International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:9, No:9, 2015

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

S. D. El Wakil, J. 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.

I. Introduction

STRUCTURAL members undergoing elastic deformation often have regions in which stresses are much higher than the nominal ones, as a result of geometric discontinuities referred to as stress raisers such as holes, notches, and threads. The maximum stress acting on the spot adjacent to a stress raiser can be determined by applying the concept of the stress concentration factor, which is the ratio between the actual stress acting on the spot adjacent to the stress raiser and the nominal stress, which is calculated based on the assumption of the absence of the stress raiser. Stress concentration factors in bodies undergoing elastic deformation, have been determined for various geometries of stress raisers, and can be found in published work [1] and [2]. Those could theoretically be determined with great accuracy by applying the theory of elasticity or the finite element method, since the relationship between the stress and the strain is linear within the elastic range. Stress concentration factors were also determined experimentally by photoelasticity, but again for stresses within the elastic range.

Structural elements which are to be designed based on the magnitude of the maximum stress in the spot adjacent to the stress raiser would be much heavier than those designed without stress raisers. The presence of a hole in a member, for example, would result in stresses around the edge of that hole which are three times higher than the nominal stress .If that member is designed to allow only elastic strains, it would be three times heavier than one without the stress raiser; something which a designer cannot afford to permit in some engineering applications such as aviation. Nevertheless, if limited plastic deformation is allowed to take place where the stress attains a maximum value, the stress concentration factor would noticeably be reduced resulting in appreciable saving in the mass of the member. The current work was, therefore, aimed at investigating the effect of the presence of a small circular hole, which is a typical stress raiser, on the stress concentration in a slab undergoing plastic deformation. The results would enable engineers make more economical design of load-carrying members having stress raisers.

II. Computational and Experimental Methods

A. Finite Element Analysis

In this part of the current work, the aim was to employ the finite element method to a model slab which was subjected to plastic deformation, and which had a small hole across its width. The dimensions of the slab and the hole were identical to those of the actual slab that was experimentally tested in order to validate the results of the analytical model. The values of the mechanical properties of the actual slab were also used in the analytical model, so as to make the parameters affecting the results identical in both cases, and a comparison of the results can hence be made with a good level of confidence.

The finite element software used was ANSYS, and an element type was selected and input in the program. For plane strain in ANSYS, the PLANE183 element was utilized. It is a

S. D. Elwakil is with the Mechanical Engineering at the University of Massachusetts Dartmouth, Dartmouth 02747 USA, (Phone: 508-999-8594; Fax: 508-999-8881; e-mail:selwakil@umassd.edu).

J. Rice is with the Emeritus at the University of Massachusetts Dartmouth, Dartmouth 02747 USA.

higher order2-d, 8-node or 6-node element with quadratic displacement behavior. The mechanical properties of the material were then introduced. A bilinear stress-strain relationship was used to define the material properties, with constants identical to those of the material used in the experimental study. Next, the slab geometry was specified and the nodes and elements were created. Because of the symmetry of the slab and the resulting stresses, strains, and displacements around the vertical central axis and the section normal to it at its middle, a model comprising only a quarter of the slab was used and is shown in Fig. 1. The boundary conditions and the pressure loading were applied and the model was run for different pressure loads in both the elastic and the plastic ranges.

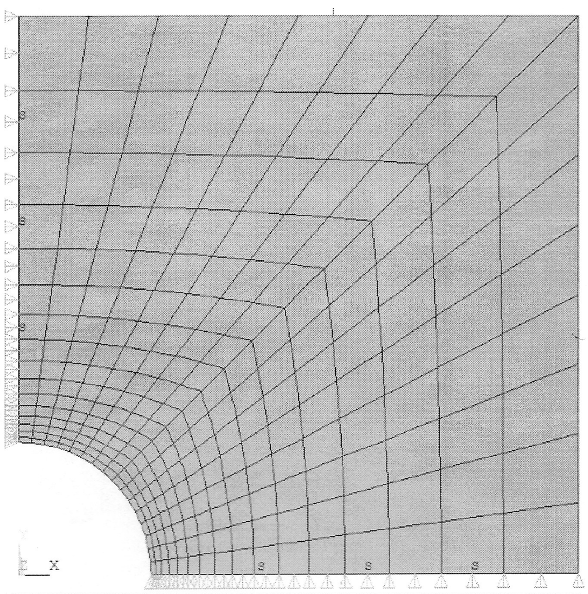

Fig. 1 Finite element mesh

B. Experimental Study

The experimental study involved employing the visioplasticity technique, which was first introduced by Thomsen [3]. A simpler version, which was used in the current work, was later used by El Wakil [4]. It involved printing a circular mesh (each circle was 0.5 mm in diameter) on the plane surface side of the aluminum slab, which was normal to the axis of no deformation. As previously mentioned, the slab had a small hole across its width acting as a stress raiser. Tensile loading was then applied to produce a small increment of plastic deformation. The strain pattern produced by this small plastic deformation step is then studied by recording the orientation and the aspect ratio of each ellipse formed from an original circle. Assuming that the material was isotropic, the directions of the principal stress and principal strain must coincide. Accordingly, the major axis of each ellipse was considered to be an indication of the direction of the maximum principal stress. We could then construct the field of the principal stress direction in the plastic deformation zone. At the center of each ellipse, the directions of the maximum and minimum shear stresses were determined by drawing short lines at 45^o to the major axis. At the center of each ellipse, the directions of the maximum and minimum shear stresses were determined by drawing short lines at 45^o to the major axis. Consequently, the slip-line field in the plastic deformation region was established, since the direction of the algebraically greatest principal stress was known. Next we were able to obtain the value of the maximum normal stress in the spot adjacent to the hole and the stress concentration factor was then determined.

