

Numerical Methods For Welding Simulation – The Next Technical Step

D.P. Bray¹, N.A. Leggatt¹, R.J. Dennis¹, M.C. Smith² and P.J. Bouchard²

1) Frazer-Nash Consultancy Limited, 1 Trinity Street, College Green, Bristol, BS1 5TE, UK

2) British Energy Generation Limited, Barnett Way, Barnwood, Gloucester, GL4 3RS, UK

Abstract: Numerical methods using Abaqus have been established to simulate welding processes, often based around the use of methods which represent the welding process in a simplified manner. Simplified methods include simultaneous deposition of weld beads and bead lumping where entire weld beads are grouped together and deposited at once. These approaches are widely accepted, however the requirement for simplified methods often results in compromises to the solution accuracy. In many cases this compromise in accuracy is acceptable providing it is well understood, however there are frequently cases where such simplifications are unacceptable and improved representation of the welding process is required. In practice this generally implies the requirement for a full moving heat source simulation. The transition from simplified simulation methods to the next technical step, full moving heat source simulations, is now possible within Abaqus for a wide variety of scenarios as will be demonstrated in this paper.

This paper presents two specific cases, a 3 pass slot weld and a multipass repair weld, where full moving heat source simulations have been considered necessary. For each of these cases the reasons why moving heat source methods are necessary and the benefits that this more demanding simulation technique offers are described. Furthermore the predicted residual stress results are compared with residual stress measurements using a variety of measurement techniques.

The work provides an extremely useful insight into how moving heat source methods are now considered a practical analysis method for a wide variety of real world problems. Of further consideration is the fact that in the 2 years since the work reported in this paper was undertaken computing performance would have at least doubled

Keywords: Welding, Residual Stress

1. Introduction

The purpose of this paper is to present two case studies where moving heat source analyses have been conducted to illustrate that the moving heat source method can be applied to complex and large welding analyses. A three pass slot weld will be investigated first with comparisons made to residual stress measurements using the contour measurement technique which is able to provide a full map of residual stress. This will highlight any specific features of the weld that would only be determined by using a moving heat source method.

The second case study will consider a multi-pass repair weld where the final capping passes of the weld are modelled using a moving heat source. The results will be compared to an analysis conducted using only a block dumped methodology.

2. Case Study 1 – 3 Pass Slot Weld

The purpose of this case study was to predict the residual stress distribution in AISI Type 316L Stainless Steel (SS) test specimens containing a three-pass slot weld. The slot weld specimen was manufactured from 50mm thick AISI Type 316L austenitic stainless steel.

Initially blanks were cut from 50mm thick parent plate to the final rectangular size of 300mm x 200mm. The 50mm thick parent plate was reduced to the specimen thickness of 25mm by first removing 10mm from both the top and bottom surfaces by milling. Stress relief was then carried out by solution heat treatment at 1050°C for one hour followed by air cooling. Subsequently, a further 2.5mm was removed, by milling, from the specimen top and bottom surfaces. Finally, 100mm long weld slots were machined in centre of the top surface of each plate to a depth of 10mm and hand-filed to obtain the 45° run out at the slot ends (see Figure 1). The final welded plate dimensions were 200mm wide by 300mm in length (parallel to the weld) and 25mm in depth as shown in Figure 1.

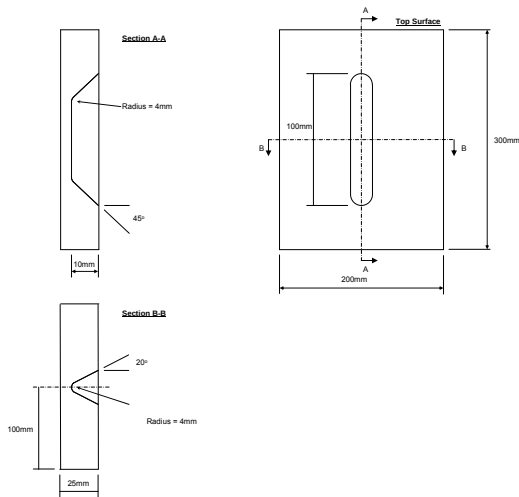


Figure 1: Slot Weld – Specimen Geometry

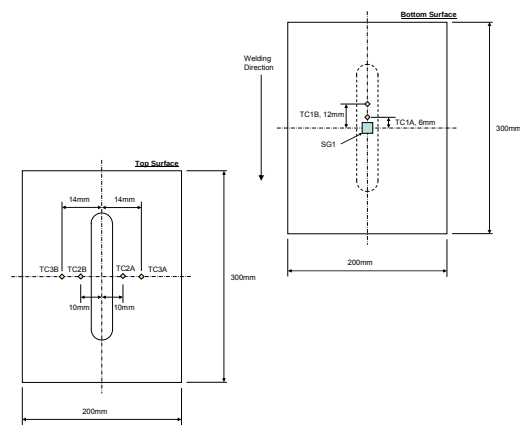


Figure 2: Slot Weld – Instrumentation

2.1 Welding Inputs

All welding was carried out using the Manual Metal Arc process. The welding electrodes were Babcock S type, batch number P0911, and were 5mm diameter by 350mm in length. The target heat input was 1.8kJ/mm and the maximum interpass temperature was set to be 80°C. The interpass temperature was monitored on a data logger. The welding parameters used are shown in Table 1.

Each slot weld specimen was welded in three layers with one weld bead in each layer. Each bead was deposited as a single stringer bead. All three beads were laid in the same direction.

Some of the slot weld specimens were instrumented with thermocouples and strain gauges. A diagram showing the instrumentation applied is shown in Figure 2.

