

# Development of an Automatic Crack Propagation Mechanism in ANSYS

Mandar M. Bidwe<sup>1</sup>, Rahul A. Darandale<sup>2</sup>

<sup>1</sup>Assistant Professor, Department of Mechanical, Dr. Vitthalrao Vikhe Patil College of Engineering, Ahmednagar, Maharashtra India.

<sup>2</sup>Assistant Professor, Department of Mechanical, Dr. Vitthalrao Vikhe Patil College of Engineering, Ahmednagar, Maharashtra India.

\*\*\*

**Abstract** - Catastrophic fracture failure of engineering structures is due to cracks which proceed beyond safe size. A crack always exists in all structures up to some extent. Cracks are due to local damage or manufacturing defect. Growth of cracks is due to fatigue, creep or stress corrosion. Structural strength gets decreased due to crack growth. Thus when the service loading cannot be barred by current strength, fracture takes place which leads towards failure of structure. In order to estimate structure durability and damage tolerance it is essential to analyze the crack within structure. By default cracks may exist in structure and hence fatigue life of structure depends upon size and location of crack. Thus in order to estimate fatigue life of any component we should study crack growth. Various FEM based commercial software are used for simulating a cracked structure. This software's are not having capacity to study automatic crack growth tracking capability. In the present work, a method will be developed for automatic crack propagation in ANSYS software. A code will be developed in APDL (ANSYS Parametric Design Language) for automatic crack propagation in ANSYS. The input to the code will be initial crack length and crack location in the structure. The output from the code will be the final path of the crack and life cycle required for the crack length.

**Key Words:** APDL, Fatigue life, ANSYS, FEM, Fracture Mechanics, Automatic crack propagation.

## 1. INTRODUCTION

It has been suggested that 50 to 90 percent of all mechanical failures are due to fatigue, and the majority of these failures are unexpected. Fatigue causes failure in many structures in automobiles, ships and aircraft as well as any other device which undergo repeated loading. In 1978, a comprehensive study indicated a cost of \$119 billion (in 1982 dollars) due to fracture in the United States. This study suggested that using proper and current technology in design could significantly reduce this cost. There are currently many approaches to fatigue design. Some are simple and inexpensive; others are extremely complex and expensive. If initially an expensive complete fatigue design procedure is implemented, this may lead to lower cost in the long run by reducing failure. Proper characterization of fatigue behavior can lead to the more competitive products. In the aircraft and automotive industries this can mean lighter structures. Fatigue life of a component can be divided into two parts:

1. Crack initiation
2. Crack propagation to critical crack size.

Catastrophic fracture failure of engineering structures is caused by cracks that extend beyond a safe size. Cracks, present to some extent in all structures, either as a result of manufacturing fabrication defects or localized damage in service, may grow by mechanisms such as fatigue, stress-corrosion or creep. The crack growth leads to a decrease in the structural strength. Thus, when the service loading cannot be sustained by the current residual strength, fracture occurs, leading to the failure of the structure. The analysis of cracks within structures is an important application if the damage tolerance and durability of structures and components are to be predicted. As part of the engineering design process, engineers have to assess not only how well the design satisfies the performance requirements but also how durable the product will be over its life cycle [1]. Often cracks cannot be avoided in structures; however the fatigue life of the structure depends on the location and size of these cracks. In order to predict the fatigue life for any component, a fatigue life and crack growth study needs to be performed. Damage tolerant methodology can be used for designing initially flawed components or determining the remaining life to failure once a flaw is detected. This is particularly of interest to the aircraft industry, where cracks are commonly detected and monitored in nearly every structural component of the aircraft. Damage tolerance assessment is a procedure that defines whether a crack can be sustained safely during the projected service life of the structure. Damage tolerance assessment is, therefore, required as a basis for any fracture control plan, generating the following information, upon which fracture control decisions can be made:

1. The effect of cracks on the structural residual strength, leading to the evaluation of their maximum permissible size.
2. The cracks growth as a function of time, leading to the evaluation of the life of the cracks to reach their maximum permissible size, from which the safe operational life of the structure is defined.

Linear elastic fracture mechanics can be used in damage tolerance analyses to describe the behavior of cracks. The fundamental assumption of linear elastic fracture mechanics is that the crack behavior is determined solely by the values of the stress intensity

factors which are a function of the applied load and the geometry of the cracked structure. The stress intensity factor, thus play a fundamental role in linear elastic fracture mechanics applications. Crack growth processes are simulated with an incremental crack-extension analysis. For each increment of the crack extension, a stress analysis is carried out and the stress intensity factors are evaluated [2]. The crack path, predicted on an incremental basis, is computed by a criterion defined in terms of the stress intensity factors. Thus, computation stress intensity factor plays a major role in fracture analysis. In the next section, computation of stress intensity factor using FEM has been discussed. Most of the commercially available FE software like ANSYS, ABAQUS and NASTRAN etc. have capabilities to model crack tip and compute the stress intensity factors. However, for the study of crack propagation, the crack tip modeling and computation of stress intensity factor has to been done at each incremental step along the crack length. This requires remodeling of the FE model to update the cracked geometries. Currently none of the above mentioned software has any automatic procedure to do the remodeling procedure. Thus, either this has to be done manually or use any third party software to do the process. The task of manually remodeling the crack tip will make the simulation process very lengthy. Using a third party software will add additional cost as well as learning a new tool. Thus, in this project an automatic procedure has been developed to do the remodeling process. This has been done in ANSYS software using APDL (ANSYS Parametric Design Language) scripting language [4].

