International Journal of Innovative Research in Science and Engineering
Vol. No.2, Issue 03, March 2016
www.ijirse.com
ISSN: 2454-9665

COMPUTATIONAL STRESS INTENSITY FACTOR OF BRASS PLATE WITH EDGE CRACK USING CINT COMMAND

Rajendra Kattimani [1], V.S.Konnur[2]

[1,2]*Assistant professor* , Mechanical *Engineering Department,*
V.P.Dr.P G Halakatti CET Vijayapur, Karnataka, (India)

ABSTRACT

The finite element modelling (FEM) for evaluation of' stress intensity factor for brass plate with edge crack an using CINT command' and for 'Sent specimen' has been investigated in this study, FE modelling is performed through which the behaviour of the material is understood at different loads an data different crack length of the specimen. Here stress intensity factor KI and the stress intensity factor KII are determined. The theoretical and ANSYS values are compared with the target solution provided in the NEFMS benchmark problems. The stress intensity factor elegantly the crack characterizer and for mode one, two, three, the stress intensity factor is expressed as KI, KII, and KII. As investigation is closer to the crack tip, often implies minute details to understand how elastic materials deform and dislocation behaviour. The project work represents the study of stress power variable of 'metal Plate with edge split utilizing CINT command of a sent specimen in various different loading condition sand similarly the study of stress intensity factor for 'rectangular section of a specimen in various loading condition under different 'crack length' using the target solution is performed

Keywords: *FEM,CINT*

I. INTRODUCTION

In these days minute details are focused more to know big things, the crystalline materials as they deform plastically, dislocations are to be understood first, as the diameters of the dislocations are very small they can be viewed in microscope, material under exist with crack front as similar to line, which runs from one region from to the form of another. So crack tip vicinity information with interesting magnitudes of the stress is extremely high component.

The wooden stick breaks at the notch by bending. High stress is created through the notch, in turn notch tip easily moved, the displacement field oranxiety is to be known in the region of split tip.. A designer who is going to modify some of the features as like notch, keyways, cut outs, so as to minimize the stress needs this minute information. An experimentalist measure strain or the stress by characterizing crack, the analysis of stress parameter is defined for crack through stress intensity factor (SIF).

Fracture mechanics is concerned with mechanics to study propagation of crack in the material, and uses analytical method (solid mechanics) to calculate material resistance to fracture. There are three modes to enable a crack by applying forces in three ways, 1)Tensile stress

2)Shear stress

3)Shear stress acting parallel to plane of crack.

The stress intensity factor K is specifically corresponding to the connected burden on the material that enhances the greatness of the connected anxiety which incorporates geometrical parameter (burden type).The KI can be exactly decided, for a sharp break to be made in a material at opening mode. KII can be connected to sliding method of a split. KIII is the stress force variable at tearing method of the specimen. Usually mode I assume a prevailing part in numerous building applications and is considered to be generally hazardous. However in specific applications, parts fall flat through the dominant parts played by mode II or mode III.

The limited component displaying (FEM) for assessment of anxiety force element for an 'edge split point and for sent example' has been examined in this study, FE modelling is performed through which the behaviour of the material is understood at different loads and at different crack length of the specimen. Here anxiety power component KI and the stress intensity element KII are resolved. The theoretical and ANSYS values are compared with the target solution provided in the NEFMS benchmark problems. The stress intensity factor elegantly the crack characterizer and for mode one, two, three, the stress intensity factor is expressed as KI, KII, and KII. As investigation is closer to the crack tip, often implies minute details to understand how elastic materials deform and dislocation behaviour.

The project work represents the study of K intensity factor of edge break point

using CINT command ‟of a specimen in various loading conditions for different „crack angles and similarly the study of stress intensity factor for „v-notch‟ of a specimen in various loading conditions under different „crack length‟ using the target solution is performed. Break is characterized as the division of a material into two or more parts affected by anxiety. Break in greater part of the cases is started by a little split which forms into a noteworthy partition to bring about disappointment.

A little break can influence the capacity and unwavering quality of the structures and segments generally. It along these lines turns out to be vital to contemplate the break's impact on the basic honesty. factor(K) is a standout amongst the most well-known parameter which is utilized for this undertaking. Yet, one of the constraining elements of the anxiety power component is that, it is pertinent in the cases in which the material shows just flexible conduct in the break tip locale.

Subsequently SIF(K) can be utilized as the important parameter just in the event of the split happening in fragile materials.