2.2 Finite Element Model

It was decided that the specimen denoted specimen 1.2-3 would be specifically modelled since this specimen had been sectioned for contour residual stress measurements. A surface profile of specimen 1.2-3 was generated using laser scanning. A three dimensional model was constructed to represent the base-plate, machined groove, and the scanned profile of specimen 1.2-3 after welding. The voltage, current and weld time are specific to specimen 1.2-3. The inputs are detailed in Table 1.

The three dimensional FE model generated is shown in Figure 3. It is a one-half model, which therefore assumes that the weld geometry is symmetrical about a line running parallel to the weld at its centre. Note that the cross sectional areas of each pass have been determined from the weld macro areas, but adjusted to take account of the difference in heat inputs between the specimen 1.2-3 heat inputs and those of the specific groove weld plates that were the source of the macrographs. The model contains 31,490 quadratic 20 noded reduced integration hybrid elements for the mechanical analysis (C3D20RH) and fully integrated 20 noded heat transfer elements for the thermal analysis (DC3D20). The same mesh is used for both the thermal and mechanical analyses to allow temperatures to be read directly from one analysis to another using the Abaqus .fil file.

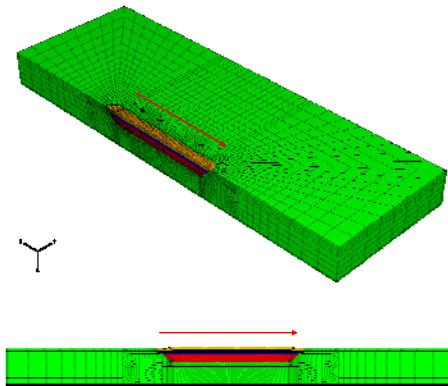


Table 1: Slot Weld – Weld Input Parameters

Specimen Reference	Run No.	Average Volts	Average Current (Amps)	Power (kW)	Arc Time (s)	Torch Travel ¹ (mm)	Weld Speed ² (mm/s)	Heat Input ³ (kJ/mm)
1.2-3	1	24.2	178	4.31	40.3	80.0	1.99	2.17
1.2-3	2	22.9	181	4.15	34.6	84.0	2.43	1.71
1.2-3	3	22.6	178	4.02	43.9	93.0	2.12	1.90

Figure 3: Slot Weld – Finite Element Model

The weld geometry was modelled taking account of the laser scanning data for the top surface. However, the height of the bead above the parent plate was idealised as being constant over the weld length at 0.8mm, approximately the average bead height from measurement data.

Using newly developed analysis techniques it is now unnecessary to explicitly model the weld to parent material boundary within the finite element mesh. Rather, to account for the predicted fusion boundary and the assignment of suitable material properties, either weld metal or parent, a ‘Dynamic Fusion Boundary’ (DFB) technique is adopted that allows each gauss point in the model to have materials assigned based upon an Abaqus Field Variable (FV) dependence. The FV’s are dynamically updated during the transient thermal analysis to represent the spread of the molten pool as the weld is deposited.

Note that analyses were conducted using small strain displacement theory.

2.3 Analysis Methodology

The analysis methodology is described in this section. The general analysis approach represents the application of a mature welding simulation methodology developed over many years, primarily in

support of British Energy. Here that methodology is applied to the analysis of a three-pass slot weld using a moving heat source approach.

2.4 Heat Source Modelling Tool

The analysis methods make use of a dedicated Heat Source Modelling Tool (HSMT), [1]. The purpose of the HSMT is to automate and improve the thermal analysis aspect of welding simulations. The mechanical analysis is always the most computationally demanding. However, there is generally greater uncertainty in the thermal solution, and several iterations may be required before a satisfactory solution is obtained. The HSMT facilitates rapid parametric studies and allows an optimised thermal solution to be obtained efficiently.

Once a satisfactory thermal solution is achieved the thermal parameters (heat source size, torch efficiency etc.) are transferred into Abaqus [2] where sequential thermal and mechanical analyses are performed.

2.5 Abaqus Thermal Analyses

It is the objective of this work to study the three-dimensional stress state associated with both the geometry of the slot weld and the influence of superimposed weld pass start and stop thermal transients on local stresses. A moving heat source thermal methodology was adopted that has been used in earlier simulations [3] and is described here also for clarity.

2.5.1 Heat Source Definition

The volumetric heat source is defined within the Abaqus user-defined DFLUX subroutine. This subroutine is provided so that Abaqus users can define complex heat fluxes, in this case dependent on spatial position and time. The heat input for the moving heat source is represented by an ellipsoidal distribution defined by the non-dimensional effective torch radius, r_e (that is the distance from the torch centre non-dimensionalised with respect to the weld pool dimensions):

$$r_e = \sqrt{\left(\frac{y - y_p}{d_t}\right)^2 + \left(\frac{x - x_p}{r_t}\right)^2 + \left(\frac{z - z_p}{r_t}\right)^2} \quad (1)$$

where (x, y, z) are Cartesian co-ordinates centred on the surface of the specimen at the torch start position, with z negative in the torch advance direction and y negative outwards and normal to the specimen top surface and (x_p, y_p, z_p) is the current torch position. Note that r_t is the torch radius and d_t is the torch depth.

The spatial volumetric heat input, q , is assigned to reduce from the torch position by a Gaussian function based on the effective torch radius:

$$q = K_q e^{-r_e^2} \quad (2)$$

where K_q is defined so that the total heat flux deposited on the plate upon integration in the finite element model equals the effective power from the torch.