### 1.1 Objective

The objective of this paper is to develop an APDL script to automatically model the crack tip and predict the crack propagation path. The project currently focuses on 2D problems. The input to the APDL script will be the initial crack location, crack orientation and crack increment. The

APDL script takes all the above and automatically model the crack tip, compute the stress intensity factors, and predict the crack propagation path.

## 2. FRACTURE MECHANICS

### 2.1 Introduction

Fracture mechanics has developed into a useful discipline for predicting strength and life of cracked structures. It can be divided into two parts: Linear Elastic Fracture Mechanics assumes that the material is isotropic and linear elastic. Based on the assumption, the stress field near the crack tip is calculated using the theory of elasticity. When the stresses near the crack tip exceed the material fracture toughness, the crack will grow. LEFM applies when the nonlinear deformation of the material is confined to a small region near the crack tip. For brittle materials, it accurately establishes the criteria for

catastrophic failure. However, severe limitations arise when large regions of the material are subject to plastic deformation before a crack propagates. Elastic Plastic Fracture Mechanics is proposed to analyze the relatively large plastic zones. EPFM assumes isotropic and elastic-plastic materials [3]. Based on the assumption, the strain energy fields or opening displacement near the crack tips are calculated. Although the term elastic-plastic is used in this approach, the material is merely *nonlinear-elastic*. In other words, the unloading curve of the so called elastic-plastic material in EPFM follows the original loading curve, instead of a parallel line to the linear loading part which is normally the case for true plastic-plastic materials [9]. Fig-1 shows the difference between LEFM, EPFM and true elastic-plastic behavior.

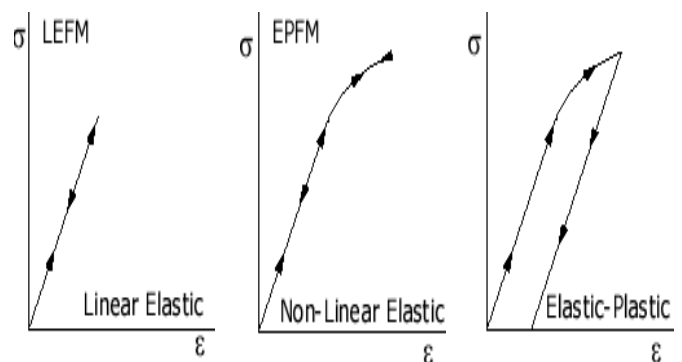


Fig-1: LEFM, EPFM and True Elastic Plastic behavior

Linear elastic fracture mechanics applied to gear tooth has become increasingly popular [4].

### 2.2 Different Fracture Modes

The three basic types of loading that a crack can

experience are shown in Fig-2. Mode I loading is normal to the crack plane, and tend to open the crack. The crack surfaces tend to separate symmetrically with respect to the crack plane.

**Mode II** corresponds to in-plane shear loading and tends to slide one crack face with respect to the other (shearing mode). The stress is parallel to the crack growth direction.

**Mode III** corresponds to out-of-plane shear, or tearing.

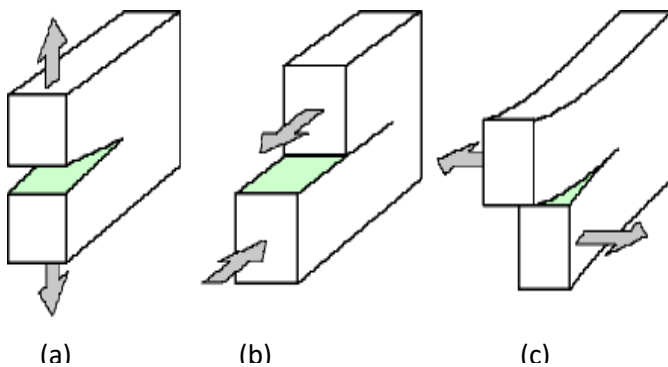


Fig - 2 Type of loading on a cracked body. (a) Mode I. (b) Mode II. (c) Mode III.

### 3. FORMULATION OF CRACK PROPAGATION

#### 3.1 Overview of formulation

An APDL code is developed for automatic modeling of crack tip and crack propagation is developed. Few plate problems defined as a 2-D plane stress problem are taken to validate the code. The code has the capability of automatic modeling of the crack at the user defined location. The code then automatically does the static analysis and then calculates the  $K_I$  and  $K_{II}$  and predicts the crack growth angle. The code then increments the crack tip, in the direction calculated, by an amount defined by the user. This procedure is repeated for the user defined number of steps.

#### 3.2 Software Requirement

The project is carried out using ANSYS as a Finite Element Software. One of the unique features of ANSYS is APDL (ANSYS Parametric Design Language). APDL is a macro language developed by Ansys Inc., to be used in conjunction with ANSYS. The language is similar to any programming language (more like FORTRAN). A parametric model, with all the boundary conditions and load conditions incorporated within, can be developed using APDL. Thus, by changing only the parameter the model changes and the effect of parameter can be studied without bothering to create a new model each time. The language also provides the looping and condition checking facilities which is necessary in many applications. It can also do matrix operations. Only because of these flexibility provided by the language, it was possible to develop the code for automatic crack propagation.

