International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2, Issue 5, May-2015

The finite element analysis of a cracked plate

Mehta GauravkumarBharatbhai [1]

[1] *Bhagvan mahavir college of engineering and technology, Surat,* gauravzzz007@gmail.com

Abstract

The finite element analysis of a cracked plate is conducted to predict stress intensity factor under various conditions using the Abaqus finite element software. The overall behaviour of plates is affected by presence of a through crack .By using finite element modelling aspects of the problem twenty four different cracked plate models have been generated and analyzed by Abaqus finite element analysis software but in this paper we will take only two cases for sample calculation .The significance of various parameters such as meshing at the crack tip, change of element type, using different number of contours, crack length, crack locations, plate material (aluminium, copper and steel), variation of loading condition are studied. FE models were created according to different condition and analyzed behaviours of the cracked plate and predict the stress intensity factors.

Keywords*— FE analysis, Abaqus, stress intensity factor, Meshing, cracked plate analysis, Linear type of meshing, Quadratic type of meshing, Fracture Mechanics.*

I. INTRODUCTION

Fracture mechanics is the study of cracks and crack-like defects in components and structure. The aim is to understand behaviour of plate and predict the tolerance of components to the presence of crack to grow and propagate in components. The broadened application of different kind of structure in engineering, it is necessary to understand fracture mechanics. The failures of engineering structures are because of fractures by cracks propagations. Now we will use fracture mechanics in numerical techniques. Most of these studies have been aimed at either the behaviour of the crack itself or the state of the plate in the close neighbourhood of the crack tip. In this section, the overall behaviour of plates as affected by presence of a through crack is the focus of the study. The overall view is important if the stability and integrity of the structure as a whole is to be investigated. Our focused on the analysis of the cracked plate using the Abaqus finite element analysis software and significance of various parameter such as the meshing at the crack tip, change of element type, using different number of contours, crack length and location, plate material and variation in loading condition are studied. Twenty four models were generated in the Abaqus finite element analysis software. All the models contain different dimension, different loading condition and different material. Material properties and dimension are given in appropriate table.

NOMENCLATURE

E - Modulus of elasticity
V - Poisson's ratio
a - Crack length
W- Width of the plate
H- Height of the plate
h- Distance of the crack tip from bottom of the plate
σ- Uniform stress in the plate
K1-Mode I stress intensity factor
K2- Mode II stress intensity factor
F1- Applied load on the plate
F2- Applied load on the plate

II .THEORY

Fracture mechanics is concerned with the study of formation of cracks in materials. The aim of fracture mechanics is to analyze cracks and crack-like defects in components and structures and also predict the tolerance of the component or structure to presence of the crack. More ever fracture mechanics also analyze the tendency of the crack to grow and propagate in various conditions like variation of component geometry, variation in loading condition, variation in crack length and crack location etc. Tendency of a crack is two types such as crack growth may be either

(i) Stable (relative slow and safe)
(ii) Unstable (virtually instantaneous and catastrophic.)

In linear elastic fracture mechanics assumes that the materials is isotropic and linear elastic and analyze the problem considering pure elastic behavior of material. However, it will apply when the nonlinear deformation of the material is confined to a small region near the crack tip.[5],[2]

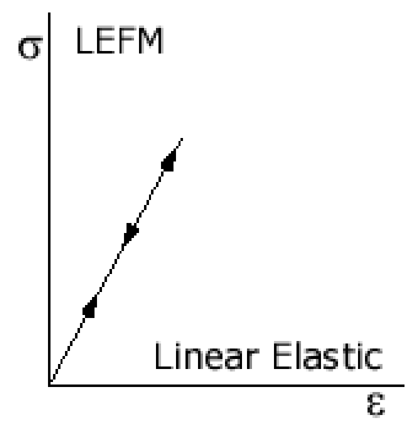

Figure 1. Linear elastic fracture mechanics

All Rights Reserved, @IJAREST-2015

There are three major modes of crack tip deformations.

