Thermomechanical modeling and experimental study of a multi-layer cast iron repair welding for weld-induced crack prediction

E.B. Farahani, A. Sarhadi, M. Alizadeh-Sh, S. Fæster, H.K. Danielsen, M.A. Eder

Department of Wind and Energy system, Technical University of Denmark, Frederiksborgvej 399, 4000 Roskilde, Denmark

ARTICLE INFO

Keywords:
Ductile cast iron
Repair welding
Thermo-mechanical modeling
Residual and thermal stresses
Crack formation

ABSTRACT

Large-scale components such as hubs in wind turbines are often made of cast iron to minimize the production costs. One of the common challenges in the casting process of such large-scale components is manufacturing defects. However, repair welding will induce residual stress which can initiate cracks in the repaired structure, especially since cast iron is not as tough as steel. The current study addresses developing a thermo-mechanical model of the cast iron repair weld validated with experiments to predict thermal and residual stresses and to identify critical locations for crack initiation. A thermo-mechanical weld model is developed, and the predicted temperature and residual stress distribution are validated against experimental data. Two repair weld experiments, one manual and one automated are carried out and are simulated using the developed thermo-mechanical model. The regions with maximum principal residual stresses are calculated by the thermo-mechanical model and the maximum principal stress method is used to predict the location and direction of the developed cracks in the repair weld. A comparison with the repair weld experiments shows good correlation with the observed cracks in the welded specimens. The outcome of this research provides a basis for repair weld optimization of large-scale cast iron components in order to reduce the carbon footprint caused by their reproduction.

1. Introduction

Engineering structures are subjected to extreme mechanical loading and simultaneously exposed to harsh environmental conditions involving thermal, hydraulic, chemical, and electric effects during their service life. Fracture may be arguably the most typical failure mode of such structures operating under such multi-physical conditions. In order to mitigate the risk of structural failure, it is crucial to accurately predict the formation and propagation of cracks in materials and quantify their negative impact on structural integrity.

Cast iron is a versatile material applied to manufacture large and geometrically complex components by casting. Because of its excellent properties cast iron alloys are developed and extensively utilized in the industry field. Several numerical and experimental studies have been conducted on cast iron including simulating the microstructure of cast iron [1], microstructure characterization [2], surface treatment using plasma beam [3], and investigating ductile failure of cast irons using uniaxial tensile tests [4].

Despite the remarkable mechanical properties of cast iron, it can have a high propensity to generate a thermally induced hard and brittle microstructure, which increases its susceptibility to crack initiation and propagation. Malek Ghaini et al. [5] performed a powder welding process to demonstrate the features of cracks in the heat-affected zone of ductile cast irons. According to their findings, cracks arise from the graphite nodules and propagate through the martensitic matrix influenced by the severity of residual stress (RS). Alizadeh-Sh et al. [6] used a linear regression analysis to establish a processing map based on empirical-statistical relationships between laser cladding track geometrical features and processing parameters. They attempted to identify the process parameters that would result in desired clad geometries, as well as to predict critical conditions leading to solidification cracking.

When considering large-scale cast iron components, such as hubs for wind turbines, complete recasting of entire components featuring a flaw or defect is time-consuming, costly and bears a significant carbon footprint. For these compelling reasons, the application of a repair welding process for defects in large scale components is desirable in order to avoid scrapping and remelting. However, repair welding can induce undesirable RS and distortions in the repaired area, which can diminish the performance and decrease the service life. To predict RS, Chang and Teng [7] examined a butt-welded joint numerically and experimentally.

* Corresponding author.
E-mail address: asar@dtu.dk (A. Sarhadi).

https://doi.org/10.1016/j.jmapro.2023.08.059
Received 20 February 2023; Received in revised form 4 July 2023; Accepted 29 August 2023
Available online 18 September 2023
1526-6125/© 2023 The Authors. Published by Elsevier Ltd on behalf of The Society of Manufacturing Engineers. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).
They aimed at improving currently employed fabrication processes in welded structures. Song et al. [8] investigated several parameters in pipe girth welds including the radius-to-thickness ratio, wall thickness, heat input, and joint preparation to identify key controlling parameters affecting residual stress distributions. Borzabadi Farahani et al. [9] introduced a three-dimensional finite element model of welding medium carbon steel pipes taking into account both the martensite transformation and the post-weld heat treatment. For the repair process, the area containing the defect is removed by grinding or milling, and then the cavity is filled with welding filler material. The repair weld experiences thermo-mechanical stresses entailing microstructure degradation owing to the repeatedly applied elevated heat during the welding processes. Extreme heating and cooling rates during the thermal process, especially in the heat-affected zone (HAZ) of multi-pass welds, can cause a significant deterioration of the ductility and toughness.