2.5.2 Weld Deposition

One issue in welding simulations is how to handle the addition of hot weld material into the model. One approach is to add a group of finite elements to the original mesh which has a volume corresponding to the feed of filler metal to the weld torch. In a moving torch calculation such as this, some method of dealing with the progressive addition of material as the weld pass progresses has to be used. In the HSMT, it is assumed that the whole bead representing a weld pass is added at the commencement of the pass. As far as the thermal calculation is concerned, this is a good approximation. Unless the weld Peclet number is very low (i.e. $Pew < 0.1$, [1]) the movement of the torch will dominate over heat conduction in the region away from the torch, and therefore the presence or not of the conducting bead material does not matter. In the region close to the torch, the hot material will in practice radiate energy ahead of the torch in the weld-prep groove, so conduction ahead of the torch in the bead material is not unrealistic. Thus adding the entire bead at the start of the pass should be reasonable in most cases, and is certainly a convenient way of doing the thermal calculations. Nevertheless in the Abaqus thermal analysis thermal isolation ahead of the heat source is enforced by reducing the conductivity of the bead material ahead of the heat source. The transition from 'behind' to 'ahead' of the torch is performed smoothly based on a simple Gaussian scaling function. This Gaussian function has been set to act over a distance of a single effective torch radius.

Thermal isolation ahead of the torch is achieved in Abaqus using field variable dependent material properties defined in the Abaqus input deck. Ahead of the torch the weld bead is defined as inactive with a field variable of 2. For all material points with a field variable of 2 the conductivity is reduced by 2 orders of magnitude to ensure adequate isolation of the inactive weld bead. For all weld material points with a field variable of 1 the material properties are not adjusted and depend solely on temperature. Where the field variable is between 2 and 1 (i.e. just ahead of the torch in the Gaussian scaling region) the properties are linearly interpolated to ensure a smooth variation. Note that when pass 1 is being deposited, passes 2 and 3 are isolated by being entirely removed from the model, and hence solution, using the *MODEL CHANGE option in Abaqus (Reference 2). Following the deposition of pass 1 and on reaching the pass 1 interpass temperature, pass 2 is added into the model solution. Pass 2 deposition subsequently occurs before the addition of pass 3.

Field variables are calculated within the Abaqus user-defined UFIELD subroutine and updated at each nodal position. This subroutine is provided so that Abaqus users can define complex field variable distributions, in this case dependent on the torch position and updated as the torch.

2.5.3 Thermal Boundary Conditions

Thermal boundary conditions consist of the application of convective and radiative heat transfer to the surfaces of the model, including the weld bead.

Convective heat transfer coefficients are applied to the top surface of the plate as a function of the metal surface temperature ranging from $4.1\text{W/m}^2\text{K}$ at 20°C to $14.6\text{W/m}^2\text{K}$ at 2500°C . The sides and bottom of the plate reach much lower temperatures than the top surface and so a fixed convective heat transfer coefficient was applied, $3.5\text{W/m}^2\text{K}$ for the bottom of the plate and $7\text{W/m}^2\text{K}$ for the sides.

Radiative heat transfer is based on radiation to a large enclosure at room temperature (i.e. 20°C), with an assumed emissivity of 0.4 for stainless steel.

During the weld deposition steps only, external convective heat transfer on the top surface of the plate is suppressed local to the torch. This accounts for the fact that the air temperature around the torch will not be at ambient conditions as a consequence of the heat from the torch. External convective heat transfer therefore cannot occur around the torch since a local sink temperature of 20°C is unrealistic.

The following suppression factor was applied, consistent with earlier analyses of the bead on plate specimen (Reference 4):

$$\begin{cases} S_{htc} = 1 - e^{-0.25r_{xz}^2} \\ r_{xz}^2 = \left(\frac{x - x_p}{r_t} \right)^2 + \left(\frac{z - z_p}{r_t} \right)^2 \end{cases} \quad (3)$$

Following the completion of the weld deposition steps, convective heat transfer to ambient conditions is reinstated across the entire top surface of the plate during the cool down phase.

2.6 Abaqus Mechanical Analyses

The mechanical modelling of the weld simulations described here is carried out using Abaqus. The only loads imposed on the model are transient thermal loads that are defined via the nodal temperature data written by the Abaqus thermal analysis (see Section 2.5).

2.6.1 Plastic Strain Annealing

The temperature transient loadings associated with simulating the welding process introduce large non-linear strains into the welded structure. These strains are treated as plastic strain in present simulation methods. However, in real stainless steel materials creep deformation begins to become important above 450°C and dominates behaviour above around 850°C. In addition, above 700°C historically accumulated plastic strains start to be annealed (annihilated) as recovery and re-crystallisation temperatures are approached and when temperatures reach the material melting point (1400°C), all historically accumulated strains are lost. At present, no constitutive models handling this complex behaviour are available. Instead the annealing function in Abaqus has been used to eliminate historical plastic strain in the analysis, in a very crude fashion, when the temperature exceeds 850°C.

2.6.2 Weld Metal Deposition

The weld metal ahead of the heat source has to be mechanically isolated from the rest of the model. This is considered to be especially important for the high constraint slot weld geometry being analysed. The mechanical isolation method used in this study follows the method developed for the Abaqus thermal analysis, utilising user-defined field variables. In order to achieve an adequate level of mechanical isolation ahead of the heat source, the inactive weld bead ahead of the torch is assigned molten plastic material properties, thereby exerting minimal shear on the parent material. This was implemented in Abaqus by assigning molten plastic material properties to all material points which have been given a field variable value of 2. The transition from 'behind' to 'ahead' of the torch was performed smoothly based on a simple Gaussian scaling function as used for the thermal isolation. This Gaussian function was set to act over a distance of a single effective torch radius.

