FINITE ELEMENT ANALYSIS OF A WELDED CRUCIFORM JOINT

Aihui Wu*, B.G.Mellor*, S.Syngellakis**

*Materials Research Group, ** Computational Engineering & Design Group, School of Engineering Sciences, University of Southampton, Highfield, Southampton. SO17 1BJ

ABSTRACT

The behaviour of a welded cruciform joint under static loads was simulated using the finite element method (FEM). The geometry, material properties and loading conditions of the joint were identical to those of a specimen for which experimental results were available[1] so that the FEM model could be validated. The effects of varying weld geometry and material properties on the developing strain and stress distributions were studied using the validated model.

INTRODUCTION

The development of modern welding technology began in the latter half of nineteenth century when electrical energy became commercially available. Due to the advantages of welded structures such as high joint efficiency, water and air tightness, weight saving, reduction in fabrication time and cost, welded joints have been widely used in construction, by both offshore and aeronautical industries since World War II[2].

Reliable structural design needs an accurate assessment of welded connection behaviour. In conventional design, although the strength of welds is designed to be higher than that of the other components of a welded structure, in practice a large percentage of fracture failures are found to occur in the welds. It is therefore evident that the behaviour of welded joints has not been fully understood and further study is necessary so that the safety and economy of the design of welded structures be improved.

FEM, as the most popular numerical technique used in engineering, has been widely applied to assess the performance of welded structures. However, there is considerable uncertainty in such finite element, static or dynamic, analyses regarding the accurate representation of the stiffness and strength of welded connections.

In this study, a finite element model of a welded cruciform joint has been constructed. For validation purposes, the geometry and loading conditions adopted are identical to those of the specimen tested by Fessler *et al*[1] who investigated experimentally the flexibility of such a connection for various types of loading. The effects of weld geometry, depth of penetration, and variation of material properties on the behaviour of the welded joint have also been examined.

JOINT SPECIMEN

The geometry of the specimens used by Fessler *et al.*[1] is shown in Fig.1a. Three crosses were loaded under pure bending and two were loaded under tension. The extent of the heat affected zone (HAZ) was determined by etching a number of prepared surfaces and this information was used in the FE model to define the HAZ size and shape. The photoelastic coating technique was used to measure the strain on one lateral surface (Fig 1a).

Table 1. Material properties of the joint specimen[1]

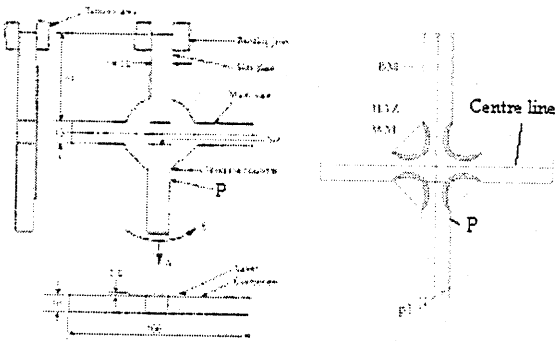
Material	Yield strength (N/mm^2)	Ultimate tensile strength (N/mm^2)	Reduction of area (%)
Steel plate*	387	536	66
Weld metal**	463	510	70
Weld metal***	398	529	63
HAZ ***	468	579	35

* Obtained from the steel maker and the Welding Institute.
** Typical manufacturer's information for this grade electrode
*** Obtained by Elliot" on same grade steel and electrode.

The material of the specimen was 50D, a typical structural steel while a E51 33B electrode was used for the weld. The material properties of the various parts of the joint were given in the paper, and reproduced in Table 1.

FE MODEL

The model shown in Fig. 1b was generated using the general purpose finite element package ANSYS[3]. Different material properties, according to the data of Table 1, were assigned to the three areas, base metal (BM), weld metal (WM) and HAZ, respectively. A bilinear kinematic-hardening stress-strain relationship was adopted to represent the elasto-plastic behaviour of these materials. In addition, a Young's modulus of 2.07×10^5 N/mm^2 and a Poisson's ratio of 0.3 were used for all areas. The element types used in this model are a 2-D 4-node quadrilateral element under plane stress (PLANE42) for BM, WM and HAZ and point-to-point contact element (CONTAC12)[3] for the contact areas between the main beam and the vertical attachments.