- Mode I (Opening or tensile mode):- In which the crack surface s move directly apart.
- Mode II (Sliding or in-plane shear mode):- In which the crack surfaces slide over one another in the direction perpendicular to the leading edge of the crack.
- Mode III (Tearing mode):- In which the crack surface move relative to one another and parallel to the leading edge of the crack.

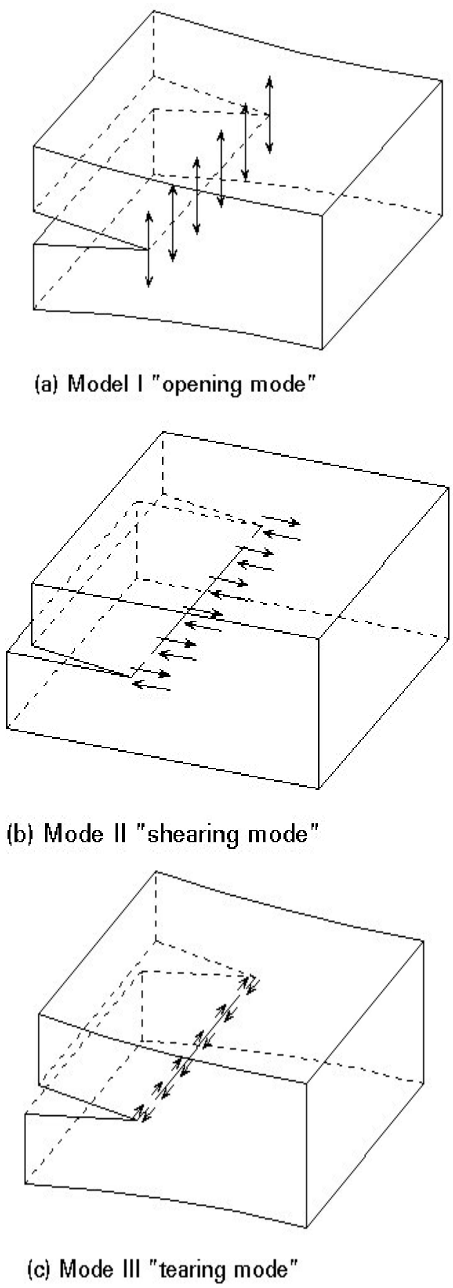

Figure 2 .Modes of crack tip deformation

Stress intensity factor:- Stress intensity factor (K) is more important in fracture mechanics to predict more accurate stress state(stress intensity) near the tip of the crack in structure or component. The magnitude of K depends on component geometry, the size and location of the crack and the magnitude and the modal distribution of load on the material.[5],[2]

KI is the stress intensity factor of model 1 (opening mode) and KII is the stress intensity factor of modal 2 (shearing mode)

It is noted that the elasticity stress distribution is always singular at crack tip which is defined by stress intensity factor based on the fracture mechanics theory for 2 dimensional problems. The stress intensity factor depends on the various parameters like loading condition, location of the crack tip, dimensions of the object & crack etc.

There are two methodologies used in Finite element method for 2 dimensional problems which are mentioned below.

2.1 Linear type of meshing

2.2 Quadratic type of meshing

2.1 Linear type of meshing: In this kind of meshing, the triangle element is used which consists of 3 or 6 nodes whose each side is straight. This kind of Methodology is useful when we analysis the object having linear displacements & stresses.

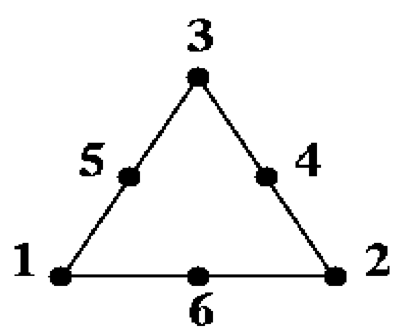

Figure 3. Triangular element

2.2 Quadratic type of meshing: In this kind of meshing, two types of element are used which are given below.

2.2.1 Triangular element: In this kind of element, the 6 nodes are used. Three nodes at mid of all sides & 3 at vertex..

2.2.2 Quadrilateral element: In this kind of element, the 8 nodes are used. i.e. 4 nodes at the each corner & 4 at mid of the each side.

