ISCA Journal of Engineering Sciences ______________________________ *ISCA J. Engineering Sci.*
Vol. **1(1),** 26-34, July **(2012)**

Analysis of Crack Initiation in Fretting Fatigue Specimen

Purkar T. Sanjay and Pathak Sunil
Swami Vivekanand College of Engineering, Indore MP, INDIA

Available online at: www.isca.in
Received 2nd June 2012, revised 15th June 2012, accepted 19th June 2012

Abstract

The study of fracture mechanisms shows that the growth rate of a crack is proportional to the square root of its length, given the same stress fluctuation and degree of stress concentration. For this reason fatigue cracks spend most of their life as very small cracks which are hard to detect. A new approach for the simulation of fatigue crack growth in two elastic materials has been developed and specifically, the concept has been applied to fretting fatigue *in a straight plate and in tubular joints. In the proposed method, the formation of new surface is represented by an interface element based on the interface potential energy. This method overcomes the limitation of crack growth at an artificial rate of one element length per cycle. In this method the crack propagates only when the applied load reaches the critical bonding strength. The predicted results compares well with experimental results.*

Keywords: Fatigue crack growth, finite element method, mixed mode, finite element method, crack path.

Introduction

MSC/FATIGUE is an advanced fatigue life estimation program for use with finite element analysis. When used early in a development design cycle it is possible to greatly enhance product life as well as reduce testing and prototype costs thus ensuring greater speed to market. However, before describing the features of the product in detail it is useful to define the term fatigue. Very often the terms fatigue, fracture, and durability are used interchangeably. Each does however convey a specific meaning. Although many definitions can be applied to the word, for the purposes of this paper, fatigue is failure under a repeated or otherwise varying load which never reaches a level sufficient to cause failure in a single application. It can also be thought of as the initiation and growth of a crack, or growth from a pre-existing defect, until it reaches a critical size, such as separation into two or more parts. Fatigue analysis itself usually refers to one of two methodologies. The stress-life (or S-N method), is commonly referred to as the total life method since it makes no distinction between initiating and growing a crack. This was the first fatigue analysis method to be developed over 100 years ago. The local-strain or strain-life (-N) method, commonly referred to as the crack initiation method, was more recently developed and concerns itself only with the 'initiation' of a crack. Fracture specifically concerns itself with the growth or propagation of a crack once it has initiated and this has given rise to many so-called crack growth methodologies. All fatigue analysis calculations are performed within the constraints of the so-called 'five-box trick.' The illustration below shows how this concept can be visualized. For any life analysis, whether it is fatigue or fracture, there are always three Inputs. The first three boxes are these inputs. Fatigue analysis has traditionally been a test-based activity. Components or models are tested with service loads, which are as close to the 'in-service' signals as possible. In a test situation loading is usually a stress signal measured remotely from a critical location. Geometry is usually a stress concentration factor to account for the separation of the critical location and measurement point and materials are the cyclic 'fatigue' properties. The biggest drawback with testing is that it cannot be undertaken until a prototype exists. If a design problem then occurs it is usually very difficult to rectify. It is also very expensive to perform fatigue tests. For these reasons FEA based fatigue analysis has been perceived as an excellent enhancement to the testing process.

