THREE-DIMENSIONAL ELASTO-PLASTIC FEM CALCULATIONS OF CT-SPECIMENS WITH SPECIAL VIEW OF INVESTIGATING THE INFLUENCE OF SPECIMEN THICKNESS ON PLASTIC ZONE SIZE

J. Redmer and W. Dahl

Institut für Eisenhüttenkunde der Rheinisch-Westfälischen Technischen Hochschule Aachen Institute of Ferrous Metallurgy, Technical University, Aachen, Federal Republic of Germany

ABSTRACT

The three-dimensional stress state is calculated using a three-dimensional elasto plastic Finite Element Method in displacement formulation. The authors have shown in a paper presented at the Numerical Conference in Swansea 1 that detailed three-dimensional calculations produce results which are contradictionary in some points to the two-dimensional models. It was shown that the plastic zone does not behave like a dog-bone. The plastic zones in the centre and on the surface of the specimen do not differ in size very much.

The goal of the present paper is to show the influence of the specimen thickness to the size of the plastic zones. The result is that the plastic zones decrease with increasing specimen thicknesses thus verifying experimental observations. For thick specimens the calculated plastic zone is in its shape and in its size very near to the plastic zone of the dog-bone model in the plane strain situation. For thinner specimens the plastic zones become bigger in the centre and on the surface of the specimens. There is no significant difference between the centre and the surface. The sizes of the plastic zones for thin specimens are in the range of the plane stress dog-bone model.

KEYWORDS

Three-dimensional Finite Element calculation; Plastic zones for different thicknesses; CT-specimens.

INTRODUCTION

The Finite Element Method (FEM) became a common instrument in structural analysis and it is by now also well established in fracture mechanics. This work wants to show some basic results of the three-dimensional elasto-plastic calculations of CT-specimens. The specimens were experimentally investigated at the 'Institut für Eisenhüttenkunde'.

The FEM programme used works with 20-node isoparametric brick elements 2 and frontal solution technique 3 . The work expressions are formulated in displacements and it is assumed that the strains and the displacements are small enough to use first order geometrical theory.

The plasticity is solved by an incremental method using initial stresses. The stiffness is kept constant so that the linear equation system has to be solved only once, namely during the first load increment.

In Figure 1 a typical FEM mesh as used in the calculations is drawn. Of course only the two-dimensional basic mesh is plotted. For the discussed three-dimensional calculations we put three 20-node isoparametric brick elements in z-direction. For there is a symmetry plane in the z-direction only one half of the specimen thickness has to be calculated. This means that in fact six quadratic elements describe the total thickness of the specimen which should be enough. The second symmetry plane (in y-direction) is well-known from all 2-d calculations so that at the end one quarter of the total specimen has to be modelled. In the 3-d calculations we used about 250 elements with about 4500 variables to model this quarter. About 110 load increments are necessary to reach the maximum load.

RESULTS

The authors showed 1 that there is no significant difference in the plastic zone size between the centre of the specimen and its surface. Figs. 2 to 4 show perspective graphs of the plastic zones in one quarter of an investigated CT-specimen. The load is increasing from Fig. 2 to Fig. 4. The z-axis scale is distorted relatively to the scale of the x-y-plane in order to arrange a proper impression of the changes in plastic zone size from the mid-plane to the surface plane. The mid-plane is on bottom of the graphs and the surface plane is on top. The plastic zones are always plotted in the Gauss-point layers.

