Chapter 1

Industrial Challenges Where Computational Welding Mechanics Becomes an Engineering Tool

The welding process is intensively used in the nuclear industry as it ensures both material continuity and containment leak tightness. However, this joining process also leads to material modifications and induces permanent deformations and residual stresses at the macroscopic level with possible consequences on damage during welding operation or under operating conditions. Numerical simulations of welding processes were first performed at the beginning of the 1970s [MAR 74]. To meet nuclear regulation authorities’ requirements and because of the strongly increasing capabilities of computers, computational welding mechanics has been used more and more in engineering studies, taking multiphysics phenomena such as metallurgy into account [MAR 74, ALB 83, KAR 86, BER 91, GOL 92, DON 05, DHI 05]. Indeed, for a long time, numerical analyses of weldability have been slowed down by the metallurgical complexity of the considered materials and the necessity to determine the material behavior in the solid state at high temperatures and during solidification. However, concerning cold cracking issues, a phenomenological approach to estimate the risk of hydrogen-assisted cold cracking using the implant test modeling has early been proposed [LEB 88]. Later, these models have been completed to account for coupled viscoplasticity and transformation-induced plasticity during the welding of low-alloy steels [BER 03]. Today, for the prevention of damage, chained approaches can be used in spite of the
difficulty to establish macroscopic fracture criteria in heterogeneous structures [ROB 09].

For manufacturing purposes, metallurgical weldability and distortions are the main issues. The former remains in the field of material science and the latter concerns solid mechanics. For design assessment of welded components under operating conditions, more attention is paid to welding residual stresses and strains. Indeed, stress corrosion cracking (SCC) can be activated due to the combination of metallurgical affection, strain hardening and tensile stresses. Fatigue crack initiation can also be sped up by these consequences of welding and its propagation under high-cycle fatigue is directly driven by the mean stress sign and level. Brittle fracture and ductile tearing are other damages for which the thermo-mechanically affected zone (TMAZ) may become more sensitive due to embrittlement or toughness reduction.

These different issues (i.e. weldability, distortions and residual stresses) can be solved by mathematical modeling using the finite element (FE) method to compute coupled transient nonlinear problems. The following sections highlight industrial challenges for which computational welding mechanics has become a decision-making tool for welding and mechanical engineers.

1.1. Reducing the risk of weld cracking

1.1.1. Implant test modeling for risk of cold cracking assessment during welding operations

Filler material used for welding operations can lead to the occlusion of hydrogen gas in the arc atmosphere into the solidifying weld metal. This amount of hydrogen as well as that originally present in the parent metal rapidly diffuses into the various regions of the weldment due to the high temperature. Diffusion is also controlled by the microstructure evolution and trapping effects. As the welded component cools down, the risk of hydrogen-assisted cold cracking in ferritic steel can arise depending on the metal microstructure in the heat-affected zone (HAZ), the concentration of hydrogen in the weld and the level of residual stresses. One of the most effective precautions against weld hydrogen cracking is to use preheating and postheating in order to reduce the hydrogen content, by diffusion in the structure and degassing, before residual stresses reach higher values at the end of cooling. The implant test is a stress-controlled test applied on small specimens during welding to assess the susceptibility of the HAZ to hydrogen cracking. It may be used to define preheating temperature and postheating duration in order to prevent nuclear component assemblies from cold cracking risk. Finite element analysis (FEA) of the problem couples hydrogen diffusion, thermometaallurgical and mechanical modeling
as shown in Figure 1.1. The simulation of the implant test can be followed by local fracture analysis of the Weibull type. A probabilistic criterion can thus be used to assess the risk of cold cracking during welding operations on components made up of low-alloy steels such as A508cl3 according to the ASME code (or 16MND5 according to the RCCM French code).

![Diagram of physical phenomena involved - couplings and interactions](image)

**Figure 1.1. Physical phenomena involved – couplings and interactions**

1.1.1.1. Computation models

The interactions between heat transfer, metallurgy and mechanics have to be taken into account. As shown in Figure 1.1, strong coupling is performed to solve the thermomechanical part and weak coupling is sufficient to predict mechanical states. Strong coupling means that temperatures and phase proportions are solved in the same system of equations unlike weak coupling that also concerns hydrogen diffusion and needs thermomechanical and mechanical results as input data.

1.1.1.1. Heat transfer analysis

The heat transfer analysis is based on the solution of the classical heat equation with appropriate boundary conditions [BER 08, FEU 07]. It is computed on a three-dimensional (3D) model in order to properly reproduce the thermal cycles in the heat-affected region. The shape of the heat sources and the input energy are fitted to experimental data (i.e. recordings of thermocouples and the dimensions of the weld pool and the HAZ obtained from macrograph transversal cuts). The precise description of the phenomena involved in the heat input such as arc-plasma interactions, and the analysis of fluid dynamics in the weld pool are not taken into account in the model. From the thermomechanical point of view, the fluid flow effect,
which leads to homogenization of the temperature in the molten area, is simply taken into account by increasing the thermal conductivity over the fusion temperature.

As far as ferritic steels such as A508cl3 are concerned, phase transformations must be included in the simulation [DEN 97]. At each time step, a material is characterized by the proportions \( p_k \) of the different phases assuming that \( \sum p_k = 1 \).

From the modeling point of view, the phase proportions are additional state variables whose evolution can be described by ordinary differential equations on time [LEB 84a]. Material properties are both phase and temperature dependent. The thermal conductivity, the density and the enthalpy of the mixture of phases are calculated from individual phase values using a linear mixture rule. For single-pass welding processes, the following metallurgical transformations are modeled for the ferritic steel A508cl3:

- Initial base metal (mixture of ferrite and bainite) \( \rightarrow \) austenite during heating.  
- Austenite \( \rightarrow \) (as-quenched) bainite during cooling.  
- Austenite \( \rightarrow \) (as-quenched) martensite during cooling.

### 1.1.1.1.2 Tempering

For temperatures lower than the austenitization temperature (Ac1 is approximately 700°C for A508cl3), tempering of as-quenched metallurgical structures also have to be taken into account. Tempering can be a consequence of multipass welding processes. It can also be the expected effect of postweld heat treatment (PWHT) or temper bead welding processes [LEB 84b]. Indeed, for metallic materials, at the end of manufacturing processes for which austenitization followed by fast cooling rates occurs, the subsequent as-quenched structure is acceptable regarding fracture toughness. For instance, the as-quenched-martensite presents a remarkably high mechanical strength but a rather low ductility. The tempering induced by a new welding thermal cycle or an appropriate heat treatment can balance these properties toward the expected values by:

- a suitable hardening precipitation in a zone where precipitates have been put in solution and the main alloy elements maintained in supersaturation;
- a softening of an excessively hardened structure.

As for as-quenched transformations, tempering transformations from as-quenched structures to fully tempered structures can be modeled using the same differential equations on time. Tempered phase proportions become new additional state variables associated with specific properties closer to the ones of the base metal. The parameters of kinetics of phase transformations during tempering have
been identified in [VIN 02] within the framework of the characterization of the A508cl3 low-alloy steel for the numerical simulation of PWHT. The tempering equivalent parameter, \( P_c \), between time and temperature, is used to identify the tempering kinetics during short thermal cycles related to welding processes. \( P_c \) is defined by the following relation [BLO 75]:

\[
P_c = \left( \frac{1 - nR}{\theta} \log \left( \frac{t}{t_0} \right) \right)^{-1}
\]

where \( \theta \) is the temperature in Kelvin, \( t \) is the time, \( t_0 \) is the time unit (generally equal to 1), \( n \) is the Naperian logarithm of 10, \( R \) is the constant of perfect gases and \( \Delta H \) is the activation energy in joules per mole.

![Figure 1.2](image_url)

**Figure 1.2.** Evolution of the hardness in the HAZ during the deposit of several welding layers. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.2 shows the tempering effect obtained during multipass welding in the HAZ of an A508cl3 ferritic steel plate using the modeling technique presented previously. The average hardness value obtained at the end of the fourth layer deposition is approximately 300 Vickers, much lower than the martensite hardness observed after the first bead deposit. Such a welding process, called temper bead, will be discussed in detail later.
1.1.1.1.3. Mechanical analysis

The mechanical analysis is based on the momentum balance equation where inertial effects are neglected. The internal heating due to plastic dissipation is neglected considering the small strain rates generated by a welding operation. As this effect, as well as the influence of stresses on metallurgical transformations, is neglected, the mechanical analysis can be uncoupled from the thermomeetallurgical simulation. The mechanical computation is thus achieved in a second stage using the temperature and phase proportions calculated previously.

Heat transfer and metallurgy are involved in the mechanical analysis through the following four effects:

- the thermal strains;
- the volume changes due to the transformations (contraction during heating and expansion during cooling);
- the influence of temperature and of the phases proportions on the behavior law (phase proportions are state variables);
- the transformation-induced plasticity.

The stress–strain relation must be temperature dependent, representative of the phase mixture, and must reproduce the transformation-induced plasticity phenomenon. Among all the models dealing with the behavior of steels during phase transformation [GRE 65, MAG 66, ABR 72, GIU 81, FIS 97, FIS 00], the model proposed by Leblond et al. [LEB 89], based on a micromechanical analysis, is widely used. It is the model chosen for the description of ferritic steel during welding simulation. It requires the temperature-dependent stress–strain relations of all the phases. The strain hardening can be either isotropic or kinematic, and viscoplastic effects can also be considered [VIN 02]. For the implant test simulation, the behavior of the different constitutive phases of A508cl3 steel is assumed to be elastoplastic with isotropic hardening.

