.33b: $m = 20$.

Figure A.33: Crack in middle of HAZ, HAZ width 0.25 mm, w_{10} , $T=0$.

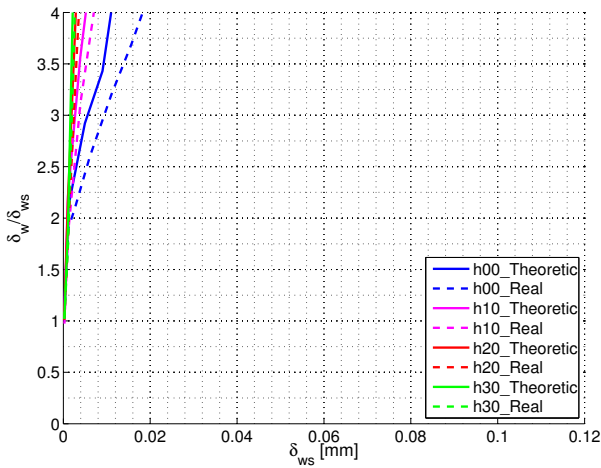


A.34a: $m = 10$.

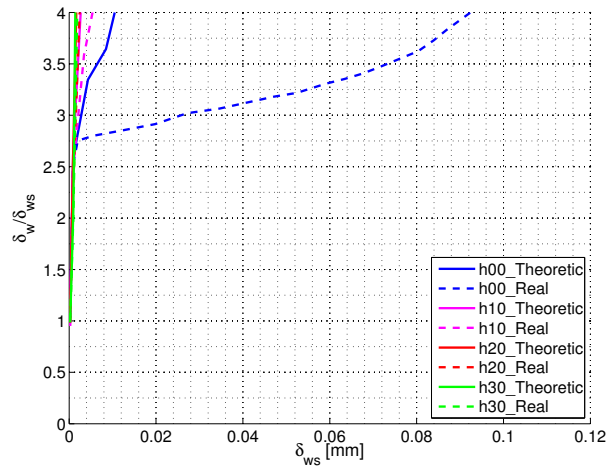


A.34b: $m = 20$.

Figure A.34: Crack in middle of HAZ, HAZ width 0.25 mm, w_{10} , $T=-0.25\sigma_{0.2}$.

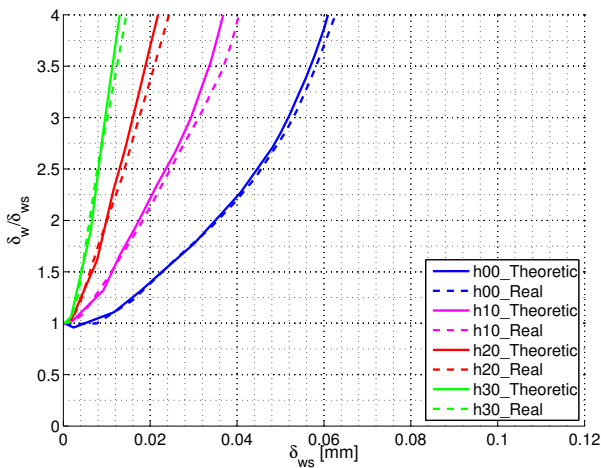


A.35a: $m = 10$.

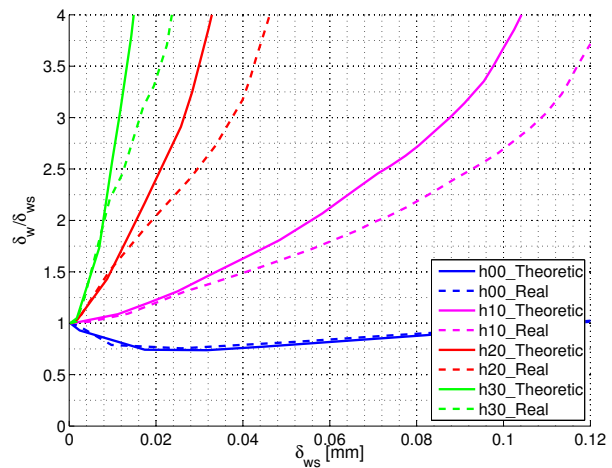


A.35b: $m = 20$.

Figure A.35: Crack in middle of HAZ, HAZ width 0.25 mm, w10, $T = -0.5\sigma_{0.2}$.

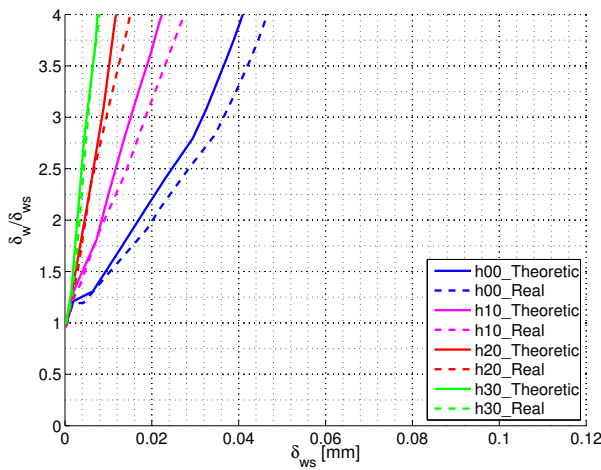


A.36a: $m = 10$.

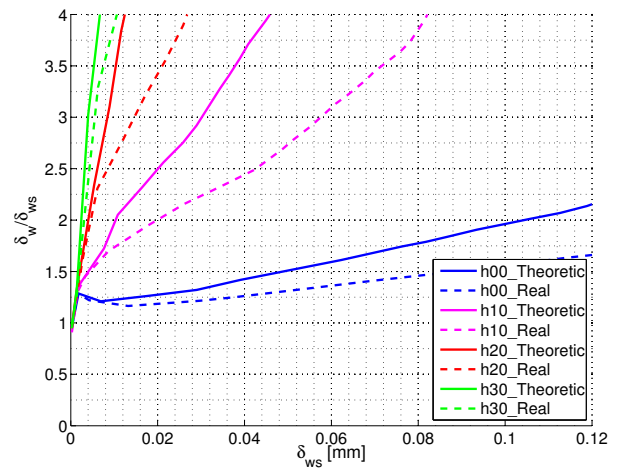


A.36b: $m = 20$.

Figure A.36: Crack in middle of HAZ, HAZ width 0.25 mm, w20, $T = 0$.

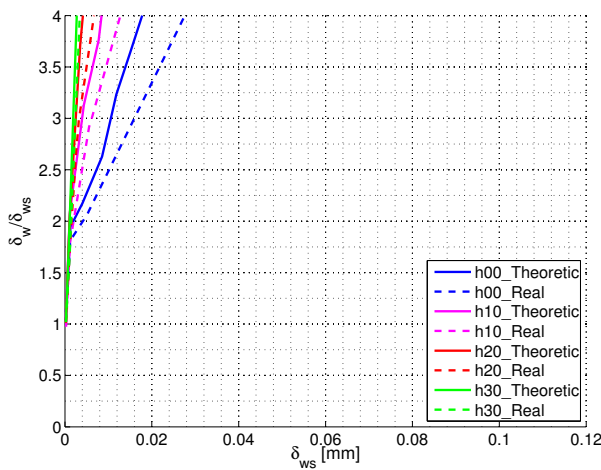


A.37a: $m = 10$.

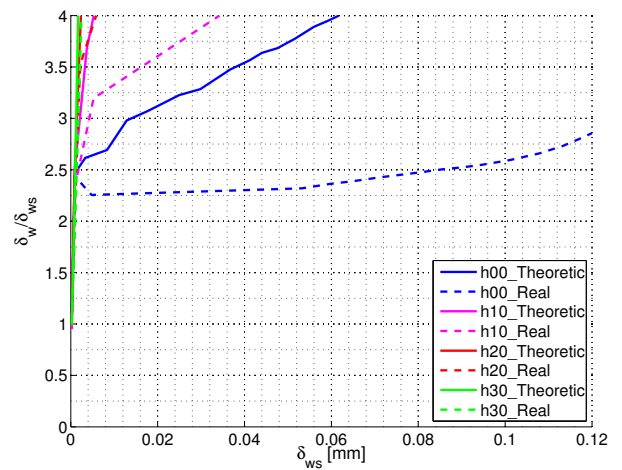


A.37b: $m = 20$.

Figure A.37: Crack in middle of HAZ, HAZ width 0.25 mm, w20, $T = -0.25\sigma_{0.2}$.

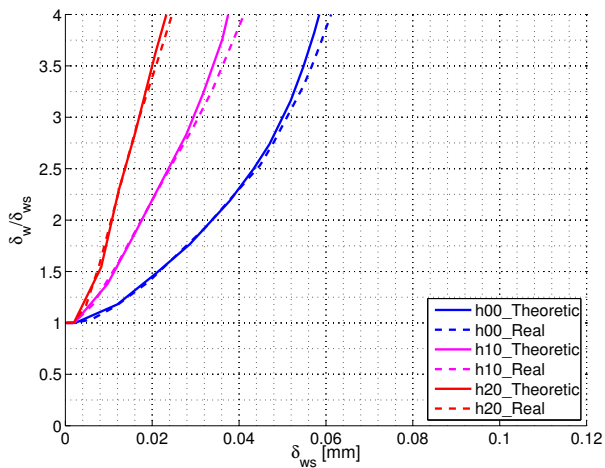


A.38a: $m = 10$.

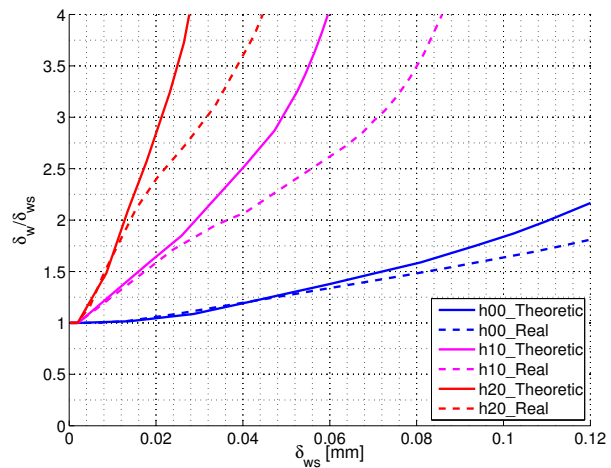


A.38b: $m = 20$.

Figure A.38: Crack in middle of HAZ, HAZ width 0.25 mm, w20, $T = -0.5\sigma_{0.2}$.

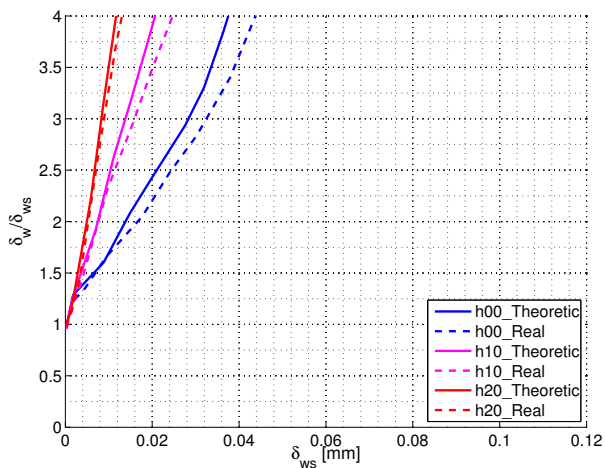


A.39a: $m = 10$.

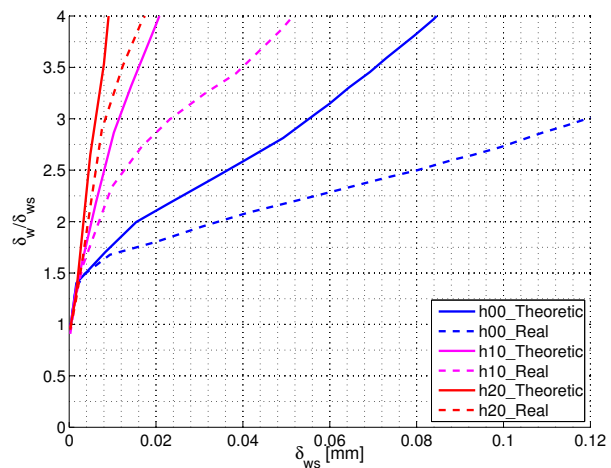


A.39b: $m = 20$.

Figure A.39: Crack in middle of HAZ, HAZ width 0.5 mm, w_{00} , $T=0$.

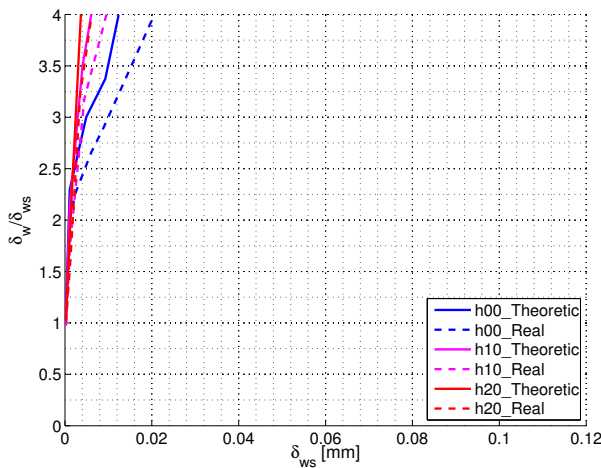


A.40a: $m = 10$.

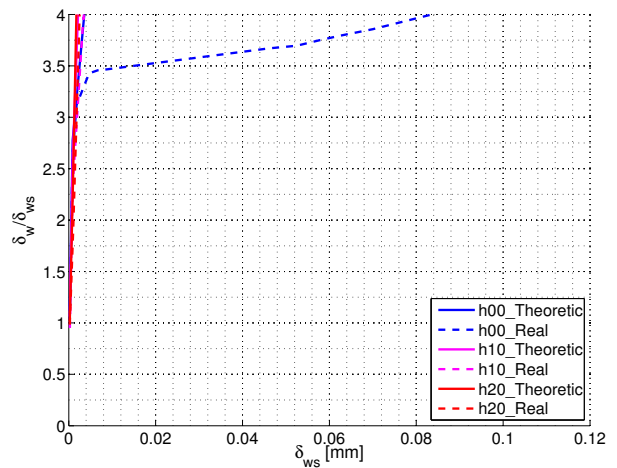


A.40b: $m = 20$.