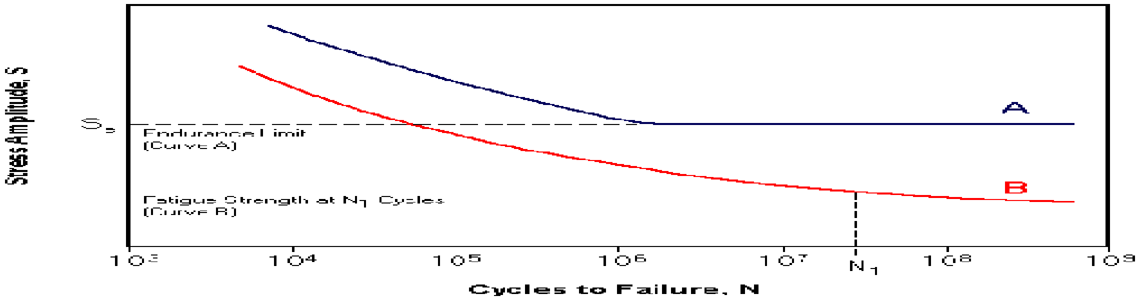

Figure-1
S-N Curve

The FEA model effectively replaces the geometry box in figure 3. Loading signals are now forces, displacements or some other driving function. Material properties still have to be obtained through test, however empirical approximations can be made based solely on the UTS and Young's Modulus of the material. The correctness and accuracy of each of these inputs is important in that an error with any of these will be magnified through the fatigue analysis procedure (the fourth box,) since this process is logarithmic. A 10% error in loading magnitude could result in a 100%, or more, error in the predicted fatigue life. The fifth box is the post-processing or results evaluation. This can take on the form of color contours on a finite element model or a tabular listing, but also quite often leads back into the three inputs to see what effect variations of these inputs will have on the life prediction. This is referred to as a sensitivity or a "what if" study. This is extremely useful at times when you are not quite sure about the accuracy of one of the inputs.

Life Prediction Methods: MSC/FATIGUE uses three life prediction methods. These are 'total life,', 'crack initiation', and 'crack propagation'. Total life is aptly named in that only the total life of the component is of concern. This is in contrast to when a crack will initiate or how quickly it will grow. The three methods can be related to each other by assuming that the total number of cycles to failure, N_f, equals the number of cycles to initiate a crack, Ni, plus the number of cycles to propagate that crack, Np. The three methods have grown out of different needs over the decades using different techniques and having different degrees of accuracy. So in theory this relationship is true, but in practice, when applying the three methods to the same problem, rarely, if ever does it add up.