1.1.1.1.4. Numerical simulation of hydrogen diffusion

The risk of cold cracking in the welded zone arises from the presence of hydrogen and welding residual stresses. The purpose of the hydrogen diffusion calculation is to quantify the evolution of the hydrogen concentration in the structure during the short duration of welding and during the cooling down to the room temperature in a longer time. The hydrogen mainly arises from the welding process as the base material procurements specify very low hydrogen contents.
The calculation of the evolution of the hydrogen concentration in the structure uses a model that is a generalization of the Fick’s second law. The equation used is the following:

\[
\frac{\partial C}{\partial t} = \text{div} \left( DS \text{grad} \left[ \frac{C}{S} \right] \right)
\]  

[1.2]

This phenomenon of classic diffusion can be extended to consider reversible and irreversible effects of trapping. At low temperature, the hydrogen trapping effects occur through various traps (dislocations, interstitial atoms, gaps, etc.) [TIS 77, LEB 84b, CWI 06, CHE 09]. A fine modeling of these phenomena needs to distinguish the hydrogen concentrations in the various types of sites (ordinary sites and different kinds of traps). Due to the lack of experimental data, the irreversible aspect of trapping is ignored. However, it is possible to take into account the irreversible effects in a simplified way by considering only a global concentration, without any distinction between the dissolved (diffusible) or trapped (incorporated) hydrogen. In practice, a modeling with a single type of site is used, and the effect of trapping is taken into account with a diffusion coefficient that depends on the temperature, the microstructure and the strain hardening (cumulative plastic strain). This last approach will be used in the numerical simulation of hydrogen diffusion presented in this chapter.

The material properties that must be considered are the solubility and the permeability. The data can be obtained from reference [PRE 88]. The permeability is deduced from the relation \( P = DS \). The solubility and the permeability are given as a function of the absolute temperature \( T_{\text{abs}} \) as follows:

\[
S = C_1 \exp\left(\frac{C_2}{T_{\text{abs}}}\right)
\]  

[1.3]

\[
P = C_3 \exp\left(\frac{C_4}{T_{\text{abs}}}\right)
\]  

[1.4]

The effect of trapping by the presence of dislocations (related to the cumulative plastic strain \( \varepsilon_{\text{eq}}^p \)) is modeled by giving the solubility as a function of \( \varepsilon_{\text{eq}}^p \):

\[
S = C_1 \exp\left(\left(\frac{C_2 + C_3 \cdot \varepsilon_{\text{eq}}^p}{T_{\text{abs}}}\right)\right)
\]  

[1.5]

\[
\varepsilon_{\text{eq}}^p = \min\left(\varepsilon_{\text{eq}}^p, \varepsilon_{\text{eq}, \text{th}}^p\right)
\]  

[1.6]
The introduction of the plastic deformation into the exponential term allows us to
decrease its effect with an increasing temperature but the effect becomes
overestimated if the plastic deformation reaches high values (at crack tips or in
singularities, for instance). Another formulation should be established but
practically, the plastic deformation dependency is simply limited to a threshold
value that appears as an additional parameter. In our problem, the plasticity in the
notch is very small (the maximal value of cumulative plastic strain is close to 2%)
and this parameter is not necessary.

1.1.1.2. Numerical results of implant test modeling

The implant test consists of welding a notched specimen made of the material to
be tested [ISO 05]. This specimen is crimped in a plate on which the weld is
deposited and then a tensile force is applied on the other extremity of the specimen
as shown in Figure 1.3. The objective of this test is to reproduce industrial welding
conditions that can lead to cold cracking: high tensile stresses in the vicinity of a
flaw, in a brittle phase where hydrogen content may rise up. All these negative
effects are concentrated in this technological test, which makes it very pessimistic
for demonstration. The process used in this research work is a single-pass arc
welding process with coated electrodes made of low-carbon steel (the mass
percentage of carbon is less than 0.05% leading to pure bainite transformation in the
molten zone considering dilution with parent metal). The level of tensile stresses in
the plane of the notch is driven by the application of a load and the level of
hydrogen is related to the process parameters. The hydrogen is brought inside the
molten pool and diffuses in the implant and the plate. At the end of the experiment,
the presence, or not, of cracks through the implant is verified on metallographic cuts
by fractographic analysis.

Figure 1.3. Implant test device with main dimensions in millimeters [ISO 05] (1: temperature
measurement; 2: welding direction; 3: implant specimen; 4: test load)
1.1.1.2.1. Numerical simulation of hydrogen diffusion

The geometry of the implant specimen and the 3D mesh used for thermomechanical computations are shown in Figure 1.4. To place FE modeling at the scale of cracking phenomena, the element size of the mesh is approximately 50 μm close to the singularity and only half of the structure is modeled for symmetry reasons. The following two configurations of welding are studied:

- Case 1 (P-H 150°C) leading to cold cracking: the preheating temperature is 150°C with postheating during 15 min.
- Case 2 (P-H 200°C) for which no crack is observed: the preheating temperature is 200°C with postheating during 15 min.

![Figure 1.4. Implant specimen with notch dimensions in millimeters [ISO 05] and three-dimensional mesh view in the plane of symmetry](image)

When the temperature decreases below the preheating temperature +50°C, a tensile load is applied on the lower surface of the specimen due to a pressure of 245 MPa, so as to obtain an average axial stress of 500 MPa in the plane of the notch.

1.1.1.2.2. Thermomechanical results

The thermomechanical computations are performed with a 3D model using a heat source moving along the weld line. The maximal temperature fields obtained through the transverse cross-section of the implant are shown in Figures 1.5(a) and (b) for case 1 (P-H 150°C) and case 2 (P-H 200°C), respectively.

For validation of heat input, some implant tests had been controlled by thermocouples. As a representative example of this validation step, a comparison is made in Figure 1.6, which gives satisfying results.
The phase proportions obtained after welding in the molten zone and the HAZ are mainly as-quenched bainite and as-quenched martensite as shown in Figures 1.7 and 1.8. The effect of the temperature of preheating is clearly visible. Indeed, in the HAZ, most of as-quenched phase is martensite (approximately 70%) for a preheating at 150°C while as-quenched bainite (approximately 65%) is mainly formed for a preheating at 200°C.

Experimental measurements made on micrographic cuts by microhardness profiles confirm that the size of the HAZ from the fusion line is approximately 3 mm in the axial direction of the implant specimen (see Figure 1.9). The level of hardness confirms also a mixture between bainite and martensite.
Figure 1.7. Bainite proportion with a preheating temperature of a) 150°C and b) 200°C. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.8. Martensite proportion with a preheating temperature of a) 150°C and b) 200°C. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.9. Hardness measurements along the implant axis: values and locations (distance between two locations = 0.5 mm)
1.1.1.2.3. Mechanical results

Two kinds of model are proposed to compute mechanics:

− A two-dimensional (2D) calculation considering rotationally symmetry conditions as proposed for the first modeling of this test [LEB 88]. The main difference here arises from the fields of temperature in the median section of the specimen, which are transferred from a 3D modeling and not directly computed with 2D assumptions. Using this transfer of physical quantities, the thermal history and the phase evolution (state variables) for the computation of strains and stresses across the section are more realistic [ROB 10].

− A 3D calculation using the temperature fields and the phase proportions calculated on the 3D model presented in the previous section.

The residual stresses along the axis of the notch, in the direction of the implant axis, namely the axial stresses, are compared between the two models to validate the relevance of the 2D approach with regard to the 3D calculation (see Figure 1.10 for the plotting direction description).

![Diagram showing residual stresses](image)

**Figure 1.10.** Positions of the lines over which opening stresses are plotted. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.11 shows that the 2D computation gives a stress profile that is very close to the profile obtained using 3D modeling of the test regarding the transverse direction (90°). Furthermore, this direction is the one that exhibits the highest stress level. Considering 2D mechanical computations relevant for thermomechanical history and pessimistic regarding stress distribution, the evaluation of the crack criteria will be based on this simplified approach.
Figure 1.11. Axial stresses plotted along the notch axis – two-dimensional results compared with three-dimensional results in several directions (preheating temperature of 150°C). For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

The highest tensile stresses are approximately 1,600 MPa and are observed near the notch surface in front of the highest strain hardened area shown in Figure 1.12.

Figure 1.12. Plastic zone in the notch at the end of cooling after loading (preheating temperature of 150°C). For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

The comparison between the 2D mechanical computations made with two different preheating temperature shows that higher stresses are predicted for case 1 (P-H 150°C) at the tip of the notch (see Figure 1.13).
1.1.1.2.4. Hydrogen diffusion analysis

Thermomechanical and mechanical results are not dependent on the hydrogen concentration in accordance with the model detailed in this chapter. On the other hand, as shown in Figures 1.14 and 1.15 for different preheating temperatures, the effects of temperature and phase proportion can obviously be observed. After 100 s, the hydrogen concentration is maximal for the lower preheating temperature. For longer times, the curve discontinuities placed at approximately 5 mm from the plate surface are due to the highest solubility of the martensite, which has the main proportion for higher cooling rates as for case 1 (P-H 150°C). The trapping effect, taken into account through the macroscopic indicator of crystallographic defects, i.e. the cumulative plastic strain, may accentuate this trend.
The hydrogen concentration remains higher as far as low preheating temperatures are concerned. Figure 1.16 shows that 3 h after the weld deposit, the hydrogen concentration close to the notch whose tip is located at $x = 3.5$ mm is two times higher when the preheating temperature is $150^\circ$C.

