World Academy of Science, Engineering and Technology
International Journal of Mechanical and Industrial Engineering
Vol:9, No:09, 2015

Computation and Validation of the Stress Distribution around a Circular Hole in a Slab Undergoing Plastic Deformation

Authors : Sherif D. El Wakil, John Rice

Abstract : The aim of the current work was to employ the finite element method to model a slab, with a small hole across its width, undergoing plastic plane strain deformation. The computational model had, however, to be validated by comparing its results with those obtained experimentally. Since they were in good agreement, the finite element method can therefore be considered a reliable tool that can help gain better understanding of the mechanism of ductile failure in structural members having stress raisers. The finite element software used was ANSYS, and the PLANE183 element was utilized. It is a higher order 2-D, 8-node or 6-node element with quadratic displacement behavior. A bilinear stress-strain relationship was used to define the material properties, with constants similar to those of the material used in the experimental study. The model was run for several tensile loads in order to observe the progression of the plastic deformation region, and the stress concentration factor was determined in each case. The experimental study involved employing the visioplasticity technique, where a circular mesh (each circle was 0.5 mm in diameter, with 0.05 mm line thickness) was initially printed on the side of an aluminum slab having a small hole across its width. Tensile loading was then applied to produce a small increment of plastic deformation. Circles in the plastic region became ellipses, where the directions of the principal strains and stresses coincided with the major and minor axes of the ellipses. Next, we were able to determine the directions of the maximum and minimum shear stresses at the center of each ellipse, and the slip-line field was then constructed. We were then able to determine the stress at any point in the plastic deformation zone, and hence the stress concentration factor. The experimental results were found to be in good agreement with the analytical ones.

Keywords : finite element method to model a slab, slab undergoing plastic deformation, stress distribution around a circular hole, visioplasticity
Conference Title : ICAMME 2015 : International Conference on Applied Mechanics and Materials Engineering
Conference Location : London, United Kingdom
Conference Dates : September 25-26, 2015