Figure A.40: Crack in middle of HAZ, HAZ width 0.5 mm, w_{00} , $T = -0.25\sigma_{0.2}$.

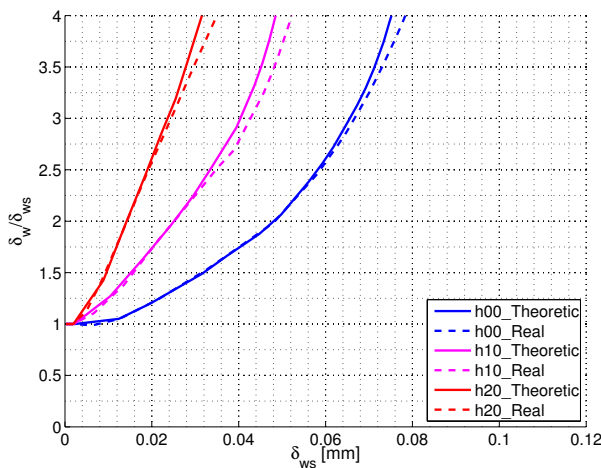


A.41a: $m = 10$.

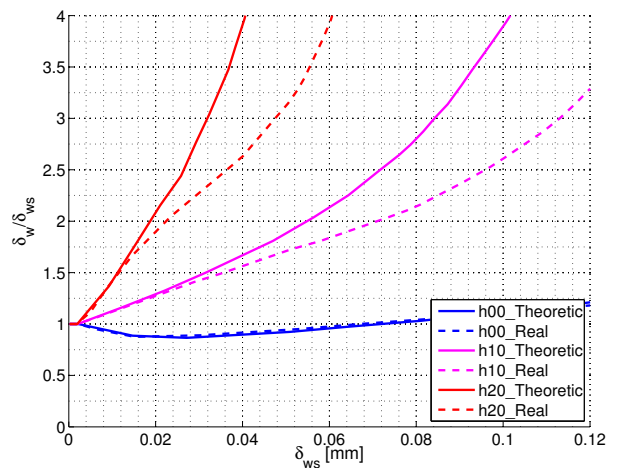


A.41b: $m = 20$.

Figure A.41: Crack in middle of HAZ, HAZ width 0.5 mm, w00, $T = -0.5\sigma_{0.2}$.

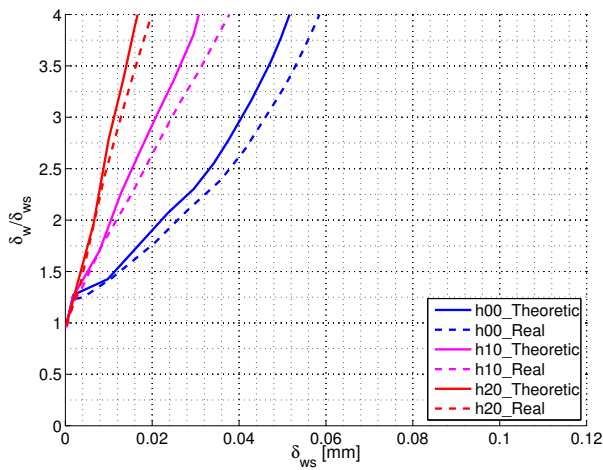
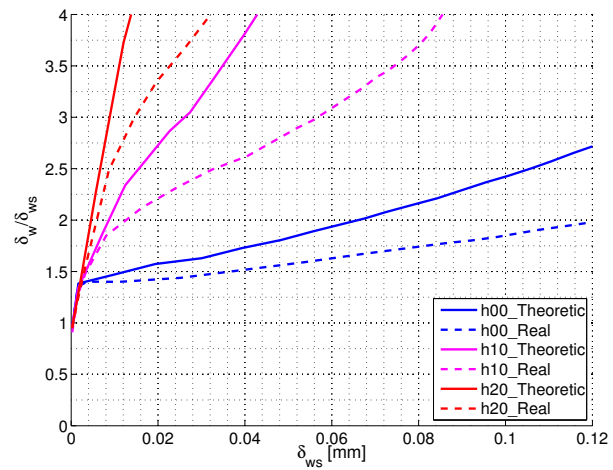
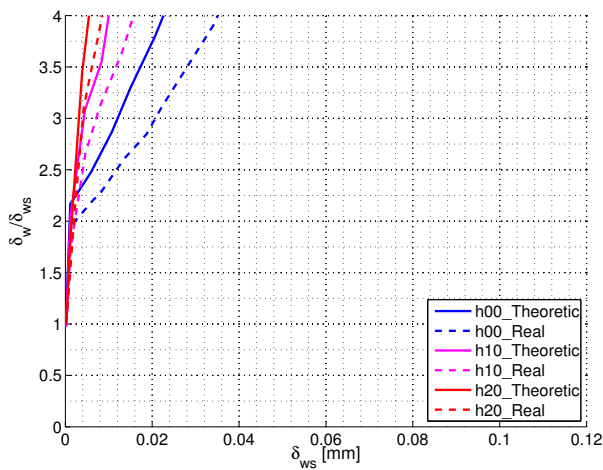
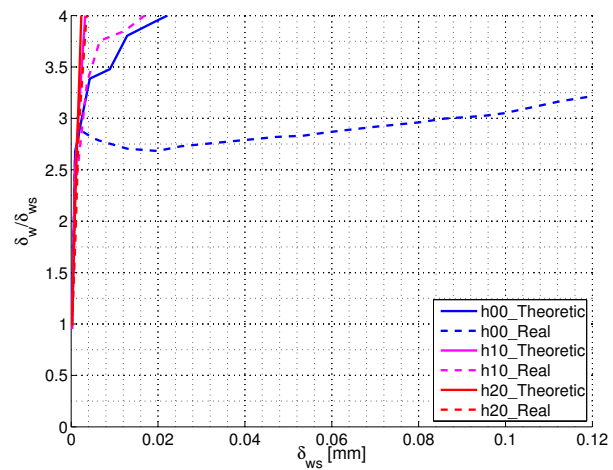


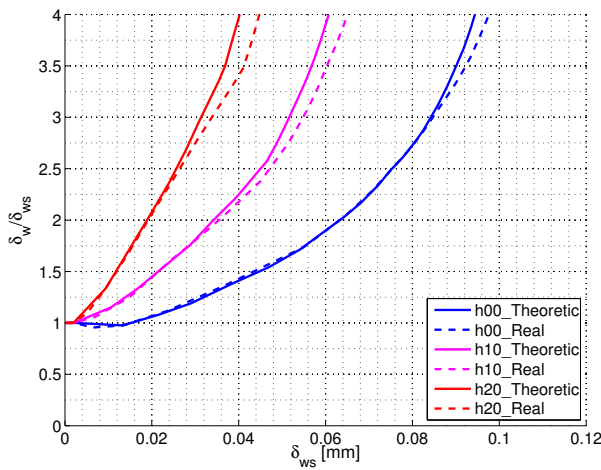
A.42a: $m = 10$.



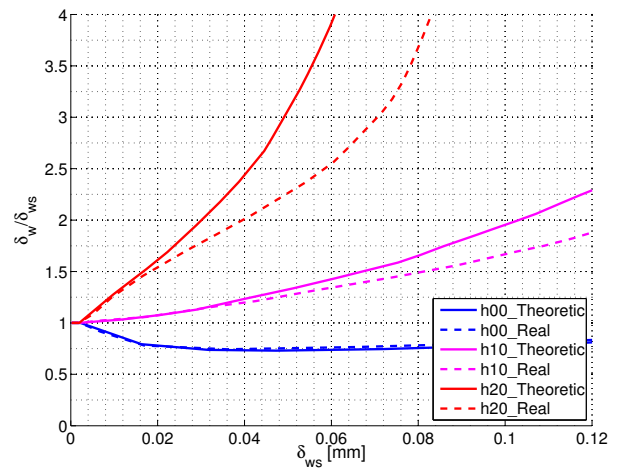
A.42b: $m = 20$.

Figure A.42: Crack in middle of HAZ, HAZ width 0.5 mm, w10, $T = 0$.

A.43a: $m = 10$.A.43b: $m = 20$.Figure A.43: Crack in middle of HAZ, HAZ width 0.5 mm, w_{10} , $T = -0.25\sigma_{0.2}$.A.44a: $m = 10$.A.44b: $m = 20$.Figure A.44: Crack in middle of HAZ, HAZ width 0.5 mm, w_{10} , $T = -0.5\sigma_{0.2}$.

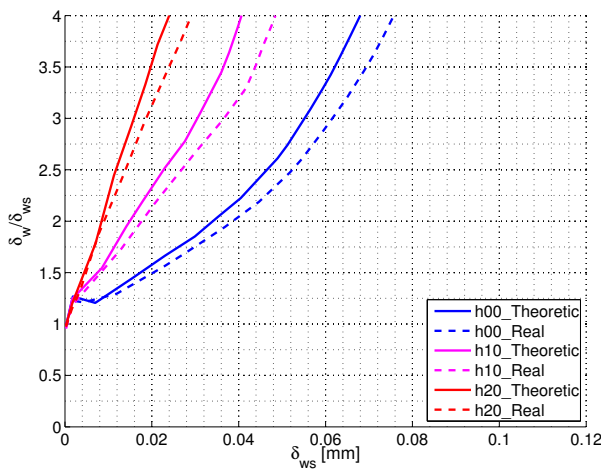


A.45a: $m = 10$.

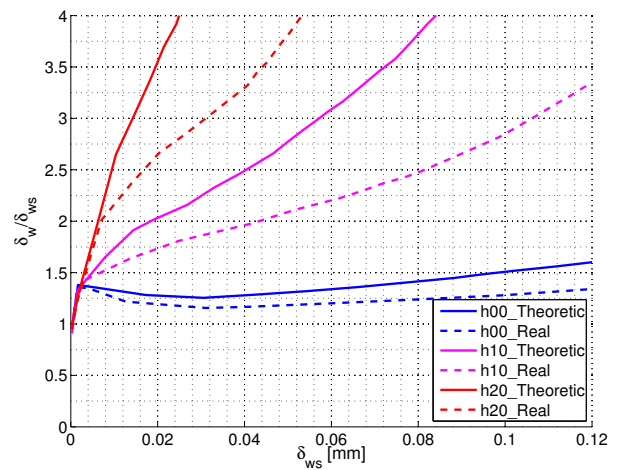


A.45b: $m = 20$.

Figure A.45: Crack in middle of HAZ, HAZ width 0.5 mm, w20, $T=0$.

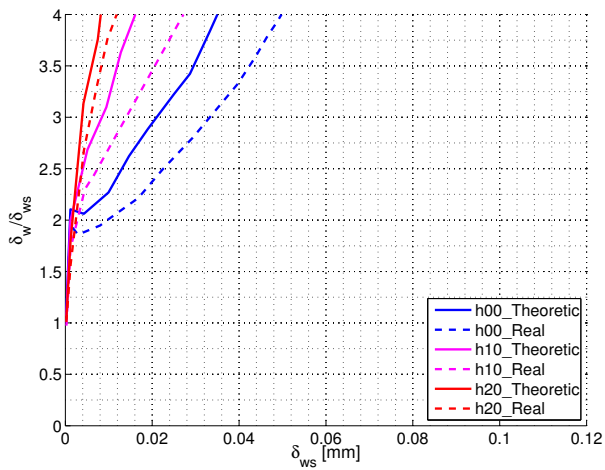


A.46a: $m = 10$.

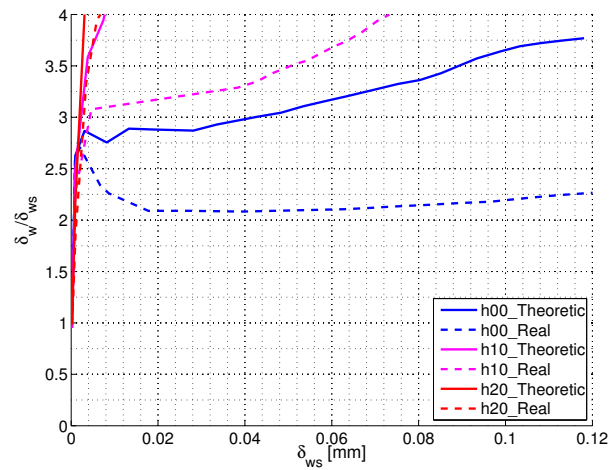


A.46b: $m = 20$.

Figure A.46: Crack in middle of HAZ, HAZ width 0.5 mm, w20, $T=-0.25\sigma_{0.2}$.

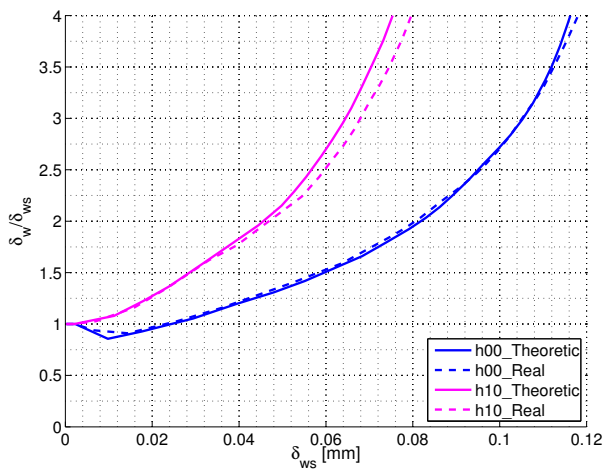


A.47a: $m = 10$.