In addition, welding RS arises as a result of the structural non-homogeneous heat distribution. Charkhi and Akbari [10] indicated the weld-related stresses in the repair weld are generally higher than those in the original weld because of welding of gouged grooves in the overall structure is subject to greater constraint conditions. These types of stresses have a detrimental effect on operational performance, by reducing the fatigue life of the cast component as a result of microcrack generation considered by Ahn et al. [11]. Therefore, to control and mitigate the formation of critical RS during repair welding, the welding parameters must be optimized. Thus, it is crucial to have a comprehensive understanding of the origin, and the temporal and spatial distribution of RS to effectively predict and control them.

It is challenging to acquire the complete distribution of welding RS in thick cast iron components by experimental methods due to the high cost and effort. Therefore, RS of welded structures are more efficiently obtained using computational approaches. Chen et al. [12] simulated the effect of the groove type in dissimilar plates considering the thermomechanical-mechanical coupling behavior and solid-state phase transformation effect, hardening and heat source conditions. They demonstrated good agreement between the numerical and experimental data. A thermomechanical model [35] solves a coupled system of equations representing the coupled thermal and mechanical responses of a process. Numerous studies on the thermomechanical analysis of the welding process have been carried out. Deng and Murakawa [13] compared the numerically simulated results of the temperature field and the welding RS field for stainless steel pipes with experimental findings and developed axisymmetric 2D and 3D models. Results indicate that a 2D axisymmetric model can accurately predict the temperature field and RS field in stainless steel pipe, except for the endpoints of the welding. Feli et al. [14] evaluated the influence of welding sequence on the distribution of RS. They predicted that the welding sequence could well have a more substantial influence on the start and end points than other locations.

Previous research focused primarily on the characteristics of the RS distribution induced by welding. However, the intensity constraint and thermal effect during repair welding differs from those during the common welding process, which considerably affects the magnitude and distribution of RS. Branza et al. [15] experimentally and numerically developed an automated TIG technique to optimize parameters in repair welding of heat-resistant alloys. Ebrahimnia et al. [16] carried out controlled cooling conditions in order to simulate the repair of large forming dies, and powder welding with nickel base self-fluxing alloys. They demonstrated that the compatibility between thermal coefficients of expansion of the weld metal and the base metal can be a major factor in the prevention of cracking. The fatigue strength characteristics of repair welding used in the maintenance of railway vehicles were investigated by Seo et al. [17]. They proved that in order to increase the fatigue life in repair welding, it is recommended to minimize the repair welding width and use a reverse welding pass in the welding direction. J. Chen et al. [18] investigated improved laser welding methods, to cope with the stresses during laser repair welding. Attalla et al. [19] studied the longitudinal and transverse RS for a total of three dissimilar welded joints repaired by using three different nickel-based welding electrodes. The effect of welding wire composition on the repair welds of sand-cast Mg-Gd-Y alloy in view of the mechanical properties and the microstructure in the TIG repair welding was studied by Hou et al. [20]. Z. Chen et al. [21] studied the RS of the thick wall pipeline. They determined the most critical parameters affecting the through-thickness RS distribution was the minimum number of weld layers, heat input, and...
geometrical characteristics.

Despite the fact that intensive research projects on welding have been conducted, there have been relatively few studies on repair welding, which have focused on the evaluation of residual stresses. The majority of these investigations have been carried out on steel, while cast iron has not yet been the subject of any detailed research. Large scale constructions such as wind turbines components, rely primarily on the cast iron material which are not economical to reproduce if they include defects such as porosity or cracks. In order to prevent the development of new defects in the structure during repair welding, it is crucial to correctly understand the behavior of cast iron, especially due to its lower toughness than steel and higher crack proneness. In this work, a numerical approach has been developed to calculate the transient thermal and residual stresses during the repair welding process. The relationship between the maximum principal stresses and directions and the locations of cracks in repair welded specimens has been addressed, and the results have been compared with the experimental observations.