In Fig. 2 plasticity has already occured in the region of the crack tip. One can see that there is no significant difference between the size of the plastic zone from the mid-plane to the surface plane. In Fig. 3 the pressure region of the specimen already started to plastify. Still the plastic zones show no variation in z-direction which is in contradiction to the two-dimensional dog-bone model. In Fig. 4 finally a difference between the mid-plane and the surface appears. We still have two plastic zones at the mid-plane whereas the plastic zones at the surface have joined one another thus establishing the state of general yield at the surface. In the vicinity of the specimen's centre of rotation one still can see a purely elastic region. Now we want to discuss the influence of the specimen thickness to the size of the plastic zones. In Figs. 5 to 10 the plastic zone for three different thicknesses and two different applied stress intensities are plotted. Figs. 5 to 7 have the same stress intensity. It is lower than the applied stress intensity of Figs. 8 to 10. In the Figures the plastic zones calculated by use of the Finite Element Method are plotted for the centre and for the surface of the specimens. For a comparison the plastic zones of the dog-bone model are drawn too.

One can see that the plastic zones are increasing with decreasing thickness. This result had been expected because of experimental observations. The plastic zone of the thickest specimen has nearly the size and the shape of the plastic zone calculated from the dog-bone model assuming the plane strain situation. Obviously the all over behaviour of the specimen is very near to the plane strain state. It is interesting that the plastic zone of the 50 mm thick specimen is already rather big. It is more plane-stress-like. The plastic zone of the 25 mm thick specimen is in x-direction even bigger than the plastic zone of the plane stress dog-bone model whereas it is smaller in y-direction. So it does not verify the shape of the plane stress plastic zone predicted by the dog-bone model. We do not want to discuss the figures too much, because they are rather self-explaining.

It has to be pointed out that the presented results are valid for a straight crack front. If the crack propagates in the centre of the specimen in form of the well known thumbnail the plastic zone at the surface becomes bigger than the plastic zone in the centre, but this is another investigation. The main result of the present paper is that the plastic zones increase with decreasing specimen thicknesses. There is no difference in the size of the plastic zones between the centre of the specimen and its surface. One can only talk about an all over behaviour of the specimens. For very thick specimens that all over behaviour is plane strain like. Thinner specimens have a much bigger plastic zone also in their centre. For this circumstance two reasons are mainly responsible. First, the plane strain situation in the centre of the specimens is only an approximate one. Additionally this approximate plane strain situation cannot be compared to the plane stress situation on the surface because both have different maximal principal stresses. Further investigations will treat the influence of the so-called thumbnail effect.

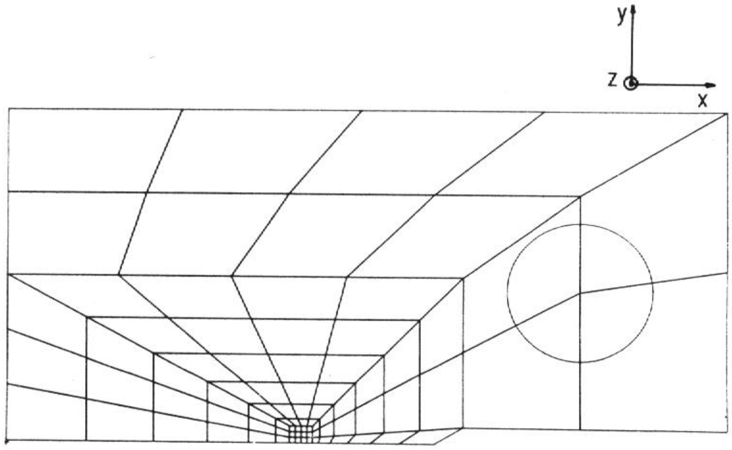
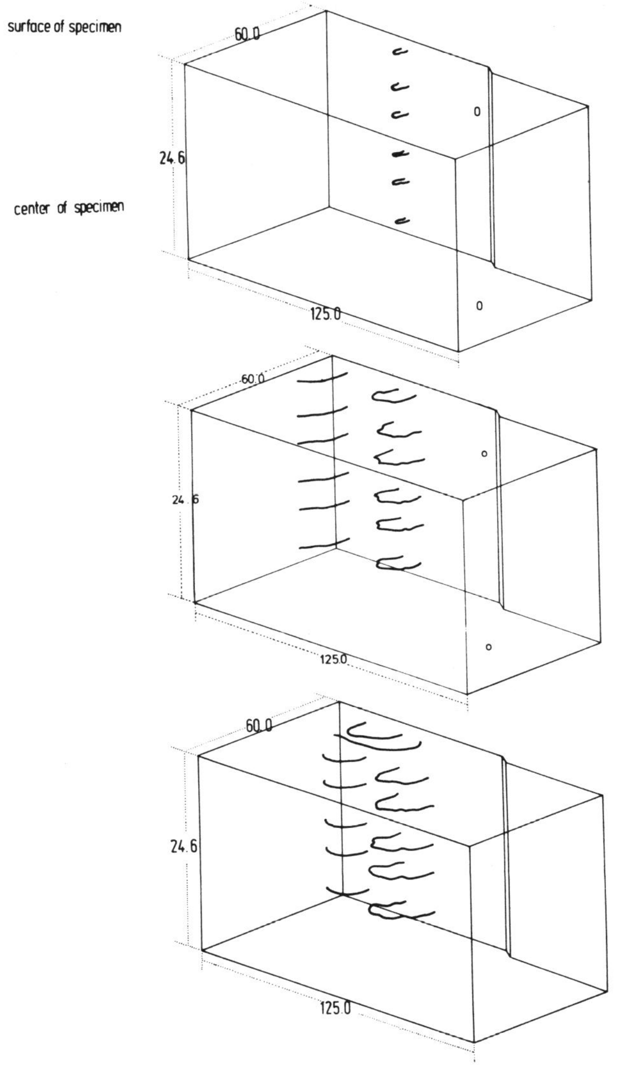