From this modeling of the implant test, we can qualitatively note that a decrease of the preheating temperature leads to a harder and thus more brittle microstructure, higher tensile stresses and greater hydrogen content. The following section describes a probabilistic model to quantify the risk of cold cracking by considering all this information.

**Figure 1.15.** Hydrogen concentration along the implant specimen axis at different times (preheating temperature of $150^\circ$C). For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

**Figure 1.16.** Hydrogen concentration along the notch axis at different times for different preheating temperatures. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip
1.1.1.3. Probabilistic cold cracking criteria

The model proposed here uses Weibull’s statistical theory, and directly results from a formulation of the critical condition for initiation of fracture by the cleavage mechanism (creation of microscopic cracks under the effect of high stresses associated with the appearance of plastic strains), and the hypothesis that initiation of unstable propagation follows the statistical conditions of the weakest point (referred to as the weakest “link”) [BER 83]. Another hypothesis is also considered, namely that elementary volumes are statistically independent. In practice, when the critical condition is reached in an elementary volume, the fracture of this element induces the total ruin of the structure.

The fracture probability is written as follows:

\[ P_\delta = 1 - \exp \left[ - \left( \frac{\sigma}{\sigma_u} \right) ^m \right] \]  \hspace{1cm} [1.7]

The critical cleavage stress \( \sigma_u \) and the exponent \( m \) are dependent on temperature, microstructure and hydrogen content. These Weibull parameters have to be calibrated through experiments on tensile specimens loaded with hydrogen. The Weibull stress is calculated from the following equation:

\[ \sigma_w = \left[ \frac{\int_{\Omega} \sigma_t \, \delta V}{V_o} \right]^{\frac{1}{m}} \]  \hspace{1cm} [1.8]

where \( \sigma_t \) = \( \frac{\sigma_t}{\sigma_w} \) if the element is strain hardened

\[ \sigma_t = \begin{cases} \sigma_t & \text{if the element is strain hardened} \\ 0 & \text{if the element is elastic} \end{cases} \]

It is assumed that all elementary volumes \( \delta V \) considered in the fracture probability computation have a characteristic dimension lower than macroscopic fluctuations of the mechanical fields and \( \delta V \leq V_o \). \( V_o \) is a characteristic volume, representative of brittle fracture. It is also a parameter of the model, which is generally related to an element size of 50 \( \mu \text{m} \). To reduce the influence of meshing regarding the results, a delocalization technique illustrated in Figure 1.17 is implemented. It consists of computing the probability of fracture not at each Gauss point, but from averaged values of all the physical quantities over a sphere characterized by its radius that is equal to 50 \( \mu \text{m} \) for this application.

The probability calculation is performed for the two welding configurations, case 1 (P-H 150°C) and case 2 (P-H 200°C), considering some Weibull parameters adjusted from experiments on specimens made of bainite. Results are shown in Figure 1.18. For case 1 (P-H 150°C), the fracture probability becomes close to 1 as soon as the load is applied at time 250 s.
These results are quantitatively in good agreement with crack observations. However, even if promising, they must be considered with care. Indeed, the microstructure surrounding the notch is a phase mixture whereas the input parameters for the criteria are established for a single-phase microstructure.

1.1.1.4. Conclusion

The implant test, which is an externally loaded cold cracking test, is designed to assess the cold cracking sensitivity of parent metals used for arc welding [ISO 05]. It allows specifying the minimum preheating temperature, heat input, and maximum diffusible hydrogen content and applied stress to prevent cold cracking. On the basis of satisfying numerical results compared to experimental work, it should be possible
now to apply the proposed probabilistic criteria on real component welds or consider multipass applications with dissimilar metal deposits that are not yet covered by standards.

However, as for accurate welding simulations in general [ROB 10], a complete material characterization would be necessary to improve the Weibull model robustness in order to cover the case of phase mixture. The reason why an important effort is made to obtain such a realistic approach is to reduce conservatism arising from the microstructure–hydrogen–stress minimal law [NEV 03] and to ensure a transferability of the model on any weld configuration.

Finally, after this additional experimental characterization, a further validation should also be required through a simple industrial mock-up. As the A508cl3 low-alloy steel following the nuclear standard for procurement and welding processes is not so sensitive to cold cracking, voluntarily bad parameters should be used to observe hydrogen cracking on the mock-up: low welding energy, high hydrogen contents and initial defects leading to stress concentration.

1.1.2. PWHT and temper bead processes

For nuclear reactor applications, AREVA has to make junctions between ferritic low-alloy steel heavy section components and austenitic stainless steel piping systems. For gas tungsten arc welding (GTAW) of dissimilar metal weld (DMW) narrow gap, AREVA has developed special manufacturing procedures guaranteeing high-quality standards and resistance in service. For a decade, AREVA has been developing the numerical simulation of welding to gain a better understanding of the physical phenomena involved and to predict residual stresses. This chapter presents numerical simulations performed by AREVA on 14" narrow gap dissimilar metal girth weld mock-ups [GOM 11]. The simulations focus on the predictions of microstructure. The analysis simulates the main steps of the mock-up manufacturing procedure. Multipass welding simulation reproduces the deposit of each bead by thermomechanical and mechanical calculations. Special attention has been paid to the buttering of the ferritic side. Generally, a PWHT is carried out after the buttering of the ferritic side in order to relieve residual stresses. For some repair operations, a PWHT is not feasible. Thus, a temper bead process can be used. During this process, a large part of the previous HAZ is tempered to guarantee a limited hardness and to reduce the risk of cold cracking.

1.1.2.1. Buttering simulation

Two 14" mock-ups are considered. One is named MC2 and has been postweld heat treated after buttering. It differs from MC3 for which no PWHT has been performed as buttering is made with a temper bead welding process. Both junctions
are made of a ferritic steel A508cl3 pipe welded to austenitic stainless steel (316L) pipe by means of a Ni base alloy 52 deposited with a GTAW narrow gap multipass process. The main steps of the manufacturing procedure are as follows:

– cladding the ferritic pipe with Ni base alloy 52;
– buttering the ferritic pipe with Ni base alloy 52, with a temper bead process for MC3 and a conventional process for MC2;
– machining the ferritic pipe;
– PWHT at approximately 600°C during 2 h for MC2 only;
– buttering of the austenitic pipe with Ni base alloy 52;
– machining the austenitic pipe;
– filling the groove (approximately 25 beads) with Ni base alloy 52;
– final machining.

The mock-ups in the final state are shown in Figure 1.19.

![Figure 1.19. Narrow gap dissimilar metal weld mock-ups](image)

Two materials are considered for the buttering of the ferritic part of MC3 mock-up (see Figure 1.20):

– A508cl3 for ferritic base metal and for backing strips;
– Ni base alloy 52 for cladding and deposited beads.

A 2D model has been used to carry out the buttering modeling. The model is meshed using second-order elements and contains 11,801 nodes. The mesh is refined in the HAZ and in the welded zone. A total of 106 beads are deposited. The
first four layers are performed using a temper bead process: the process parameters lead to small beads overlapping each other in order to temper a large part of the previous HAZ. The five other layers are deposited in a conventional manner.

![Figure 1.20. Materials for buttering](image)

For MC2, the materials are identical to the ones used during MC3 fabrication. The buttering is made up of 49 beads that are deposited with a conventional GTAW process. After the buttering, the PWHT is modeled. Below 400°C, the material behavior is considered purely elastic–plastic and above 400°C, viscous effects are taken into account leading to stress relaxation as shown in Figure 1.22.

![Figure 1.21. Mesh for buttering (MC3 mock-up)](image)

The comparison between MC2 and MC3 mock-ups is first made regarding microstructure in the ferritic HAZ. The microstructure of the HAZ of MC3 is more homogeneous than the microstructure of MC2 before PWHT: the amplitude and the length of oscillations are smaller in the radial direction (see Figure 1.23). Moreover, the tempering effect is reached: the level of tempered phase is higher than the level obtained on MC2 before and even after PWHT along line 1 (see Figure 1.24). However, after PWHT, the HAZ of MC2 mock-up is more homogeneous, especially in the axial direction (see Figure 1.25), even if locally, the proportion of the tempered phase of MC3 is higher than that of MC2.
Figure 1.22. MC2 hoop stress distribution after buttering and after PWHT. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.23. Temper phase proportions. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.24. Comparison of tempered phase proportion in the radial direction between MC2 and MC3 mock-ups – line 1.
1.1.2.2. Results after welding

It is difficult to conclude on the efficiency of the temper bead process as no hardness measurement has been carried out yet through these girth welds. This kind of measurement is made on a temper bead mock-up presented in the next section. Regarding stress distributions in the weld at the end of the manufacturing of the mock-ups, simulation results give similar trends (see Figures 1.26 and 1.27). However, for the MC3 mock-up, the final level of hoop stresses in the ferritic HAZ is quite high contrary to the level observed on the MC2 mock-up. Indeed, viscous effects that lead to stress relaxation during MC2 PWHT have no time to operate during the MC3 temper bead process.
A focus is made on axial stresses that are the opening stresses for feared circumferential cracks. FE results are compared with two different measurement techniques. Neutron diffraction [HUT 05] is used to measure stresses through the MC2 mock-up and the deep hole drilling (DHD) technique is applied as a semi-invasive method based on the mechanical stress relief through MC3 thickness [LEG 96, KIN 08, COU 09]. Both experimental and numerical axial stress profiles correlate to each other as shown in Figures 1.28 and 1.29. It confirms tensile axial stresses on the outer surface and compressive stresses on the outer surface for 14" dissimilar metal girth welds.
1.1.3. Validation of residual stress prediction on a temper bead mock-up

The temper bead mock-up was approximately 960 mm long, 124 mm high and 189 mm wide. The temper bead mock-up was manufactured from a ferritic steel type A508cl3/16MND5 plate (960 mm × 149 mm × 86 mm) with an austenitic stainless steel type 308L and 309L butting (960 mm × 40 mm × 86 mm) and a Ni base alloy 82 temper bead (960 mm × 154 mm × 40 mm). The following are the manufacturing steps:

− buttering of the ferritic plate;
− postheating;
− PWHT;
− machining of two faces of the mock-up;
− cladding with a temper bead process;
− postheating;
− machining of the buttering and of the cladding.