The paper is organized as follows. First, the experimental study is presented. Then, the theory of the finite element model is described. Validation of the thermo-mechanical simulation will be presented, afterward. The thermal and mechanical results calculated by the thermo-mechanical model are validated with the experimental results using the available experimental data. In the next section, a discussion of multi-pass cast iron repair welding simulation follows. Finally, the developed thermo-mechanical model will be used to predict the directions and locations of cracks in the repair welded specimens.

2. Experimental investigation

GJS-500-14 ductile cast iron plates measuring $500 \times 100 \times 50$ mm were considered as the base metal. The experimental study is performed on two samples whose conditions are nominally identical (Fig. 1). The only difference between these two samples is the welding method: The first sample was produced by manual welding (sample 1) while the second one involved an automated welding process (sample 2). On the base metal plate, a $145$ mm long U-shaped groove was milled to a $9$ mm depth and a $36$ mm width to resemble repair circumstances to fill up the groove using several welding passes (Fig. 2).

The plate was preheated to $200^\circ$ C using a conventional electrical furnace prior to welding, and this temperature is consequently stipulated as the inter-pass temperature throughout the welding operation.

Two samples were welded in 10 passes (Fig. 1) using a Metal Cored Arc Welding (MCW) technique, the parameters of which are provided in Table 1. For characterization of the weld beads shape, weld geometry as well as the microstructure, standard metallography procedures including cutting, grinding, polishing, and etching with Nital 2 % were performed on a transverse cross-section of the weldments.

3. Finite element model

Using the finite element approach, the temperature fields and the evolution of the residual stresses are examined. The thermo-elastic-plastic behavior of the weldment is modeled utilizing a sequentially coupled formulation because the mechanical work accomplished is insignificant when compared to the heat energy from the welding arc. To acquire temperature history, the heat transfer problem is addressed independently of the stress problem. Meanwhile, the formulation considers the contributions of the transient temperature field to the stress analysis via thermal expansion and temperature-dependent thermo-physical and mechanical properties. The solution consists of two steps. In the first step, the thermal model computes the temperature distribution and history. Then, in the second step, the temperature history is applied in the subsequent mechanical analysis as a thermal load (Fig. 3).

All analyses in this work are conducted using ABAQUS code [24].

3.1. Heat source model and thermal analysis

Modeling the heat source is the most challenging part of the thermal analysis of the welding process. For the simulation of various welding processes, various heat source models, such as the double ellipsoid model and the cone-shaped volumetric heat source with Gaussian distribution, are typically used [25]. Nonetheless, the designated model is entirely dependent on the size of the weld zone and the temperature field in the HAZ and Fusion Zone (FZ). This study assessed the volumetric heat flow distribution by adapting Goldak’s double-ellipsoid model to the moving heat source. This model includes two distinct ellipses, one in the front and the other in the back quadrants of the heat source, as depicted in Fig. 4. The Goldak equations have been presented as following [25]:
Welding parameters for samples 1 and 2 [22, 23].

<table>
<thead>
<tr>
<th>Pass No.</th>
<th>Current (A)</th>
<th>Voltage (V)</th>
<th>Speed (mm/ min)</th>
<th>Heat input (RJ/ mm)</th>
<th>Deposition efficiency</th>
<th>Arc efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>234</td>
<td>27</td>
<td>413</td>
<td>0.92</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>2</td>
<td>234</td>
<td>27</td>
<td>448</td>
<td>0.85</td>
<td>0.99</td>
<td>0.9</td>
</tr>
<tr>
<td>3</td>
<td>229</td>
<td>27</td>
<td>405</td>
<td>0.92</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>4</td>
<td>233</td>
<td>27</td>
<td>413</td>
<td>0.92</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>5</td>
<td>235</td>
<td>27</td>
<td>375</td>
<td>1.02</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>6</td>
<td>228</td>
<td>27</td>
<td>324</td>
<td>1.14</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>7</td>
<td>227</td>
<td>27</td>
<td>378</td>
<td>0.97</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>8</td>
<td>228</td>
<td>27</td>
<td>257</td>
<td>1.44</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>9</td>
<td>225</td>
<td>27</td>
<td>281</td>
<td>1.30</td>
<td>0.98</td>
<td>0.9</td>
</tr>
<tr>
<td>10</td>
<td>227</td>
<td>27</td>
<td>291</td>
<td>1.26</td>
<td>0.98</td>
<td>0.9</td>
</tr>
</tbody>
</table>