In bendable materials the flexible locale is joined by the plastic's vicinity area close to the split tip zone. Henceforth it shows versatile plastic conduct which can't be spoken to by the SIF. An alternate break mechanics parameter is to be utilized. J- necessary is one such parameter which is broadly utilized. It considers both flexibility and versatility present around the break tip. It can likewise be utilized as a part of situations where just versatile conduct is watched.

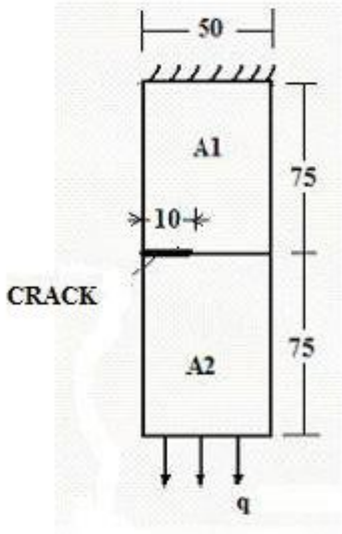

Fig 1: Plate of edge crack specimen

In this project ANSYS software is used for analysing the stress intensity factor KI and KII for the rectangular specimen with the crack at an angle, the typical propagation from an initial crack is a combination mode 1 and mode 2, but the crack tends to propagate normal to the applied stress, resulting in pure mode 1 loading. The typical scenario for an angle crack, when fracture occurs, the crack tends to propagate orthogonal to the applied normal stress that is mixed mode crack becomes mode 1 crack and the propagation of the crack seeks the path of least resistance and need not be confined to its initial plane.

If the material is isotropic and homogeneous the crack will propagate in such a way as to maximize the energy release rate, which follows in an evaluation of energy release rate as a function of propagation of direction in mixed mode problems. Only mode 1 and mode 2 are considered here, but the basic methodology can, in principle, be applied

to a more general case where all three mode‟s are present, this analysis is based on similarwork

II. BRASS MATERIAL (CU70ZN30, ANNELEAD)

Metal is typically the first-decision material for a significant number of the parts for gear made in the general, electrical and exactness designing commercial ventures. Metal is determined due to the exceptional blend of properties, coordinated by no other material, that make it crucial where a long, financially savvy administration life is required.

The non specific term „brass ‟ covers an extensive variety of copper-zinc

compounds with varying blends of properties, including: • Strength • Machinability •

Ductility • Wear resistance • Hardness • Color • Conductivity • Corrosion resistance

Brasses can without much of a stretch be thrown to shape or manufactured by expulsion ,

moving, drawing, hot stamping and cool framing. • The machinability of metal sets the standard by which different materials are judged. • Brasses are perfect for an extensive variety of uses. • Brass is every now and again the least expensive material to choose. •

The right decision of metal is essential if producing and working necessities are to be met.

Table1: Mechanical Properties

BRASS (70CU30ZN, ANNEALED)

Density (R ,Mg/m3)	8.4
Young's Modulus (E , GPa)	130
Shear Modulus (G , GPa)	39
Poisson's Ratio (n)	0.33
Yield Stress (S Y , MPa)	75
Breaking strain (E F ,%)	70
UTS (S F ,MPa)	325
Fracture Toughness (K C ,MN m(-3/2))	2400 N mm(-3/2)
Thermal Expansion (A ,10-6/C)	20

III. METHODOLOGY

3.1 Methodology for crack at edge

The evaluation are carried and performed using the ANSYS software 14.5 version, the Material property chosen for this evaluation, is structural, linear, elastic, isotropic material. As it a solid material structural mass, in the library of element it was defined as 8 node plane 183 as element type.

The material model was defined as linear elastic isotropic material^0 young"s^0 ^{0} modulus 207 Gpa, as Poisson"s ratio of 0.3, as the angle of the crack 22.5 , 67.5 , 90

was chosen, the loading for uniform stress was chosen as 100N/mm², the boundary condition as Uy=0 at line AD and Ux=0 at point D.

Using ANSYS the key points are created in active coordinated system, as the coordinates are created as shown in the fig, the key points are joined through the straight lines , these straight lines are further chosen through the no of areas are selected through the commands, as the loading is done through analysis command by activating new analysis command for which an static loading analysis is chosen, further loads are defined as structural, displacement, on lines, on nodes are selected.

As pressure of yy=100 N/mm² Is chosen, further solution of analysis is carried through, new analysis static. And solution control as automatic time stepping ON is chosen as no of sub steps= 10 and max no of =100 is chosen. The solution is carried in the current LS, solution is done, as KI and KII are obtained, the stress intensity factors are compared with the target solutions.