Throughout the current study, each aluminum slab was 12.7mm (0.5 inch) in thickness, 38.1mm (1.5 inch) in width, and 400mm (16 inches) in length. That large value of length was chosen to insure that the pressure exerted by the grips of the testing machine on the slab, would not affect the state of stress around the hole at the middle of the slab. The hole was 3mm in diameter, and the thickness of the wall next to it on each side was, therefore, only 4.85mm. Accordingly, that fulfilled the condition for the plane strain deformation since the width –to-thickness ratio was almost 8.

The computer was employed to draw small circles making each circle just touching the neighboring ones, thus generating a circular mesh. The hard copy of the later was photographically reduced to produce a 35 mm black and white negative image.

After careful polishing and degreasing of one side of the slab in the area around the hole, the surface was coated with a thin uniform film of cold enamel (a kind of ultra-violet curing polymer provided by Roth Tech, in New Bedford, and Massachusetts.) by spinning. When the film of polymer was sufficiently cured, it was exposed to very high intensity ultra-violet light for 10 seconds through a high-contrast master negative of the grid mesh, which was placed and taped over the film. Spots of the enamel that were under the opaque areas of the negative slide, and were therefore not exposed to the ultra-violet light, were washed off with water. The slab was then baked at 180^{o}C for 30 minutes, and the slab with the required grid was ready to be subjected to tensile loading.

The slab was then subjected to tension using a universal testing machine Tinius Olsen. The applied load was gradually and slowly increased and the mesh around the hole was observed. When the circles of the mesh started taking the shape of ellipses, the applied load, which reached 33.8 kN (7, 600 pounds) was then removed and the slab was released from the testing machine for documentation and analysis of the results. The deformed mesh was photographed using a high-resolution camera (Nikon D1X) at the Photography Department of the University of Massachusetts Dartmouth and an enlarged printout on a high-contrast film was finally obtained and was used for further analysis. Measurements were made at a magnification of X15.

The mechanical properties of the slab material had to be determined experimentally since they are used in both the finite element analysis and the experimental study. A standard test piece was cut from a slab and was subjected to tensile testing in order to obtain the true stress- true strain curve and therefore determine the required properties, which are as follows:

Yield stress, Y	100 MPa
Elasticity modulus, E	70 GPa
Tangent modulus, E_t	33 MPa
Poisson's ratio	0.3

III. Results and Discussion

As previously mentioned, we ran the finite element model for different applied tensile stresses, namely 20 MPa, 60 MPa, 70 MPa, and 85 MPa. Each time, the stress concentration factor was obtained and we also looked for the presence and spread of plastic strains in the vicinity of the small hole. When the applied stress was 20 MPa, there was no evidence of any plastic strain and the model was totally undergoing elastic deformation. The stress concentration factor was 2.7, slightly less than the theoretical value of 3.0. This might be due to that the ratio of the slab thickness to the diameter of the hole was not large enough as the case in the elasticity theory analysis. It might also be due to the need for further refining of the finite element mesh. When the value of the applied stress was 70 MPa, plastic strain was evident in the area adjacent to the hole, as shown in Fig. 2. The stress concentration factor was only 1.39. That can be explained if we bear in mind that the material was almost rigidly plastic (very little work hardening beyond the yield stress), with the outcome that the yield was a limiting value or a ceiling and the stress could not be higher. That resulted in leveling of the stress or making the stress more evenly distributed across the section, which meant lower stress concentration factor.

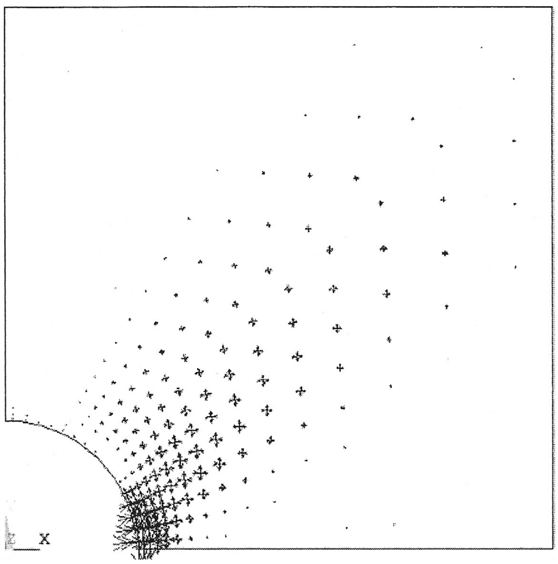

Fig. 2 Plastic principal strains (ANSYS)

A photograph of the slab with the distorted mesh after being subjected to limit tensile loading in the plastic zone is shown in Fig. 3. We used a magnified version of it on a high-contrast film to construct the slip-line field in the plastic deformation region as shown in Fig. 4. The state of stress at any point in that region could therefore be determined by applying Henky's equations [5].