#### 3.3 Modeling of the crack

As we know that, the main difficulty in modeling the crack in FEM is to model the singularity ( $1/\sqrt{r}$ ) behavior of the crack tip. Traditionally a very fine mesh was used at the crack tip to capture the singularity behavior. But this makes the computational time expensive. A special type of

element is, therefore, used in this formulation. This special type of element is known as quarter point element. An introduction to this kind of element is given below [3].

#### 3.3.1 Quarter point elements

As mentioned earlier, one of the major difficulties in fracture mechanics using FEM is the modeling of the singularity ( $1/\sqrt{r}$ ) in the stress field formed at the crack tip. Henshell and Shaw discovered that it was possible to represent the ( $1/\sqrt{r}$ ) stress singularity using quadratic isoparametric elements. Barsoum formulated a quadratic isoparametric element able to represent the ( $1/\sqrt{r}$ ) stress singularity moving the mid side node of the two sides to the quarter point position [8]. A brief formulation of the element is given. Fig-3 shows the quarter point 8-noded element.

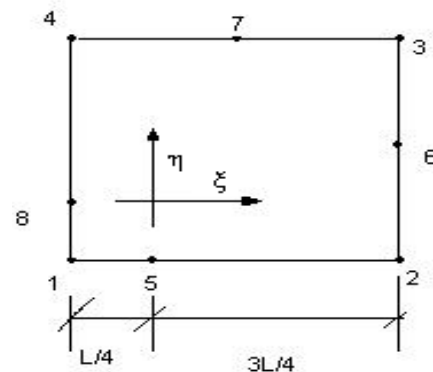


Fig-3 Quarter point 8-noded element

The geometry of an 8-noded plane isoparametric element can be mapped into a normalized square space ( $\xi, \eta$ ), ( $-1 \leq \xi \leq 1, -1 \leq \eta \leq 1$ ) through the following transformation:

$$x = \sum_{i=1}^8 N_i(\xi, \eta) x_i$$

$$y = \sum_{i=1}^8 N_i(\xi, \eta) y_i$$

The shape functions for the three nodes can be written as,

$$N_1 = -\frac{1}{2} \xi (1 - \xi)$$

$$N_2 = \frac{1}{2} \xi (1 + \xi)$$

$$N_5 = (1 - \xi^2)$$

From Equation,

$$x = -\frac{1}{2} \xi (1 - \xi) x_1 + \frac{1}{2} \xi (1 + \xi) x_2 + (1 - \xi^2) x_5$$

Choosing,  $x_1=0, x_2=L$  and  $x_5=L/4$

$$x = \frac{1}{2} \xi (1 + \xi) L + (1 - \xi^2) \frac{L}{4}$$

Thus, 
$$\xi = \left( -1 + 2\sqrt{\frac{x}{L}} \right)$$

And in the Jacobian,  $\frac{\partial x}{\partial \xi}$  is given by:

$$\frac{\partial x}{\partial \xi} = \frac{L}{2} (1 + \xi) = \sqrt{\frac{x}{L}}$$

Which makes the determinant of the Jacobian vanish at the crack tip when  $x=0$

Regarding the displacement along the line 1-5-2:

$$u = -\frac{1}{2} \xi (1 - \xi) u_1 + \frac{1}{2} \xi (1 + \xi) u_2 + (1 - \xi^2) u_5$$

and writing in terms of  $x$ :

$$u = -\frac{1}{2} \left( -1 + 2\sqrt{\frac{x}{L}} \right) \left[ 2 - 2\sqrt{\frac{x}{L}} \right] u_1 + \frac{1}{2} \left( -1 + 2\sqrt{\frac{x}{L}} \right) \left[ 2\sqrt{\frac{x}{L}} \right] u_2 + \left( 4\sqrt{\frac{x}{L}} - 4\frac{x}{L} \right) u_5$$

So, the strain in the  $x$  direction is given by:

$$\begin{aligned} \epsilon_x &= \frac{du}{dx} = J^{-1} \frac{du}{d\xi} \\ &= \frac{d\xi}{dx} \frac{du}{d\xi} \\ &= -\frac{1}{2} \left[ \frac{3}{\sqrt{xL}} - \frac{4}{L} \right] u_1 + \frac{1}{2} \left[ -\frac{1}{\sqrt{xL}} + \frac{4}{L} \right] u_2 + \left[ \frac{2}{\sqrt{xL}} - \frac{4}{L} \right] u_5 \end{aligned}$$

The singularity achieved for the strain filed in above Equation is of order  $1/\sqrt{x}$ , therefore, at the crack tip, the stress is infinite which, is the same singularity achieved beyond the elastic plane analysis. A 6-noded triangular element is shown in the Fig- 4. In this case the singularity is investigated along the  $x$ -axis, where  $\eta=0$ , and for this line we have,

$$x = -\frac{1}{4}(1+\xi)(1-\xi)l_1 - \frac{1}{4}(1+\xi)(1-\xi)l_1 + \frac{1}{2}(1-\xi^2)\frac{l_1}{4} + \frac{1}{2}(1+\xi)l_1 + \frac{1}{2}(1-\xi^2)\frac{l_1}{4}$$