ISSN: 2454-9665

$$K_I = \sigma\sqrt{\pi a}\left[1.12 - 0.23\left(\frac{a}{b}\right) + 10.6\left(\frac{a}{b}\right)^2 - 21.7\left(\frac{a}{b}\right)^3 + 30.4\left(\frac{a}{b}\right)\right.$$

$$K_I = \sigma\sqrt{\pi a}\left[\frac{1+3\frac{a}{b}}{2\sqrt{\pi\frac{a}{b}}\left(1-\frac{a}{b}\right)^{3/2}}\right].$$

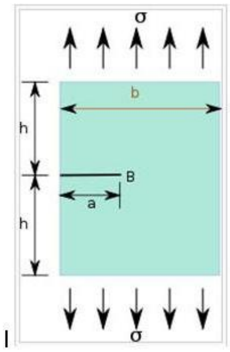

Fig 2:Edge crack in a plate under uniaxial stress

Table-1: Loading points in X and Y direction

Key points #	X	Y
1	40	0
2	0	0
3	-10	0
	-10	0
5	40	75
6	-10	75
7	40	-75
8	-10	-75

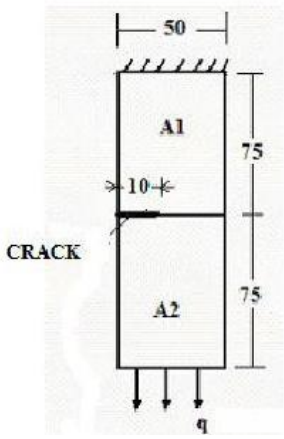

Fig 3: Plate Dimensions

For a plate of dimensions $h \times b$containing an edge break of length , if the plate's measurements are such that $h/b \geq 1$ and $a/b \leq 0.6$, the stress intensity factor at the crack tip under an uniaxial stress σ is

For the situation where $h/b \geq 1$ and $a/b \geq 0.3$, the anxiety force element can be approximated by

Examples of this setup are normally utilized as a part of break durability testing.[[

4.1 Edge split in a limited plate under uniaxial stress.

Key points

The below figure shows the specimen model of the we need to divide the model into two parts as say A1 creating key points in active CS line select area A1 arbitary by lines then elemental type for solid 8 node 183 which means it will saved as type 1 plane 183.

3.2 Model Development

The dimensions of the model are previously calculated and tabulated. The model is created in the software using Key Points in the active co ordinate system. The coordinates of the key-points are calculated and tabulated as shown below.

Table2: key points data

The Fig below shows nodes at crack length (a=10mm), as the nodes have gone under Displacement due to initiation of the crack, which help in determining the displacement of nodes

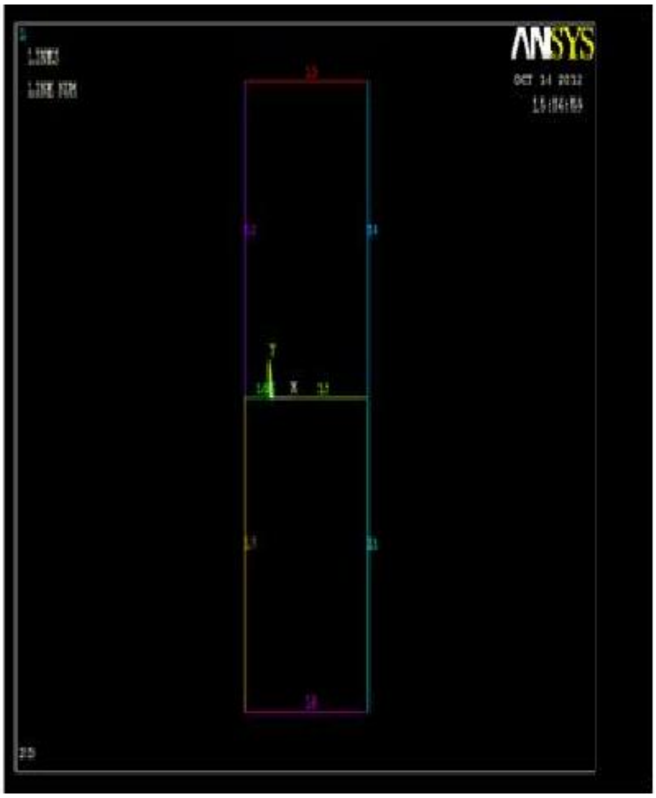

Fig 4: SENT specimen key points

International Journal of Innovative Research in Science and Engineering
Vol. No.2, Issue 03, March 2016
www.ijirse.com
ISSN: 2454-9665