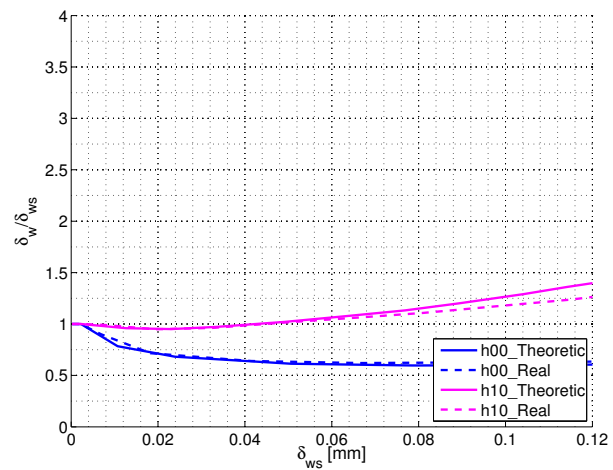


A.47b: $m = 20$.

Figure A.47: Crack in middle of HAZ, HAZ width 0.5 mm, w20, $T = -0.5\sigma_{0.2}$.

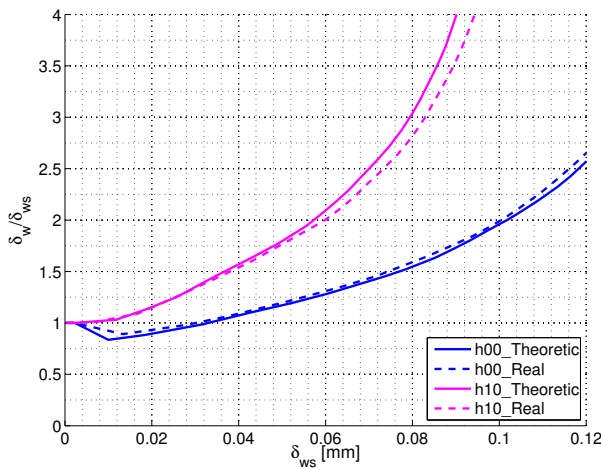


A.48a: $m = 10$.

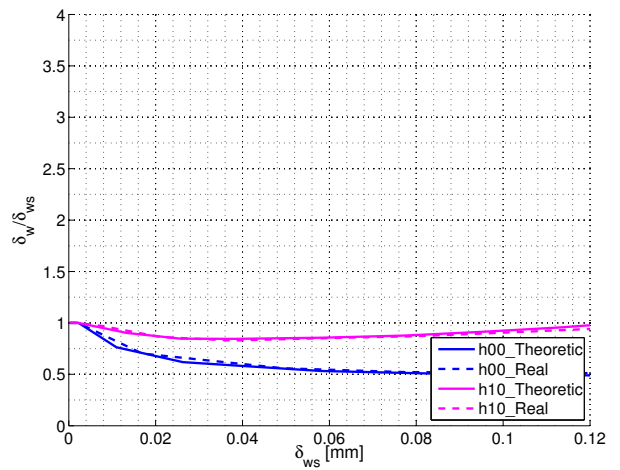


A.48b: $m = 20$.

Figure A.48: Crack in middle of HAZ, HAZ width 0.5 mm, w30, $T = 0$.

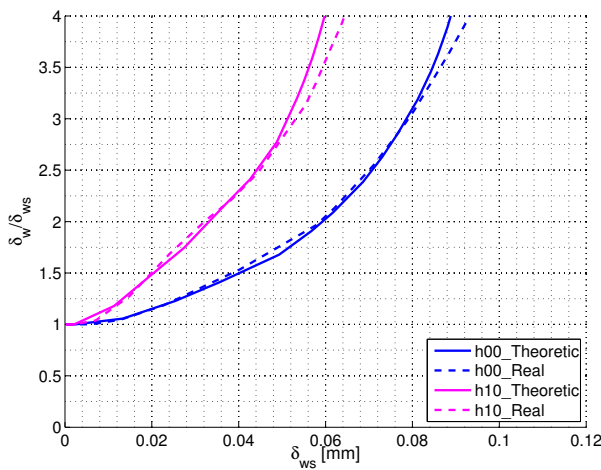


A.49a: $m = 10$.

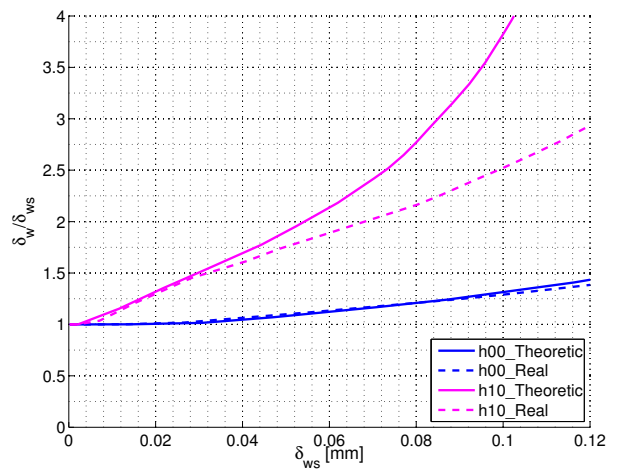


A.49b: $m = 20$.

Figure A.49: Crack in middle of HAZ, HAZ width 0.5 mm, w40, $T=0$.

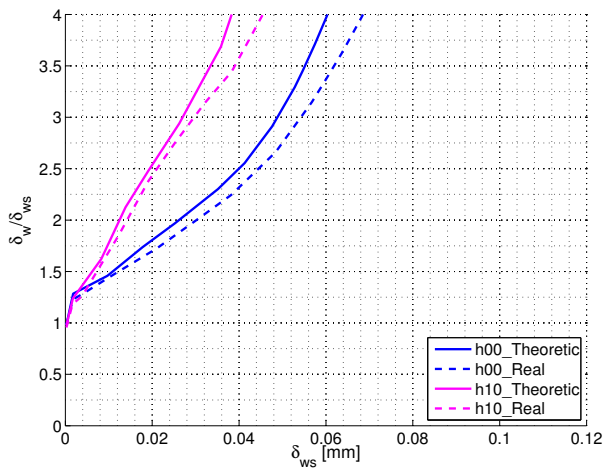
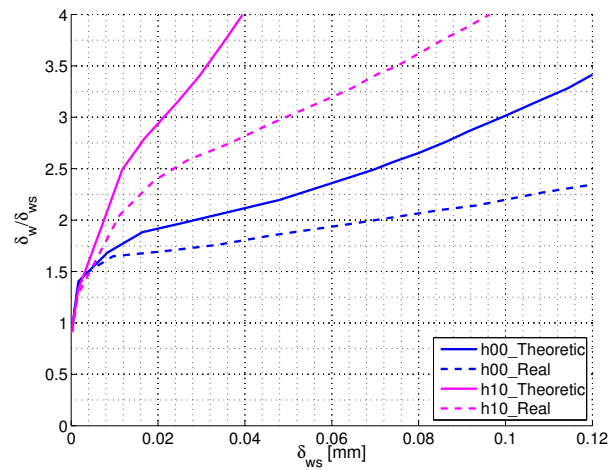
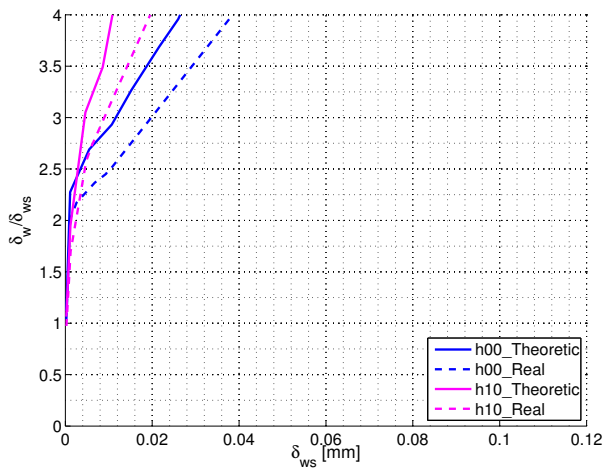
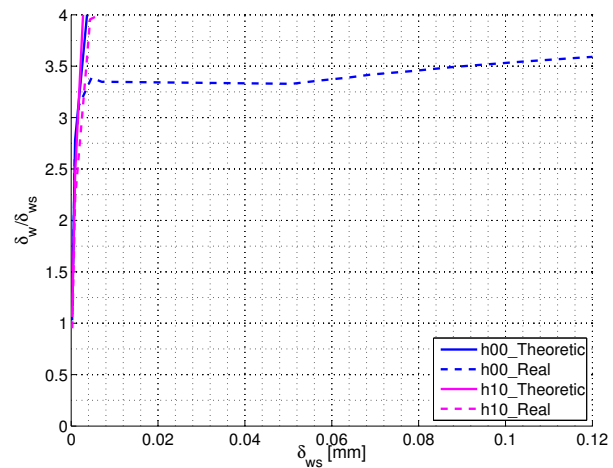


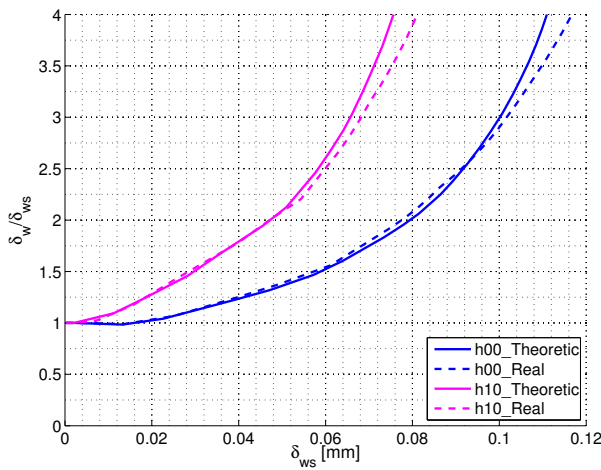
A.50a: $m = 10$.



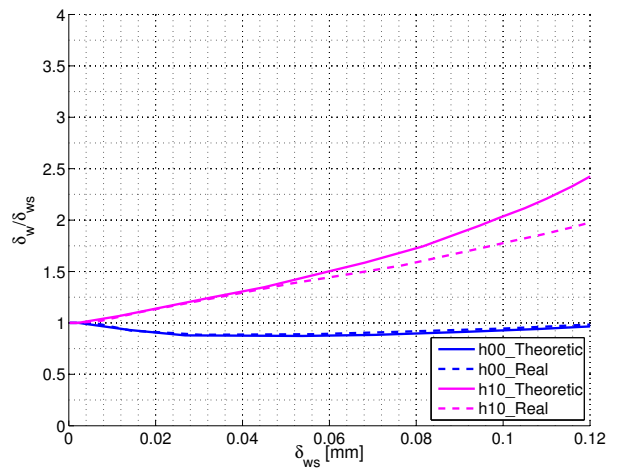
A.50b: $m = 20$.

Figure A.50: Crack in middle of HAZ, HAZ width 1 mm, w00, $T=0$.

A.51a: $m = 10$.A.51b: $m = 20$.Figure A.51: Crack in middle of HAZ, HAZ width 1 mm, $w00$, $T = -0.25\sigma_{0.2}$.A.52a: $m = 10$.A.52b: $m = 20$.Figure A.52: Crack in middle of HAZ, HAZ width 1 mm, $w00$, $T = -0.5\sigma_{0.2}$.

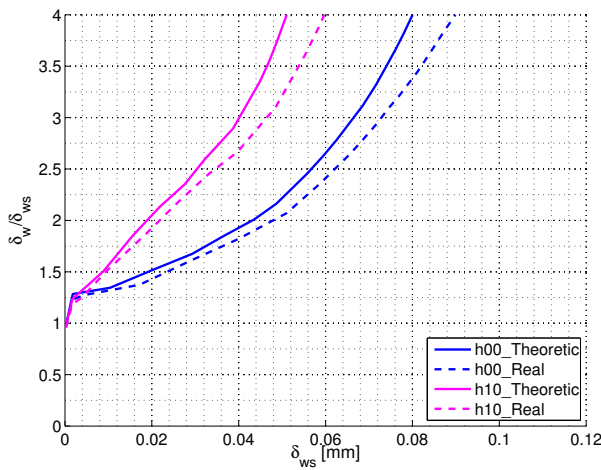


A.53a: $m = 10$.

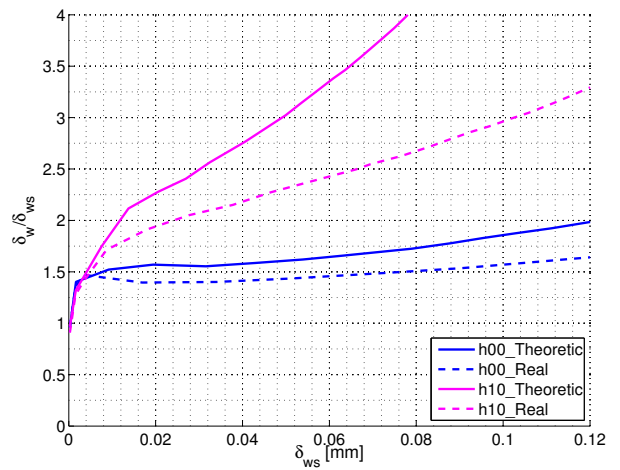


A.53b: $m = 20$.

Figure A.53: Crack in middle of HAZ, HAZ width 1 mm, w10, $T=0$.

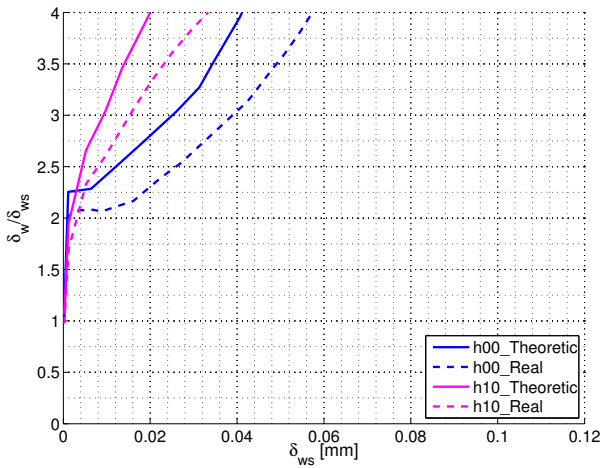


A.54a: $m = 10$.