2.6.3 Mechanical Boundary Conditions

The slot weld model is assumed to be unconstrained during welding. Free thermal expansion in all principal directions is allowed with reference to the symmetry plane(s). The constraints applied in the Abaqus models are:

- All nodes on the geometric symmetry plane (parallel to the welding direction) have x (1) symmetry conditions applied

- One node (on the bottom surface of the specimen, on the symmetry plane, at the maximum z (3) position constrained in the y (2) (normal) and z (3) (longitudinal) directions.
- One node (on the bottom surface of the plate on the bead symmetry plane at the minimum z (3) position) constrained in the y (2) (normal) direction.

These boundary conditions remove rigid body translational and rotational motions without introducing a state of stress.

2.7 Material Properties

The parent plate is manufactured from AISI Type 316L stainless steel, while the welding electrodes are type Babcock S. In the analysis presented here the so-called ‘Dynamic Fusion Boundary’ (DFB) deposition technique was applied. This approach utilises a single material definition for both parent and weld metal with ‘Field Variable’ (FV) dependence to determine whether weld material or parent material is appropriate. Both parent and weld metal properties are defined under a single material definition with a FV of 0 signifying parent and a FV of 1 signifying weld metal. To include the thermal isolation a FV of 2 is assigned to the isolated weld bead ahead of the torch. During the welding phase the material temperature is monitored to establish which regions of the model have become molten (i.e. exceeded the melting temperature, 1400°C in this case). Should melting of parent metal occur, i.e. the temperature has exceeded 1400°C, then on cooling, weld material properties are assigned. The following simple rules are utilised in this process:

- Weld metal deposited always has a FV of 1, and is therefore always weld metal with appropriate properties.
- If the weld metal is ahead of the torch then it has a FV of 2 which represents isolated weld metal. As the torch moves along the weld the material is activated by switching the FV to 1 as it approaches the torch.
- Parent metal a significant distance away from the weld always has a FV of 0, and is therefore always parent metal with appropriate properties.
- Parent metal in the HAZ region is monitored. Where the temperature exceeds 1400°C the field variable is changed from 0 to 1 signifying the change from parent to weld metal upon cooling. Note that this process acts as a switch. Each Gauss point is either parent plate or weld metal.

The thermal properties used are summarised in Table 2.

Table 2: Slot Weld – Thermal Material Properties

Temp °C	Weld		Parent and Weld			Parent and Weld (Isotropic)		
	Specific Heat kJ/kg/K	Specific Heat kJ/kg/K	Conductivity W/m/K	Density kg/m ³	Thermal Exp x 10 ⁻⁶ mm/mm/K	Parent Young's Modulus GPa	Weld Young's Modulus GPa	Poisson's Ratio
20	0.488	0.492	14.12	7966	14.56	195.6	168.2	0.294 (weld) & 0.290 (parent)
100	0.502	0.502	15.26		15.39	191.2	164.1	
200	0.520	0.514	16.69		16.21	185.7	158.3	
300	0.537	0.526	18.11		16.86	179.6	152.0	
400	0.555	0.538	19.54		17.37	172.6	144.9	
500	0.572	0.550	20.96		17.78	164.5	136.9	
600	0.589	0.562	22.38		18.12	155.0	127.7	
700	0.589	0.575	23.81		18.43	144.1	117.4	
800	0.589	0.587	25.23		18.72	131.4	105.8	
900	0.589	0.599	26.66		18.99	116.8	92.8	
1000	0.589	0.611	28.08		19.27	100.0	78.3	
1100	0.589	0.623	29.50		19.53	80.0	62.4	
1200	0.589	0.635	30.93		19.79	57.0	44.5	
1300	0.589	0.647	32.35		20.02	30.0	23.4	
1400	0.589	0.659	32.78		20.21	2.0	1.6	

The mechanical analysis used an advanced constitutive material hardening law based upon the Lemaitre-Chaboche formulation provided within Abaqus. Mixed isotropic-kinematic hardening models for both the parent 316 SS and weld materials were employed based upon monotonic and isothermal cyclic tensile test programmes. This type of hardening law enables realistic predictions of residual stress in materials that rapidly strain harden.

The Lemaitre-Chaboche hardening model implemented within Abaqus is divided into kinematic and isotropic hardening components. The kinematic portion is defined using the relationship (in the absence of field variable or temperature dependence):

$$\dot{\alpha} = C \frac{1}{\sigma_0} (\sigma - \alpha) \dot{\varepsilon}^{pl} - \gamma \alpha \dot{\varepsilon}^{pl} \quad (4)$$

where:

$\dot{\alpha}$	=	rate of change in kinematic backstress
σ_0	=	is the size of the yield surface (MPa)
$\dot{\varepsilon}^{pl}$	=	equivalent plastic strain rate
C, γ	=	fitting coefficients (MPa and no units respectively)

C is the initial kinematic hardening modulus, and γ determines the rate at which the hardening modulus decreases with increasing plastic deformation.

The isotropic portion (i.e. the size of the yield surface, σ^0) is defined via a simple exponential law:

$$\sigma^0 = \sigma|_0 + Q_\infty (1 - e^{-b\bar{\varepsilon}^{pl}}) \quad (5)$$

Where Q_∞ and b are material parameters, $\bar{\varepsilon}^{pl}$ is the equivalent plastic strain and $\sigma|_0$ is the size of the yield surface at zero plastic strain (MPa).

A mixed hardening material model, referred to as the “mixed-j” model, was derived from monotonic and isothermal cyclic tensile data for both parent and weld metal by British Energy.

The mechanical properties used are summarised in Table 3 and Table 4 for parent and weld metal respectively.