The quadratic functions are used to define the sides of quadratic elements. These kinds of elements are more appropriate to fit the curved type shape which is responsible for accuracy of the results for stress intensity factors. The quadratic function is passing through the nodes along with sides which satisfy the compability equations & also the boundary conditions. This kind of meshing element is also called as higher order elements. The benefit of this higher

All Rights Reserved, @IJAREST-2015

order element is that the shape function can easily represent the stress variation accurately [1],[4]

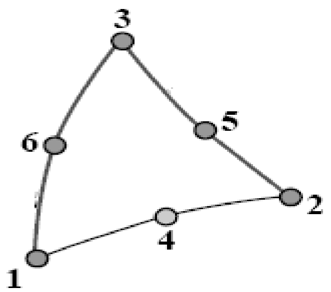

Figure 4 Triangular type quadratic element

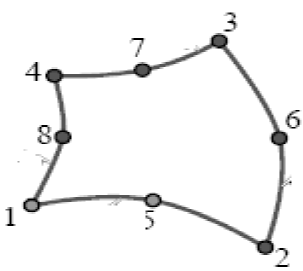

Figure 5 Quadrilateral type quadratic element

III Finite element analysis treatment of Fracture problems:

- It is very essential to use maximum numbers of elements relatively small elements with maximum nodes in the area of the stress concentration to solve the changing stress fields. In most of the analysis the quadrilateral types elements are used for 2 dimensional problems.[5],[2]
- The elastic stresses are always singular at the crack tip. In this course work the tri element is used in the region of the smallest circle because the quadrilateral elements are collapsed into the triangles occupying the same position which gives good numerical results of stress.
- The reason behind the use of the Quadrilateral type meshing is that it gives accurate results than the triangular type meshing as quadrilateral elements consist of more elements which provides accurate results.[5],[2]
- The node at the mid side of the quadratic elements always shift by quarter i.e. 0.2 to the quarter points near to the crack tip in order to satisfy the singularity of the stress for accurate results..
- Individual work: In the individual work, the total number of contours taken into account is six which is responsible for accuracy of the results for SIFs. The triangular type is considered in small circle as explained above that the quadrilateral element is collapsed into the same & occupy the same points in the space. This is appropriate to fit the curved shape & resulting accurate stress results. As the number of contour increases, the accuracy of the results for stress also increases..

Group work: In the group work, the number of contours increases case wise to see the effects of the stress concentration. The small contour is taken tri element & rest contours considered quad by choosing the plate area as tri & quad elements alternatively to see the impact on the SIFs at the crack tip.

The stress intensity factor (SIF) is non- dimensanalised as below for mode I.

$K_I = KI / \sigma(\pi a)^{\wedge 1/2}$

Where,

K_I = The non-dimenstionalised stress intensity factor.

KI = The stress intensity factor in Pa.m$^{1/2}$.

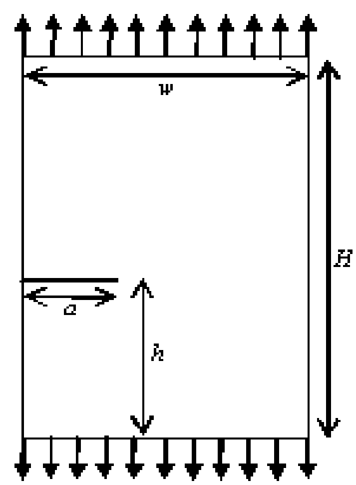

Figure 6 plate

PROCEDURE:-

STEP 1- PART – Click on create part, create part window is open :-

- Enter the name of the part
- Select 2D planar.
- Select Deformable and shell
- Enter the approximate size as per the model and click on continue…

Click on the create lines:- rectangular, plate shape can be drawn by using the entering the co-ordinates of two opposite corner points of the rectangle. After the completion of the drawing of the plate Click done at the bottom of the drawing window[3]

STEP-2- PROPERTY- Click on create material

- Enter the name of the material
- Click on the mechanical and choose elasticity- elastic.