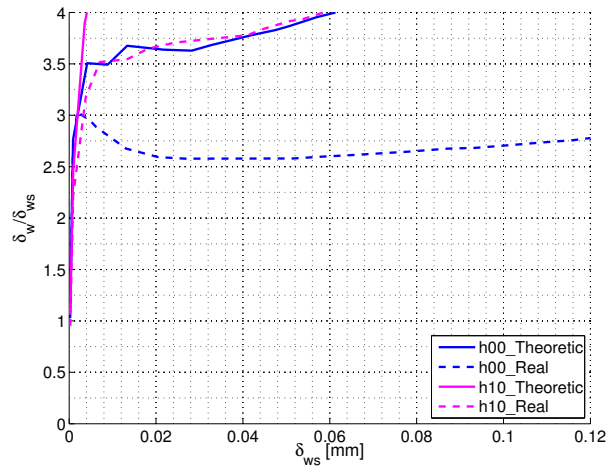


A.54b: $m = 20$.

Figure A.54: Crack in middle of HAZ, HAZ width 1 mm, w10, $T = -0.25\sigma_{0.2}$.

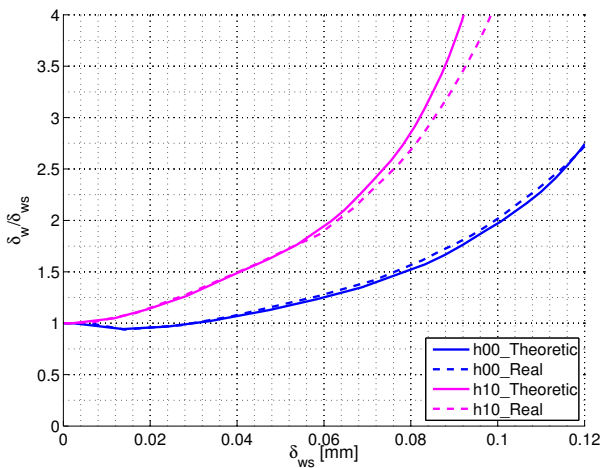


A.55a: $m = 10$.

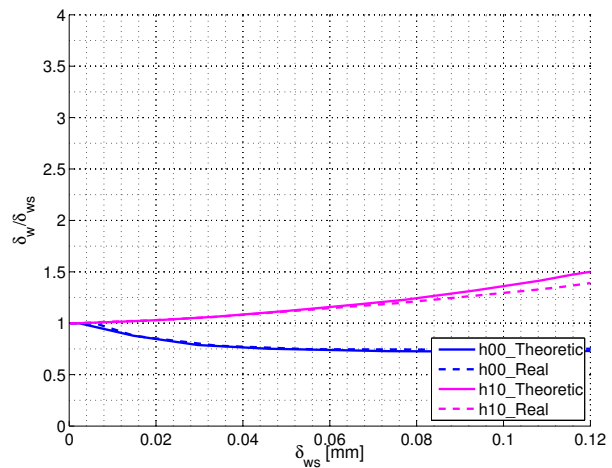


A.55b: $m = 20$.

Figure A.55: Crack in middle of HAZ, HAZ width 1 mm, w_{10} , $T = -0.5\sigma_{0.2}$.

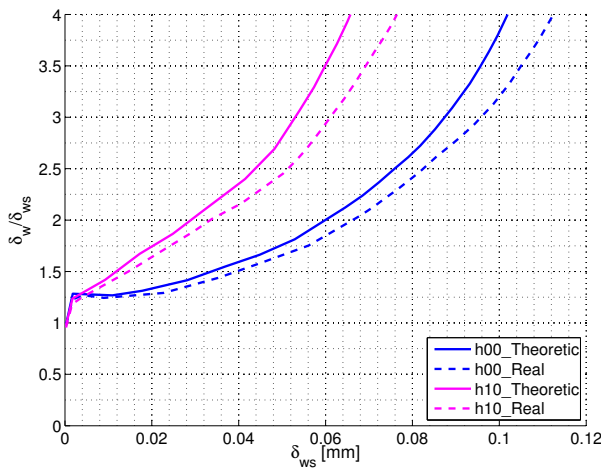


A.56a: $m = 10$.

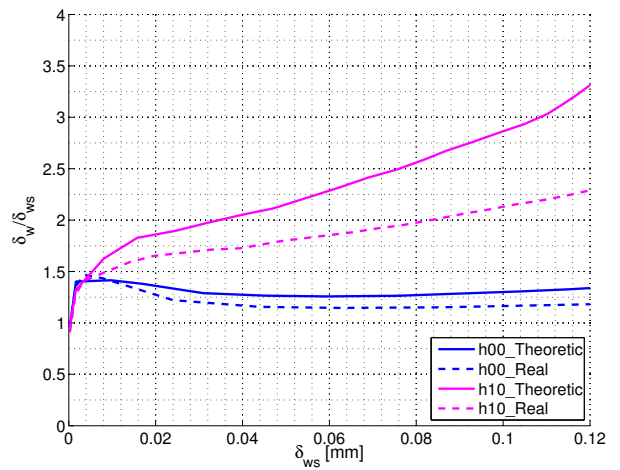


A.56b: $m = 20$.

Figure A.56: Crack in middle of HAZ, HAZ width 1 mm, w_{20} , $T=0$.

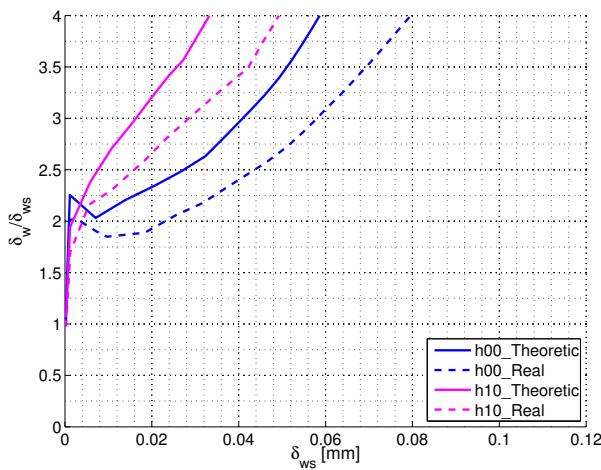


A.57a: $m = 10$.

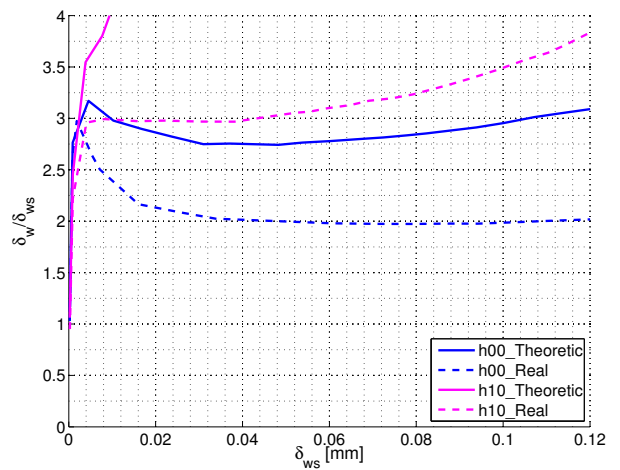


A.57b: $m = 20$.

Figure A.57: Crack in middle of HAZ, HAZ width 1 mm, w20, $T = -0.25\sigma_{0.2}$.



A.58a: $m = 10$.



A.58b: $m = 20$.

Figure A.58: Crack in middle of HAZ, HAZ width 1 mm, w20, $T = -0.5\sigma_{0.2}$.

Appendix B First Python code

This Python code makes the basis model. It first makes the model then partition, seed and mesh it:

```

1 from abaqus import *
2 from abaqusConstants import *
3 backwardCompatibility.setValues(includeDeprecated=True, reportDeprecated=
  False)
4
5 import sketch
6 import part
7 import fields
8 import mesh
9
10 from part import *
11 from material import *
12 from section import *
13 from assembly import *
14 from step import *
15 from interaction import *
16 from load import *
17 from mesh import *
18 from job import *
19 from sketch import *
20 from visualization import *
21 from connectorBehavior import *
22
23 #Input:
24 A=0.5 #Length of HAZ on crack
25
26 ###For s=0.01 and Z=1.92
27 N=int(192) #Number of elements along one edge
28 n=int(5) #Number of scales from fine to coarse mesh
29 S=0.01 #Element size on fine mesh
30 Z=N*S #Approximately half size of fine mesh region
  approximately 1.92mm
31 ###
32
33 B=2*Z-A #Length of HAZ on ligament
34 r=5000 #Radius MBL
35 C=Z #Half thickness of fine region
36 R=0.01 #Radius of crack tip
37
38 ##PART:
39
40 #Make model: MBLmodell
41 MBLmodel = mdb.Model(name='Model-1')
42
43 #Make sketch of geometry
44 rr=sqrt(r**2-R**2)
45 MBLsketch = MBLmodel.ConstrainedSketch(name='MBLs', sheetSize=2*r)
46 MBLsketch.Arc3Points(point1=(-rr,R), point2=(r,0), point3=(0,r))
47 MBLsketch.Arc3Points(point1=(-rr,-R), point2=(r,0), point3=(0,-r))
48 MBLsketch.Arc3Points(point1=(0,R), point2=(0,-R), point3=(R,0))
49 MBLsketch.Line(point1=(-rr,R), point2=(0,R))

```

```
50 MBLsketch.Line(point1=(-rr,-R), point2=(0,-R))
51
52 #Make part from sketch
53 MBLpart = MBLmodel.Part(name='MBLp', dimensionality=TWO_D_PLANAR, type=
    DEFORMABLE_BODY)
54 MBLpart.BaseShell(sketch=MBLsketch)
55
56 ##Partition: Generate lines for meshing:
57
58 # The mesh consist of five zones
59 # The first zone is the area around the crack that have a circular mesh in
    front of the crack tip and a square mesh behind. This zone has a radius
    defined by the variabel "le" and starts form zero.
60 # The second zone is the zone where the square mesh behind the crack
    becomes a circular mesh. This zone is defined from "le" to a square
    around the crack with size "Si" times "Si"
61 # The third zone is the square zone that goes form x=-A to x=B and from y=-
    C to y=C
62 # The fourth zone is a transition zone from the fine square mesh to the
    coarse circular. The zone is square.
63 # The fifth zone is the circular mesh out to the surface of the model.
64
65 #Crack tip area, zone one and two
66
67 CRACKTIPsketch = MBLmodel.ConstrainedSketch(name='CRACKTIP',sheetSize=2*r)
    #Define sketch
68
69 Si=0.05          #size of transition sone to square mesh, zone two.
70 ant=int(Si/S)    #half the number of elements along a side of zone two
71
72 b=pi*R/(ant*4)   #element size at crack tip
73 le=0
74 for i in range(ant):
75     bn=pi*(R+le)/(ant*4) #element size outwards
76     le=le+bn           #length of transition zone to round mesh
77 biasratio=bn/b       #Bias ratio for meshing
78
79 #lines around zone one
80 CRACKTIPsketch.Line(point1=(0,R+le), point2=(-le,R+le))
81 CRACKTIPsketch.Line(point1=(-le,R), point2=(-le,R+le))
82 CRACKTIPsketch.Line(point1=(0,-R-le), point2=(-le,-R-le))
83 CRACKTIPsketch.Line(point1=(-le,-R), point2=(-le,-R-le))
84 #CRACKTIPsketch.Arc3Points(point1=(0,R+le), point2=(0,-R-le), point3=(R+le,
    0))
85
86 xone=R+le
87 yone=0
88 xtwo=(R+le)/sqrt(2)
89 ytwo=(R+le)/sqrt(2)
90 xthree=R+le
91 ythree=0
92 xfour=(R+le)/sqrt(2)
93 yfour=-(R+le)/sqrt(2)
94 for i in range(ant):
95     xstart=R*cos((i+1)*pi/(ant*4))
96     ystart=R*sin((i+1)*pi/(ant*4))
```

```

97  xsi=Si
98  ysi=(i+1)*S
99  xend=xstart+le*(xsi-xstart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
100 yend=ystart+le*(ysi-ystart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
101     CRACKTIPsketch.Line(point1=(xstart, ystart), point2=(xend, yend))
102 CRACKTIPsketch.Line(point1=(xone, yone), point2=(xend, yend))
103 xone=xend
104 yone=yend
105
106 xstart=R*cos((i+1+ant)*pi/(ant*4))
107 ystart=R*sin((i+1+ant)*pi/(ant*4))
108 xsi=Si-(i+1)*S
109 ysi=Si
110 xend=xstart+le*(xsi-xstart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
111 yend=ystart+le*(ysi-ystart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
112     CRACKTIPsketch.Line(point1=(xstart, ystart), point2=(xend, yend))
113 CRACKTIPsketch.Line(point1=(xtwo, ytwo), point2=(xend, yend))
114 xtwo=xend
115 ytwo=yend
116
117 xstart=R*cos((i+1)*pi/(ant*4))
118 ystart=-R*sin((i+1)*pi/(ant*4))
119 xsi=Si
120 ysi=-(i+1)*S
121 xend=xstart+le*(xsi-xstart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
122 yend=ystart+le*(ysi-ystart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
123     CRACKTIPsketch.Line(point1=(xstart, ystart), point2=(xend, yend))
124 CRACKTIPsketch.Line(point1=(xthree, ythree), point2=(xend, yend))
125 xthree=xend
126 ythree=yend
127
128 xstart=R*cos((i+1+ant)*pi/(ant*4))
129 ystart=-R*sin((i+1+ant)*pi/(ant*4))
130 xsi=Si-(i+1)*S
131 ysi=-Si
132 xend=xstart+le*(xsi-xstart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
133 yend=ystart+le*(ysi-ystart)/sqrt((xsi-xstart)**2+(ysi-ystart)**2)
134     CRACKTIPsketch.Line(point1=(xstart, ystart), point2=(xend, yend))
135 CRACKTIPsketch.Line(point1=(xfour, yfour), point2=(xend, yend))
136 xfour=xend
137 yfour=yend
138 CRACKTIPsketch.Line(point1=(R, 0), point2=(R+le, 0))
139
140 #lines in front of crack, circular through zone one.
141 #for i in range(2*ant):
142 # xstart=R*cos((i+1)*pi/(ant*4))
143 # xend=
144 # xtemp=R*cos((i+1)*pi/(ant*4))
145 #     CRACKTIPsketch.Line(point1=(R*cos((i+1)*pi/(ant*4)), R*sin((i+1)*pi/(
146 #         ant*4))), point2=((R+le)*cos((i+1)*pi/(ant*4)), (R+le)*sin((i+1)*pi/(ant
147 #         *4))))
148 #     CRACKTIPsketch.Line(point1=(R*cos((i+1)*pi/(ant*4)), -R*sin((i+1)*pi/(
149 #         ant*4))), point2=((R+le)*cos((i+1)*pi/(ant*4)), -(R+le)*sin((i+1)*pi/(
150 #         ant*4))))))
151
152 #box around zone two