It reduces to,

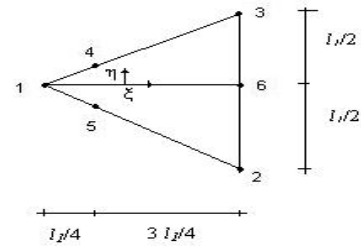


Fig-4 Quarter point 6-noded element

$$x = \left( \xi^2 + 2\xi + 1 \right) \frac{l_1}{4}$$

Thus,

$$\xi = \left( -1 + 2\sqrt{\frac{x}{l_1}} \right)$$

Which is the same singularity achieved in 8-noded element. Quadrilateral quarter point element is used less frequently in practice than the triangular version. These may be because of fewer number of this element being able to place around the crack tip. With fewer elements, the circumferential variations of the stress and displacement fields around a crack tip may be can less accurately represented then the triangular case where more elements can be placed around the tip.

### 3.3.2 Modeling crack tip using Quarter point element

In the present work, 6-noded quarter point element is used to model the crack tip. The APDL macro developed, *clear.mac*, has the capability to automatically model the crack tip with 6-noded quarter point element. The user has to input the crack initiation node and the crack tip location. Most of the time, the crack initiates at maximum principles stress [6]. The procedure is shown below in the Fig- 5. Once the crack tip is remeshed with QP element, static analysis is carried out. The stress intensity factor and propagation angle are found out. Based on these values the crack is incremented by the APDL code *iaclear.mac*.

### 3.4 Extracting SIF using FEM

Displacement correlation technique is used in the present work for extracting the stress intensity factor [6]. As the present work is for 2-D analysis, only  $K_I$  and  $K_{II}$  values have to be calculated. This is done by the macro *sifl.mac*. We can calculate the  $K_I$  and  $K_{II}$  values by using formulae given below. This method was because of the simplicity and less computational time required. The Table -1 shows

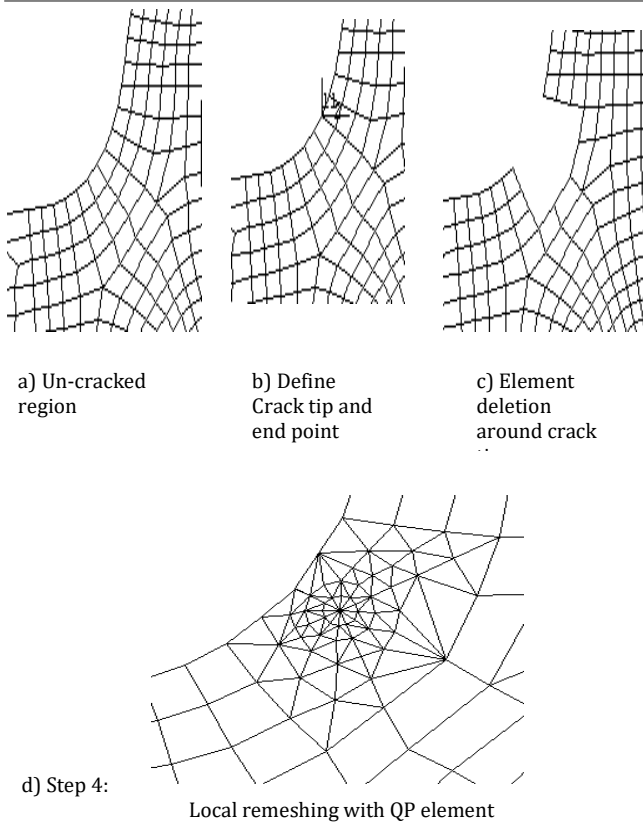


Fig - 5 Automatic Modeling of crack

Nodal displacement of nodes behind crack tip.

$$K_I = \frac{\mu}{\kappa + 1} \sqrt{\frac{2\pi}{L}} [4(v_b - v_d) + v_e - v_c]$$

$$K_{II} = \frac{\mu}{\kappa + 1} \sqrt{\frac{2\pi}{L}} [4(u_b - u_d) + u_e - u_c]$$

$$\mu = \frac{E}{2(1+\nu)}$$

$\kappa = 3 - 4\nu$  For plane stress condition

$\kappa = \frac{3 - 4\nu}{1 + \nu}$  For plane strain condition

For example, after doing the static analysis on the initial crack in, let the nodal displacement values of the element on the crack edge be

Let  $E = 2.1E6$ ,  $\nu = 0.3$  and  $L = 0.02$ . Then,

$$\mu = \frac{E}{2(1+\nu)}$$

Table -1 Nodal displacement of nodes behind crack tip

Node position	UX (mm)	UY (mm)
Crack Tip	0.48331 E-3	0.22029 E-2
Quarter Point (Upper side) <i>b</i>	0.4992E-3	0.23636E-2
End Point (Upper side) <i>c</i>	0.52207E-3	0.25313E-2
Quarter Point (Lower side) <i>d</i>	0.47856E-3	0.20550E-2
End Point (Lower Side) <i>e</i>	0.48373E-3	0.19266E-2

$$= \frac{2.1E05}{2(1+0.3)}$$

= 80769.2307 (Shear modulus)

For plane stress condition

$$\kappa = \frac{3 - 4\nu}{1 + \nu}$$

$$= \frac{3 - 4 \cdot 0.3}{1 + 0.3}$$

= 1.384615

The relative nodal displacement due to the displacement of the crack tip can be calculated as below.