3.3. Creating lines

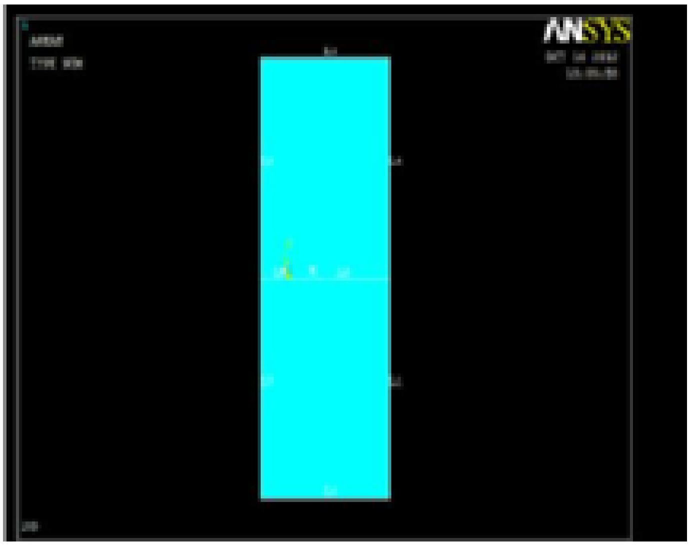

Fig 5: Lines on Plates

Using ANSYS the key points are created in active coordinated system, as the coordinates are created as shown in the above fig.

The key points are joined through the straight lines , these straight lines are further chosen through the no of areas are selected through the commands.

3.4. Creating areas

This figure shows that the areas are created through lines this comes in analysis procedure then after the creating key points and then through this area again it will going to applying for the load on it then it changes to an resulting crack on it as we can see in next slide figure.

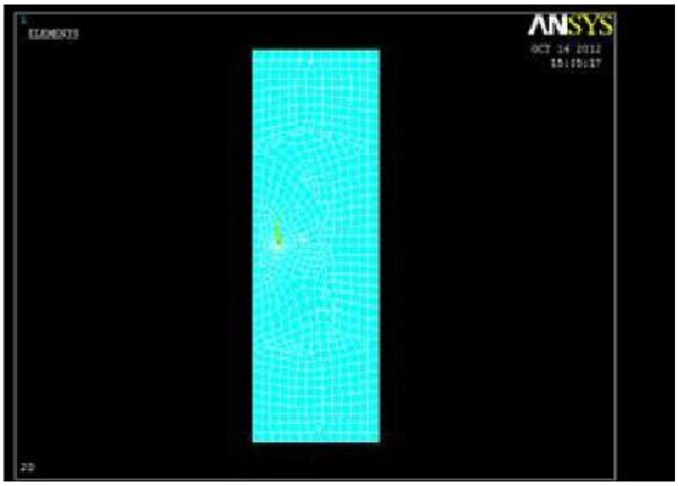

Fig 6: Areas on plates

3.5. Meshing

So far in this step as we discussed in the above two steps after creating the lines through key points after that select the whole lines through which area is going to come for he specimen. The next step is to be meshing part can be done in this step which is necessary for the analysis work of model.

We are having the command in ansys software go to meshing select the model area and click on it which results in meshing this can be done likely as fallows meshing – mesh tool – line - set- select the above 75 mm line and say ok. No. of element division say 10 and den enter the command. Go back to mesh tool , line – set for 50 mm line den click OK. Give no. of elements 10 to it den enter. Lastly mesh tool say quad den free mesh select area say ok meshing is done as shown in above figure.

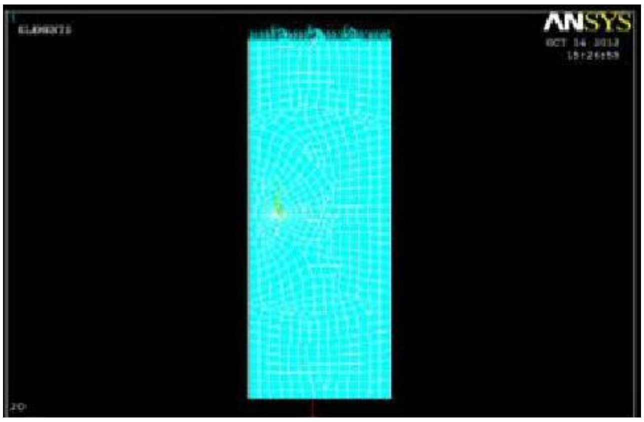

Fig 7: Meshing

3.6 Applying Loads and Boundary Conditions

Next part is the only applying loads and the boundary conditions go to command take load define loads n applying loads on the model say structure displacement as we can see in above figure.