```

```

149 CRACKTIPsketch.Line(point1=(-Si,R), point2=(-Si,Si))
150 CRACKTIPsketch.Line(point1=(Si,Si), point2=(Si,-Si))
151 CRACKTIPsketch.Line(point1=(-Si,-Si), point2=(-Si,-R))
152 CRACKTIPsketch.Line(point1=(-A,Si), point2=(B,Si))
153 CRACKTIPsketch.Line(point1=(-A,-Si), point2=(B,-Si))
154
155 #Lines throught zone two behind crack
156 CRACKTIPsketch.Line(point1=(-le,R+le), point2=(-Si,Si))
157 CRACKTIPsketch.Line(point1=(-le,-R-le), point2=(-Si,-Si))
158
159 #Lines throught zone one and two in front of crack
160 CRACKTIPsketch.Line(point1=(R,0), point2=(Si,0))
161 CRACKTIPsketch.Line(point1=(0,R), point2=(0,Si))
162 CRACKTIPsketch.Line(point1=(0,-R), point2=(0,-Si))
163 CRACKTIPsketch.Line(point1=((R)/sqrt(2),(R)/sqrt(2)), point2=(Si,Si))
164 CRACKTIPsketch.Line(point1=((R)/sqrt(2),-(R)/sqrt(2)), point2=(Si,-Si))
165
166 MESHpartiton = MBLpart.PartitionFaceBySketch(faces=MBLpart.faces, sketch=
    CRACKTIPsketch) #Partiion faces by sketch, zone one and two
167
168 #Lines between fine and coarse mesh, zone four
169 MESHsketch = MBLmodel.ConstrainedSketch(name='MESH',sheetSize=2*r)
    #Define sketch
170 k=int(N) #Old number of nodes
171 Ss=S #Old element size
172 Aa=A #Old position A
173 Bb=B #Old position B
174 Cc=C #Old position C
175 for i in range(n):
176     l=int(k/2) #New number of nodes
177     SS=Ss*2 #New element size
178     AA=Aa+SS #New length A
179     aa=Aa+Ss #Between new and old A
180     BB=Bb+SS #New length B
181     bb=Bb+Ss #Between new and old B
182     CC=Cc+SS #New length C
183     cc=Cc+Ss #Between new and old C
184     LINES=((-AA,-C),(-AA,-R),(-AA,-R),(-AA,C),(BB,-C),(BB,C),(-A,CC),(B,CC)
        ,(-A,-CC),(B,-CC)) #Position for main lines, except first
185     for p in range(5):
186         MESHsketch.Line(point1=LINES[int(2*p)], point2=LINES[int((p*2+1)]))
            #Make main lines, except first
187     for j in range(k):
188         js=Ss+(j-1)*SS
189         JS=(1+j-1)*SS
190     cJ=-A+Ss+j*SS
191     CJ=-A+j*SS
192     JSA=JS
193     jsA=js
194     if i==0:
195         if j==1:
196             JSA=(1+j-1)*SS+Ss/2 # Get better element shape at crack
197         elif j==1-2:
198             JSA=(1+j-1)*SS-Ss/2
199     elif i==1:
200         if j==1:

```

```

201     jsA=Ss+(j-1)*SS+Ss/4
202     elif j==l-1:
203         jsA=Ss+(j-1)*SS-Ss/4
204         LINES=((-Aa, jsA),(-aa, jsA),(-Aa, JS),(-AA, JSA), (bb, js), (Bb, js), (Bb,
                JS), (BB, JS), (cJ, Cc), (cJ, cc), (CJ, Cc), (CJ, CC), (cJ, -Cc), (cJ, -cc), (
                CJ, -Cc), (CJ, -CC))#Lines normal to main line
205         for p in range(len(LINES)/2):
206             MESHsketch.Line(point1=LINES[int(2*p)], point2=LINES[int((p
                *2+1))])
207     for j in range(1):

        #Lines parallel to main line
208     MESHsketch.Line(point1=(-A+j*2*SS,CC), point2=(-A+Ss+j*2*SS, cc))
209     MESHsketch.Line(point1=(-A+2*SS+j*2*SS,CC), point2=(-A+3*Ss+j*2*SS,
        cc))
210     MESHsketch.Line(point1=(-A+Ss+j*2*SS, cc), point2=(-A+3*Ss+j*2*SS, cc
        ))
211     MESHsketch.Line(point1=(-A+j*2*SS,-CC), point2=(-A+Ss+j*2*SS,-cc))
212     MESHsketch.Line(point1=(-A+2*SS+j*2*SS,-CC), point2=(-A+3*Ss+j*2*SS
        ,-cc))
213     MESHsketch.Line(point1=(-A+Ss+j*2*SS,-cc), point2=(-A+3*Ss+j*2*SS,-
        cc))
214     for j in range(1/2):

        #Sloping lines
215     if i==1:
216         if j==0:
217             MESHsketch.Line(point1=(-aa, Ss+Ss/4), point2=(-aa, 3*Ss+j*2*SS))
218             MESHsketch.Line(point1=(-aa, -Ss-Ss/4), point2=(-aa, -3*Ss-j*2*SS
                ))
219         else:
220             MESHsketch.Line(point1=(-aa, Ss+j*2*SS), point2=(-aa, 3*Ss+j*2*SS
                ))
221             MESHsketch.Line(point1=(-aa, -Ss-j*2*SS), point2=(-aa, -3*Ss-j*2*
                SS))
222     else:
223         MESHsketch.Line(point1=(-aa, Ss+j*2*SS), point2=(-aa, 3*Ss+j*2*SS))
224         MESHsketch.Line(point1=(-aa, -Ss-j*2*SS), point2=(-aa, -3*Ss-j*2*SS
                ))
225
226         MESHsketch.Line(point1=(-AA, 2*SS+j*2*SS), point2=(-aa, 3*Ss+j*2*SS))
227         MESHsketch.Line(point1=(-AA, -2*SS-j*2*SS), point2=(-aa, -3*Ss-j*2*SS
                ))
228     if j==0:                                     #Stop loop lines from going into the crack
229         if i==1:
230             MESHsketch.Line(point1=(-AA, R), point2=(-aa, Ss+Ss/4))
231             MESHsketch.Line(point1=(-AA, -R), point2=(-aa, -Ss-Ss/4))
232         else:
233             MESHsketch.Line(point1=(-AA, R), point2=(-aa, Ss+j*2*SS))
234             MESHsketch.Line(point1=(-AA, -R), point2=(-aa, -Ss-j*2*SS))
235     else:
236         MESHsketch.Line(point1=(-AA, j*2*SS), point2=(-aa, Ss+j*2*SS))
237         MESHsketch.Line(point1=(-AA, -j*2*SS), point2=(-aa, -Ss-j*2*SS))
238
239         MESHsketch.Line(point1=(bb, Ss+j*2*SS), point2=(bb, 3*Ss+j*2*SS))
240         MESHsketch.Line(point1=(bb, -Ss-j*2*SS), point2=(bb, -3*Ss-j*2*SS))

```



```

241     MESHsketch.Line(point1=(BB,2*SS+j*2*SS), point2=(bb,3*Ss+j*2*SS))
242     MESHsketch.Line(point1=(BB,-2*SS-j*2*SS), point2=(bb,-3*Ss-j*2*SS))
243     MESHsketch.Line(point1=(BB,j*2*SS), point2=(bb,Ss+j*2*SS))
244     MESHsketch.Line(point1=(BB,-j*2*SS), point2=(bb,-Ss-j*2*SS))
245
246     Aa=AA           #Update Aa
247     Bb=BB           #Update Bb
248     Cc=CC           #Update Cc
249     Ss=SS           #Update Ss
250     k=int(1)       #Update k
251
252     MESHsketch.Line(point1=(-A,-CC), point2=(-A,CC)) #
        Inner main lines
253     MESHsketch.Line(point1=(B,-CC), point2=(B,CC))
254     MESHsketch.Line(point1=(-AA,C), point2=(BB,C))
255     MESHsketch.Line(point1=(-AA,-C), point2=(BB,-C))
256
257     MESHpartiton = MBLpart.PartitionFaceBySketch(faces=MBLpart.faces, sketch=
        MESHsketch) #Partiion faces by sketch, zone four
258
259     #Corners in the the area between fine and coarse mesh, zone four
260
261     CORNERSketch = MBLmodel.ConstrainedSketch(name='CORNER',sheetSize=2*r) #
        Define sketch
262     s=2*S
263
264     CORNERSketch.Line(point1=(-A-s,C), point2=(-A-s,C+3*s)) #upper left corner
265     CORNERSketch.Line(point1=(-A-3*s,C), point2=(-A-3*s,C+3*s))
266     CORNERSketch.Line(point1=(-A-7*s,C), point2=(-A-7*s,C+7*s))
267     CORNERSketch.Line(point1=(-A-15*s,C), point2=(-A-15*s,C+15*s))
268     CORNERSketch.Line(point1=(-A-31*s,C), point2=(-A-31*s,C+31*s))
269     CORNERSketch.Line(point1=(-A,C+s), point2=(-A-3*s,C+s))
270     CORNERSketch.Line(point1=(-A,C+3*s), point2=(-A-3*s,C+3*s))
271     CORNERSketch.Line(point1=(-A,C+7*s), point2=(-A-7*s,C+7*s))
272     CORNERSketch.Line(point1=(-A,C+15*s), point2=(-A-15*s,C+15*s))
273     CORNERSketch.Line(point1=(-A,C+31*s), point2=(-A-31*s,C+31*s))
274     CORNERSketch.Line(point1=(-A-3*s,C+3*s), point2=(-A-31*s,C+31*s))
275     CORNERSketch.Line(point1=(-A-3*s,C+s), point2=(-A-7*s,C+7/2*s))
276     CORNERSketch.Line(point1=(-A-7*s,C+7/2*s), point2=(-A-31*s,C+31/2*s))
277     CORNERSketch.Line(point1=(-A-s,C+3*s), point2=(-A-7/2*s,C+7*s))
278     CORNERSketch.Line(point1=(-A-7/2*s,C+7*s), point2=(-A-31/2*s,C+31*s))
279
280     CORNERSketch.Line(point1=(B+s,C), point2=(B+s,C+3*s)) #upper right corner
281     CORNERSketch.Line(point1=(B+3*s,C), point2=(B+3*s,C+3*s))
282     CORNERSketch.Line(point1=(B+7*s,C), point2=(B+7*s,C+7*s))
283     CORNERSketch.Line(point1=(B+15*s,C), point2=(B+15*s,C+15*s))
284     CORNERSketch.Line(point1=(B+31*s,C), point2=(B+31*s,C+31*s))
285     CORNERSketch.Line(point1=(B,C+s), point2=(B+3*s,C+s))
286     CORNERSketch.Line(point1=(B,C+3*s), point2=(B+3*s,C+3*s))
287     CORNERSketch.Line(point1=(B,C+7*s), point2=(B+7*s,C+7*s))
288     CORNERSketch.Line(point1=(B,C+15*s), point2=(B+15*s,C+15*s))
289     CORNERSketch.Line(point1=(B,C+31*s), point2=(B+31*s,C+31*s))
290     CORNERSketch.Line(point1=(B+3*s,C+3*s), point2=(B+31*s,C+31*s))
291     CORNERSketch.Line(point1=(B+3*s,C+s), point2=(B+7*s,C+7/2*s))
292     CORNERSketch.Line(point1=(B+7*s,C+7/2*s), point2=(B+31*s,C+31/2*s))
293     CORNERSketch.Line(point1=(B+s,C+3*s), point2=(B+7/2*s,C+7*s))