The nonlinear isotropic Fourier heat flux constitutive equation has been implemented:

\[
\dot{q}_{\text{cond}} = -k^\text{eff} \nabla T
\]  

Convective and radiative heat dissipation were properly considered in the FE model due to the heat exchange between the weldments and the environment. Both Stefan-law Boltzmann’s and Newton’s law (Eq. (4)) can be used to explain heat loss

\[
\begin{align*}
q_{\text{conv}} &= -h_c(T_w - T_b) \\
q_{\text{rad}} &= -\sigma_0(T_w)^4 - (T_b + 273.15)^4
\end{align*}
\]  

For the purpose of simulating the combined thermal boundary condition, a user subroutine was developed. Eq. (5) provides the overall temperature-dependent heat transfer coefficient [14]. The latent heat of fusion is also employed to describe the thermal effects produced by the weld pool solidification [14].

\[
h_{\text{eq}} = \begin{cases} 
0.068T \ (W/m^2) & 0 \leq T \leq 500 \ ^\circ C \\
0.231T - 82.1 \ (W/m^2) & T \geq 500 \ ^\circ C
\end{cases}
\]  

3.2. Mechanical analysis

Except for the element type and boundary conditions, identical finite element models are applied in both the thermal and the mechanical evaluations. The temperature history obtained from the thermal analysis is provided as the input data for the investigations. Based on the following governing partial differential equation of force equilibrium, the mechanical model is developed.

\[
\nabla \sigma = 0
\]  

In this study, the effect of rapid cooling was neglected which leads to a small proportion of non-equilibrium phases generated during the welding. Since the phase transformation has a negligible effect on the RS and deformation [26], the total strain can be divided into the following components.

\[
\varepsilon_{\text{total}} = \varepsilon^p + \varepsilon^d + \varepsilon^n
\]  

Using isotropic Hooke’s law, temperature-dependent Young’s modulus, and Poisson’s ratio, elastic strain is modeled. The thermal strain is modeled using a temperature-dependent thermal expansion coefficient. To consider the plastic behavior temperature-dependent mechanical properties is implemented. The yield criterion is the von Mises yield surface. A linear kinematic hardening rule is chosen to account for the Bauschinger effect during multiple thermal loading.
3. Kinematic hardening is considered an important feature since the material experiences both loading and unloading throughout the welding [13, 14, 27].

3.3. Finite element discretization

Based on the theory of virtual work, it is possible to discretize the fundamental differential equilibrium Eqs. (2) and (6), hence addressing the physical problem on a specific domain using finite elements. For the discretized structure, the unknowns for the heat transfer and mechanical analyses, respectively, are the nodal temperatures and nodal displacements. Using the shape functions of the element, point-wise fields are interpolation based on the nodal values. The principle of virtual temperatures has been applied to discretize the heat flux balance in terms of the nodal temperatures for the heat transfer problem. This results in an overall equilibrium of heat fluxes by the summation of element integrals. The definition of the temperature gradient within an element is [28].

\[ T^* = B_{th}^e T \]  

Matrix \( B_{th}^e \) is derived by differentiating the shape functions appropriately. Fourier's law relating fluxes to temperatures in terms of the thermal properties.

Fig. 3. Algorithm of the sequentially coupled solution for a thermomechanical analysis. Obtaining the nodal temperature history from the thermal analysis to set as a predefined field for the mechanical evaluation.

Fig. 4. Double ellipsoid heat source model presented by Goldak et al. [25] consisting of two separated parts: the back of the heating source (blue) and the front section (purple). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)
formation of the thermal finite element equation will be as follows:

\[ D^{th} \ddot{T} + K^{th} T = R^{th}_e + R^{th}_S \]  \hspace{1cm} (9)

The stresses in a finite element are connected to the element strains in terms of the constitutive law that describes the material behavior of the structure under consideration, in matrix form:

\[ \sigma = C^e \varepsilon \]  \hspace{1cm} (11)

\[ K^e u = R^e + R^p \]  \hspace{1cm} (12)

4. Results and discussion

4.1. Validation of the simulation results against the experimental data

In this section of the study, a comparison has been made between the results predicted by the current numerical model and the available experimental data. A 60-degree V-groove butt weld was performed using submerged arc welding to join two rectangular plates with dimensions of 240 mm × 480 mm × 10 mm (Fig. 5a) and mesh models (Fig. 5b and c). The welding conditions are listed in Table 2. The thermal and mechanical solutions obtained using 3D and 2D models of the butt weld joint have been validated against experimental data. The material used for validating the numerical model is ST 37 steel which has temperature-dependent thermal and mechanical properties from room to melting temperature.