As the loading is done through analysis command by activating new analysis command for which an static loading analysis is chosen, further loads are defined as structural, displacement, on lines, on nodes are selected. as pressure ofyy=100 N/mm² Is to be chosen .symmetric boundary condition on the selected line 40 mm pressure of amount 230Mpa on the line 50mm top line click as OK give load pressure value as -230. al propertiesThe evaluation are carried and performed using the ANSYS software 14.5 version, the Material property chosen for this evaluation, is structural, linear, elastic, isotropic material. As it a solid material structural mass, in the library of element it was defined as 8 node plane 183 as element type.

The material model was defined as linear elastic isotropic material young‟s modulus 207 Gpa, as Poisson‟s ratio of 0.33,

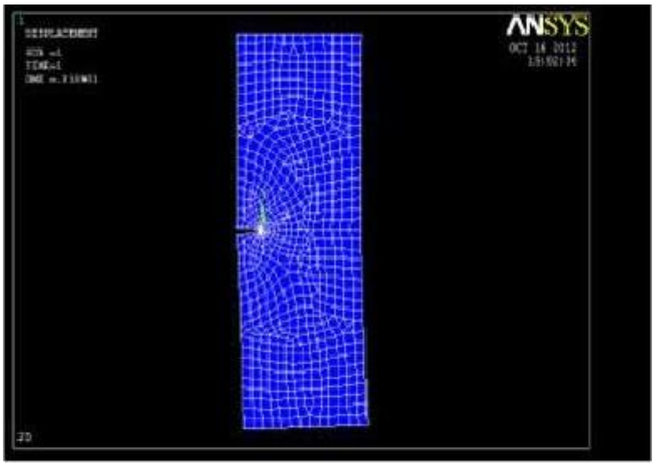

Fig 8: Stress distribution

3.7 Deformed Shapes

As far the analysis work has been done after applying loads of 100N/mm2 and B.C on 40mm line the resulting specimen model gets deformed in shape as gets cracked as shown in above figure.

New analysis can be done with static remember that this project we are going to solve with CINT commands using Ansys all these commands copied in the upper command front.

```
NSEL,S,LOC,X,0

NSEL,R,LOC,Y,0


CM,CRACKTIP,NODE          ! DEFINE CRACK TIP NODE COMPONENT

ALLSEL,ALL

CINT,NEW,1   ! DEFINE CRACK ID

CINT,TYPE,SIFS   ! DEFINE CRACK TYPE

CINT,CTNC,CRACKTIP        ! DEFINE CRACK TIP NODE COMPONENT

CINT,SYMM,ON  ! SYMMETRY ON

CINT,NCON,6   ! NUMBER OF COUNTOURS

CINT,NORM,0,2
```

CINT COMMANDS

KSCON 1,0.002,0.8

KSCON,1,0.002,0,8

These commands which are mentioned in above should copied drag into upper command front this can be done in pre-processor after executing these CINT commands crack propagation takes place so far analysis is done we need to determine the stress intensity factor for material model

For the post processor we use CINT command as PRCINT,1,1,K1

Table -2: Geometric properties of model

Material Properties	Geometric Properties (Dimension as shown in Fig. 3.10)	Loading
E = 70 GPa	a = 5 mm	σ = 100 MPa
v = 0.3	b = 25 mm	
$K_I = 35\ MN/m^{3/2}$	h = 25 mm	
	Thickness (t) = 5 mm (3-D) (Fig. 3.12)	

The basic 2-D model utilizing PLANE82 is made via programmed (free) work era. The PREP7 KSCON order, which allocates component division sizes around a key- point, is especially valuable in a 2-D model to make 2-D split tip components with nodal peculiarity. This charge uproots the nodal peculiarity at the break tip. POST1 is utilized to get break mechanics stress force component (KI) by dislodging extrapolation (KCALC charge). The measurement of 2-D demonstrating is appeared beneath with hubs.

International Journal of Innovative Research in Science and Engineering
Vol. No.2, Issue 03, March 2016
www.ijirse.com
ISSN: 2454-9665

In break mechanics, the KI worth speaks to the imperviousness to disappointment. This anxiety power calculate that is given comparison (1.1), is connected with the geometry of demonstrating, the split size and the connected anxiety.