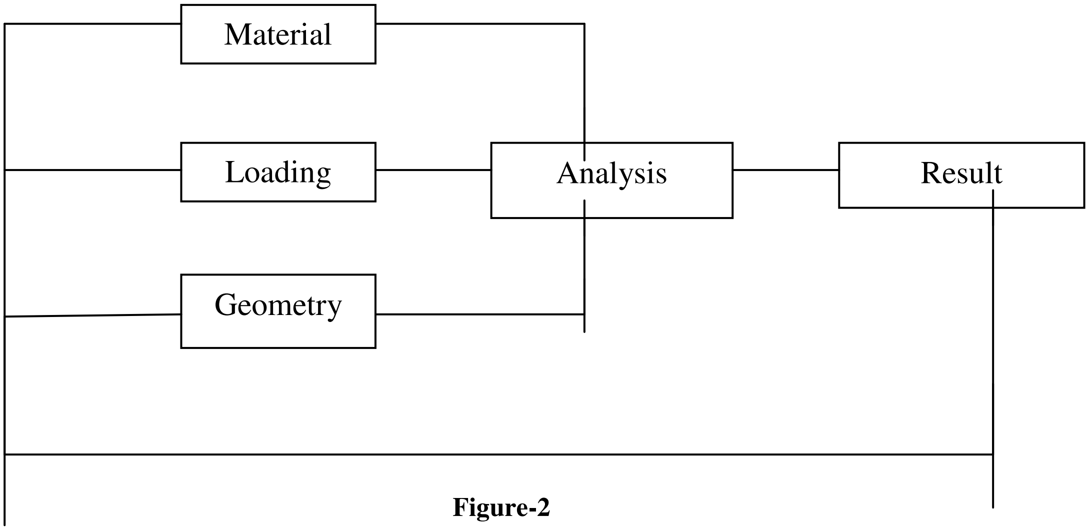

Figure-2
The 'fatigue' 5 box trick

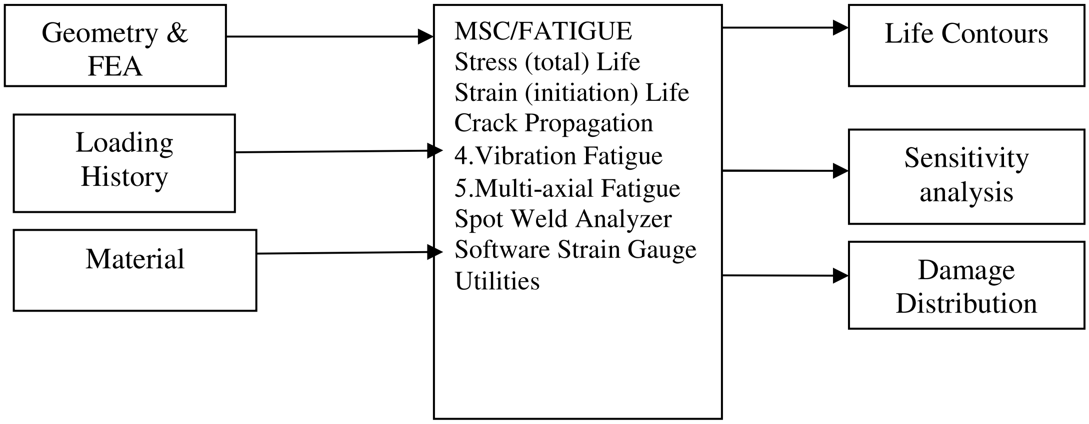

Figure-3
An overview of an FEA based fatigue analysis

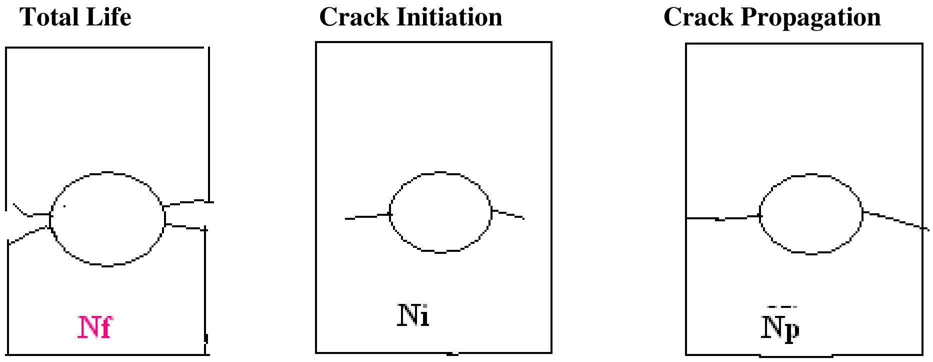

Figure-4
An 'idealization' of the fatigue design process

FEA based stress analysis options: There are several FEA based methods for obtaining the stress information that is required to perform a fatigue life calculation.

Static structural (and fatigue) analysis can be undertaken utilising MSC/FATIGUE's superposition capabilities for combining multi load application inputs. Unit inputs of load are applied to all desired load application points. The resultant stresses (caused by the unit load cases) are then factored by the actual time history of loading for that load application point. This process is repeated for all load application points and the results are linearly superimposed. Fatigue life calculations are then performed using these combined stress histories. This method ignores dynamic influences such as mass effects.

Dynamic transient analysis: If this approach is used, the stress histories are produced at each point of interest using a FE transient analysis method. These stress histories are also superimposed to obtain the required combined stress histories, but the FE solver handles this. Fatigue life calculations are then performed on these stress time histories. This method accounts for all dynamic effects but is less versatile in that all loads must be combined in a Single FE analysis.

Frequency Response analysis: In this approach the transfer functions are produced using the desired solver. These transfer functions are then resolved onto the desired stress axis system (usually principal stress). The response caused by multiple random loading inputs is then obtained using standard random process techniques. The effect of correlation between inputs can be dealt with by including cross power spectral density functions in the input loading data. This method accounts for all dynamic effects and is quite versatile.

Random Vibration analysis: In this approach the response power spectral density function is determined directly from the FE solver. Effects due to multiple load inputs must be dealt with in the FE analysis as with a transient analysis approach. All dynamic effects are accounted for but this method has the limitation that fatigue life can only be computed for a single component direction. Stress response results are not resolved onto a desired stress axis system by the FE analysis.