$v_b = 0.23636E-2 - 0.22029E-2 = 0.1607E-3$

$u_b = 0.4992E-3 - 0.48331E-3 = 0.1589E-4$

$v_d = 0.20550E-2 - 0.22029E-2 = -0.1479E-3$

$u_d = 0.47856E-3 - 0.48331E-3 = -0.475E-5$

Similarly,  $v_e = 0.3284E-3$   $u_c = 0.3876E-4$

$v_e = -0.2763E-3$   $u_c = 0.42E-6$

Using the above values,  $K_I$  and  $K_{II}$  can be calculated as,

$$K_I = \frac{\mu}{\kappa + 1} \sqrt{\frac{2\pi}{L}} [4(v_b - v_d) + v_e - v_c]$$

$$= \frac{80769.2307}{1.384615 + 1} \sqrt{\frac{2\pi}{0.02}} [4(0.1607E-3 - (-0.1479E-3)) + (-0.2763E-3) - 0.3284E-3]$$

= 378.13 MPa  $\sqrt{\text{mm}}$

$$K_{II} = \frac{\mu}{\kappa + 1} \sqrt{\frac{2\pi}{L}} [4(u_b - u_d) + u_e - u_c]$$

$$= \frac{80789.2307}{1.384515 + 1} \sqrt{\frac{2\pi}{0.02}} [4(0.1589E-4 - (-0.475E-5)) + 0.42E-6 - 0.387E-4]$$

$$= 26.5898 \text{ MPa}\sqrt{\text{mm}}$$

### 3.5 Calculating Crack Propagation Angle

Crack propagation angle is calculated using the maximum tangential stress theory. This method can only be applied to 2-D problem and the problem defined here is 2-D. Also, the simplicity of this method is that only  $K_I$  and  $K_{II}$  values are required [3]. Thus, this method is computationally inexpensive. While the other two methods, are computationally expensive. This method has been implemented in the macro *sif1.mac*. The Equation for calculating the angle is given as,

$$\theta_m = 2 \tan^{-1} \left[ \frac{\frac{K_I}{K_{II}} \pm \sqrt{\left(\frac{K_I}{K_{II}}\right)^2 + 8}}{4} \right]$$

Here,  $\theta_m$  is the angle calculated with respect to x-axis of the coordinate system with its origin at the crack tip. Note that if  $K_{II}$  is positive then subtraction should be done in the equation and vice-verse [5].

For example, let  $K_I = 378.13 \text{ MPa}\sqrt{\text{mm}}$  and  $K_{II} = 26.5898 \text{ MPa}\sqrt{\text{mm}}$ , then  $\theta_m$  can be calculated as,

$$\theta_m = 2 \tan^{-1} \left[ \frac{\frac{378.13}{26.5898} \pm \sqrt{\left(\frac{378.13}{26.5898}\right)^2 + 8}}{4} \right]$$

$$= -7.967^\circ$$

### 3.6 Crack growth rate model

Paris model was used for calculating the crack growth rate and thus the life for the crack growth. Paris model is quite simple and less material parameter is required. Due to non availability of other material properties for fracture mechanics, Paris equation has to be used [8]. We know that, the equation for crack growth rate according to Paris model is given as,

$$\frac{da}{dN} = C (\Delta K)^n$$

Where, C and n are constants that are found by a curve fit to experimental data [7].  $\Delta K$  is the range of stress intensity factor for each cycle.  $\Delta K$  can be calculated as shown in the

Fig - 6. For the present model  $K_{min}$  is zero. Hence for the present formulation,

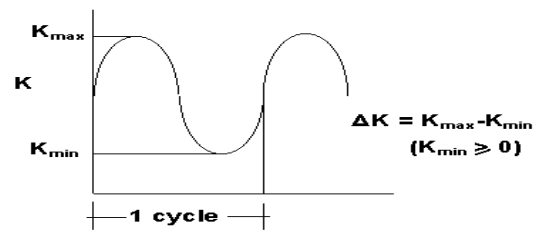


Fig - 6 Range of K

$$\Delta K = K_{max}$$

$K_{max}$  and  $K_{min}$  are the mixed mode stress intensity factor. Mixed mode stress intensity factor can be calculated as shown below,

$$K = \sqrt{(K_I^2 + K_{II}^2)(1 - \nu^2)}$$

Where K is the mixed mode stress intensity factor.  $K_I$ ,  $K_{II}$  are the stress intensity factor due to mode I and mode II loading conditions.  $\nu$  is the Poisson's ratio. Thus,  $K_{max}$  can be calculated by substituting  $K_I$  and  $K_{II}$ . This has been implemented in the code *lif.mac*.