```

```

294 CORNERsketch.Line(point1=(B+7/2*s,C+7*s), point2=(B+31/2*s,C+31*s))
295
296 CORNERsketch.Line(point1=(-A-s,-C), point2=(-A-s,-C-3*s)) #lower left
    corner
297 CORNERsketch.Line(point1=(-A-3*s,-C), point2=(-A-3*s,-C-3*s))
298 CORNERsketch.Line(point1=(-A-7*s,-C), point2=(-A-7*s,-C-7*s))
299 CORNERsketch.Line(point1=(-A-15*s,-C), point2=(-A-15*s,-C-15*s))
300 CORNERsketch.Line(point1=(-A-31*s,-C), point2=(-A-31*s,-C-31*s))
301 CORNERsketch.Line(point1=(-A,-C-s), point2=(-A-3*s,-C-s))
302 CORNERsketch.Line(point1=(-A,-C-3*s), point2=(-A-3*s,-C-3*s))
303 CORNERsketch.Line(point1=(-A,-C-7*s), point2=(-A-7*s,-C-7*s))
304 CORNERsketch.Line(point1=(-A,-C-15*s), point2=(-A-15*s,-C-15*s))
305 CORNERsketch.Line(point1=(-A,-C-31*s), point2=(-A-31*s,-C-31*s))
306 CORNERsketch.Line(point1=(-A-3*s,-C-3*s), point2=(-A-31*s,-C-31*s))
307 CORNERsketch.Line(point1=(-A-3*s,-C-s), point2=(-A-7*s,-C-7/2*s))
308 CORNERsketch.Line(point1=(-A-7*s,-C-7/2*s), point2=(-A-31*s,-C-31/2*s))
309 CORNERsketch.Line(point1=(-A-s,-C-3*s), point2=(-A-7/2*s,-C-7*s))
310 CORNERsketch.Line(point1=(-A-7/2*s,-C-7*s), point2=(-A-31/2*s,-C-31*s))
311
312 CORNERsketch.Line(point1=(B+s,-C), point2=(B+s,-C-3*s)) #lower right
    corner
313 CORNERsketch.Line(point1=(B+3*s,-C), point2=(B+3*s,-C-3*s))
314 CORNERsketch.Line(point1=(B+7*s,-C), point2=(B+7*s,-C-7*s))
315 CORNERsketch.Line(point1=(B+15*s,-C), point2=(B+15*s,-C-15*s))
316 CORNERsketch.Line(point1=(B+31*s,-C), point2=(B+31*s,-C-31*s))
317 CORNERsketch.Line(point1=(B,-C-s), point2=(B+3*s,-C-s))
318 CORNERsketch.Line(point1=(B,-C-3*s), point2=(B+3*s,-C-3*s))
319 CORNERsketch.Line(point1=(B,-C-7*s), point2=(B+7*s,-C-7*s))
320 CORNERsketch.Line(point1=(B,-C-15*s), point2=(B+15*s,-C-15*s))
321 CORNERsketch.Line(point1=(B,-C-31*s), point2=(B+31*s,-C-31*s))
322 CORNERsketch.Line(point1=(B+3*s,-C-3*s), point2=(B+31*s,-C-31*s))
323 CORNERsketch.Line(point1=(B+3*s,-C-s), point2=(B+7*s,-C-7/2*s))
324 CORNERsketch.Line(point1=(B+7*s,-C-7/2*s), point2=(B+31*s,-C-31/2*s))
325 CORNERsketch.Line(point1=(B+s,-C-3*s), point2=(B+7/2*s,-C-7*s))
326 CORNERsketch.Line(point1=(B+7/2*s,-C-7*s), point2=(B+31/2*s,-C-31*s))
327
328 MESHcorner = MBLpart.PartitionFaceBySketch(faces=MBLpart.faces, sketch=
    CORNERsketch) #Partition face by sketch, zone four.
329
330 # Lines form the coarse square to the surface, zone five
331
332 LINESsketch = MBLmodel.ConstrainedSketch(name='LINES', sheetSize=2*r) #
    Define sketc
333 s=2*S
334 for i in range (7):
335     LINESsketch.Line(point1=(-A-31*s,16*s*(i)), point2=(-r*cos((i)*pi/32)
        ,r*sin((i)*pi/32)))
336     LINESsketch.Line(point1=(B+31*s,16*s*(i)), point2=(r*cos((i)*pi/32),r
        *sin((i)*pi/32)))
337     LINESsketch.Line(point1=(-A-31*s,-16*s*(i)), point2=(-r*cos((i)*pi
        /32),-r*sin((i)*pi/32)))
338     LINESsketch.Line(point1=(B+31*s,-16*s*(i)), point2=(r*cos((i)*pi/32)
        ,-r*sin((i)*pi/32)))
339 for i in range (7):
340     LINESsketch.Line(point1=(-A+s*16*(i),C+31*s), point2=(-r*cos((i+10)*
        pi/32),r*sin((i+10)*pi/32)))

```

```

341     LINESsketch.Line(point1=(-A+s*16*(i),-C-31*s), point2=(-r*cos((i+10)*
        pi/32),-r*sin((i+10)*pi/32)))
342 for i in range(6):
343     LINESsketch.Line(point1=(B-s*16*(i),C+31*s), point2=(r*cos((i+10)*pi
        /32),r*sin((i+10)*pi/32)))
344     LINESsketch.Line(point1=(B-s*16*(i),-C-31*s), point2=(r*cos((i+10)*pi
        /32),-r*sin((i+10)*pi/32)))
345
346
347 LINESsketch.Line(point1=(-A-31*s,C+31/2*s), point2=(-r*cos(7*pi/32),r*sin
        (7*pi/32)))
348 LINESsketch.Line(point1=(-A-31*s,C+31*s), point2=(-r*cos(8*pi/32),r*sin(8*
        pi/32)))
349 LINESsketch.Line(point1=(-A-31/2*s,C+31*s), point2=(-r*cos(9*pi/32),r*sin
        (9*pi/32)))
350
351 LINESsketch.Line(point1=(B+31*s,C+31/2*s), point2=(r*cos(7*pi/32),r*sin(7*
        pi/32)))
352 LINESsketch.Line(point1=(B+31*s,C+31*s), point2=(r*cos(8*pi/32),r*sin(8*pi
        /32)))
353 LINESsketch.Line(point1=(B+31/2*s,C+31*s), point2=(r*cos(9*pi/32),r*sin(9*
        pi/32)))
354
355 LINESsketch.Line(point1=(-A-31*s,-C-31/2*s), point2=(-r*cos(7*pi/32),-r*sin
        (7*pi/32)))
356 LINESsketch.Line(point1=(-A-31*s,-C-31*s), point2=(-r*cos(8*pi/32),-r*sin
        (8*pi/32)))
357 LINESsketch.Line(point1=(-A-31/2*s,-C-31*s), point2=(-r*cos(9*pi/32),-r*sin
        (9*pi/32)))
358
359 LINESsketch.Line(point1=(B+31*s,-C-31/2*s), point2=(r*cos(7*pi/32),-r*sin
        (7*pi/32)))
360 LINESsketch.Line(point1=(B+31*s,-C-31*s), point2=(r*cos(8*pi/32),-r*sin(8*
        pi/32)))
361 LINESsketch.Line(point1=(B+31/2*s,-C-31*s), point2=(r*cos(9*pi/32),-r*sin
        (9*pi/32)))
362
363 MESHlines = MBLpart.PartitionFaceBySketch(faces=MBLpart.faces, sketch=
        LINESsketch) #Partition face by sketch, zone five.
364
365 ##Mesh:
366
367 #Define general seeds in the fine region, zone one, two and three
368 deltaL=0
369 for i in range(n+1):
370     deltaL=deltaL+pow(2,i)*S
371 Aend=A+deltaL
372 Bend=B+deltaL
373 Cend=C+deltaL+0.05
374
375 FINEmesh=MBLpart.edges.getByBoundingBox(yMax=Cend,yMin=-Cend,xMax=Bend,xMin
        =-Aend)
376 MBLpart.seedEdgeByNumber(edges=FINEmesh, number=1, constraint=FIXED) #
        Seed all edges within the area with one element on every edge, then
        change the edges which will hav more elements
377

```

```

378 ##Seed zone one:
379
380 for i in range(4*ant+1):
381     MBLmesh=MBLpart.edges.findAt((((R+0.000002)*cos((i-2*ant)*pi/(ant*4)),(R
        +0.000002)*sin((i-2*ant)*pi/(ant*4)),0),),) #Seed by bias the
        circular part of zone one
382     MBLpart.seedEdgeByBias(end1Edges=MBLmesh, number=ant-1, ratio=biasratio,
        constraint=FIXED)
383
384 MBLmesh=MBLpart.edges.findAt(((R+0.000002,0,0),),(0,R+0.0002,0),),(0,-R
        -0.0002,0),),(R,R,0),),(R,-R,0),),)
385 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, number=ant-1, ratio=biasratio,
        constraint=FIXED)
386
387 MBLmesh=MBLpart.edges.findAt(((0.000002,R,0),),(0.000002,-R,0),),)
        #Seed fist line at upper and lower side of the crack by bias
388 MBLpart.seedEdgeByBias(end2Edges=MBLmesh, number=ant, ratio=biasratio,
        constraint=FIXED)
389
390 MBLmesh=MBLpart.edges.findAt(((0.000002,R+le,0),),(0.000002,-R-le,0),),)
        #Seed box around zone one by number
391 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, number=ant, ratio=biasratio,
        constraint=FIXED)
392
393 MBLmesh=MBLpart.edges.findAt(((0.000002,R+le,0),),(0.000002,-R-le,0),),)
        #Seed box around zone one by number
394 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, number=ant-1, ratio=biasratio,
        constraint=FIXED)
395
396 ## Sees zone two:
397
398 MBLmesh=MBLpart.edges.findAt(((0,Si-0.005,0),),(0,-Si+0.005,0),),(Si
        -0.005,0,0),),) #Seed normal and parallel lines in zone two
        by bias
399 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, minSize=pi*(R+le)/(ant*4),
        maxSize=S, constraint=FIXED)
400
401 MBLmesh=MBLpart.edges.findAt(((0.000002,R,0),),(0.000002,-R,0),),)
        #Seed crack lines in zone two by bias
402 ratio=(Si-le)/(Si-le-R) #Ratio to get correct
        number of elements
403 MBLpart.seedEdgeByBias(end2Edges=MBLmesh, minSize=ratio*pi*(R+le)/(ant*4),
        maxSize=ratio*S, constraint=FIXED)
404
405 MBLmesh=MBLpart.edges.findAt(((0.000002,Si-0.000002,0),),(0.000002,-Si
        +0.000002,0),),) #Seed digonal lines in left side
        of zone two by bias
406 ratio=sqrt((Si-le-R)**2+(Si-le)**2)/(Si-le-R) #Ratio
        to get correct number of elements
407 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, minSize=ratio*pi*(R+le)/(ant*4),
        maxSize=ratio*S, constraint=FIXED)
408
409 MBLmesh=MBLpart.edges.findAt(((Si-0.005,Si-0.005,0),),(Si-0.005,-Si
        +0.005,0),),) #Seed digonal lines in rigth side ofzone two
        by bias, but free

```

```

410 MBLpart.seedEdgeByBias(end1Edges=MBLmesh, minSize=sqrt(2)*pi*(R+le)/(ant*4)
    , maxSize=sqrt(2)*S, constraint=FREE)
411
412 MBLmesh=MBLpart.edges.findAt((( -Si,R+0.000002,0), ),((-Si,-R-0.000002,0), ),)
    #Seed outer edges in the left par of zone two by number
413 MBLpart.seedEdgeByNumber(edges=MBLmesh, number=ant-1, constraint=FIXED)
414
415 MBLmesh=MBLpart.edges.findAt(((0.000002,Si,0), ),((Si,R+0.000002,0), )
    ,((0.000002,-Si,0), ),((Si,-R-0.000002,0), ),((-0.000002,Si,0), )
    ,((-0.000002,-Si,0), ),)#Seed outer edges in the righth par of zone two by
    number
416 MBLpart.seedEdgeByNumber(edges=MBLmesh, number=ant, constraint=FIXED)
417
418 ## Seed zone three:
419 MIDDLEmesh=MBLpart.edges.findAt((( -A+0.005,R,0), ),((-A+0.005,-R,0), ),((-A
    +0.005,Si,0), ),((-A+0.005,-Si,0), ),((B-0.005,Si,0), ),((B-0.005,-Si,0), )
    ,)
420 MBLpart.seedEdgeBySize(edges=MIDDLEmesh, size=S, constraint=FIXED)
    #Seed the lines which goes through zone three
421
422 #bug fixing for two lines in zone one and two:
423 #MBLmesh=MBLpart.edges.findAt(((Si-0.000002,-Si+0.000002,0), ),)
424 #MBLpart.seedEdgeByBias(end2Edges=MBLmesh, minSize=pi*(R+le)/(ant*4),
    maxSize=S, constraint=FREE)
425 #MBLmesh=MBLpart.edges.findAt(((R+0.000002,-R-0.000002,0), ),)
426 #MBLpart.seedEdgeByBias(end2Edges=MBLmesh, number=ant, ratio=biasratio,
    constraint=FIXED)
427
428 #Defines seed by bias for the lines from square mesh to boundary, zone five
429 for i in range(63):
430     MBLmesh=MBLpart.edges.findAt((((r-0.000002)*cos((i-31)*pi/32),(r
        -0.000002)*sin((i-31)*pi/32),0), ),)
431     MBLpart.seedEdgeByBias(end1Edges=MBLmesh, minSize=32*S, maxSize=pi*r/32,
        constraint=FIXED) #circular lines in zone five by bias
432
433 MBLmesh=MBLpart.edges.findAt((( -r+0.002),R,0), ),((-r+0.002),-R,0), ),)
434 MBLpart.seedEdgeByBias(end2Edges=MBLmesh, minSize=32*S, maxSize=pi*r/32,
        constraint=FIXED) #crack lines by bias
435
436 #Define element type and region
437 elemType = mesh.ElemType(elemCode=CPE4, elemLibrary=STANDARD)
438 areaset = MBLpart.Set(name='areaset', faces=MBLpart.faces)
439 MBLpart.setElementType(regions=areaset, elemTypes=(elemType,))
440
441 #Define mesh technique
442 MBLpart.setMeshControls(elemShape=QUAD, regions= MBLpart.faces, technique=
    STRUCTURED)
443 #Generate mesh
444 MBLpart.generateMesh(regions=MBLpart.faces)
445
446 ##Make assembly:
447 MBLassembly=MBLmodel.rootAssembly
448 MBLinstance=MBLassembly.Instance(name='MBLinstance', part=MBLpart, dependent
    =ON)