All Rights Reserved, @IJAREST-2015

- Enter material property than click ok
- For, steel E= 200 e9 and v= 0.30
- For, aluminium E= 70 e9 and v=0.33
- For, copper E=120 e9 and v= 0.36

Click on create section – enter a name – choose solid-homogeneous- and click on continue …
Click on assign section-select the part-click done

STEP -3- ASSEMBLY- Click on instance part-a new window open –choose independent

STEP-4- STEP – Click on create step- enter a name-select static, general.-click on continue.

STEP-5- ASSEMBLEY- Now we need to draw the crack line: click on partition face: sketch click on create lines: connected- enter the co-ordinate of the start point of the crack and press enter, than enter the end co-ordinate of the end point of the crack and enter – esc from the command and click on done [3]

STEP-6- INTERACTION – Click on special-choose crack and click on assign seam-select the crack line and click on done. Click again on special, choose crack and click on create: enter the name click on continue..
Select the crack front by clicking on the crack tip point. Click on done. Click on q-vectors; enter the start point and end point of the q – vectors , a new window opens –click on singularity tab-enter 0.25 for midsize node parameter –select collapsed element side ,single node.

STEP 7- STEP- Click on history output manager –click on edit-a new window opens: click on the domain and choose contour integral – enter number of contours –select stress intensity factors.

STEP 8- LOAD- Click on create load: enter the name –click on the pressure-click on continue – select the top and bottom boundary of the plate-click on done –enter the magnitude of the load.

STEP 9 – MESH –click on the partition face: sketch. If geometry of the plate is not aligned with the grid, then click on sketcher option icon. Remove the tick from auto in front of 'grid spacing'. Type appropriate grid spacing for align the model. Using create line rectangle – click on bottom left corner and draw a one unit square. Click on ' linear pattern ' icon and select right and top edge - click on done – new window opens using spacing and number of boxes do the partition.
Select four squares near the crack tip and draw the six circles cantered at the crack tip and draw the two diagonal lines through the crack tip as in fig. 7

STEP-10 MESH – click on seed part instance – enter the approximate global size
For, individual task
Click on assign mesh control- select whole plate by dragging cursor around the drawing-click on done-select quad. And ok – select element around the crack tip –click done-select tri and ok.
For, group task
Click on assign mesh control- select whole plate by dragging cursor around the drawing –click done and select quad or tri as per the case and ok. Select element around the crack tip –click done and select tri and ok.
Click on assign element type –select whole plate-click done
Select the following option: - standard (in element library), quadratic (in geometric order),plane stress (in family),and unselect reduced integration. Click on tri tab and unselect modified formulation.
Click on mesh part instance and click on yes once prompted the plate is mashed.[3]

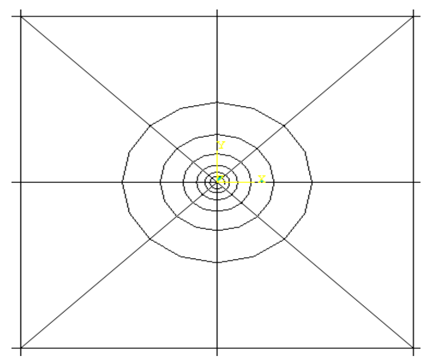

Figure 7 meshing element of plate

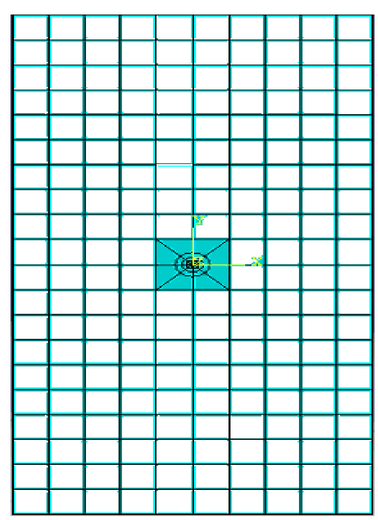

Figure 8 meshing of plate

STEP-11-JOB- click create –enter the name –click on continue and submit the job.