Fig. 1. Two-dimensional basic mesh, as used in the calculations



Figs. 2 to 4. Development of the plastic zone with increasing load

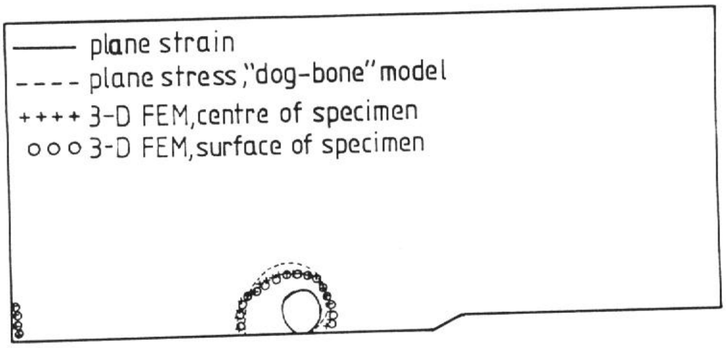


Fig. 5. 25 mm 2CT specimen, medium stress intensity, FeE 460, room temperature

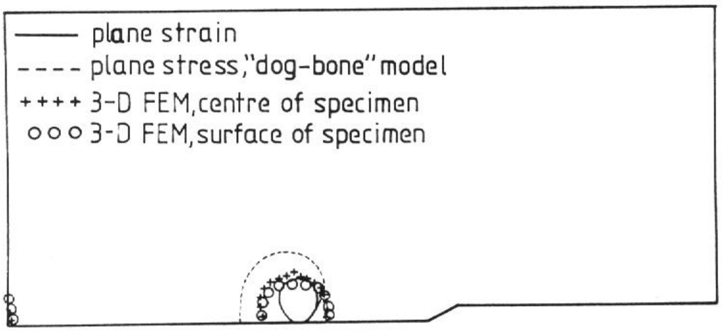


Fig. 6. 50 mm 2CT specimen, medium stress intensity, FeE 460, room temperature

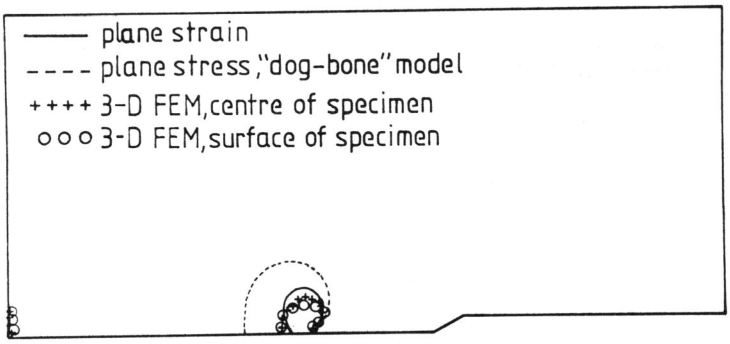


Fig. 7. 200 mm 2CT specimen, medium stress intensity, FeE 460 room temperature

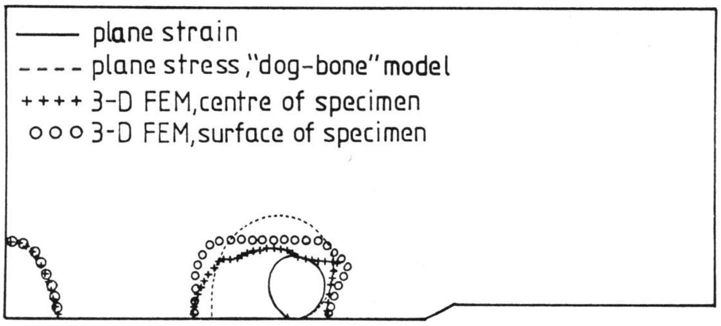


Fig. 8. 25 mm 2CT specimen, high stress intensity, FeE 460 room temperature