Table 3: Slot Weld – Tensile Properties - Parent

Parameter	Mixed Hardening Parameters (mixed-j model)											
	20°C	100°C	200°C	275°C	300°C	350°C	400°C	450°C	500°C	550°C	600°C	650°C
C (MPa)	4097.0	3288.3	1676.2	970.0	570.2	369.7	230.2	140.1	90.9	54.0	3.0	0.0
γ	34.0	34.0	34.0	34.0	34.0	34.0	34.0	34.0	34.0	34.0	34.0	34.0
Q_∞ (MPa)	120.0	130.0	130.0	137.0	141.0	144.0	146.0	147.0	148.0	149.0	150.0	150.0
b	6.0	6.0	6.0	6.0	6.0	6.0	6.0	6.0	6.0	6.0	6.0	6.0
0% Proof Stress (MPa)	232.0	188.0	153.0	140.0	137.0	132.0	126.0	121.0	116.0	111.0	106.0	101.0

Table 4: Tensile Properties - Weld

Parameter	Mixed Hardening Parameters (mixed-j model)						
	20°C	200°C	400°C	600°C	850°C	1000°C	1400°C
C (MPa)	18800.0	17241.0	12752.0	12374.0	0.0	0.0	0.0
γ	150.0	150.0	150.0	150.0	150.0	150.0	150.0
Q_∞ (MPa)	69.1	80.5	81.8	99.75	0.0	0.0	0.0
b	53.2	46.1	12.5	37.7	0.25	0.25	0.25
0% Proof Stress (MPa)	280.0	210.0	200.0	175.0	108.0	55.0	4.5

2.8 Results

A comparison of the fusion boundary at weld mid-length with geometrically similar groove weld macrographs (Figure 4) suggests that the typical fusion boundary shape has been captured. The total heat inputs that were used to generate the groove weld specimens, and hence macrographs, differ by as

a maximum of 15% (slot weld being greater) when compared to the specimen 1.2-3 slot weld heat inputs. Furthermore the groove weld showed significant transverse contraction resulting in what appears to be a thinner weld profile, but note that the fusion boundaries are plotted in an undeformed state.

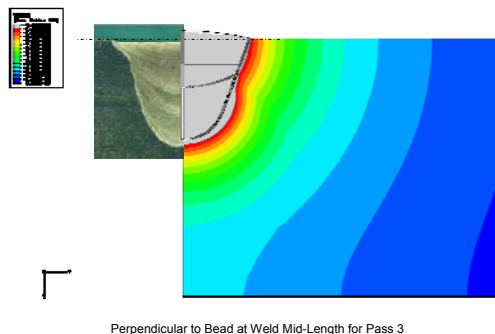


Figure 4: Slot Weld – Comparison of Fusion Boundary at Weld Mid-Length

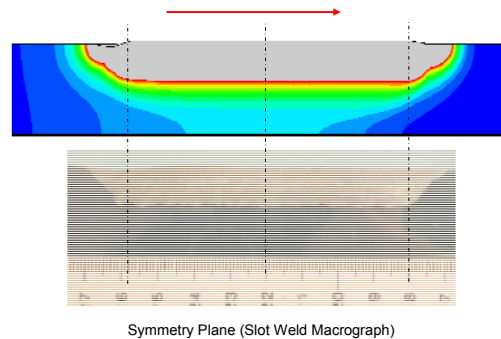


Figure 5: Slot Weld – Comparison of Fusion Boundary on Weld Symmetry Plane

A comparison of the fusion boundary at the slot weld symmetry plane (Figure 5) suggests that the fusion boundary shape and length as shown in the macrograph has been captured adequately.

During the deposition of pass 1 the predicted metal temperatures were recorded and the temperature difference calculated (Figure 6). The temperature difference is the absolute difference in temperature between the start of the deposition step and the temperature at any point during the deposition step. In multipass analyses this removes the effect of the interpass temperature and simplifies the comparison with thermocouple data. For pass 1 the predicted temperature peak for TC2 is approximately 5% less than the average recorded temperatures at that location. Note that TC3a is significantly lower than TC3b, possibly due to a contact problem, and has been disregarded as has TC1a which appears to be non-functional. The recorded thermal response of the specimen, for pass 1, has therefore been captured accurately with an efficiency of 70%. Comparisons with weld passes two and three were also deemed acceptable.

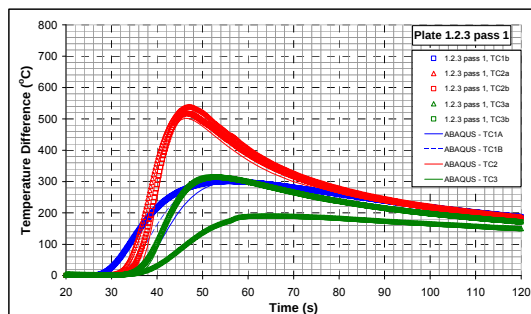


Figure 6: Slot Weld – Comparison of Thermocouple Predictions for Pass 1

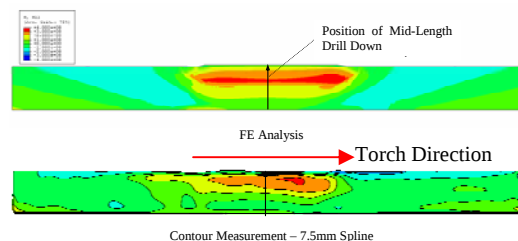


Figure 7: Slot Weld – Comparison of Transverse Stress

In terms of residual stresses, comparisons of transverse stress are presented here. Transverse stress is of interest as this allows a comparison to be made between a contour measurement carried out on the

slot weld. This comparison is shown in Figure 7. It can be seen that the transverse stresses predicted by the FE model, following deposition of pass 3, are generally constant along the length of the weld and approximately 300MPa in magnitude, but rise sharply to about 350 to 400MPa at the stop end. This represents a stop end increase in stress of 50 to 100MPa, when compared to the mid-length. Peak stresses on the top surface of the parent plate are biased towards the stop end. This is consistent with previous studies [3].