For example, let  $K_I = 378.13 \text{ MPa}\sqrt{\text{mm}}$ ,  $K_{II} = 26.5898 \text{ MPa}\sqrt{\text{mm}}$  and let the crack be incremented in the direction at angle  $-7.967^\circ$ . Let  $C = 1.33559e-13$  and  $n = 2.954$ . Then for the crack to grow 0.1 mm length, number of cycles required can be calculated as below,

$$K = \sqrt{(K_I^2 + K_{II}^2)(1 - \nu^2)}$$

$$= \sqrt{(378.13^2 + 26.5898^2)(1 - 0.3^2)}$$

$$= 361.6037 \text{ MPa}\sqrt{\text{mm}}$$

As  $\Delta K = K_{max}$ , therefore  $\Delta K = 361.6037 \text{ MPa}\sqrt{\text{mm}}$ . Using these values in Paris equation, the number of cycles can be calculated as shown in equation.

$$dN = \frac{da}{C (\Delta K)^n}$$

$$= \frac{0.1}{1.33559e-13 (361.6037)^{2.954}}$$

$$= 20763.8 \text{ cycles}$$

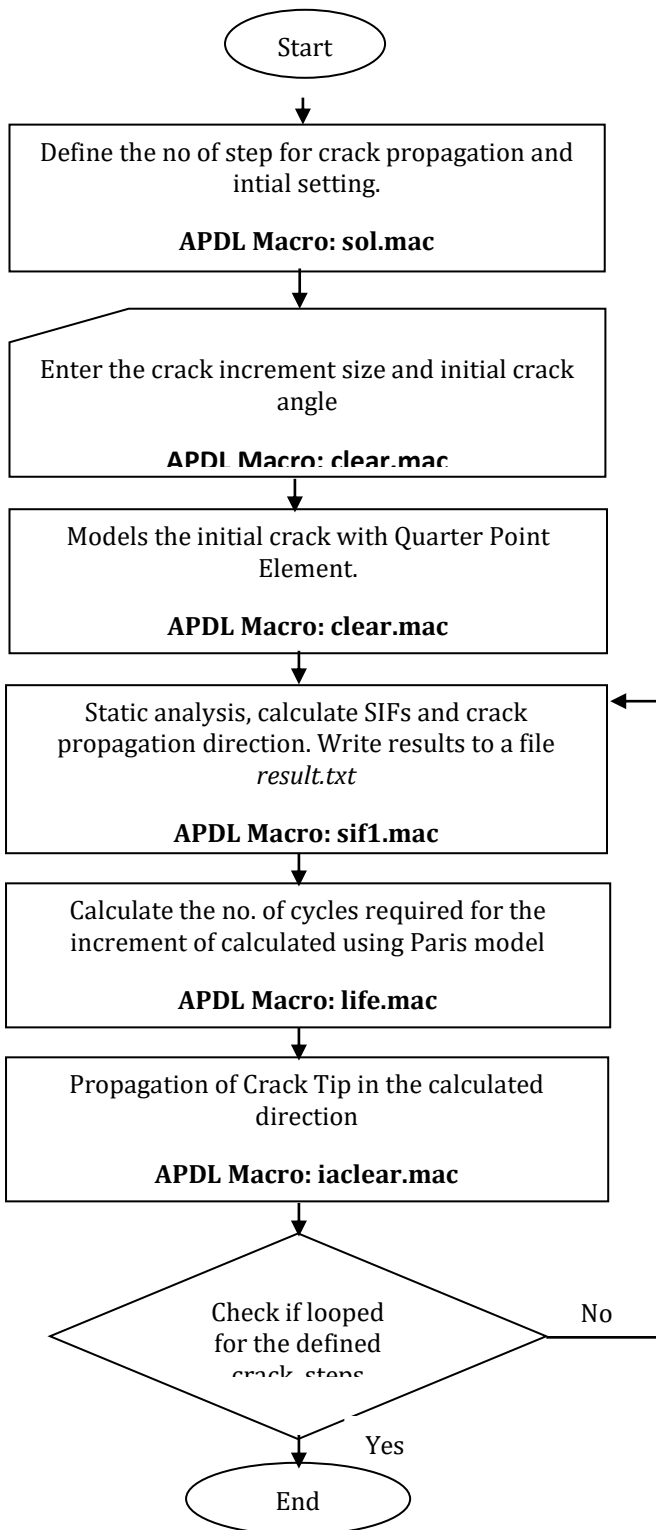
A flowchart showing the work flow in the code is given in Fig - 7. The functionality of all the macros in the code has also been described in Table -2.