Design Philosophies: There are three main fatigue design philosophies. Each centres on one of the fatigue life estimation methodologies. To illustrate the three consider the design of a stool.

Safe Life: The safe life philosophy is a philosophy adopted by many. Products are designed to survive a specific design life. Full scale tests are usually carried out with margins of safety applied. In general, this philosophy results in fairly optimized structures such as a stool with three legs. Any less than three legs and it would fall over.

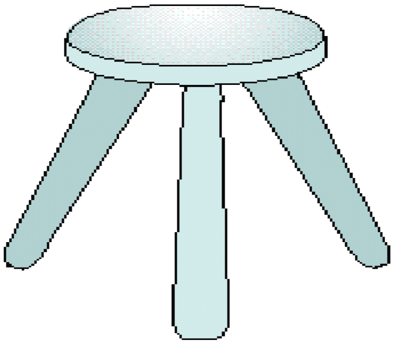

Figure-5(a)
Safe Life

Fail Safe: On the other end of the spectrum of design philosophies is that of failsafe. This is where a failure must be avoided at all costs. And if the structure were to fail it would fall into a state such that it would survive until repairs could be made. This is illustrated with our stool now having six legs. If one were to fail the stool would remain standing until repairs could be made. This philosophy is heavily used in safety critical items such as in the aerospace or offshore industries.

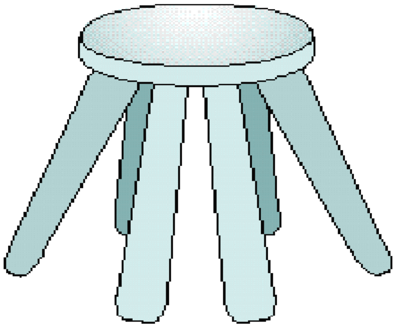

Figure-5(b)
Fail Safe

Damage Tolerant: The middle ground philosophy is that of damage tolerance. This philosophy, adopted heavily in the aerospace community and nuclear power generation, relies on the assumption that a flaw already exists and that a periodic inspection schedule will be set up to ensure that the crack does not propagate to a critical state between inspection periods. As implied, this philosophy adopts the crack growth method. This is illustrated using our stool (now with four legs) but with someone inspecting it. This particular design philosophy is generally used in conjunction with the fail safe philosophy first to try and design such that no failure is expected but then to assume that, for whatever reason, a flaw does exist and must be monitored.

Figure-5(c)
Damage Tolerant

Integrated Durability Management: Durability management is the control and organization of design, test, and production, to ensure products are developed to meet the required life within cost and on time. The process has evolved over the last 150 years since fatigue failures were first recognized. While there are many technologies that have contributed to the understanding of fatigue and to the solution of fatigue problems, two major procedures are used in durability management: fatigues testing and fatigue modelling.

Fatigue Testing: The first fatigue tests were carried out on full-scale components to establish their safe working stress. Later, the more complete relationship between cyclic stress and strain and fatigue life was established. Small-scale specimens were tested to study component life and also fatigue mechanisms. In more recent times, as tests had to become increasingly realistic, special test techniques were developed such as remote parameter control. Today, testing is still the most common way of confirming the fatigue life of a product prior to releasing it onto the market. However, testing often reveals weaknesses, which necessitate re-design. Assessing the suitability of particular design modifications using fatigue testing alone can be time consuming and cost far more than just a delayed product.

Fatigue Modelling: The estimation of fatigue life using mathematical modelling techniques was developed to assist the engineer in solving fatigue problems without always having to physically test all the options. For this reason, techniques such as local strain or crack initiation modelling have become widely used. Improvements in the power of computers have enabled the effective use of these techniques. Today, most major companies designing mechanical structures will use a fatigue life estimation tool such as MSC/FATIGUE in conjunction with testing. The late 1980s had established the use of finite element analysis (FEA) as a tool for stress analysis. At the same time the integration of FEA and fatigue life estimation through the MSC/FATIGUE product began to provide new benefits by assessing fatigue earlier in the development process.