The contour measurement shows a similar trend with a definite peak in transverse stress towards the stop end of the weld. A further comparison between the FE model and the contour measurement is shown in Figure 8 where the stresses at mid-length are shown. Both measurement and FE model show exactly the same trend in stress with a peak occurring approximately 15-20mm from the plate bottom surface. Generally the FE model over predicts stresses compared to the contour measurement.

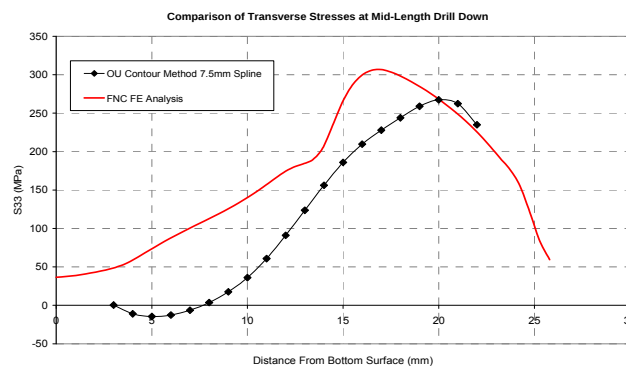


Figure 8: Slot Weld – Comparison of Transverse Stress at Mid-Length

This case study demonstrates that a moving heat source approach is required to capture the behaviour at the start and stop ends of welds. Historically this type of welding analysis would be carried out using a block dumping methodology due to it being a multi-pass, relatively long weld which would not pick up the start and stop effects to the same degree. Reference [3] shows some comparisons between the moving heat source methodology and a block dumped methodology.

3. Case Study 2 – Multi-Pass Repair Weld

3.1 Introduction

Repair welds are usually introduced into structures either to remedy initial fabrication defects found in castings or welds by routine inspection, or to rectify in-service degradation of components and thereby extend the life and economic operation of ageing engineering plant. The type of repair can range from filling a very localised shallow excavation using standard weld procedures, to welding deep excavations that can extend around a significant proportion of a structure. Repairs can be further categorised into those centred on the original weld and those that are offset from the weld centre-line. The need to rectify defects caused by lack of side-wall fusion, or degraded heat affected zone material, typically leads to offset repairs or centred repairs encompassing material beyond the original fusion boundary.

Structural integrity assessments require an accurate description of the through-wall residual stress field in the component. However, reliable characterisation of residual stresses at non-stress-relieved welds

is notoriously difficult. Some recommended upper bound residual stress profiles can be found in the R6 Revision 4 defect assessment procedure, as well as alternative structural integrity codes. Definition of 'best estimate' residual stress profiles requires high quality experimental measurements coupled with an understanding of component structural behaviour and non-linear analytical modelling of the welding processes responsible.

The work reported in this document is a development of the welding simulation methodology presented above, but extended and applied to multi-pass 3-dimensional repair welds. The repair weld simulations are based on a repair weld made on a test component manufactured from ex-service power station steam headers. Three analyses were undertaken, which simulated the deposition of the original girth weld and the deposition of the repair weld. The repair weld was modelled using two different methods. Initially the entire repair weld was modelled using a block dumped methodology. However recent studies have suggested that the capping passes are the most influential in the development of residual stresses within a weld. Therefore for the second analysis the capping passes only are modelled using a moving heat source whilst the remaining filler passes in the repair are modelled using a block dumped methodology.

A 'long' repair, with an outer surface arc length of 262.6mm and positioned 12mm offset from the original girth weld was modelled. The actual weld repair preparation was ground into position and the final repair made to 70-75% of the cylinder wall thickness. In all, twelve passes were performed to fill the repair weld.

For the analysis where the capping passes are modelled using a moving heat source method the first nine weld passes are deposited using a block dumped approach. The remaining three weld passes are deposited using a moving heat source. The capping passes are made up of a number of weld runs. Each of these weld runs will be modelled explicitly using a moving heat source. Therefore intermediate start/stops along the weld length can be assessed as well as the start/stops at the ends of the weld passes. Ten weld runs make up the final three capping passes.

3.2 Methodology

The general steps in completing a girth and repair weld analysis of this type are as follows:

1. Build a 2D axisymmetric mesh of the original girth weld.
2. Read the girth weld mesh into the HSMT and conduct an initial quasi-steady state moving heat source thermal analysis.
3. Ensure the thermal solution is appropriate, i.e. approximately 200% of the bead area is melted and/or by comparison with suitable macrographs and thermocouples.
4. If the thermal solution is satisfactory then go to step 5, otherwise repeat Steps 2 and 3 and revise the definition of the heat source, e.g. efficiency or weaving parameters.
5. Run the Abaqus sequential thermal-mechanical analysis using the calibrated heat source parameters obtained in Steps 2 and 3. Using the block-dumped approach transient heat fluxes are read into Abaqus from the HSMT. Abaqus solves for the transient thermal elastic-plastic stress/displacement solution.
6. Build a three-dimensional quarter model mesh of the repair weld.
7. Read the repair weld mesh into the HSMT and conduct an initial quasi-steady state moving heat source thermal analysis.