**Table – 2** Functionality of all the macros

Macro	Description
<i>clear.mac</i>	Takes crack increment size and initial crack angle as input and models initial crack in the FE model.
<i>sif1.mac</i>	Solves the model and calculates the KI and KII values and the crack growth direction
<i>iaclear.mac</i>	Increments the crack tips in the crack growth direction calculated by a user defined amount. It does so by deleting and remeshing elements locally. For remeshing it calls the macro <i>ac.mac</i>
<i>ac.mac</i>	Remeshes the region deleted by <i>clear.mac</i> and <i>iaclear.mac</i> . Places the quarter point element around crack tip.
<i>bound1.mac</i>	Determines the boundary of the region deleted by <i>clear.mac</i> and <i>iaclear.mac</i> .
<i>life.mac</i>	Calculates the number of cycles required for the crack propagation.
<i>Sol1.mac</i>	Loops all the macro, until the no. of steps for crack propagation is achieved. Writes results of each loop in a file <i>result.mac</i>

**4 VALIDATION OF CODE**

Based on the formulation described in the previous section an APDL script was developed needs to validate by some means. Thus, the results of the crack propagation code are compared with the FRANC2D. This software is available free. Franc2D was developed by the Cornell Fracture Group at Cornell University. The program is designed for two dimensional problems and uses the principle of linear elastic fracture mechanics. It is capable of analyzing plane stress, plain strain or axis-symmetric problems. A unique feature of the program is the ability to model a crack in a structure. For this, it uses a method called “delete and fill”. It has been widely used by different organizations and institutions for the study of crack growth. A number of international papers on fracture mechanics have reference to this software. One problem is taken to compare the results predicted by APDL with FRANC2D. In the problem mixed mode crack propagation occurs. Thus the prediction of K1 and K2 mode can be compared [3].



**Fig – 7** A flowchart describing the working of the APDL code

### 4.1 Problem Description

**Problem:** In this problem, a hole of radius 4 mm is made at top part of the plate. The geometrical dimensions of the plate is same as problem 1. The intention of making a hole was to introduce a weak region in the plate and to study the effect of this on the crack propagation path. Fig - 8 shows the dimensions for the problem.

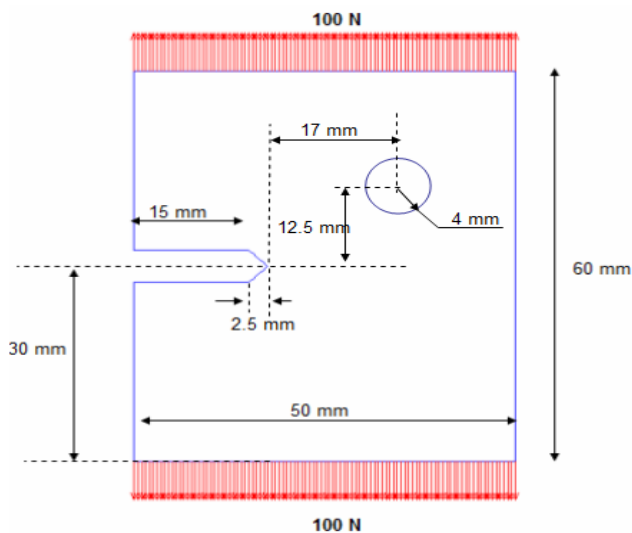


Fig - 8 Dimensions for the problem

The problem is solved using APDL and FRANC2D software. Initial crack of 0.5 mm length is initiated at the notch. The crack is propagated for 80 increment step with a crack increment size of 0.5 mm. Thus the crack propagates for a length of 40 mm. The results are discussed below. The Fig -9 shows the crack path predicted by APDL and FRANC2D software.

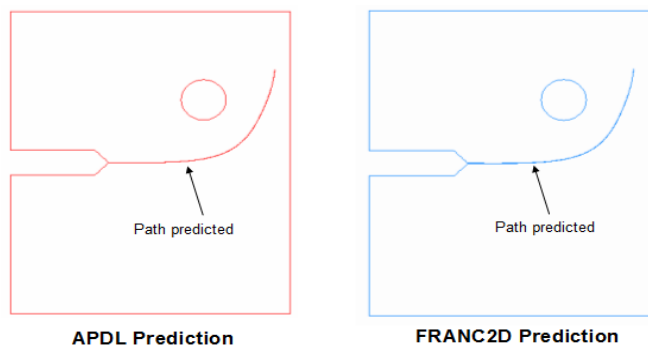


Fig - 9 Crack Propagation Path predicted by APDL and FRANC2D

Again, the path predicted by ANSYS APDL and FRANC2D is the same. This suggests that the calculation made by the APDL script is correct. Thus, it validates the APDL script. The predicted path in this problem is curvilinear, whereas in the simple plate problem it was straight line. This suggests that the crack path is affected by any nearby weak region. Also, as this is a curvilinear path, both K1 mode and

K2 mode play important role in deciding the crack propagation path. In the simple plate problem (Problem 1), there was no role played by K2 mode and its value was negligible compared to K1.

In the Fig - 10 & Fig - 11 shown below, the prediction made by APDL and FRANC2D for K1 and K2 has been plotted.