In centre focus break example, the geometry variable "Y" is 118.1)2/sec(==baYπ ASTM Standard E 647 (ASTM, 95) (3.1)

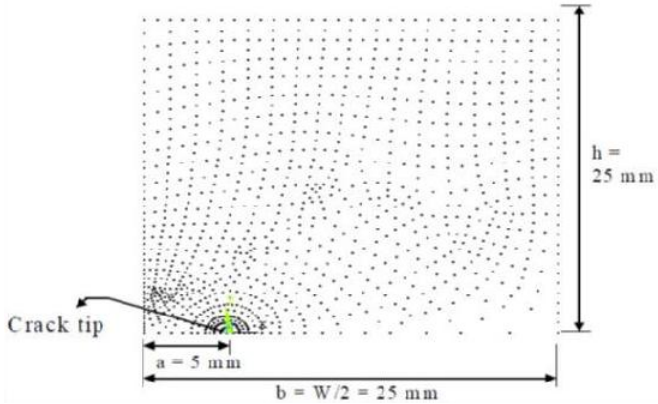

Fig 8: Dimension of finite element modelling

The connected anxiety (σ) is 100 MPa, the break measure (an) is 5 mm and the half width (b) is 25 mm. Henceforth, the figured anxiety power variable at a middle split example (limited width plate) is 14 MN/m3/2

Figure indicates: Dimension of finite element modelling

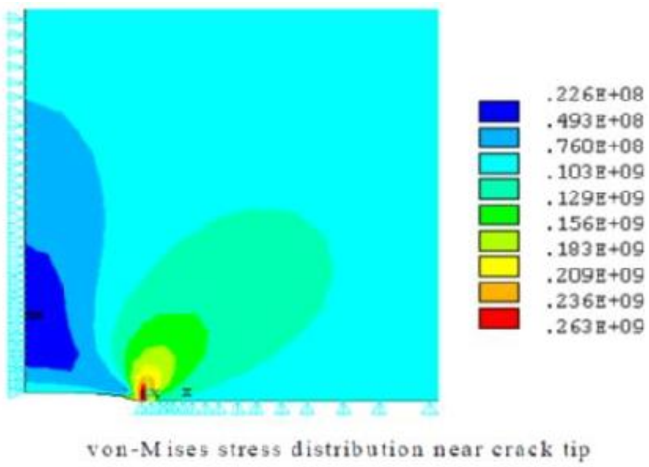

Fig 8: Von mises stress distribution

IV. RESULTS AND DISCUSSION

(1) 2-D modelling

Figure 3.13 von-Mises stress dispersion close to the split tip in 2-D displaying and KI = 11.96 MN/m3/2

Actually, the split tip components to assign peculiarity ought to be made by large scale documents to uproot the peculiarity at the break tip in 2-D displaying. In any case, in this study, peculiarity point at the split tip was not uprooted. At the break tip, the components are incoherent.

International Journal of Innovative Research in Science and Engineering
Vol. No.2, Issue 03, March 2016
www.ijirse.com
ISSN: 2454-9665

The reason is that the similarity couldn't be kept up at the break tip. Since the split tip components are not doled out to evacuate the peculiarity at the break tip in 2-D demonstrating. This outcome demonstrates the peculiarity's significance at the split tip component.

Go to plot results select the option deform + undeform shape then counter plots –nodal solutions we will get the von-mises deform shape results for stress distribution near crack tip as shown in above figure

The problem is solved first using the two-dimensional PLANE183 element with plane-strain behaviour. After the pre-process (/PREP7) part of the APDL code, the crack- tip node or nodes can be introduced in the solution part (/ SOLU) with CINT command as follows: CINT,NEW,1 CINT,TYPE,MFOR CINT, CTNC, C Node Comp CINT, N CON,NumCon CINT,NORM,0,2 CINT,SYMM,OFF Here, CNodeComp is the node component at the break tip and NumCom is the integration contours.

To get the results, PRCINT command can be used as PRCINT,2,,MFRX PRCINT,2,,MFRY Since the crack normal is introduced as the second component of the

global basis vector with the command CINT,NORM,0,2, the first command gives the tangential component of the material force, whereas the second one is the normal component to the crack surface. Since the problem is two-dimensional, CNodeComp includes only the crack-tip node. In Figure 2.2, von Mises stresses and nodal material force vectors are shown. High stress concentration occurs around the crack tip.

Since there is a r-0.5 stress singularity at the crack tip, the discrete material forces mainly appear in the crack-tip domain. The tangential component of the material-force vector to the crack surface presents the required energy to create a new crack surface. In other words, if the energy release rate or crack-driving force are large enough to overweight fracture toughness of the material, the crack will propagate in the opposite direction of the material force as it is introduced in brittle fracture theory.