STEP-12 VISULIZATION- To view the stress intensity factors, click on Results in the menu bar & click on History

All Rights Reserved, @IJAREST-2015

Output. A new window will open. Select the stress intensity factor for mode I & II & plot them for each contour.

- Note down the values of stress intensity factors for each contour & prepare the table for different case as given in the course work for further study /analysis of the results

IV RESULT

By using finite element modelling aspects of the problem twenty four different cracked plate models have been generated and analyzed by Abaqus finite element analysis software

In this paper we are taking only one sample for our understanding.

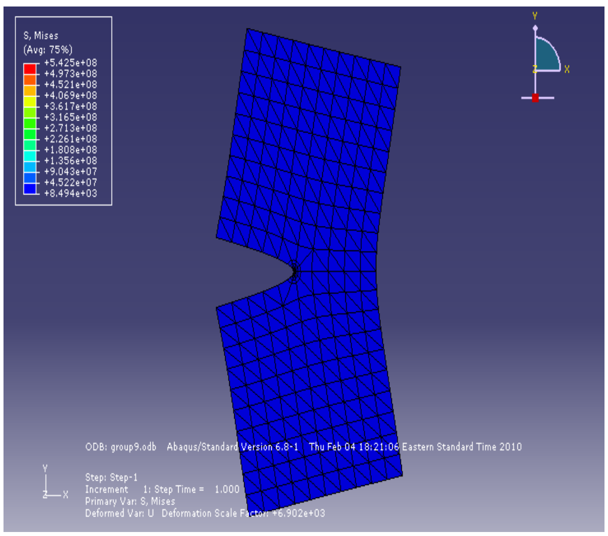

Figure 9 Case 1 Fe analysis

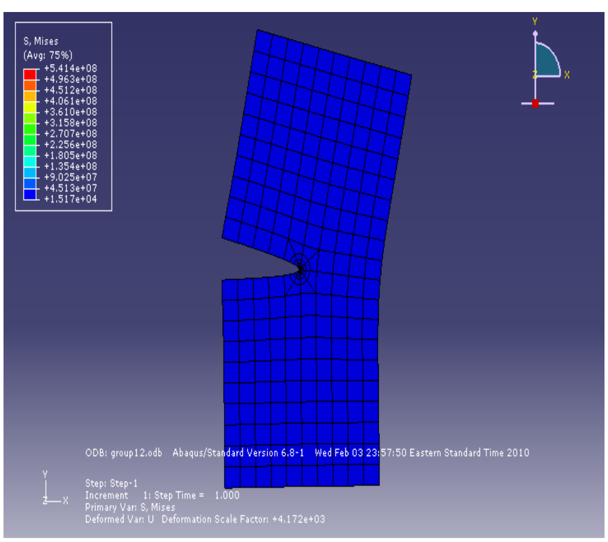

Figure 10 Case 2 Fe analysis

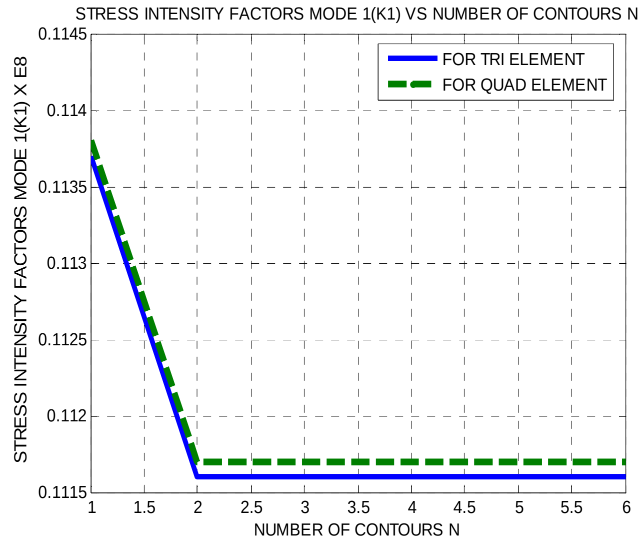

Figure 11 Graph for Case 1 and 2 Fe analysis

DISCUSSION: - By observing the above results, the discussion can be seen as follows.

- The magnitude of Stress intensity factor (K1-tensile mode) is high at the crack tip and becomes constant as we moving far away from the crack tip. Above graph will support the discussion.
- In FEM we are trying to reach nearly accurate solution of our problem. The accuracy of our solution (stress intensity factor) depends upon meshing. According to above result, if we increase the number of contours than accuracy of our solution will be high & the same can be compared from the above case one to twelve.
- In our case we are using the tri element nearest to the crack tip. And changing the element type rest of the plate to find which element type gives more accurate result.
- Tri and quad are element types which give the nearly accurate result but use of element type is depending upon component geometry.
- In our case the accuracy of quad type element is high compare to tri type element. Above result will support the discussion.

All Rights Reserved, @IJAREST-2015

Table :1 FE analysis Result

Case 1 - Element type -Tri						
Number of contours	1	2	3	4	5	6