```

Appendix C Second Python code

This Python code uses the basis model as input. It then define boundary conditions and material regions before writing input files:

```

1 from abaqus import *
2 from abaqusConstants import *
3 backwardCompatibility.setValues(includeDeprecated=True, reportDeprecated=
    False)
4
5 import sketch
6 import part
7 import fields
8 import mesh
9 import regionToolset
10 import visualization
11
12 from part import *
13 from material import *
14 from section import *
15 from assembly import *
16 from step import *
17 from interaction import *
18 from load import *
19 from mesh import *
20 from job import *
21 from sketch import *
22 from visualization import *
23 from connectorBehavior import *
24
25 #Ys=((0),(0.5),(0.75),(1),(1.42),(-0.5),(-0.75),(-1),(-1.42))
26 #Yss=(('0'),('0,5'),('0,75'),('1'),('1,42'),('-0,5'),('-0,75'),('-1')
    ,(' -1,42'))
27 #Ds=((0.125),(0.25),(0.375),(0.5),(0.96))
28 #Dss=(('1'),('3'))#('2'),('3'),('4'),('5'))
29 #Ds=((0.12),(0.37))#(0.25),(0.37),(0.5),(0.96))
30 Ts=((0),(-0.25),(-0.5),(-0.75))
31 Tss=(('0'),('mi025'),('mi05'),('mi075'))
32 Temps=((466.4),(480.3),(507.84),(541.04))
33 Tempss=(('temp0'),('tempmi30'),('tempmi60'),('tempmi90'))
34
35 #for tt in range (len(Ts)): # For making mulitple files
36 # for TT in range (len(Temps)):
37 #Input:
38 Name= Tempss[TT]+'_T'+Tss[tt]
39 D=0.5 #Approximately half thickness of HAZ
40 Y=-D #Approximately distance form center of CRACK to center of HAZ
41
42
43 A=0.5 #Length of HAZ on crack
44 KI_stress=42000 #Stress intensity factor mode I
45 KII=0 #Stress intensity factor mode II
46 T_stress=Temps[TT]*Ts[tt]#T stress
47 ###For s=0.01 and Z=1.92
48 N=int(192) #Number of elements along one edge
49 n=int(5) #Number of scales from fine to coarse mesh

```

```

50 S=0.01      #Element size on fine mesh
51 Z=N*S      #Approximately half size of fine mesh region approximately 1.92
      mm
52 ###
53
54 B=2*Z-A    #Length of HAZ on ligament
55 r=5000    #Radius MBL
56 C=Z       #Half thickness of fine region
57 R=0.01    #Radius of crack tip
58
59 Si=0.05    #size of transition zone to square mesh, zone two.
60 ant=int(Si/S) #half the number of elements along a side of zone two
61
62 b=pi*R/(ant*4) #element size at crack tip
63 le=0
64 for i in range(ant):
65     bn=pi*(R+le)/(ant*4) #element size outwards
66     le=le+bn #length of transition zone to round mesh
67 biasratio=bn/b #Bias ratio for meshing
68
69
70
71 #Update variables
72 MBLmodel = mdb.models[ 'Model-1' ]
73 MBLpart = mdb.models[ 'Model-1' ].parts[ 'MBLp' ]
74 MBLassembly=mdb.models[ 'Model-1' ].rootAssembly
75 MBLinstance=mdb.models[ 'Model-1' ].rootAssembly.instances[ 'MBLinstance' ]
76
77 #Materialdefinsion
78 #WM
79 WME=208.E3      #Young's modulus
80 WMv=0.3         #Poisson's ratio
81 WMtable=((461.4397891 , 0),(472.5635995 , 0.000197972))
82 #BM
83 BME=208.E3
84 BMv=0.3
85 BMtable=((469.4515098 , 0),(481.2133191 , 0.000253191))
86 #HAZ
87 HAZE=208.E3
88 HAZv=0.3
89 HAZtable=((462.3343537 , 0),(473.2364703 , 0.000701631))
90
91 ##Property:
92
93 #WM
94 #Material
95 WMmaterial = MBLmodel.Material(name='WMm') #Make material
96 WMelasticproperties = (WME, WMv) #Make elastic table
97 WMmaterial.Elastic(table=(WMelasticproperties, )) #Give elastic
      properties
98 WMmaterial.Plastic(table=WMtable) #Give plastic properties
99 #Section
100 WMsection = MBLmodel.HomogeneousSolidSection(material='WMm', name='WMsec',
      thickness=1)
101 WMarea = MBLpart.elements.getByBoundingBox(yMin=-0.0001,xMax=-A+0.0001),
      MBLpart.elements.getByBoundingBox(yMin=-0.0001,xMin=B-0.0001),MBLpart.

```

```

elements.getByBoundingBox(yMin=Y+D-0.0001,xMin=-A-0.0001, xMax=B+0.0001)
,MBLpart.elements.getByBoundingBox(yMin=C-0.0001)
102 WMareaset = MBLpart.Set(name = 'WMset', elements = WMarea) #Define area
for WMsection
103 MBLpart.SectionAssignment(offset=0.0, offsetField='',offsetType=
MIDDLE_SURFACE, region=WMareaset, sectionName='WMsec',
thicknessAssignment=FROM_SECTION) #Define Section with material and
area
104
105 #BM
106 #Material
107 BMmaterial = MBLmodel.Material(name='BMm')
108 BMelasticproperties = (BME, BMv)
109 BMmaterial.Elastic(table=(BMelasticproperties, ))
110 BMmaterial.Plastic(table=BMtable)
111 #Section
112 BMsection = MBLmodel.HomogeneousSolidSection(material='BMm', name='BMsec',
thickness=1)
113 BMarea = MBLpart.elements.getByBoundingBox(yMax=0.0001,xMax=-A+0.0001),
MBLpart.elements.getByBoundingBox(yMax=0.0001,xMin=B-0.0001),MBLpart.
elements.getByBoundingBox(yMax=Y-D+0.0001,xMin=-A-0.0001, xMax=B+0.0001)
,MBLpart.elements.getByBoundingBox(yMax=-C+0.0001)
114 BMareaset = MBLpart.Set(name = 'BMset', elements = BMarea)
115 MBLpart.SectionAssignment(offset=0.0, offsetField='',offsetType=
MIDDLE_SURFACE, region=BMareaset, sectionName='BMsec',
thicknessAssignment=FROM_SECTION)
116
117 #HAZ
118 #Material
119 HAZmaterial = MBLmodel.Material(name='HAZm')
120 HAZelasticproperties = (HAZE, HAZv)
121 HAZmaterial.Elastic(table=(HAZelasticproperties, ))
122 HAZmaterial.Plastic(table=HAZtable)
123
124 #Section
125 HAZsection = MBLmodel.HomogeneousSolidSection(material='HAZm', name='HAZsec
',thickness=1)
126 HAZarea = MBLpart.elements.getByBoundingBox(yMin=Y-D-0.0001,yMax=Y+D
+0.0001,xMin=-A-0.0001,xMax=B+0.0001)
127 HAZareaset = MBLpart.Set(name = 'HAZset', elements =HAZarea)
128 MBLpart.SectionAssignment(offset=0.0, offsetField='',offsetType=
MIDDLE_SURFACE, region=HAZareaset, sectionName='HAZsec',
thicknessAssignment=FROM_SECTION)
129
130 ##Load:
131 #Surfaces
132 #Find edges in the WM area to set boundary
133 WMedge=MBLinstance.edges.findAt(((r*cos((1-0.5)*pi/32),r*sin((1-0.5)*pi/32)
,0)), ((r*cos((2-0.5)*pi/32),r*sin((2-0.5)*pi/32),0)),((r*cos
((3-0.5)*pi/32),r*sin((3-0.5)*pi/32),0)),((r*cos((4-0.5)*pi/32),r*sin
((4-0.5)*pi/32),0)),((r*cos((5-0.5)*pi/32),r*sin((5-0.5)*pi/32),0)),((
r*cos((6-0.5)*pi/32),r*sin((6-0.5)*pi/32),0)),((r*cos((7-0.5)*pi/32),r*
sin((7-0.5)*pi/32),0)),((r*cos((8-0.5)*pi/32),r*sin((8-0.5)*pi/32),0)),
((r*cos((9-0.5)*pi/32),r*sin((9-0.5)*pi/32),0)),((r*cos((10-0.5)*pi
/32),r*sin((10-0.5)*pi/32),0)),((r*cos((11-0.5)*pi/32),r*sin((11-0.5)*
pi/32),0)),((r*cos((12-0.5)*pi/32),r*sin((12-0.5)*pi/32),0)),((r*cos

```



```

((13-0.5)*pi/32),r*sin((13-0.5)*pi/32),0),),(r*cos((14-0.5)*pi/32),r*
sin((14-0.5)*pi/32),0),),(r*cos((15-0.5)*pi/32),r*sin((15-0.5)*pi/32
,0),),(r*cos((16-0.5)*pi/32),r*sin((16-0.5)*pi/32),0),),(r*cos
((17-0.5)*pi/32),r*sin((17-0.5)*pi/32),0),),(r*cos((18-0.5)*pi/32),r*
sin((18-0.5)*pi/32),0),),(r*cos((19-0.5)*pi/32),r*sin((19-0.5)*pi/32
,0),),(r*cos((20-0.5)*pi/32),r*sin((20-0.5)*pi/32),0),),(r*cos
((21-0.5)*pi/32),r*sin((21-0.5)*pi/32),0),),(r*cos((22-0.5)*pi/32),r*
sin((22-0.5)*pi/32),0),),(r*cos((23-0.5)*pi/32),r*sin((23-0.5)*pi/32
,0),),(r*cos((24-0.5)*pi/32),r*sin((24-0.5)*pi/32),0),),(r*cos
((25-0.5)*pi/32),r*sin((25-0.5)*pi/32),0),),(r*cos((26-0.5)*pi/32),r*
sin((26-0.5)*pi/32),0),),(r*cos((27-0.5)*pi/32),r*sin((27-0.5)*pi/32
,0),),(r*cos((28-0.5)*pi/32),r*sin((28-0.5)*pi/32),0),),(r*cos
((29-0.5)*pi/32),r*sin((29-0.5)*pi/32),0),),(r*cos((30-0.5)*pi/32),r*
sin((30-0.5)*pi/32),0),),(r*cos((31-0.5)*pi/32),r*sin((31-0.5)*pi/32
,0),),(r*cos((32-0.5)*pi/32),r*sin((32-0.5)*pi/32),0),),)
134 WMsurfaceset = regionToolset.Region(edges=WMedge)
135 #Find edges in the BM area to set boundary
136 BMedge=MBLinstance.edges.findAt(((r*cos((1-0.5)*pi/32),-r*sin((1-0.5)*pi
/32),0),),(r*cos((2-0.5)*pi/32),-r*sin((2-0.5)*pi/32),0),),(r*cos
((3-0.5)*pi/32),-r*sin((3-0.5)*pi/32),0),),(r*cos((4-0.5)*pi/32),-r*sin
((4-0.5)*pi/32),0),),(r*cos((5-0.5)*pi/32),-r*sin((5-0.5)*pi/32),0),)
,(r*cos((6-0.5)*pi/32),-r*sin((6-0.5)*pi/32),0),),(r*cos((7-0.5)*pi
/32),-r*sin((7-0.5)*pi/32),0),),(r*cos((8-0.5)*pi/32),-r*sin((8-0.5)*pi
/32),0),),(r*cos((9-0.5)*pi/32),-r*sin((9-0.5)*pi/32),0),),(r*cos
((10-0.5)*pi/32),-r*sin((10-0.5)*pi/32),0),),(r*cos((11-0.5)*pi/32),-r*
sin((11-0.5)*pi/32),0),),(r*cos((12-0.5)*pi/32),-r*sin((12-0.5)*pi/32
,0),),(r*cos((13-0.5)*pi/32),-r*sin((13-0.5)*pi/32),0),),(r*cos
((14-0.5)*pi/32),-r*sin((14-0.5)*pi/32),0),),(r*cos((15-0.5)*pi/32),-r*
sin((15-0.5)*pi/32),0),),(r*cos((16-0.5)*pi/32),-r*sin((16-0.5)*pi/32
,0),),(r*cos((17-0.5)*pi/32),-r*sin((17-0.5)*pi/32),0),),(r*cos
((18-0.5)*pi/32),-r*sin((18-0.5)*pi/32),0),),(r*cos((19-0.5)*pi/32),-r*
sin((19-0.5)*pi/32),0),),(r*cos((20-0.5)*pi/32),-r*sin((20-0.5)*pi/32
,0),),(r*cos((21-0.5)*pi/32),-r*sin((21-0.5)*pi/32),0),),(r*cos
((22-0.5)*pi/32),-r*sin((22-0.5)*pi/32),0),),(r*cos((23-0.5)*pi/32),-r*
sin((23-0.5)*pi/32),0),),(r*cos((24-0.5)*pi/32),-r*sin((24-0.5)*pi/32
,0),),(r*cos((25-0.5)*pi/32),-r*sin((25-0.5)*pi/32),0),),(r*cos
((26-0.5)*pi/32),-r*sin((26-0.5)*pi/32),0),),(r*cos((27-0.5)*pi/32),-r*
sin((27-0.5)*pi/32),0),),(r*cos((28-0.5)*pi/32),-r*sin((28-0.5)*pi/32
,0),),(r*cos((29-0.5)*pi/32),-r*sin((29-0.5)*pi/32),0),),(r*cos
((30-0.5)*pi/32),-r*sin((30-0.5)*pi/32),0),),(r*cos((31-0.5)*pi/32),-r*
sin((31-0.5)*pi/32),0),),(r*cos((32-0.5)*pi/32),-r*sin((32-0.5)*pi/32
,0),),)
137 BMsurfaceset = regionToolset.Region(edges=BMedge)
138
139 ##Step:
140 MBLmodel.StaticStep(name='T', previous='Initial',timePeriod=1.0, initialInc
=0.1,maxInc=0.1,maxNumInc=6000, nlgeom=ON)
141 MBLmodel.StaticStep(name='Load', previous='T',timePeriod=1.0, initialInc
=0.001,maxInc=0.1,minInc=0.00000000001,maxNumInc=6000, nlgeom=ON) #
Make step
142
143 #Output for the .odb file
144
145 CTODnodes1=MBLassembly.Set(name='CTODnodes1', nodes=MBLinstance.nodes.
getByBoundingBox(xMin=-0.00005,yMax=R+0.00005,yMin=-R-0.00005,xMax
=0.00005)) #CTOD first node definition