The space assessment can be got by considering a resultant configurationally power vector of an impact area that encompasses the split tip. In ANSYS software, this procedure can be accomplished by contours. The user-defined contour, by the command CINT,NCON,NumCon, represents domain evaluation.

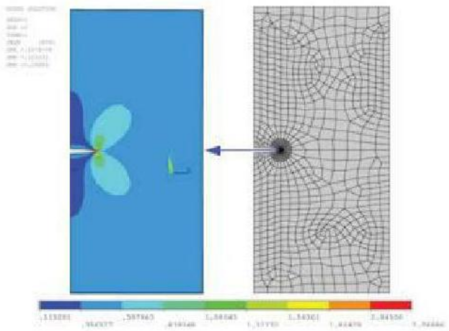

Fig 9: Two-dimensional central crack specimen: Von Mises stress (left) andnodal material-force distribution (right)

The first contour presents the individual Figure 2.2. Two-dimensional central crack specimen: VonMises stress (left) and nodal material-force distribution (right) 13 Material Forces material force at the crack tip node where subsequent contours take the resultant material-force vectors from the neighboring nodes into the calculation. As shown in Table 2.1, the crack-tip material force, which is equal to the first domain result from the material-force calculation by ANSYS software, gives more than 90 percent of the total energy release rate for this example.

In addition to the single-edge crack problem under plane-strain condition, the same problem is studied for the three-dimensional case. The same APDL commands are used as explained above. The nodal von Mises stress contour and nodal material force distribution can be found in Figure 2.3. Results are presented in Table 2.2 with a comparison to the ANSYS J-integral tool.

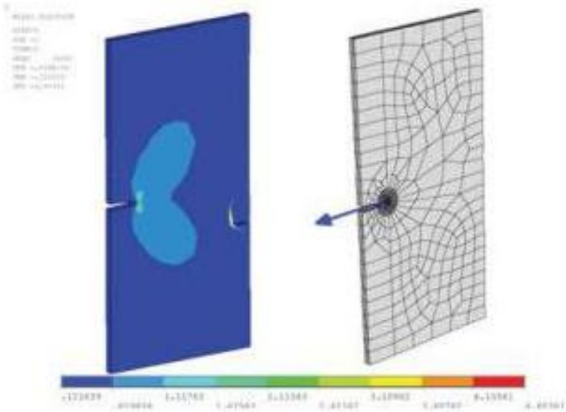

Fig 10:. Three-dimensional central crack specimen: von Mises stress (left) andnodal material force distribution (right)

At the point when the model is pushed over yield point 200 MPa, however the anxiety strain bend ends up being unreasonable. The reason was a bilinear inelastic isotropic solidifying (BISO) bend (a versatile immaculate plastic model) was considered as a material conduct. Really, the BISO(material model conduct) can't be utilized as a part of request to reenact precise rate-subordinate versatility and additionally cyclic solidifying or softening. In this manner, cautious consideration ought to be given to choices like component sort, material model conduct. Further work should be did later on. Keeping in mind the end goal to recreate the genuine break development, network at the split tip should be re-fit at whatever point the break progresses along the split plane. In Figure 3.6, the yield has happened at the break tip because of the anxiety fixation when over-burden of 70 MPa was connected. In this way, the plastic disfigurement happened at the break tip. This plastic distortion at the break tip was crushed after over-burden and after that, compressive remaining anxiety field has been actuated at the split tip.

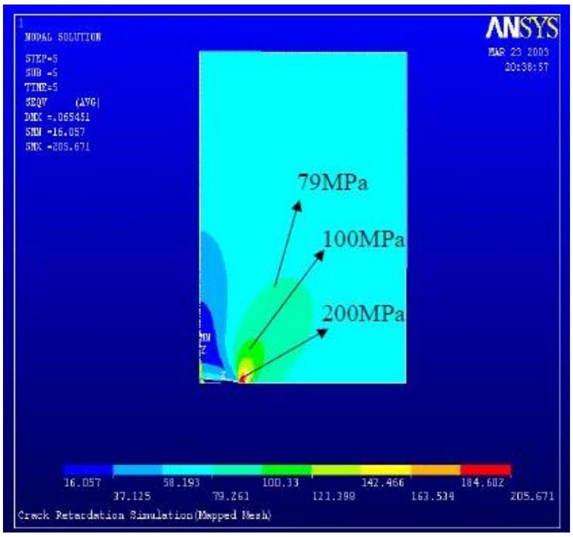

Fig 11:indicates von-Mises Stress amid over-burdening (after burden step 5)

V. CALCULATIONS

By theoretical Method :