(a) (b)

Fig.1 Geometry and loading condition of the joint: (a) tested specimen[2], (b) FE model.

After a mesh sensitivity study, the final mesh of the model was adopted as shown in Fig.2.

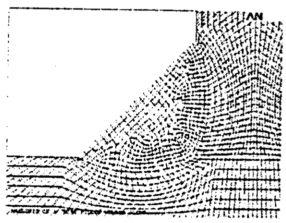
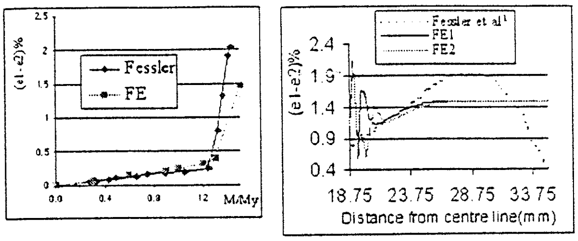


Fig. 2 Mesh map of the weld

RESULTS AND DISCUSSION

The difference of principal strains measured at position P in the side plates (Fig.1a) is plotted versus the applied bending moment in Fig.3a, while the strain distribution along the tensile edge of the side plate under bending is plotted in Fig.3b. Results from the FE model are shown in the same figures so that they can be compared with the respective experimental measurements.

Fig.3a shows good agreement between the FE and the experimental results. The strain differences in the plate are proportional to the applied load up to the bending moment $M = 1.25M_y$ in both cases. The FE model predicts a lower strain after the load exceeds $1.25M_y$. This is because, although the assumed Young's modulus is expected to be close to that of the specimen, the values of maximum elongation entered into the FE model were probably considerably lower. As a consequence, the FE model had a higher tangent modulus and therefore stiffer post-yield behaviour than the specimen tested.



(a) Strain versus bending moment plot

(b) Strain distribution along tensile edge of side plate

Fig.4 Comparison between the FE and experimental model

In Fig.3(b) the strain distribution reaches the peak value at a similar position in both the FE and experimental results, but the FE model predicts lower peak values than the experimentally obtained peak. As explained above, this may be due to the high tangent modulus adopted in the FE model, which results in a stiffer behaviour than the observed one. To the left of this peak point, FE predictions and test data show the same increasing trend. The initial sharp peaks in the FE curves are due to the stress rise at the weld toe causing localised plasticity in the HAZ and the sudden change of properties across the HAZ-base metal interface. The drop of the strain difference to the right of the peak point in the experimental result may be due to the edge effect of the photoelastic layer used in the experiment.

PARAMETRIC STUDIES

These studies were facilitated by the use of ANSYS Parametric Design Language. The effect of varying material properties and the size of the weld on the joint behaviour were examined. The results of the parametric studies gave some idea of how significantly different parameters affect the behaviour of the welded joint, which will prove very useful in subsequent work on generating a simplified model for the joint.

Effect of material properties

Micro-hardness test results reported by Fessler *et al.* show clearly that there is, in reality, a gradual change of plastic material properties across the BM,WM and HAZ rather than the sudden change across their idealised interfaces adopted in the original FE model. The effect of considering a gradual change of material properties across these three areas in the FE model was studied by assigning linearly extrapolated different material property values to different layers of elements.