1.1.3.1. Finite element analysis

The FE modeling methodology (the weld deposition, materials models, etc.) is the same as the methodology used for the dissimilar girth weld computation, which is detailed in [GOM 11]. Welding parameters are not given for proprietorial reasons. Thermal and mechanical analyses are treated successively. The nonlinear transient mechanical calculation is achieved using the temperature fields computed previously in the transient thermal analysis. 2D plane strain or rotational symmetry
assumptions can be made for welding computations. As the structure is similar to a plate, a section perpendicular to the welding direction is used with a generalized plane strain assumption. The temperature field is computed on a 3D model in order to properly reproduce the thermal cycles in the heat-affected region. The shape of the heat sources and the input energy are fitted to experimental data (i.e. recordings of thermocouples and the dimensions of the weld pool and the HAZ obtained from macrograph transversal cuts). Adjusted thermal cycles obtained from 3D modeling are directly applied as prescribed temperature for each bead of the 2D model. For each pass position, these beads are identified by grouping the elements of the mesh. These elements are activated incrementally when the fusion temperature is reached to simulate the material deposit according to the welding sequence. The FE meshes are shown in Figure 1.30.

![Figure 1.30. Finite element models used for buttering + PWHT and cladding simulation](image)

The material properties come from AREVA private database. The following three materials are considered:

- A508cl3 for ferritic steel;
- 308L/309L for austenitic stainless steel buttering;
- Ni base alloy 82 for temper bead cladding.

As far as temper bead process is concerned, metallurgical transformations in the ferritic base metal, i.e. in the HAZ, become a key issue. These transformations are simulated using the description of a continuous cooling transformation (CCT) diagram for as-quenched phases and equivalent time/temperature functions for tempering [ROB 12, LEB 84a]. The equivalent time/temperature cycles are
determined by tempering experiments conducted at different temperatures at different times. The tempering kinetics parameters are set according to hardness measurements obtained from these experiments [ROB 09, VIN 02]. Figure 1.31 shows the tempered phase fraction at the end of the manufacturing process, before the DHD measurement campaign. The tempering effect is reached: the level of tempered phase under the cladding is very close to the level obtained after PWHT on the butting side. This is confirmed in Figure 1.32 where hardness measurements are compared with computation results. These curves are obtained by averaging the hardness values along 10 profiles starting from the fusion line between the weld metal and the base metal. The distance between each profile is 0.5 mm as shown on the right of the figure. Points are covering the HAZ and more.

Figure 1.31. Tempered phase fraction in heat-affected zones of the mock-up. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.32. Comparison between hardness measurements and computation in the HAZ under the cladding
Figure 1.33 shows the longitudinal stresses on the complete mock-up before cutting and on the section for incremental deep hole drilling (iDHD) measurements after cutting. Residual stress FEA results are discussed in the next section by comparisons with iDHD7 measurements made on this 160 mm long residual part.

Figure 1.33. Longitudinal stresses on the complete mock-up before cutting and on the part used for iDHD measurements. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

1.1.3.2. Measurements and FEA comparison

Residual stresses were measured at a number of locations using the standard DHD technique [KIN08]. Figure 1.34 shows the diagram and a picture of the mock-up with measurement locations.

Figure 1.34. Diagram of the shortened temper bead mock-up showing the dimensions and measurement locations (all dimensions in millimeters)
High tensile residual stresses appear just at 20 mm from the interface (see Figures 1.35 and 1.36). The presence of these peaks was not predicted by FEAs and the quality of measurements had to be questioned. It showed that the first series of measurements using the standard DHD technique led to some errors due to plastic deformation during the measurement process [FIC 13]. The temper bead mock-up was then cut to 160 mm long for new measurements using iDHD measurements [MAH 09]. iDHD7 measurement was performed through the temper bead cladding and ferritic steel, at 55 mm from the buttering interface, and results were in very good agreement with FEA.

Figure 1.35. Comparison of longitudinal residual stresses at locations 1–7. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.36. Comparison of transverse residual stresses at locations 1–7. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip
This highlights the need to use the iDHD technique, which would avoid incorrect results in the event of plastic relaxation. It also shows how FEA can complement experimental approaches for a better understating of residual stresses in welds and a good and realistic assessment of stress profiles.

1.2. Welding and distortion issues

Welding processes induce residual stresses and distortions in the welded joint and the connected components. For manufacturing purpose, distortions are the main issues and up to now the problem has been handled by postwelding corrective actions. Numerical simulation has become an efficient tool for considering distortions at the earliest stages of manufacturing engineering. For instance, computational welding mechanics can be used to design the shape and the dimensions of the groove for multipass processes and to optimize the lowest distortions welding sequence of large assemblies.

1.2.1. Local shrinkage prediction

For multipass narrow gap girth welds, local axial shrinkage may be an important issue. The groove width can be reduced by a factor of 2 during welding due to the pass-by-pass shrinkage accumulation, which leads to difficulties in driving the welding device and to troubles in the component alignment. From the numerical point of view, such problems can be analyzed using a 2D axisymmetric assumption as is proposed for the following configuration. The results are achieved using computational welding mechanics of a narrow gap dissimilar metal girth weld made up of 25 beads. The approach is validated in two steps. First, the results obtained with the 2D model are discussed according to those resulting from a 3D analysis but for the first six passes only. The results of the 2D axisymmetric computation for the whole manufacturing process are then compared with experimental measurements of residual stresses and axial shrinkage.

1.2.1.1. Presentation of the models

A 14" pipe mock-up is considered. This pipe is made of an assembly of A508cl3 ferritic part welded to a 316L austenitic pipe by means of a Ni base alloy 52 GTAW narrow gap weld [COU10]. Figure 1.37 illustrates the configuration of the mock-up, called MC1, within the framework of a larger R&D program regarding DMWs [GIL 13]. We can also note the presence of a weld-deposited cladding with coated electrodes on the ferritic internal edge. There is no buttering on the ends of the two pipes. The following are the main steps of the mock-up manufacturing procedure:
- weld-deposited cladding on the ferritic edge;
- machining of the two ends before narrow gap welding;
- narrow gap welding with automatic GTAW process, approximately 25 beads being deposited in the groove;
- final machining of the mock-up.

Figure 1.37. a) The final diameters after machining and the weld centerline position. b) Start/stop welding position for the stress measurements at the sixth pass

The FE method is well suited for the computation of residual stresses and strains due to welding processes. But the main difficulty arises from the extremely high gradients around the heat source for the temperature and consequently for the stresses. The welding heat source is moving and the mesh must be refined along the weld path leading to very large FE models. One solution to reduce the number of elements of the model is to use an adaptive meshing procedure [DUR 04, ROB 07a]. This approach consists of refining the mesh around the heat source where the gradients are high and unrefining it when the heat source has moved, i.e. when gradients are lower. This procedure has yet to be successfully applied to multipass welding simulation [ROB 10, FEU 11] but still leads to high computation times. Figure 1.38(a) shows the adaptive 3D mesh considered for the present study, which contains approximately 115,000 nodes and 155,000 elements (the precise size of the mesh depends on the heat source position).
Another approach to reduce the computation time, historically the first [LEB 88, DEV 00], is to use a rotational symmetry assumption. The heat input is assumed to be deposited in one time. This assumption drastically reduces the size of FE models as the calculation is then performed on a 2D mesh, using a cylindrical coordinate system on a meridian section of the pipe. It can be justified by the fact that experimentally, the HAZ and residual stresses present a rather good rotational symmetry. But, we have to take care to apply adequate boundary conditions so as to account for the clamping conditions induced by the weld solidification (self-clamping effects). If this effect is not taken into account, big discrepancies in residual distortions when compared to experimental measurements are generally obtained. To reproduce this self-clamping effect, the structure is axially constrained during heating (weld deposit) and then unclamped during cooling [COU 10]. It has been shown that this assumption is relevant for the residual stresses as well as for the in-plane distortions [ROB 07a]. Figure 1.38(b) shows the mesh used for the 2D axisymmetric calculations in the present study, which contains 2,026 nodes and 2,096 elements.

1.2.1.2. Validation of the 2D model

The filling of the groove is partially simulated for comparison issues between 2D axisymmetric and 3D results. Only the first six passes are computed so as to limit the computation cost of the 3D simulation. The principle of the FE simulation consists of identifying each pass position in the mesh by grouping the elements and of activating them incrementally to simulate the material deposit according to the welding sequence. The “block dumped” method with a prescribed thermal cycle is applied on a 2D model with rotational symmetry conditions. The results obtained by the 3D simulation in a current region are very close to those obtained with the 2D axisymmetric calculation as can be seen in Figure 1.39 showing the residual hoop stresses after the sixth pass for both calculations.
Figure 1.39. Hoop stress distribution after the sixth pass: a) two-dimensional axisymmetric and b) three-dimensional results in a current region. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figures 1.40 and 1.41 show the stress distribution after the sixth pass at the weld centerline (Figure 1.37) in the axial and hoop directions. 3D results are very close to those obtained with the 2D axisymmetric model. The main difference arises from the stresses plotted in the cross-section of the overlapping region of the sixth pass (180°).