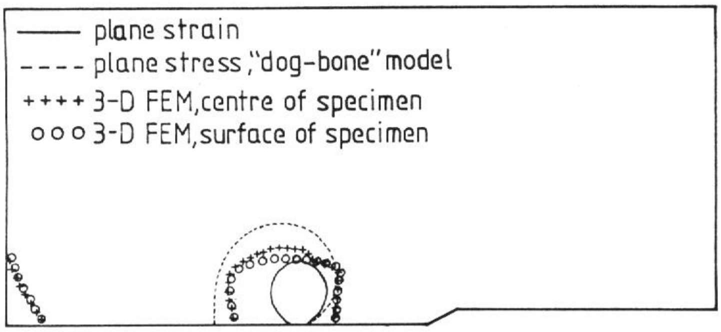


Fig. 9. 50 mm 2CT specimen, high stress intensity, FeE 460 room temperature

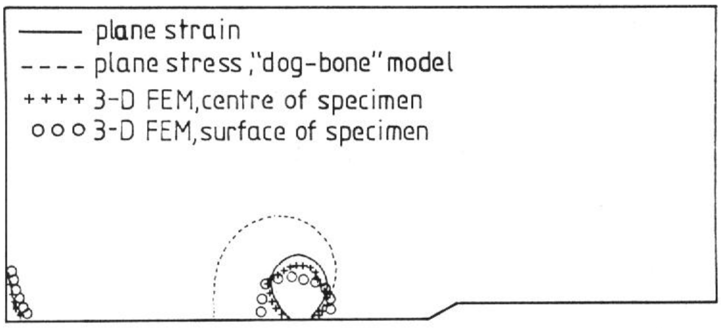


Fig. 10. 200 mm 2CT specimen, high stress intensity, FeE 460 room temperature

REFERENCES

1. **REDMER, J., W. DAHL** - Two- and three-dimensional elasto-plastic FEM-calculations of CT-specimens
in: Numerical Methods in Fracture Mechanics. Conf. Proc., Swansea (1980)

2. ZIENKIEWICZ, O.C. - The Finite Element Method in Engineering Science, McGraw-Hill, London (1971)

3. IRONS, M.B. - A Frontal Solution Program for Finite Element Analysis. Int.J.Num. Meth.Eng.Sc., Vol.2, No.1 (1970)