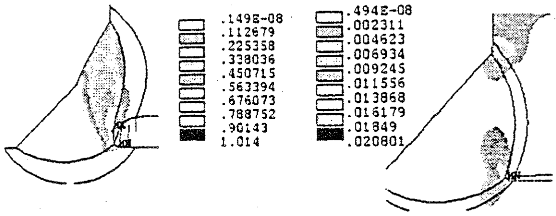
Strain distributions along the tensile edge of the side plate from FE models with sudden and gradual property changes across the three areas are shown in Fig.3(b) as curves FE1 and FE2, respectively. It is seen that a more realistic strain distribution across the HAZ-base metal interface is achieved but the high strain peak within HAZ persists. Inspection of the Von Mises stress plot showed that a slightly higher value for the yield stress would have removed this localised plastic deformation. This is further evidence of the sensitivity of the FE predictions to assigned material properties. This may have a great effect on fatigue analysis of a welded joint because a fatigue crack is most likely to initiate from the weld toe and the stress state near the toe affects greatly crack propagation.

Effect of weld size

The weld leg length was changed to 8 mm from 12.5 mm in the original FE model. The combined weld throat is then less than the width of the side plate. Fig.4(a) shows the contour plot of the Von Mises stress in the model under tension. While the original specimen was observed to fail by excessive plastic deformation of the base metal, in the reduced weld size model, strain exceeding the fracture strain of the weld metal was predicted in the weld metal area. It may be concluded that failure will occur in the weld with a failure plane at around 20°. Conventional weld design assumes that the weld throat is the failure plane and stress is uniformly distributed along the weld throat. However, test results have shown that the actual plane of failure does not coincide with the throat plane[4-7] and is positioned at a certain angle relative to the base of the weld. Furthermore, Solakian *et al.*[8] concluded that the stress distribution over the weld throat area is not uniform as the conventional design method assumes. The present FE study produced results that support the above argument.

Fig.4(b) shows the result of equivalent strain distribution for the same leg length joint with a 1 mm penetration under the same loading conditions. Comparing the results shown in Fig.4(b), it is clear that the introduction of penetration increased the load carrying capability significantly.

The stress concentration factor (SCF) at the weld toe under tension in this FE study was found equal to 2.9 for a weld leg length of 12.5 mm and 3.93 for a weld leg length of 8 mm. The SCF increases with decreasing weld size and decreases with increasing load magnitude. The SCF is found to be smaller under bending than under tension.



(a) Without penetration

(b) With 1 mm penetration

Fig.4 Contour plot of Von Mises equivalent strain for the 8 mm weld leg model under tension

CONCLUSIONS AND FUTURE WORK

A cruciform welded joint has been analysed using FEM. The FE model gives results in reasonable agreement with published experimental data. Parametric studies were carried out on this model to study the sensitivity of the FE predictions to material property variations, as well as the effect of weld size and loading condition on the behaviour of the welded joint.

For welded joints with a combined weld throat larger than the thickness of the plates they join, failure is likely to occur in the plates. For smaller welds, failure is likely to occur in the weld with the failure plane located at an angle less than that of the throat to the weld edge. The SCF for a smaller weld size is larger than that for a larger weld size. The strength capability of the welded joint is increased by the introduction of weld penetration.

Further research will be carried out to study the fracture behaviour of the joint including the effects of material mismatch, depth of penetration and residual stresses.

REFERENCES

1. Fessler H. & Pappalettere G., 5^{th} *International Conference on Pressure Vessel Technology*, 15, 1984

2. Masubuchi K., Analysis of Welded Structures, MIT, Pergamon Press, 1980

3. ANSYS 5.6, SAS IP, Inc., Canonsburg, PA, 1999.

4. Higgins T.R. & Preece F.R., *Welding Research supplement*, **15**(10), 429s, 1968

5. Fessler H. & Pappalettere C., *J. Strain Anal.*, **24**(1), 1989

6. Barshi, O.A., Zaitsev, N.L., *Svar. Proiz*, No. 9, 3, 1977

7. Kamtekar A.G., *J. Constr. Steel Res.*, **2**(2), 33. 1982

8. Solakian A.G, Claussen G.E., The Engineering Fundation Welding Research Committee, *Supplement to the Journal of the American Welding Society*, 1. 1937

9. Elliot K.S., PhD Thesis, Nottingham University, 1987