Figure 1.40. Axial residual stresses at the weld centerline after the sixth pass: comparison between 2D axisymmetric and 3D results in different cross-sections
We can note that 2D axisymmetric results and 3D results are in very good agreement in the current region (45°) and in the overlapping region of the fifth pass (0°). These observations confirm that the start and stop effect is annealed by the next weld deposit.
Distortions are also a source of concern regarding narrow gap welding or when the joined components are not fully clamped. Computed distortions are compared with measurements. The comparison between numerical simulations and measurements of the chamfer width reduction at the top surface are in very good agreement and thus show the ability of both 3D and 2D computations to predict axial shrinkage. The computation time per pass for the 2D simulation is approximately 8,000 times lower than for the 3D simulation.

1.2.1.3. Final welding results and comparison with experiments

To model the whole manufacturing process, the 2D axisymmetric welding computation has been performed up to the complete filling of the groove. The machining steps have also been simulated according to the manufacturing procedure by removing the elements: Young’s modulus is decreased to a very low value and Poisson’s ratio is set to zero. It is assumed that, except in a thin surface layer, the residual stress field due to welding is only modified by the stress redistribution induced by the removal of material and not by the surface heating and deformation during machining [VAL 13]. The numerical results are compared with stress measurements obtained by neutron diffraction [HUT 05] and DHD technique [LEG 96, KIN 08]. Residual stresses are observed at various locations away from the weld centerline, through the depth of the pipe. Hoop residual stresses are plotted as a function of the depth from the outer surface of the pipe in Figure 1.43.

Figure 1.43. Hoop residual stresses in the austenitic stainless steel a) and in the ferritic steel b) at 9 mm from the weld center line – measurements and 2D axisymmetric simulation

Comparisons are made both in the austenitic stainless steel (316L) and in the ferritic steel (A508c13) because the material behaviors are very different due to the metallurgical transformations that occur in the HAZ of the ferritic steel during welding. Numerical results are in good agreement with the measurements. These good results are obtained despite the fact that the assumption of axisymmetry usually leads to an overestimation of the hoop stresses close to the outer surface
Moreover, purely isotropic strain hardening behavior assumptions may also generate overestimated residual stresses [GIL 09a]. Comparisons were also made for the axial stresses and the results are even better [COU 09, COU 10].

Regarding distortions, the axial shrinkage has been measured by the welding operator in two ways. For the first, the measurement called pure axial shrinkage is made far from the groove \((A-A_0)\), where \(A_0\) is the initial distance between measurement locations. The second is measured next to the groove and is called the top width groove \((L-L_0)\), where \(L_0\) is the initial top groove width measurement. \(L-L_0\) value is influenced by the radial shrinkage that accumulates until the end of the welding. Numerical results and measurements correlate to each other. The comparison is shown in Figure 1.44.

\[\text{Figure 1.44. Axial shrinkage during multipass welding – comparison between experiments (points) and simulation (lines)}\]

This other example also shows that engineering approaches based on FE computations have reached an acceptable level of maturity for welding residual stress and distortion predictions. To obtain quantitatively good results, these techniques based on the representation of material behavior at the macroscopic scale require the definition of properties from room temperature up to high temperature as close as possible to the fusion point and have to take into account metallurgical aspects in a phenomenological manner. Concerning material characterization, recent works [GIL 09a, GIL 13] have also shown that for multipass welding applications, the material cyclic behavior needs to be considered through a kinematic hardening
model with recovery of strain hardening at high temperature. This work also highlights the fact that the material testing for mechanical behavior law identification has to be done as close as possible to the welding conditions considering possible viscous effects.

The phase transformations and strain hardening or recovery effects are usually taken into account through the constitutive equations used for the numerical simulation of welding. To validate the specific rules for material characterization under welding conditions (i.e. transformation kinetics, mixture law, annealing temperature and viscous effects), it is possible to compute a tensile test from computed state variables at the final manufacturing stage. For instance, the FE results obtained at the end of MC1 welding have been transferred into a model of a plate specimen centered in the weld as shown in Figure 1.49. The notched specimen is considered free from residual stresses before a transverse tensile test.

Figure 1.45. MC1 mesh and tensile notch specimen model centered in the weld. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Four computations have been performed. One 3D computation without any data transfer and three plane stress 2D computations considering the following initial state (thickness = 2 mm):

- no state variable (same condition than 3D computation);
- only metallurgical state variables (HAZ in the ferritic steel is considered);
- complete state variable transfer (phase fractions and strain hardening for all materials).

The different results are plotted in Figure 1.46. It shows that, as expected, the notch create a strain localization in the material to be tested. Indeed, the phase fractions in the HAZ, which is outside the notch on the ferritic side, have no effect on the global behavior of the specimen. The plane stress assumption seems to be relevant but the 3D model could be preferred if local thinning becomes important for
large strains. Finally, strain hardening that occurs during welding increases up to more than 15% the weld material resistance, which makes this quantity relevant for weld characterization validation.

![Figure 1.46](https://www.iste.co.uk/bergheau/thermomechanical.zip)

**Figure 1.46. Force versus displacement curves – numerical experiments; temperature = 300°C. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip**

Such advanced testing devices [BOU 12] are used to determine local mechanical behavior for toughness assessment in weld junctions. They are useful for obtaining local behavior in heat and mechanically affected material of the welded joint in order to estimate the mismatch effect in comparison with homogeneous base metals. Both numerical and experimental approaches could complement each other to check the reliability of computational welding mechanics results and also to assess the mechanical behavior of dissimilar weld junctions under operating conditions.

### 1.2.2. Global distortions

The numerical simulation of welding is growing within the nuclear industry because of several projects trying to solve welding issues such as weldability related to the process itself or their consequences in terms of by residual stresses and distortions. Numerical modeling can help to predict stresses and distortions in structures in order to optimize or develop new manufacturing processes or weld repair techniques. To calculate final distortions at the scale of the structure, 3D models considering all the welding operations must be computed. A solution to predict welding distortions of large components is then to use a multiscale approach [SOU 02, FAU 04, ROB 07b] in which the welding simulation is accurately performed on a small part of the welded component (local model) and then...
transferred onto the global model made up of 3D solid or shell elements for distortion calculation. The inner forces computed in the local model and due to residual inelastic strain field are prescribed as initial strains in the global mesh by means of welding macroelements having their own stiffness matrix and load vector. The computed welding macroelements can also be inserted step by step in the global model in order to calculate the evolution of the component distortions during the welding sequence. Figure 1.47 shows how challenging the multiscale problem of the large electron beam welded structure is, due to the size and number of welds to be simulated. Optimized solutions can be found through local/global approaches by changing the sequence and/or the clamping conditions and solving new linear elastic problems. This approach dedicated to distortion issues allows the modeling of long and numerous welds with a reasonable calculation time.

![Figure 1.47. Illustration of a multiscale problem: the assembly of the ITER vacuum vessel](image)

Many projects include experimental mock-ups to validate models and software. The work presented in the following section was performed within the framework of the International Thermonuclear Experimental Reactor (ITER). It deals with the computation of the mock-up manufacturing process of an element of the vacuum vessel (VV).

1.2.2.1. Mock-up description

ITER is an advanced fusion project that faces a lot of challenges at every technical level, especially on the manufacture of the VV that has strict tolerances (~10 mm) compared to the global dimensions of the structure (~10 m) [JON 03,
To study the manufacturing process of the VV, different welding technologies have been investigated. The electron beam welding appears to be more appropriate for numerous important welds because of the low distortions induced by this process. A coupon of an element of the VV (1 m × 1 m), called the validation of procedures on e-beam-welded coupon (VEC), has been developed by DCNS in order to validate the Electron Beam (EB) welding of austenitic stainless steel plates (316L(N) ITER GRADE) [MAR 09, GAL 10].

The VEC assembly process is composed of 13 welds. Before being welded, the shells are bent by rolling. The rolling process consists of the displacement of a plate between three cylinders to obtain the required curved shape. The plate is deformed in several passes (displacement from one side to the other) between three cylinders. After the rolling, the overlengths are water-jet cut to obtain the final dimensions of the plates. Finally, the holes are drilled by means of water-jet cutting to get the final product. The residual stresses induced by the manufacturing process are not taken into account in the welding simulation since the study of their influence on the welding distortions has been shown negligible [GAL 10].
The first welding operation consists of a circular melt run on the middle of the inner shell, which represents the assembly of another component with the inner shell. In this chapter, this step is called “key welding”. This component is not present in the mock-up. The second step is the welding of the four components (two ribs, the inner shell and the outer shell) by longitudinal welds in horizontal position. Before the welding operation, the structure is tack-welded with partial penetration electron beam welding of 5 mm depth. This operation makes the structure stiffer and fixes the assembly of the four components before the full penetration welding operation. The last step consists of welding the housings onto the structure using circular welds joining the housings with the inner and outer shells.

1.2.2.2. Distortion measurements

Normal distortion measurements are based on the photogrammetry method. Several spots are placed on the plates and different pictures from different viewpoints are taken in order to have a 3D perspective of the plates. Finally, the comparison is performed with the computer-aided design (CAD) geometry with a gauging. The latter corresponds to the application of the least squares method between the deformed and the CAD structure.