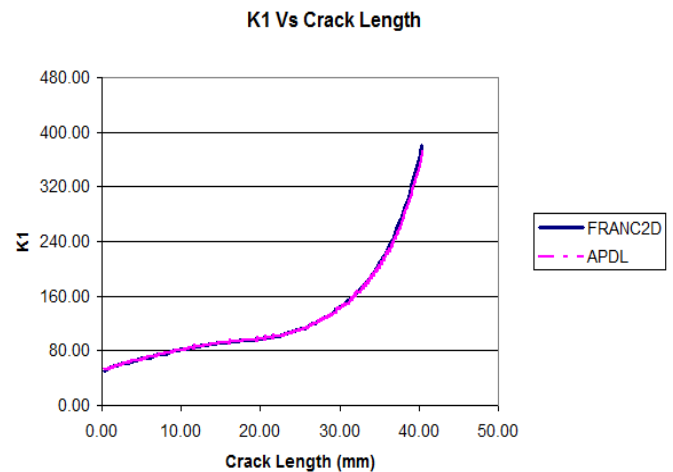


Fig - 10 K1 Vs Crack Length

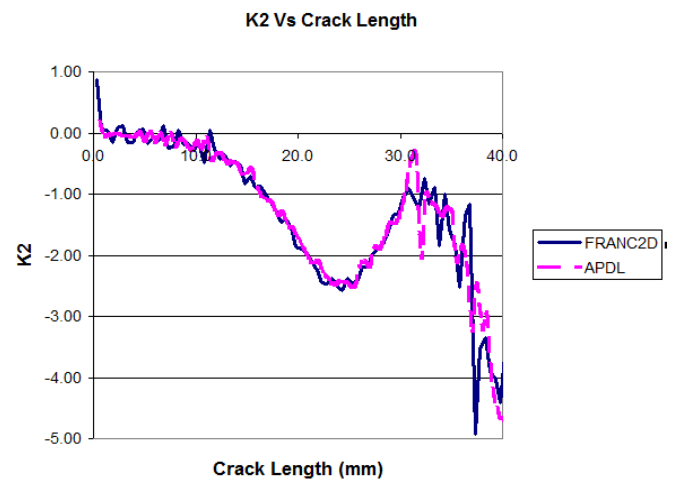


Fig - 11 K2 Vs Crack Length

Fig - 10 shows that the K1 prediction along the crack

length is same in APDL and FRANC2D. In Fig - 11, K2 predictions are plotted. The plot shows that the K2 prediction for the APDL and FRANC2D matches very well. This again confirms that the predictions made by APDL and FRANC2D are matching. Initially the K2 values are very small. At that point of time the crack is moving along a straight line. This can be seen in the Fig - 9. Once the crack reaches near the hole (weak region in the plate), there is increase in the value of K2. Thus, now there is crack propagation due to mixed mode, i.e., the crack path prediction is taking place due to K1 and K2 mode. This



results in a curvilinear crack path. In the above problem, comparison was made between the APDL predictions and FRANC2D predictions. Very good correlation is found between the predictions of APDL and FRANC2D. It is also found that the predictions are correct in pure K<sub>I</sub> mode problems as well as in the mixed mode (K<sub>I</sub> and K<sub>II</sub> mode) problems. Thus the APDL script is validated through this problem.

## 5. CONCLUSIONS

An APDL code is developed for the automatic modeling as well as propagation of crack. The code is validated by comparing its result with one of the widely used software, Franc2D. Based on the numerical investigations carried out on different plate problems, following conclusions can be deduced.

1. The results predicted by APDL for K<sub>I</sub> and K<sub>II</sub> modes match very well with the results predicted by FRANC2D. Thus the developed APDL script is validated.
2. The results show that when K<sub>I</sub> mode is dominant (Pure K<sub>I</sub> mode), the crack moves along a straight path. As the K<sub>II</sub> mode start playing role in the crack propagation the curve takes a curvilinear path.
3. The study on the crack increment size showed that the K<sub>I</sub> mode is not affected by higher increment size but the K<sub>II</sub> mode is affected by it. There is change in crack propagation path with different crack increment size in mixed mode problems.

## REFERENCES

- [1] Maitra G. M., "Handbook of Gear Design", Tata McGraw-Hill Publishing Company, 1988.
- [2] Sharma P. C., Aggarwal D. K., "Machine Design", S. K. Kataria & Sons, 2001.
- [3] Carlos V. Carvalho, Tereza D. de Araújo, Joaquim B. Cavalcante, Luiz F. Martha and Túlio N. Bittencourt "Automatic Fatigue Crack Propagation Using A Self-Adaptative Strategy" semantic scholar 1999.
- [4] Lewicki D. G., "Effect of speed (centrifugal load) on gear crack propagation direction".
- [5] Golde S., Jelaska D., Kramberger J., "A computational model for calculation of load capacity of gears". International Journal of Fatigue, Vol.24, No. 10, 2002, pp. 1013-1020.
- [6] Cavalcanti Marcus, Luciano M., Partridge P., "A comparative analysis between three different approaches on obtaining stress intensity factor", Computational Mechanics: New trends and applications, Barcelona, Spain, 1998.
- [7] Ciavarella M., Demelio G., "Numerical methods for the optimization of specific sliding, stress

concentration and fatigue life gears", International Journal of Fatigue, Vol. 21, 1999, pp. 465-474.

- [8] Pathan S, Hellen T., Flasket J., Glodez S., "Numerical methods for determining stress intensity factors vs. crack depth in gear tooth roots", International Journal of Fatigue, Vol. 19, No. 10, 1997, pp. 673-685.
- [9] MackAlender M., "Tooth Interior Fatigue Fracture & Robustness of Gears", PhD Thesis, 2001, Royal Institute of Technology, Sweden.
- [10] Andersen M., "Fatigue crack initiation and growth in ship structures", PhD Thesis, 1998, Technical University of Denmark, Denmark..