Thermocouples were placed at different distances with respect to the weld line to measure the temperature history. Residual stresses were measured using neutron scattering method [29].

The elements for the 3D thermal and mechanical models were 8-node brick elements Abaqus type DC3D8 and C3D8, respectively. 4-node linear elements DC2D4 and CPE4 were used for the 2D thermal and mechanical models, respectively. The size and distribution of the elements have been shown in Fig. 5. Convective and radiative heat transfer boundary conditions have been considered for the surfaces in contact with the environment. Free boundary conditions (in accordance with the welding experiments) were considered for the mechanical model. In order to prevent rigid body motion of the weldment, the rigid body degrees of freedom were constrained.

The heat source movement, the development of FZ, and HAZ have been simulated using the birth and death finite element technique and are depicted schematically in Fig. 6.

Fig. 7a and b show the comparison between the numerical and experimental temperature histories of two points at distances of 14 mm and 42 mm perpendicular to the weld centerline, respectively. Assessing the results confirms that both the 2D and 3D simulations predict the thermal behavior of the welding process reasonably well. The difference between the peak temperature of the numerical and experimental results can be explained by the discrepancy between the approximated radiative heat transfer coefficient in the model and the actual value.

In the current step, the simulated RS are compared with the experimentally measured ones. The distribution of the axial and the transverse RS in the 2D and 3D simulations of the welding process are indicated in Fig. 8b, c, e and f. As depicted in Fig. 8, the directions of the numerical stress results are consistent with the theory (Fig. 8a and Fig. 8d) [30].

In order to compare the results more precisely, the transverse and axial RS have been plotted versus the distance from the weld centerline. Fig. 9 illustrates the comparison of the 2D and 3D simulations with the experimental measurements [29]. Comparison of the predictions with linear hardening and a perfectly plastic assumption showed little effect.
Fig. 6. The gradual formation of the welding pass, FZ, HAZ and temperature distribution in 2D and 3D simulations from (a) to (d) for the butt weld joint.

Fig. 7. Comparison between the numerical nodal temperature history and the experimental measurements [29] for two points at (a) 14 mm and (b) 42 mm from the center line in the butt weld joint.
Fig. 8. (a) Distributions of the transverse RS according to the theory [30], (b) 3D and (c) 2D numerical simulations for the butt weld joint. (d) The distributions of the axial RS according to the theory [30], (e) 3D and (f) 2D numerical simulations for the butt weld joint. Compared to the 3D model, the 2D numerical simulation of the transverse (c) and axial (f) stresses will significantly reduce computational time.
of the hardening on the residual stress response for the welded plate, therefore a linear elastic perfectly plastic assumption was considered in the weld model.

Evaluation of the results (see Fig. 9) signifies that the RS predicted by the 2D model agrees well with those predicted by the 3D model as well as the experimental measurements. Due to easier implementation and much shorter computational time of the 2D model, it is preferred over the 3D model especially for sensitivity analysis and optimization applications. Therefore, after the validation of the 2D model for both thermal and mechanical simulations in this section, it is employed for repair welding in the following section.

4.2. Cast iron repair welding

This section presents temperature histories and thermal stresses calculated by thermo-mechanical modeling of the cast iron repair welding. The relation between the numerical results and experimental observations of the cracks in the repair weld has been analyzed. The numerical model (Model 2) has been developed according to the process parameters and geometry of the repair weld in the experiments. The material properties (Figs. 10 and 11) for the considered cast iron were extracted from JMatPro [31] which is in good agreement with the available reference Li et al. [32]. Fig. 12a depicts the mesh quality for
the model, and Fig. 12b shows the temperature distribution and generation of the weld passes in the numerical model as well as in the weld experiment described in Section 2. As shown in the figure, the weld passes generated in the model match very well with those generated in the experiment.

The value of the latent heat was 220KJ/kg. \( T_S \) and \( T_L \) values are 1120°C and 1160°C, respectively [33].