The measurements are carried out on a grid of points on the inner and outer shells. Comparisons between the experiments and the computed displacements are performed at the location of these points. The measurements are represented by points and the numerical ones by curves. Their positions on shells are shown in Figure 1.49.

Figure 1.49. Grid for the inner a) and the outer b) shells
Comparisons are performed according to the normal direction to the plate surface. Relative values of distortions between two welding steps are compared: the compared values correspond to the difference of the values between the state after and the state before welding.

1.2.2.3. Simulations

For the first and last steps of welding, which are circular welds, a 2D model with rotationally symmetry conditions is used. For the second step, which deals with longitudinal welds, a 3D local model is mandatory to obtain reliable results. Figure 1.50 shows the distribution of the cumulative plastic strain near the weld line for the key welding.

![Figure 1.50. Cumulative plastic strains of the key welding local model. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)

The boundary conditions for the global simulation are set up in order to prevent rigid body motions and the distortions are computed after each sequence by a step-by-step insertion of the local model results. The welding distortions due to the key welding lead to a saddle shape for the inner shell. The longitudinal welds cause the inner and outer shells to move toward the interior of the VEC mock-up. The welding operation of the housings assembly is divided into two stages: one for the welding of the first two housings, and other for the welding of the last two housings. After the first stage, there is a translation of the welded housings toward the concave side that causes a flexion of the inner and outer shells. The second stage leads to the same phenomenon.
1.2.2.4. Comparison between experimental and numerical results

This section shows some examples of comparisons between the experiments and the computation results. The main welding results are shown in Figure 1.51 and the longitudinal welding results in Figure 1.52. Figure 1.53 shows the distortions due to the welding of the housings with the inner shell.

**Figure 1.51.** Comparison of displacements for the key welding along lines L7 and L12 in the direction normal outward from the inner shell.

**Figure 1.52.** Comparison of displacements for the longitudinal welding along lines L7 and L12 in the direction normal outward from the inner shell.
Figure 1.53. Comparison of displacements for the housing welding along lines L3 and L11 in the direction normal outward from the inner shell

Numerical results obtained on the VEC mock-up show that the global trend of the deformed shape is well represented and the amplitude of the distortions is in good agreement with the measurements. This work demonstrates that the quality of the local model and its boundary conditions play a role in the global distortions: 3D local models are mandatory for longitudinal weld simulation and 2D axisymmetric models are acceptable for circular weld local plastic strain calculations. After each welding operation, distortions have been measured at some points of the mock-up. The comparison between computation and experiments is performed using these measurements. The main trends of the experimental deformed shape are well represented by the simulation. Comparisons are good on the two main components of the mock-up.

Concerning assembly distortions, the local/global approach is very promising, even for thick components and multipass welds [PON 11]. The advantage of this method is that it allows the investigation of numerous welding sequences in short calculation time. To give numerical modeling an additional level of confidence, the measurement methods are also improving. Here also, both approaches complement each other, measurements can present local results for a particular configuration and simulation gives a complete cartography of macroscopic quantities under idealized conditions. In the manufacturing industry, this is an important issue when safety is related to high requirements leading to a low margin. In the nuclear industry for instance, integrity assessments with low margins are sometimes conducted and statements can be based on best-estimate computations validated through experimental techniques [GIL 09b].
1.3. Integrity assessment of welded structures

1.3.1. DMW junction

In pressurized water reactors (PWR) or boiling water reactors (BWR), heavy section components made up of low-alloy steel are in most cases connected with stainless steel piping systems. The DMW junctions are created between ferritic nozzle ends and austenitic stainless steel piping, following a special manufacturing procedure to ensure a good quality and resistance of the joint. PWHT is applied to reduce residual stresses in the HAZ and increase its ductility, but whatever the process is, the difference in thermal expansion coefficients of the dissimilar materials induces residual stresses during the cooling stage. Furthermore, differences in tensile properties (yield limit mismatch) may cause strain concentration at the weld to ferritic steel interface, which enhances the risk of crack initiation and extension. Since welds exhibit a lower toughness than the base metal, investigations of the influence of residual stresses on DMW resistance to ductile fracture have been performed through several successive European Community Research and Development Programmes. Two of these former projects BIMET and ADIMEW contributed to the development and verification of analysis methods, which describe the behavior of an external circumferential defect in a DMW [GIL 09b]. In both projects, ductile fracture tests were conducted on real dimension pipes in which a large crack was machined close to the interface between ferritic and weld materials. The BIMET project has been launched to examine the feasibility of the ADIMEW project. The BIMET mock-ups were much smaller (6" instead of 16" for ADIMEW as shown in Figure 1.54) and have been tested at room temperature, whereas the ADIMEW test has been carried out at 300°C.

![Figure 1.54. Macrographs, materials and thicknesses of DMW mock-ups BIMET and ADIMEW](image)

1.3.1.1. The BIMET mock-up: global approach of fracture

In bimetallic welds, the type of fracture is ductile, but the question of the influence of the residual stress fields on tearing initiation has been raised since the
Material resistance is lowered when the crack is close to the interface between the ferritic steel pipe and the weld [DEV 87]. It has been shown that this lowering of the local toughness is due to the high degree of stress triaxiality at the interface [GIL 06]. In defect assessments, tensile residual stress fields have to be taken into account, except if their influence may be proven as being negligible.

The assessment of the fracture resistance can be made by a global approach. It consists of calculating the crack driving force $J$ through the energy release rate computation based on the $G–\theta$ method and comparing this value to measurements conducted on crack specimens [ROB 09]. The criterion used to assess the risk of initiation in mode I is commonly called $J_{IC}$. Below this value, the crack remains stable and does not propagate. The fracture resistance has been measured on a three-point bending test performed on a single-edge notched bend (SENB) cracked specimen taken close to the interface in the first layer of the buttering. For this material and this weld configuration, the fracture resistance is between 135 and 165 kJ/m² as shown in Figure 1.55.

![Figure 1.55. Load-crack mouth opening displacement (CMOD) curve and resistance curve $J–\Delta a$ obtained on an SENB taken from the material where the crack is placed in BIMET mock-up](image)

All the manufacturing stages have been simulated to obtain the residual stress state. For this mock-up, the PWHT was done after the buttering of the ferritic pipe and not at the end of the weld manufacturing. The crack whose tip is closer to a notch is electro-discharge machined at the outer surface, parallel and very close to the ferritic steel/buttering layer interface as shown in Figure 1.56.
Residual stress measurements were performed using the neutron diffraction technique across the piping thickness in the butting, the weld and the HAZ of the base material. The best agreements with the measurements are obtained for the viscoplastic and the combined strain hardening models [GIL 09b]. Figure 1.57 shows the residual state after machining of the notch and Figure 1.58 compares computed and measured residual stresses.

The effect of residual stresses is greater on the fracture parameter $J$ since stresses and strains are combined as a product in the $J$ computations. Figure 1.59 clearly shows that the kinematic strain hardening model gives much better prediction of initiation than the computation without residual stresses and the isotropic model. This is evidenced not only by comparing $J$ to $J_e$ but also by the comparison of the
measured moment values at initiation. Since the crack is opened by the residual stresses, $J$ is not equal to zero for the computations considering a residual initial state.

![Axial stresses at 3 mm from external surface](image1)

**Figure 1.58.** Comparison of axial residual stress profile at 3 mm from the outer surface between measurements and computations.

![J (kJ/m²) versus applied moment](image2)

**Figure 1.59.** Energy release rate versus applied moment considering residual stresses (*_RS) or not (*_NoRS)

The crack driving force is higher in the range of the initiation detected experimentally (between 155 and 170 kN·m) when considering residual stresses with any type of strain hardening. In any case, complementary analysis [GIL 09b]
48 Thermomechanical Industrial Processes

have shown that the effect of residual stresses on ductile fracture becomes negligible as soon as the resistance parameter has a sufficiently high value to obtain an initiation load higher than 80–90% of the limit load of the cracked section. Regarding propagation, high deformations at the crack tip decrease the effect of residual stresses that are also redistributed when the crack propagates.

Considering the global approach of fracture, the resistance curve \( J - \Delta a \) is used to determine the critical condition reached when the crack propagation becomes unstable. It relates the increasing fracture resistance energy \( J \) applied to a cracked structure that is required to increase the crack length by \( \Delta a \). Although these curves may present discrepancies according to specimen geometries or loading conditions, they are used as intrinsic material properties for integrity assessment. Another method to solve this issue and to answer the question of transferability from the specimen to the structure is to consider a more efficient theory, which is the local approach of fracture. Within this frame, it is possible to analyze both initiation and propagation in case of ductile tearing.

1.3.1.2. The ADIMEW mock-up: local approach of fracture

The ADIMEW mock-up was postweld heat treated at 600°C for 6 h at the end of the weld manufacturing. The dimensions of the weld as well as the crack description are given in Figure 1.60.