K, = α σ √πa α = 1.12 - 0.23(a / w) + 10.55 (a / w)2 - 21.71 (a / w)3 + 30.38 (a / w)4

where α = break geometric connection variable

σ = Pressure =crack length

w = testing material width

α = 1.12 - 0.23(10/50)+10.55(10/50)2 – 21.71(10/50)3 + 30.38(10/50)4 = 1.12-0.23(0.2) + 10.55(0.2)2 - 21.71(0.2)3 + 30.38(0.2)4 = 1.12-0.046+0.422-0.17368+0.048608 = 1.590-0.21968 = 1.37002 K, =1.37002 * 230 √π*10

K,= 1289.147 N\mm(-3/2)

K=Yσ√πa

Y=1.118, a=10

K=1.118*230√π*10

K=1441.26Nmm.

Table-3:Specified Loads At Different Crack Length

SIF	AT A LOAD OF 50 N/mm² & CRACK LENGTH				AT A LOAD OF 150 N/mm² CRACK LENGTH			
	a=5	a=10	a=15	a=20	a=5	a=10	a=15	a=20
K_I	554.46	603.29	650.76	695.09	1663.3	1809.87	1952.3	2085.27
K_0	198.16	280.24	343.23	396.33	594.49	840.73	1029.69	1188.99
K_I/K_0	2.79	2.15	1.89	1.75	2.79	2.15	1.89	1.75

SIF	AT ALOAD OF 200 N/mm² CRACK LENGTH			
	a=5	a=10	a=15	a=20
K_I	2217.8	2413.15	2603.05	2780.37
K_0	792.66	1120.98	1372.92	1585.33
K_I/K_0	2.79	2.15	1.89	1.75

The above$_0$ table gives the result of stress intensity$_0$ factor for the corresponding values of KI, K and Normalized SIF mode1 i.e. KI/K at different crack
length (a=5, 10, 15, 20) for different load conditions of 50 N/mm², 150 N/mm² and 200 N/mm².

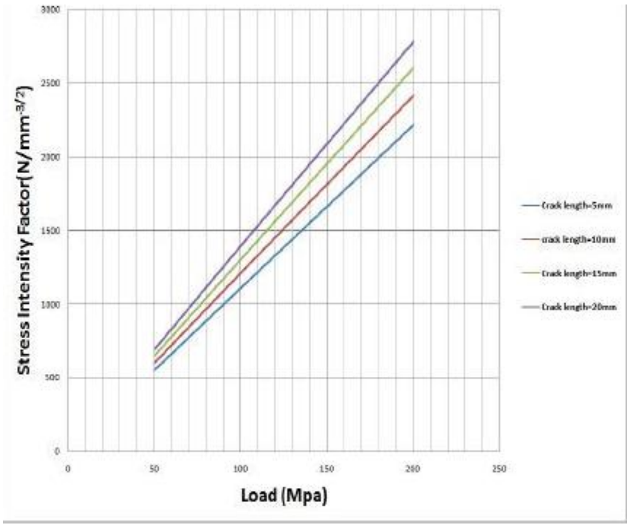

Chart-1:SIF KI versus load. The variation of stress intensity factor KI

plotted for different load conditions for different crack length i.e. for 5mm, 10mm, 15mm, & 20mm respectively, and the load as varied from 50, 100, 150 & 200 (Mpa).

The graph clearly shows the variation is linear and the stress intensity factor (SIF) increases with an increase in load.

VI. CONCLUSION

The variation of stress intensity factor KII is also studied and it is observed that variation is linear for all load condition with negative slope.The graph clearly shows the variation is linear and the stress intensity factor (SIF) increases with an increase in load and this result we are getting as comparing with j-integral technique.

REFERENCES

[1] ABACUS 6.12 BENCHMARKS MANUEL 2012

[2] T.L.ANDERSON, TAYLOR AND FRANCIS 1989: International Standard Book Number-13: 978-1-4200-5821-5.

[3] RAFAEL PICON & FEDERICO PARIS 1995: fracture mechanics

[4] Zejin Xu, Yulong Li Impact of stacking rate on mode 1 element break durability of high steel Vol 42(9) pp 965-970.

[5] F. A. DIAZ, E. A. PATTERSON, P. SIEGMANN 2010, A novel experimental approach for calculating stress intensity factor from isochromatic data , experimental mechanics vol 52(2) : pp 273- 282.

[6] ENGINEERING FRACTURE MECHANICS 1988: 'The impact of geometry of edge gruff indented plates on split spread under blended mode stacking conditions' vol 30(5)pp547-733

[7]. Ansys 13.0 Theory and Reference Manual, 2010.