8. Ensure the thermal solution is appropriate, i.e. approximately 200% of the bead area is melted and/or by comparison with suitable macrographs and thermocouples.
9. If the thermal solution is satisfactory then go to step 10, otherwise repeat Steps 7 and 8 and revise the definition of the heat source, e.g. efficiency or weaving parameters.
10. Map the residual stresses from the original girth weld onto the repair weld mesh .
11. Run the Abaqus sequential thermal-mechanical analysis using the calibrated heat source parameters obtained in Steps 7 and 8. Using the block-dumped approach transient heat fluxes are read into Abaqus from the HSMT. Abaqus solves for the transient thermal elastic-plastic stress/displacement solution. For the block dumped analysis this is the final step.

For the moving heat source analysis only:

12. Build a three-dimensional half model mesh of the repair weld (see Figure 9).
13. Map the residual stresses from the quarter repair weld mesh onto the half model repair weld mesh.
14. Run the Abaqus thermal moving heat source analysis using the calibrated heat source parameters obtained in Steps 7 and 8 for the remaining capping passes and the weld runs.
15. Run the Abaqus mechanical moving heat source analysis using the temperature output from Step 14 for the remaining capping passes.

Details of the block dumped methodology can be found in used to determine the residual stresses in the girth weld and the initial weld passes of the weld repair can be found in [5]. The moving heat source methodology is the same as that described in Section 2.5 and 2.6 above.

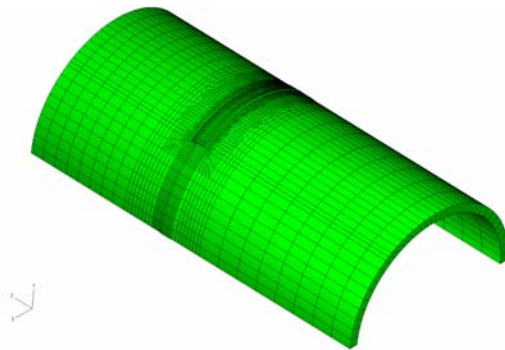


Figure 9: Repair Weld – Finite Element Model

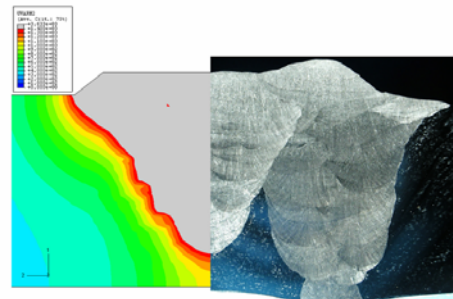


Figure 10: Repair Weld –Comparison of Weld Fusion Boundary

3.3 Material Properties

A mixed isotropic-kinematic hardening model was used for this analysis as described in Section 2.7. The model was modified slightly to take into account the different yield strengths of the materials in question otherwise the model implementation was the same.

3.4 Results

A comparison of the predicted repair weld fusion boundary with that from available macrographs (as shown in Figure 10) suggests that the extent of the fusion boundary is predicted well.

A review of the mechanical analysis indicates that significant residual stresses are predicted within the repair weld and surrounding area. High residual stresses are predicted in the parent/HAZ region as well as weld metal with radial stresses being as high as 60MPa, axial as high as 300MPa and hoop as high as 500MPa.

3.4.1 Comparison – Block Dumped and Moving Heat Source

Contour plots of transverse and longitudinal stress for both block dumped and moving heat source analyses are shown in Figure 11 to Figure 14. The plane chosen to present these plots is located on a plane of peak stress at the approximate last repair weld capping pass centreline. Towards the end of the repair the stress distributions between models is similar with an apparent peak in stress near the start / stop ends of the weld. However the block dumped model does not predict the peaks in transverse stress near the weld mid-length. This effect is similar to that observed in [5]. This is investigated further by plotting the linearised distribution through the thickness of the pipe on this plane of peak stress.



Figure 11: Repair Weld –Transverse Stress on Plane of Peak Stress –Moving Heat Source

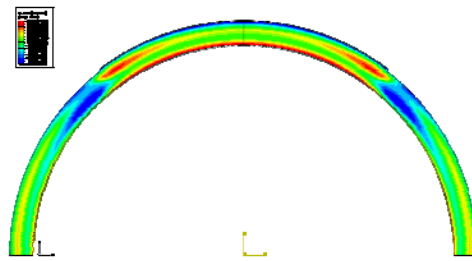


Figure 12: Repair Weld –Transverse Stress on Plane of Peak Stress –Block Dumped

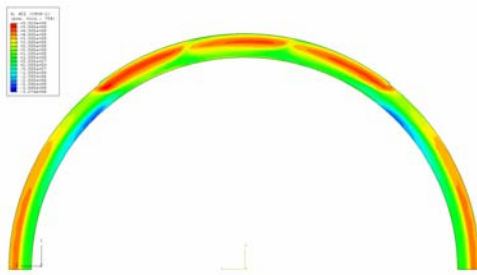


Figure 13: Repair Weld – Longitudinal Stress on Plane of Peak Stress –Moving Heat Source

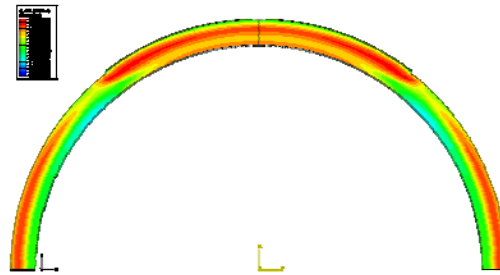


Figure 14: Repair Weld – Longitudinal Stress on Plane of Peak Stress – BD

The distributions of the linearised transverse membrane stress component along the length of the weld (Figure 15) display significantly contrasting profiles for the block-dumped and moving torch analysis. This is due to the simulation of three end to end weld beads for the moving torch analysis, whereas the block-dumped approach simulates the entire length of the repair weld pass being deposited at once. The transverse membrane stress component is considerably higher for the moving torch analysis at the

repair weld mid-length and weld stop locations, by as much as three times at mid-length. At the weld start location the two models are in agreement with each other. The longitudinal component of membrane stress (Figure 16) for the block-dumped results bound those for the moving torch throughout the length of the repair weld.