![Figure 1.60](image)

As for BIMET mock, the ADIMEW mock-up has been submitted to a four-point bending pipe test. The ADIMEW test has been conducted at 300°C on the piping assembly shown in Figure 1.61. Crack propagation has been measured during the experiment using the potential drop technique, which has been calibrated in the same manner as for BIMET tests.
Many damage models exist to model the ductile tearing of metallic structures. In this section, we focus on the Wilkins model. The Wilkins fracture model [WIL 99] is a cumulative strain damage model, which postulates plasticity as a relevant measure of damage for metals. A damage functional $D$, uncoupled from the behavior law, is used. When it exceeds a critical value $D_c$ over a critical material volume $R_c$, it leads to discontinuous macrocrack creation and stepwise growth. The damage toughness criterion $D$ is suitable to predict both fracture initiation and crack propagation. This model takes into account a local strain history representation based on McClintock’s theory of ductile fracture [MCC 68] that incorporates:

- microvoid growth through hydrostatic tension;
- microvoid coalescence through asymmetric strains;
- rupture size effect through critical material volume that allows us to control localization effects.

$$D = \int w_1 w_2 \, d\varepsilon_{eq}^p$$ \hspace{1cm} [1.9]$$

where $\varepsilon_{eq}^p$ is the equivalent plastic strain and $w_1$ is a hydrostatic pressure weighting term calculated as in the following equation:

$$w_1 = \left(1 - \frac{P}{P_{lim}}\right)^a$$ \hspace{1cm} [1.10]$$
$P$ being the hydrostatic pressure, $D = D_c$ when $P \geq P_{lim}$

$w_2$ is a deviatoric stress weighting term and its expression is given in following equation where $s_1 > s_2 > s_3$:

$$w_2 = (2 - A)^\beta$$

$$A = \max \left( \frac{s_2}{s_1}, \frac{s_2}{s_3} \right)$$

$s_1$, $s_2$ and $s_3$ are the principle stresses of the deviatoric stress tensor. Parameters $D_c$, $P_{lim}$, $\alpha$ and $\beta$ are adjusted from experimental results, ideally performed under multiaxial loading conditions.

The delocalization distance, called $R_c$, is an intrinsic material property as it is linked to the intercavity distance. Each set of parameters $D_c$, $P_{lim}$, $\alpha$ and $\beta$ is associated to this characteristic distance. For 2D modeling, the delocalization volume corresponds to a disc. In the case of 3D modeling, this volume is a sphere. Figure 1.62 shows the delocalization area for two Gauss points in a 2D mesh. This technique makes the criteria non-local, independent from the mesh size if it is below $R_c$ and can be used for any crack geometry in 2D and 3D. When an integration point reaches $D_c$ value, its stiffness vanishes and the crack initiates and then propagates.

![Delocalization technique used to compute fracture criteria](https://www.iste.co.uk/bergheau/thermomechanical.zip)

**Figure 1.62**. Delocalization technique used to compute fracture criteria. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Wilkins model parameters have been identified using a precracked side groove SE(B) specimen after fitting force–displacement measurements with simulation results.
The Wilkins model that has been previously detailed is dedicated to predict both crack initiation and propagation in case of large deformations as those occurring in crash simulation. A delocalization technique was used to simulate the crack
deviation and extension without any pre-orientation of the mesh. To simulate the bending test of the mock-up, one-half of the component was modeled for symmetry reasons. The mesh density is very high at the front of crack tip as required for the Wilkins local approach (see Figure 1.65(a)). The 3D calculation was stopped after 12 mm of crack propagation corresponding to the dimension of the locally refined mesh, which represents one-half of the experimental extension (see Figure 1.65(b)). The bending test was performed until the crack propagated though a distance of 28 mm. This is the reason why comparisons between numerical results and experiments focus on the crack path direction and its shape.

![Figure 1.65 a) Mesh of one-half of the pipe in the symmetry plane with special attention to the crack area. b) Computed and measured ductile crack path. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)

Figure 1.65 also demonstrates that the crack deviates toward the interface and remains along it. The shape of the crack extension is also well simulated as shown in Figure 1.66: the crack grows in the depth of the pipe and not on the surface, which is in good agreement with the experiment. The larger crack extension is observed in the plane of symmetry.

![Figure 1.66. Measured and computed crack extension through the thickness. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)
1.3.2. **Ductile tearing prediction in welds considering hydrogen embrittlement**

Some flaws may appear in metal components, in the weld region and, more specifically, in the case of electron beam girth weld in the slope area of the process (start and stop of the welding operation). These initial flaws can grow with delay even without any external loads. Indeed, close to the junction, the material undergoes the combination of high tensile residual stresses due to welding operation and the presence of hydrogen brought by manufacturing process. Hydrogen-assisted cracking is then suspected to explain the origin of crack growth through hydrogen embrittlement of the base metal.

To understand by numerical modeling, at least qualitatively, the scenario of appearance of such cracks and their evolution, without any external load or underpressure load, the proposed approach first consists of simulating the welding process and its consequences on residual stress distribution and hydrogen concentrations. As shown in Figure 1.1, strong coupling is performed to solve the thermomechanical part and weak coupling is sufficient to predict mechanical states (residual stresses and distortions) and hydrogen diffusion. Thus, the hydrogen diffusion computation is pursued after the welding operation simulation in order to highlight the most critical moment at which macroscopic defects may appear. Then, a macroscopic defect is created in the thus-determined critical zone, the stability of which is studied by estimating the energy release rate at the crack front and by comparing these values with experimental data such as the critical energy release rate at initiation and the tearing resistance curves, which may depend on the hydrogen content. So, it is numerically possible to propagate the defect in the time, considering hydrogen diffusion and residual stress rebalancing, by successive crack front definition performed as the crack tip region exceeds the critical energy release rate [ROB 13a, GIL 11]. Finally, the evolution of the defect is estimated in the same way under pressure test loading conditions. Results and discussions are presented to propose an engineering approach for the design assessment of such specific weld junctions with a low and hydrogen-dependent toughness.

1.3.2.1. **Welding simulation**

1.3.2.1.1. **Welding process simulation**

The cylindrical structure is welded by a high-energy electron beam process. The structure is made up of three materials that must remain confidential. A section with the rotational axis is shown in Figure 1.67. Material 1 and material 2 are connected by the weld performed without filler metal. The HAZ, which is of interest, is placed in material 1. Material 3 is too far from the weld to be heat affected. The structure is large enough for the welding to quickly reach a thermal and mechanical steady state. This specificity is used for the simulation. More than 90% of the welding joint is solved in one step by using a steady state computation [BER 99, BER 00]. The
remaining sector is solved by a standard transient step-by-step method. This scenario allowed us to predict the mechanical state in the current zone as well as in the closure area (electron beam overlap and slope down).

Figure 1.67. Structure and components around the weld

1.3.2.1.2. Adjustment with thermocouples and macrographs

From the simulation point of view, the electron beam input energy is modeled by a running heat source represented by a power density function applied along the welding trajectory. The parameters of this function have to be adjusted in a way that the numerical macrograph (left picture in Figure 1.68) fits as closely as possible with the experimental macrograph (right picture in Figure 1.68).

Figure 1.68. Heat source adjustment
This calibrating stage ensures that the computed thermal gradients are representative of what happens in the structure during the welding operation. This observation has also been correlated by thermal cycle measurements at different locations. The graph in Figure 1.68 shows the comparison between thermocouple and simulated maximum temperatures at these locations. Figure 1.69 highlights the distribution of temperatures on the structure around the heat source and through the thickness. The conical molten zone shape is typical of electron beam welding process. The different materials of the joint are subjected to metallurgical transformations that are modeled with a first-order differential equation where $p$ is the new phase fraction and $\theta$ is the temperature [LEB 84a]:

$$
\dot{p} = f(p, \theta, ...) \tag{1.13}
$$

**Figure 1.69.** Temperature fields in the current zone. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

1.3.2.1.3. Computational welding mechanics

The mechanical simulation is conducted assuming an elastoplastic behavior of the materials with isotropic hardening [FEU 11]. Due to phase transformation, it is possible to consider recovery effects in the HAZ. A nonlinear mixture law between the two parent metals is used for the molten zone in order to fit hardness measurements. The hoop stress contours shown in Figure 1.70, which is a cut in the welding direction in the middle of the joint, highlight the 3D effects in the electron beam closure area where no steady state exists.
The area of high tensile stress is the location where potential crack may appear. Figure 1.71 shows the distribution of the stress fields in the current and in the slope areas. The current area is the portion of the structure where a thermal and mechanical steady state exists. It covers approximately 90% of the structure circumference. The slope area is the transient region created during beam welding. It corresponds to the portion of weld pass that is remelted and subjected to the slope down (the energy of the electron beam is progressively switched off). Significant tensile hoop stresses are observed in the HAZ of the slope area, which promotes the creation and growth of axial cracks as those revealed by non-destructive examination made on the component.

Figure 1.72 shows two mechanical quantities (stress triaxiality and cumulated plastic strains) that are key parameters for crack behavior and for hydrogen diffusion into the structure.

1.3.2.1.4. Numerical simulation of hydrogen diffusion

The risk of cold cracking in the welded zone arises from the combined presence of hydrogen, welding residual stresses and cumulated plastic strains. Hydrogen is introduced in material 1 during its manufacturing. The structure is not environmentally exposed to hydrogen. The purpose of the hydrogen diffusion calculation is to quantify the evolution of the hydrogen concentration in the structure during the life of the component. The initial hydrogen concentration arises from the manufacturing process of the component.

The calculation of the evolution of the hydrogen concentration in the structure uses a model that is a generalization of Fick’s second law. The model is described in section 1.1.1.1.4.
The introduction of the plastic deformation into the exponential term of equation [1.14] allows us to decrease the trapping effect with an increasing temperature but the effect becomes overestimated if the plastic deformation reaches high values (at crack tips or in singularities, for instance). Another formulation should be established but practically, the plastic deformation dependency is simply limited to a
threshold value that appears as an additional parameter. In our problem, the plasticity at the crack tip may become high and unrealistic due to discontinuous crack propagation and this parameter is thus arbitrary set to 6% considering that damage occurs when this value is reached.