K1(**Pa.m**$^{1/2}$)	0.1137E+08	0.1116E+08	0.1116E+08	0.1116E+08	0.1116E+08	0.1116E+08
KI non Dimensionalised	2.82					
K2(**Pa.m**$^{1/2}$)	4773.	4680.	4681.	4679	4679.	4697.
Case 2- Element type -Quad						
K1(**Pa.m**$^{1/2}$)	0.1138E+08	0.1117E+08	0.1117E+08	0.1117E+08	0.1117E+08	0.1117E+08
KI non Dimensionalised	2.82					
K2(**Pa.m**$^{1/2}$)	-19.22	0	0	0	0	13.94

V CONCLUSION

By observing the all FE results, the discussion of the conclusion can be seen as follows.

- The magnitude of Stress intensity factor(K) depends on the Size, shape, location, and orientation of the crack.
- The magnitude of Stress intensity factor(K) depends on the loading condition and the Size and shape (Geometry) of the part.
- The magnitude of Stress intensity factor(K) is independent of the component/plate length and material property.
- The magnitude of Stress intensity factor(K) is high at the crack tip and becomes constant as we moving far away from the crack tip.
- In FEM we are trying to reach nearly accurate solution of our problem. The accuracy of our solution (stress intensity factor) is depends upon number of contours means meshing.
- Tri and quad are element type which gives the nearly accurate result but use of element type is depending upon component geometry.
- We obtain solution of our problem using Abaqus (finite element analysis software), LEFM and aspect of finite element method.

REFERENCE

[1] Fenner.R.T (1986). Engineering Elasticity(1st).USA:John wiley & sons.

[2] Dr. S O Oyadiji (2008-2010) point out that 'Fracture mechanics' hand out (p.5).UK.

[3] Estekanchi. H.E. .(1999) A parametric finite element study of cracked plates and shells ,fracture mechanics paper,Sharif University of Technology, Iran

[4] Timoshenko,S. and Goodier, J.N.(1951). Theory of Elasticity(2nd).USA: McGraw-Hill

[5] Perez.N (2004), Fracture Mechanics (USA):Kluwer academic Publishers.

[6] [Material property] last accessed February 6th,2010,from http://www.efunda.com/materials/common_matl/common_matl.cfm?MatlPhase=Solid&MatlProp=Mechanical

All Rights Reserved, @IJAREST-2015