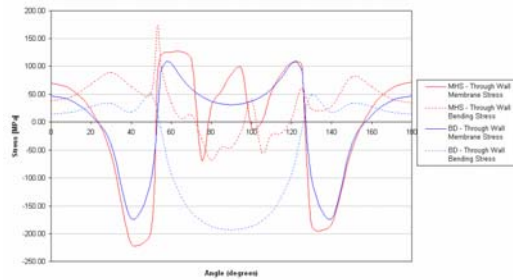


Figure 15: Repair Weld – Comparison of Linearised Transverse Stress on Plane of Peak Stress – BD and Moving Heat Source

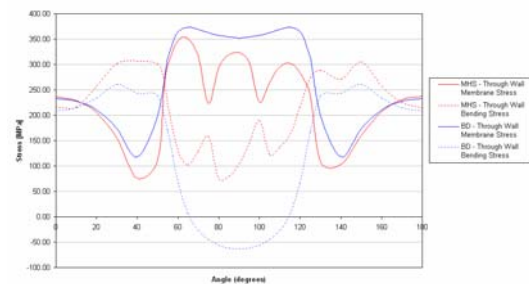


Figure 16: Repair Weld – Comparison of Linearised Longitudinal Stress on Plane of Peak Stress – BD and Moving Heat Source

3.4.2 Start and Stop Effects

The use of a moving heat source technique for the capping beads has been successfully applied to capture individual start and stop stresses. Examination of the residual stresses on the outer surface of the pipe and repair can be used to determine which capping passes influence the residual stresses. Further investigation indicates that the stresses are dominated by runs 1 to 3 of pass 12 (the final capping pass). The stresses from the previous capping passes are not so clearly defined and it is judged that the stresses from these passes have been reduced by the deposition of subsequent weld beads. Therefore the final capping pass layer dominates the residual stress distribution.

There is also a variation in stresses between the individual weld runs for the final capping pass. By examining the contour plots (Figure 11 to Figure 14) the variation between weld runs for the final capping pass can be determined. A review of the transverse and longitudinal stresses at this location reveals that the final capping run produces the highest residual stresses. The slot weld analysis presented in case study 1 have demonstrated that the peak stresses occur at the stop end of the final weld run. The final capping pass run supports this with the highest stresses occurring at the stop end of the final run. However, at the intermediate stop ends the peak stresses are reduced. Therefore it can be concluded that the subsequent weld runs remove the previous stop position peak stresses possibly due to annealing. This is particularly evident for the transverse and longitudinal stress components. Beyond the weld excavation there are still peak stresses associated with the start position of the first capping run pass and the stop position of the final capping pass run. The stresses at the stop end are greater in magnitude.

In terms of analysis run times, the moving heat source analyses for the capping passes took in the region of four to five times as long compared to the equivalent block dumped analysis. This still represents a significant time penalty, however in the two years since this work was carried out computing speeds have improved considerably. Therefore this type of analysis could be used efficiently for a given model size.

4. Conclusions

Finite element studies have been carried out to demonstrate that the moving heat source approach can be used to model complex and large structures. The following conclusions can be drawn from this work:

- The methodology adopted has proven itself to be very flexible, robust and efficient and is readily extendible to larger and more complex analyses.
- Residual stress predictions at the mid-length of a multi bead weld pass in a repair weld are sensitive to assumptions made with respect to heat source modelling approach and bead deposition sequence. In particular, modelling each run of a multi-bead weld pass can increase the transverse stresses at mid-length relative to a simulation that deposits weld using a block dumped approach. However longitudinal stresses are conservatively predicted by a block dumped approach.
- Moving heat source analyses enable the analyst to capture stress concentrations around 3D features and start / stop effects. The stresses around start / stop locations for the slot weld case study is in good agreement with full maps of measured residual stresses.
- Whilst run times for moving heat source analyses are still longer than for block dumping simulations, the improvements in computing hardware capability allow complex models to be assessed in realistic time frames.

5. References

- [1] Feat Weld Heat-Source Modelling Tools – Part 2. Guides and Tests, FEATPLUS Limited, FP/CR/0016/04, Issue 1, January 2004.
- [2] Abaqus/Standard User's Manual, Abaqus Inc., Version 6.5, 2004.
- [3] Dennis R J, Leggatt N A, Gregg A, Optimisation of Weld Modelling Techniques – Bead-on-Plate Analysis, Proceedings of PVP2006-ICPVT-11: 2006 ASME Pressure Vessels and Piping Division Conference July 23-27, 2006, Vancouver, BC, Canada.
- [4] Leggatt N A, Dennis R J, Bouchard P J, Smith M C, Numerical Methods to Predict Distortion in Welded Components, Proceedings of 2008 ASME Pressure Vessels and Piping Division Conference July 27-31, 2008, Chicago, USA.
- [5] Elcoate C.D., Dennis R.J., Bouchard P.J. and Smith M.C., 3-Dimensional Repair Weld Simulations – Bead Sequencing Studies, ASME PVP 2004, San Diego, USA.

6. Acknowledgements

The authors acknowledge the work funded by British Energy Generation Ltd. The authors are grateful to Chris Watson and Paul Hurrell at Rolls-Royce for their support in producing this paper.