Figure 1.73 shows the distribution of the hydrogen content in the component after 1 year. It highlights the trapping effects of the strained hardened area. It also shows the effects of the difference of hydrogen solubility at the interface between components.

![Figure 1.73. Hydrogen distribution 1 year after welding without crack. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)

1.3.2.2. Fracture analysis

1.3.2.2.1. Methodology

To study the behavior of a defect located at the root of the weld due to the presence of porosities (spikes due to the electron beam welding process as shown in Figure 1.74), it is necessary to define its shape and its initial position at a macroscopic scale: the size of the defect to be modeled has to be greater than 1 mm. For modeling, we consider an initial defect in an axial plane. It is a millimeter-length size defect with a circular shape placed in the HAZ on the soft material side (material 1).
In the plastically deformed zone, the circumferential residual stresses are in tension and we may expect higher hydrogen contents. After calculation of the crack energy release rate $J$ by the $G - \theta$ method at the initial crack front [GIL 93], the defect is discontinuously propagated by steps of maximum size 0.5 mm at the point presenting the highest value of $J$ greater than $J_{IC}$ considered as equal to $J_{0.2}$ (the $J$ value at crack initiation). This crack growth value must be large enough to represent a potential increase of $J$ but not too large to have a discrete propagation close to a continuous process). At points where $J < J_{0.2}$, the crack does not propagate. Then, a new crack front is defined. The physical quantities are transferred on the new mesh containing a line of nodes at the new crack front position. The faces of the crack are released and residual stresses are redistributed until a new inner force balance is reached. If values of $J$ at the new crack tip remain greater than $J_{0.2}$, this crack continues to propagate by maximum steps of 0.5 mm at the point presenting the highest value of $J$. This operation is repeated until the defect stabilizes. Finally, a calculation of hydrogen diffusion is performed on the successive cracked meshes in order to update the hydrogen concentration due to plasticity that develops at crack tips and plays a role on trapping effect (see Figure 1.87). As the critical energy release rate depends on the hydrogen ($J_{0.2}(H)$) that diffuses around the crack, the stability has to be examined: the increase of hydrogen concentration must satisfy the relation $J < J_{0.2}(H)$. 

Figure 1.74. Micrographic examination of the fusion line highlighting porosities of the order of some dozens of micrometers
The defect arrests under the combined effect of residual stress relaxation and stabilization of the crack growth resistance. The pressure test at room temperature can now be simulated. The characteristic time of diffusion of hydrogen in the soft material at this temperature is very small. The characteristic diffusion distance \( \delta = \sqrt{D \cdot t} \) for a period of 24 h is approximately 0.05 mm without considering trapping effects. This distance is too small to justify updating the hydrogen distribution in the structure during the crack growth under pressure test, unlike what was proposed for studying the crack growth in storage conditions. The pressure inside the structure is gradually increased up to 180, 216, 232, 241, 253 and finally 265 bar. If the crack does not break through the wall thickness, the simulation will be carried on by increasing the pressure until it breaks through. At each point of the crack front, the crack growth \( \Delta a \) is calculated from the resistance curves. As soon as the maximum crack growth reaches 1 mm, the position of the new crack front is defined on a new mesh. The state variables and physical quantities are transferred on to the new mesh containing a line of nodes at the new crack front position. The faces of the crack are released and the stresses are redistributed under a same level of pressure. The pressure load is increased starting from the level before the previous crack growth. Then the propagation–remeshing–result transfer stages are repeated until the maximum pressure of the test is reached or the wall is broken through. The position of the device used to perform the pressure test is shown in Figure 1.75.

![Figure 1.75. Pressure test modeling – pressure application and boundary conditions. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)

1.3.2.2.2. \( J \) assessment on specimen

\( J-R \) resistance curve assessment is made by simulating ductile tearing tests performed on SE(B) specimen taken in the soft HAZ (material 1). The specimen
geometry is shown in Figure 1.76. J-R testing can be conducted in air as diffusion of hydrogen in material 1 is very slow. Experimental force versus displacement curves obtained for specimen with different levels of hydrogen content are shown in Figure 1.77. These experimental results are used to fit the 3D FE modeling of the tests (see Figure 1.78). Simulations are performed with different static crack lengths and J is calculated following the standards [AST 01]. To have a good agreement between the ASTM formula and the $G - \theta$ method for $J$ value, the geometric factors $\eta$ of the ASTM formula are adjusted for each computation. Comparisons between experiments and static crack computations are shown in Figure 1.79. $J$ versus $\Delta a$ points are obtained where simulation and experimental curves intersect. $J_{0.2}$ is obtained using the blunting line. The resulting $J$-$R$ curves used for the rest of the simulation are shown in Figure 1.80.

---

**Figure 1.76.** Specimen definition (ASTM E 1820-01): SE(B) 10 × 10 × 55 [AST 01]

**Figure 1.77.** Force versus displacement for different specimen
Figure 1.78. Finite element model used for the test simulation (Von Mises stresses). For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.79. Force versus displacement for static cracks ($a_0, a_0 + 0.5, 1, 1.5$ and $2$ mm)

Figure 1.80. $J$-$R$ curves for two levels of hydrogen concentration ($10$ and $17$ ppm)
1.3.2.2.3. Crack propagation

The shape of the first crack designed for the propagation analysis is elliptical, with a minor axis equal to 1 mm and the major axis equal to 2 mm. The hoop stress redistribution after opening the defect is shown in Figure 1.81. Cumulative plastic strain increases at the crack tip (see Figure 1.82).

Figure 1.81. Hoop stress redistribution in the crack plane at the first crack position. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.82. Cumulative plastic strain at the crack tip. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip
The subsequent crack propagations are shown in Figure 1.84. $J$ values along the crack tip for each of these crack positions are shown in Figure 1.85.

We can note that the propagation is rapidly slowed down at the root of the weld and that the highest $J$ values are placed at the upper crack front (the part of the front that is closer to the outer surface). This evolution may be basically explained by the hoop stress distribution, which becomes compressive below the root of the weld and whose tensile components tend to decrease toward the outer surface (see Figures 1.85 and 1.86).
Figure 1.85. $J$ values along the crack front at the 12 first crack positions. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

Figure 1.86. Qualitative relation between crack propagation kinetics and hoop stress distribution. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip
From the sixth crack position, the maximum $J$ value along the crack tip, on the opposite side to the weld, starts to decrease. At the 12th crack position, the $J$ value becomes lower than at the beginning and it is decided to continue the simulation with the pressure test in order to see if approaching in-service loading conditions may lead to break through the wall.

During the pressure test, there is no hydrogen diffusion computation, as the hydrogen displacement in the low toughness material is too small at the time scale of the test. The hydrogen distribution considered for the pressure test is the one computed after 26 months as shown in Figure 1.87.

![Figure 1.87. Hydrogen distribution 26 months after welding with crack propagation. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip](image)

The new crack front prediction (crack front number 12 $\rightarrow$ crack front number 13) at 60 bars and the definition of the smoothed crack front are shown in Figure 1.88. As hydrogen concentration at the crack tip is higher than 17 ppm (19 ppm), the maximum $J$ values lead to a maximum crack propagation of 1.6 mm as shown on $J$-$R$ curves in Figure 1.89.

![Figure 1.88. Crack front number 13 (prediction in light color, manual smooth front in dark)](image)
The following front prediction (crack front number 13 → crack front number 14) at 160 bar and the definition of the smoothed crack front are shown in Figure 1.90. As hydrogen concentration at the crack tip is lower than 10 ppm (9.7 ppm), the maximum crack propagation is approximately 0.8 mm according to $J$-$R$ curves as shown in Figure 1.91.

Consecutive crack front positions 15, 16 and 17 are obtained for a pressure of 220, 260 and 300 bar, respectively. At 300 bar, the crack does not break through the wall, whereas in the slope area, perforations were observed experimentally before 260 bar. Figure 1.92 shows all the crack front positions. The line gives the crack front prediction at 340 bar and we can note that this time, the crack front predicted from the previous position goes out to the external surface. The crack propagation decelerates in the region where the H content is less than 10 ppm (residual tensile stresses are also lower) and the crack propagation becomes higher in the smallest residual ligament.
Figure 1.91. Crack front number 14 – maximum propagation determination with 10 ppm of H and less at the crack tip.

\[ \Delta a_{\text{max}} = 0.8 \text{ mm} \]

Figure 1.92. Consecutive crack front positions until the pressure test reaches 340 bar. For a color version of this figure, see www.iste.co.uk/bergheau/thermomechanical.zip

1.3.2.3. Discussion

The fracture mechanics simulation properly predicts the locations of the cracks and their shapes. These ones are very similar to non-destructive examination results and other experimental observations. Indeed, the crack evolution and the external surface break-through are predicted at the correct location. The level of maximum acceptable pressure is overestimated by simulation due to a lack of accurate data for the resistance curve. Indeed, J-R curve assessment should also be performed with
other crack specimen representatives of the heat-affected material under a controlled level of hydrogen.

Nevertheless, the different digital tools and models used for the computations presented in this chapter are now applicable for industrial issues. The welding simulation results are quite consistent with the experiments. The methodology can be applied with good confidence with respect to residual stresses that mainly drive the crack propagation. Coupled with simulation of specific toughness tests (dissimilar material, non-standard cracked specimen), this chained approach gives new prospects for fracture analysis based on J criteria for large ductile tearing [GIL 11].

Complementary to such stress and fracture mechanics analysis, manufacturing engineering can be supported by numerical simulation tools to propose relevant mitigation or repair procedures in case of manufacturing non-conformity [ROB 13b, FIC 13].

1.4. Bibliography