```

```

146 #CTODnodes2=MBLassembly.Set(name='CTODnodes2', nodes=MBLinstance.nodes.
    getByBoundingBox(xMin=-2*S-0.00005,yMax=0.00005,yMin=-0.00005,xMax=-S
    -0.00005)) #CTOD second node definition
147
148 MBLmodel.FieldOutputRequest(name='F-Output-1', createStepName='Load',
    variables=('S', 'PEEQ', 'U', 'EVOL', 'PE'), timeInterval=0.05) #Redefine
    field output
149 #MBLmodel.FieldOutputRequest['F-Output-1'].setValuesInStep(stepName='Load',
    timeInterval=0.05)
150
151 #Displacementfield on WM surface
152 #U1 direction
153 T=T_stress
154 MBLmodel.ExpressionField(description='', expression='((1+WMv)*sqrt(r/(2*pi)
    )/WME)*((0*cos(acos(X/r)/2)*(3-4*WMv-X/r))+KII*sin(acos(X/r)/2)*(5-4*WMv
    +X/r))+T*(1-pow(WMv,2))*X/WME', localCsys=None, name='WMu1.T') #
    Expression
155 #U2 direction
156 MBLmodel.ExpressionField(description='', expression='((1+WMv)*sqrt(r/(2*pi)
    )/WME)*((0*sin(acos(X/r)/2)*(3-4*WMv-X/r))-KII*cos(acos(X/r)/2)*(1-4*WMv
    +X/r))-T*r*(WMv+pow(WMv,2))*sin(acos(X/r))/WME', localCsys=None, name='
    WMu2.T')
157 #Displacementfield on BM surface
158 #U1 direction
159 MBLmodel.ExpressionField(description='', expression='((1+BMv)*sqrt(r/(2*pi)
    )/BME)*((0*cos(acos(X/r)/2)*(3-4*BMv-X/r))-KII*sin(acos(X/r)/2)*(5-4*BMv
    +X/r))+T*(1-pow(BMv,2))*X/BME', localCsys=None, name='BMu1.T')
160 #U2 direction
161 MBLmodel.ExpressionField(description='', expression='((1+BMv)*sqrt(r/(2*pi)
    )/BME)*((-0*sin(acos(X/r)/2)*(3-4*BMv-X/r))-KII*cos(acos(X/r)/2)*(1-4*
    BMv+X/r))+T*r*(BMv+pow(BMv,2))*sin(acos(X/r))/BME', localCsys=None, name
    ='BMu2.T')
162
163 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='T',
    distributionType=FIELD, fieldName='WMu1.T', fixed=OFF, localCsys=None,
    name='WMdispu1.T', region=WMsurfaceset, u1=1.0, u2=UNSET, ur3=UNSET)
164 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='T',
    distributionType=FIELD, fieldName='WMu2.T', fixed=OFF, localCsys=None,
    name='WMdispu2.T', region=WMsurfaceset, u1=UNSET, u2=1, ur3=UNSET)
165 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='T',
    distributionType=FIELD, fieldName='BMu1.T', fixed=OFF, localCsys=None,
    name='BMdispu1.T', region=BMsurfaceset, u1=1.0, u2=UNSET, ur3=UNSET)
166 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='T',
    distributionType=FIELD, fieldName='BMu2.T', fixed=OFF, localCsys=None,
    name='BMdispu2.T', region=BMsurfaceset, u1=UNSET, u2=1, ur3=UNSET)
167
168 MBLmodel.boundaryConditions['WMdispu1.T'].deactivate('Load')
169 MBLmodel.boundaryConditions['WMdispu2.T'].deactivate('Load')
170 MBLmodel.boundaryConditions['BMdispu1.T'].deactivate('Load')
171 MBLmodel.boundaryConditions['BMdispu2.T'].deactivate('Load')
172
173 KI=KI_stress
174 #Displacementfield on WM surface
175 #U1 direction
176 MBLmodel.ExpressionField(description='', expression='((1+WMv)*sqrt(r/(2*pi)
    )/WME)*((KI*cos(acos(X/r)/2)*(3-4*WMv-X/r))+KII*sin(acos(X/r)/2)*(5-4*

```

```

    WMv+X/r))+T*(1-pow(WMv,2))*X/WME', localCsys=None, name='WMu1') #
    Expression
177 #U2 direction
178 MBLmodel.ExpressionField(description='', expression='((1+WMv)*sqrt(r/(2*pi)
    )/WME)*((KI*sin(acos(X/r)/2)*(3-4*WMv-X/r))-KII*cos(acos(X/r)/2)*(1-4*
    WMv+X/r))-T*r*(WMv+pow(WMv,2))*sin(acos(X/r))/WME', localCsys=None, name
    ='WMu2')
179 #Displacementfield on BM surface
180 #U1 direction
181 MBLmodel.ExpressionField(description='', expression='((1+BMv)*sqrt(r/(2*pi)
    )/BME)*((KI*cos(acos(X/r)/2)*(3-4*BMv-X/r))-KII*sin(acos(X/r)/2)*(5-4*
    BMv+X/r))+T*(1-pow(BMv,2))*X/BME', localCsys=None, name='BMu1')
182 #U2 direction
183 MBLmodel.ExpressionField(description='', expression='((1+BMv)*sqrt(r/(2*pi)
    )/BME)*((-KI*sin(acos(X/r)/2)*(3-4*BMv-X/r))-KII*cos(acos(X/r)/2)*(1-4*
    BMv+X/r))+T*r*(BMv+pow(BMv,2))*sin(acos(X/r))/BME', localCsys=None, name
    ='BMu2')
184 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='Load',
    distributionType=FIELD, fieldName='WMu1', fixed=OFF, localCsys=None, name
    ='WMdispu1_KI', region=WMsurfaceset, u1=1.0, u2=UNSET, ur3=UNSET)
185 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='Load',
    distributionType=FIELD, fieldName='WMu2', fixed=OFF, localCsys=None, name
    ='WMdispu2_KI', region=WMsurfaceset, u1=UNSET, u2=1, ur3=UNSET)
186 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='Load',
    distributionType=FIELD, fieldName='BMu1', fixed=OFF, localCsys=None, name
    ='BMdispu1_KI', region=BMsurfaceset, u1=1.0, u2=UNSET, ur3=UNSET)
187 MBLmodel.DisplacementBC(amplitude=UNSET, createStepName='Load',
    distributionType=FIELD, fieldName='BMu2', fixed=OFF, localCsys=None, name
    ='BMdispu2_KI', region=BMsurfaceset, u1=UNSET, u2=1, ur3=UNSET)
188
189 ##Make set for output data to .dat fil. Define the output in .inp file.
190 #CRACKfront = MBLassembly.Set(name = 'CRACKfront', nodes = MBLinstance.
    nodes.getByBoundingBox(xMin=0.05,yMin=0.1,xMax=B+0.1,yMax=0.1)) #
    j smal
191 #CRACKfront = MBLassembly.Set(name = 'CRACKfront', nodes = MBLinstance.
    nodes.getByBoundingBox(xMin=-A-0.005,yMin=-C-0.005,xMax=B+0.005,yMax=C
    +0.005)) #j ABC
192 #CRACKfront_ABC = MBLassembly.Set(name = 'CRACKfront_ABC', nodes =
    MBLinstance.nodes.getByBoundingBox(xMin=-A-62*S-0.005,yMin=-C-62*S
    -0.005,xMax=B+62*S+0.005,yMax=C+62*S+0.005)) #j ABC+
193 #CRACKfront_far = MBLassembly.Set(name = 'CRACKfront_far', nodes =
    MBLinstance.nodes.getByBoundingBox(xMin=-100,yMin=-100,xMax=100,yMax
    =100)) #j far out
194 #CRACKtip=MBLassembly.Set(name = 'CRACKtip', vertices= MBLinstance.vertices
    .getByBoundingBox(xMin=-0.00005,yMin=-0.00005,xMax=0.00005,yMax=0.00005)
    )
195 #ALL = MBLassembly.Set(name = 'ALL', elements =MBLinstance.elements)
196
197 MBLassembly.Set(name = 'WMele', elements = (MBLinstance.elements.
    getByBoundingBox(yMin=-0.0001,xMax=-A+0.0001), MBLinstance.elements.
    getByBoundingBox(yMin=-0.0001,xMin=B-0.0001), MBLinstance.elements.
    getByBoundingBox(yMin=Y+D-0.0001,xMin=-A-0.0001, xMax=B+0.0001),
    MBLinstance.elements.getByBoundingBox(yMin=C-0.0001)))
198 MBLassembly.Set(name = 'BMele', elements =(MBLinstance.elements.
    getByBoundingBox(yMax=0.0001,xMax=-A+0.0001), MBLinstance.elements.
    getByBoundingBox(yMax=0.0001,xMin=B-0.0001), MBLinstance.elements.

```

```
    getByBoundingBox(yMax=Y-D+0.0001,xMin=-A-0.0001, xMax=B+0.0001),
    MBLinstance.elements.getByBoundingBox(yMax=-C+0.0001)))
199 MBLassembly.Set(name = 'HAZeLe', elements =(MBLinstance.elements.
    getByBoundingBox(yMin=Y-D-0.0001,yMax=Y+D+0.0001,xMin=-A-0.0001,xMax=B
    +0.0001)))
200
201 ligamentelements=MBLinstance.elements.getByBoundingBox(xMin=Si-0.00005,yMin
    =-0.0005,xMax=B-0.005,yMax=0.015)
202 #inner=MBLinstance.faces.findAt(((R+0.005,0.00005,0),),)
203 #ligamentelements=ligamentelements, MBLinstance.elements.sequenceFromLabels
    ((156567,156568,156569,156570,156571,156756,156761,156766,156771,156776,156781,156786,1
    ,) #R=0.0005
204 ligamentelements=ligamentelements, MBLinstance.elements.sequenceFromLabels
    ((156463,156462,156461,156460,156579,156584,156589,156594,156599,),) #R
    =0.001
205
206 ligament=MBLassembly.Set(name = 'ligament', elements=ligamentelements)
207 elall=MBLassembly.Set(name = 'elall', elements=MBLinstance.elements)
208 #change work dir
209 os.chdir(r'/home/gaute/Gaute.sommer.2011/simuleringer/3_material_T_10/')
210 #Make job and write input file to WRD
211 job=mdb.Job(model='Model-1', multiprocessingMode=THREADS, name=Name,
    numCpus=8, numDomains=8)
212 job.writeInput()
```



```

148 for n=1:N
149     readtemp1=dlmread(strcat(placein,'out',int2str(n),'.txt'),'',[157132 1
        157133 1]); % Read CTOD
150     if n==1
151         ctod=zeros(N+1,1);
152     end
153     ctod(n+1,1)=abs(readtemp1(1,1)-readtemp1(2,1));
        % Calculate CTOD
154 end
155 fit_J_ctod = polyfit(J, ctod, 1);
156 %%% Write: Time, KI, J, CTOD, WM and HAZ Sigmax: 7.5, 10, 15, 20 %%%
157
158 write=zeros(N+1,12);
159 write(:,1)=KI(:,1)/KI_max;
160 write(:,2)=KI(:,1);
161 write(:,3)=J(:,1);
162 write(:,4)=ctod(:,1);
163 write(:,5)=real(HAZsigmax(:,1));
164 write(:,6)=real(HAZsigmax(:,2));
165 write(:,7)=real(HAZsigmax(:,3));
166 write(:,8)=real(HAZsigmax(:,4));
167 write(:,9)=real(WMsigmax(:,1));
168 write(:,10)=real(WMsigmax(:,2));
169 write(:,11)=real(WMsigmax(:,3));
170 write(:,12)=real(WMsigmax(:,4));
171
172 dlmwrite(strcat(placeout,'Ti_KI_J_CTOD_sigmax'), write, '\t');
173 %%% Write_lig: r, S22 %%%
174
175 write_lig=zeros((N+1)*2,L);
176 for n=1:N
177     for p=1:L
178         write_lig(2*n+1,p)=Lig(n+1,p,1);
179         write_lig(2*n+2,p)=Lig(n+1,p,2);
180     end
181 end
182 write_lig(1,:)=write_lig(3,:); % set length for
    first increment
183 dlmwrite(strcat(placeout,'r.S22'), write_lig, '\t');
184
185 con=con+1;
186 end
187 end
188 end
189 end

```


Appendix E Materials

This Matlab script makes a stress strain relationship for a theoretic material.

```

1 clc
2 clear all
3 close all
4
5 sigma02=450;
6 E=208000;
7 n=0.07;
8
9 sig=(420:0.001:440);
10
11 sigy=sig(1);
12 for i=1:length(sig)
13     sigtemp=sigma02/(1+0.002*E/sigy)^n;
14     if sigtemp>sigy
15         sigy=sig(i+1);
16     end
17 end
18
19 ept=[0:0.0005:0.002,0.003:0.001:0.01,0.02:0.01:0.15,0.3,0.6,1,2];
20 mat0=zeros(length(ept),2);
21 mat0(:,2)= transpose(ept);
22 mat1=mat0;
23 mat2=mat0;
24 mat3=mat0;
25 mat4=mat0;
26
27 for i=1:length(ept)
28     mat0(i,1)=sigy*(1+mat0(i,2)*E/sigy)^n;
29     mat1(i,1)=sigy*(1+mat1(i,2)*E/sigy)^n+0.1*sigma02;
30     mat2(i,1)=sigy*(1+mat2(i,2)*E/sigy)^n+0.2*sigma02;
31     mat3(i,1)=sigy*(1+mat3(i,2)*E/sigy)^n+0.3*sigma02;
32     mat4(i,1)=sigy*(1+mat4(i,2)*E/sigy)^n+0.4*sigma02;
33 end
34
35 hold on
36 plot(mat0(:,2),mat0(:,1))
37 plot(mat1(:,2),mat1(:,1))
38 plot(mat2(:,2),mat2(:,1))
39 plot(mat3(:,2),mat3(:,1))
40 plot(mat4(:,2),mat4(:,1))
41 hold off
42
43
44 dlmwrite('00r', mat0, ',');
45 dlmwrite('10r', mat1, ',');
46 dlmwrite('20r', mat2, ',');
47 dlmwrite('30r', mat3, ',');
48 dlmwrite('40r', mat4, ',');

```