It is clear that the magnitude of the normal stress right at the edge of the hole was equal to the yield stress of the slab (100 MPa) because that area was starting to yield. On the other hand, the nominal stress could be obtained by dividing the load acting on the slab (33.8 kN) by the cross sectional area of the slab if there was no stress raiser or actually one that is far away from the hole (12.7 x 38.1mm), and was found to be 69.8 MPa. The stress concentration factor was easily calculated and was found to be 1.43 which is close to that one obtained by the finite element method, thus validating the results obtained by employing that method .It is also clear from this simple analysis that increasing the applied nominal load would enlarge the plastic deformation zone, but would decrease the stress concentration factor since the normal stress right at the edge of the hole is constant and is equal to the yield stress. The mesh here helped only to identify the start of yielding. It can be used, however, to determine the state of stress at any point in the plastic deformation zone.

As can be seen in Fig. 4, the slip-line field was actually a fan-shaped region, intersecting with the free surface of the hole along a very short arc that was probably a singularity stress point when plastic deformation first took place. At the extreme point of the circular hole on the transverse axis, the line of the field makes 45^o with the free surface. The value of each of the maximum and minimum shear stresses is therefore equal to the yield stress in shear K (50 MPa). The maximum principal stress is parallel to the axis of the specimen and is equal to the yield stress in tension Y (100 MPa). Also, the hydrostatic tension component at that point is equal to K. The state of stress at any point in the plastic deformation zone can therefore be obtained, and the magnitude and direction of the maximum principal stress can then be readily calculated. This is achieved by noting the difference in the inclinations of the slip-line at the free boundary and at the desired point then applying Henky's equation as previously mentioned. The effective plastic strain at any point can also be calculated, and then the iso-strain contours can be constructed. Those would help predict and study the ductile failure as a result of the stress raiser.

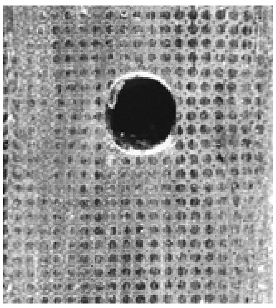

Fig. 3 A photograph of the slab with the distorted mesh

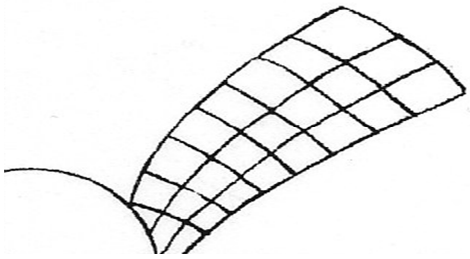

Fig. 4 Experimental slip-line field

IV. Conclusion

1. The visio-plasticity method was successfully used to obtain the stress distribution in the plastic deformation region adjacent to a stress raiser in a slab subjected to tensile loading.
2. The stress concentration factor as a result of the presence of the stress raiser in a body having a plastic deformation region was experimentally determined and was in good agreement with that obtained by employing the finite element method.
3. When the stress acting on a rigid body having a stress raiser such as a small hole results in a plastic deformation zone, the stress concentration factor would be far less than that when the deformation is purely elastic. The higher the applied stress and the larger the resulting plastic deformation zone, the lower would be the stress concentration factor.
4. Load–carrying member with stress raisers can be more economically designed if limited plastic deformation is allowed to occur.

Acknowledgment

S. D. El Wakil would like to express his gratitude to the late Fredrick Roth of Roth Tech in New Bedford for his support and indispensible help in printing the circular mesh on the aluminum slab. Some of the experimental raw data are from the graduate work of Dr. Tetsuro Osishi, for him acknowledgment must also go.

References

[1] W. D. Pilkey, *Peterson's Stress Concentration Factors*, 2nd Ed. New York: Wiley, 1997, ch.4.
[2] W. C. Young and R.G. *Budynas, Roark's Formulas for Stress and Strain*, 7th Ed. New York: McGraw-Hill, 2002, ch. 17.
[3] E.G. Thomsen, J.T. Lapsley, and J.B. Bierbower, "Experimental stress determination within a metal during plastic flow, "in *1954 Proc.Am. Soc. Exper. Stress Analysis*, vol .VII, no.2, pp.59-68.
[4] S.D. El Wakil, "Deformation in bar cropping investigated by the visio-plasticity", *Journal of Mechanical Working Technology*, vol 1, pp.85-98, 1977.
[5] W. Johnson, and P.B. Mellor, *Plasticity for Mechanical Engineers*. London: Van Nostrand, 1970, ch.12.