Three regions of the model (A, B and C) have been selected to investigate the temperature history during the repair welding in more detail. Fig. 13 depicts the temperature history of these areas throughout the development of the weld passes and their cooling. The sudden jumps in the temperatures happen when new weld passes are added to the repair area.

The following section describes a detailed examination of the developed thermal stresses during the repair welding and its correlation with the observed experimental outcomes. In order to investigate the correlation between the experimental observations and numerical results, transient thermal stresses and RS have been visualized in four critical areas as seen in Fig. 14.

Fig. 14 depicts the distribution of the axial RS in the material as well as the evolution of thermal stresses during the welding until it cools down to ambient temperature. The calculated thermal stresses (see Fig. 14b), indicate that areas 1, 2, and 4 experience the highest thermal stresses and thus have a high risk of fracture. The distribution of transverse transient thermal and RS shown in Fig. 15, again highlights these areas as critical places with the highest value of the transverse stresses throughout the welding process.

Micrographs of cross sections of the sample 1 and sample 2 repair welds (c.f. Section 2) have been demonstrated in Fig. 16. As seen in the figure, the directions and locations of the cracks observed in certain areas of both samples are close to being identical.

The maximum principal stress theory [34] is one of the approaches which can be employed to analyze the locations and directions of the crack. Based on this criterion, potential fractures occur at places with the highest principal stresses. Furthermore, if a crack develops in these areas, it tends to propagate perpendicular to the principal stress direction.

Fig. 17 illustrates visually how the location and direction of the cracks developed in the repair weld agree well with the maximum principal stress criterion. As shown in the figure, the numerical model has predicted the regions with the highest principal stresses (i.e., regions 1, 2 and 4) and thus a higher probability of crack formation. This prediction agrees well with the locations of the cracks observed in both experimental weld samples. Furthermore, as seen in Fig. 17, the directions of the developed cracks are consistent with the directions predicted by the numerical model which are perpendicular to the principal stresses. These results show that the numerical model developed in this study is capable of accurately predicting the location and direction of the cracks generated during the repair welding process.

5. Conclusion

In this study, a thermomechanical model of repair welding in cast iron components has been developed to predict weld-induced thermal and RS. The results of the thermo-mechanical model were validated with experimental data. The thermo-mechanical model was used to predict the location and direction of cracks in repair welds and the results were compared with experimental observations. Following are the major points which have been concluded based on the results of this study:

- Comparing the results of 2D and 3D thermal and mechanical analysis with the experimental results shows a good agreement. Since the weld repair can be simulated in 2D with reasonable accuracy, this approach can be used for sensitivity analysis and optimization of the repair weld, whereas a 3D simulation involves several challenges such as high computational times.
- The regions of high potential for crack development in the repair welded structure have been predicted using the developed thermo-
Fig. 12. (a): Mesh model, (b): Numerical prediction of FZ and HAZ in the thermal model (Model 2) according to the experiment (Sample 1).
Fig. 13. Tracking the temperature history of the three selected points during the repair welding development to investigate the thermal behavior of the material in critical areas of the repair weld.
Fig. 14. Distribution of the axial RS (a) and average axial thermal stresses (b) in the four specified regions of the repair weld.
Fig. 15. Distribution of the transverse RS (a) and average transverse transient thermal stresses (b) in the four specified regions of the repair weld.
Fig. 16. Locations of the cracks in sample 1 (a) and sample 2 (b).
mechanical model and maximum principal stress criterion. The critical areas predicted through the numerical model are confirmed to be the regions where fractures were identified in the experimental samples.

- Besides predicting crack locations, the application of the maximum principal stress criterion has resulted in a relatively accurate prediction of the crack direction in the repair welded specimen. The close agreement between the predicted crack directions in the repair welded specimen and the observed experimental findings proves the reliability of the thermo-mechanical model developed in this study.

Future work will address the optimization of the repair welding parameters in cast iron components for minimizing the amount of induced thermal and RS and achieving better quality repair welds.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

The presented results are derived from the WeldCast project, which was funded by the Innovation Fund Denmark grant number 0224-00116A, the support is gratefully acknowledged. The authors would like to thank Rian Holdstock and Peter Tommy Nielsen from FORCE Technology for sharing the experimental data of the repair weld. Furthermore, the authors appreciate the practical and scientific advice provided by Anders Dahl Kragh from Vestas and Per Hessellund Lauritsen from Siemens Gamesa.

References