Integrated Durability Management: Understanding and effective implementation of durability management strategies requires a partnership between tests and design analysis. It can reduce product lead-time by focussing the use of fatigue testing to the essential correlation and sign-off tests. The use of fatigue modelling, at the design analysis stage, allows more options to be assessed for little incremental cost. Integrated durability management can produce better products more quickly and cheaply.

Terminology Used: 2a= projected crack length, 2c= actual crack length, 2w= plate width, T= plate thickness, E= young's modulus, σ_0= stress amplitude, θ= direction of crack extension, υ = Poisson's ratio, K_{I} , K_{II} and K_{III} = stress intensity factor for mode I, mode II and mode III respectively. X= symmetric axis of specimen in horizontal direction, Y= symmetric axis of specimen in vertical direction, x, y = crack tip coordinate parallel to X and Y.

Related Work: Finite element analysis (FEA) has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of mechanics of materials. In spite of the great power of FEA, the disadvantages of computer solutions must be kept in mind when using this and similar methods. There is a lot of software available for finite element analysis, such as algor (FEMPRO), ANSYS, ABAQUS, and MSC Patran. All this software widely used nowadays, in order to analyze the part. In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause there sulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The finite element method is a good choice for solving partial differential equations over complicated domains. Finite element method; FEM is a numerical technique for finding approximate solutions of partial differential equation as well as of integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problem), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method[1] and Runge-Kutta[1]. Skorupa and Skorupa[1] were studied crack growth predictions for structural steel using constraint factors. At positive stress ratios, structural steel shows significant crack growth retardation under variable amplitude. It also studied load interaction effects in crack growth are negligible. The crack growth response of structural steel in some of the experiments is very different from Al-alloys used in the aircraft industry were studied two different geometries were used on this finite element model in order, to analyze the reliability of this program on the crack propagation in linear and nonlinear elastic fracture mechanics. These geometries were namely; a rectangular plate with crack emanating from square-hole and double edge notched plate (DENT). Where, both geometries are in tensile loading and under mode I conditions. Predict the crack propagations directions and calculate the stress intensity factors. And the results are the application of the source code program of 2-D finite element model showed a significant result on linear elastic fracture mechanics detail level.

Fatigue Rate Curve: A typical fatigue rate curve, commonly referred to as a da/dN versus Δ*K* curve, is illustrated by figure 3 The curve is defined by regions A, B and C which are commonly referred to as region I, II and III respectively

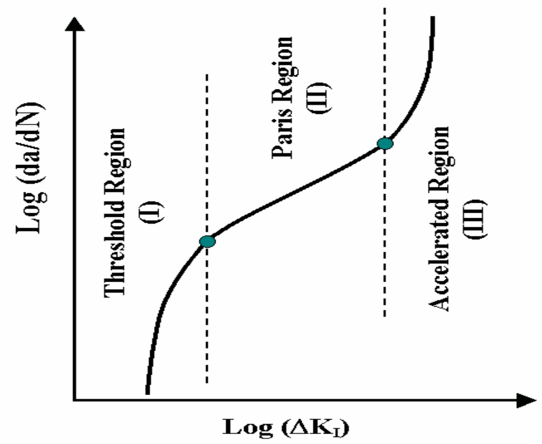

Figure-6
A typical fatigue crack growth rate Curve (FCG curve)

Region I represents the early development of a fatigue crack and the crack growth rate; da/dN is typically of the order 10-6 mm/cycle or smaller of the test data result from ASTM E647. This region is extremely sensitive and is largely influenced by the microstructure features of the material such as grain size, the mean stress of the applied load, the operating temperature and the environment present.

The most important feature of this region is the existence of a stress intensity factor range below which fatigue cracks should not propagate. This value is defined as the fatigue crack growth threshold and is represented by the symbol ΔKth. Its value is experimentally determined by using the decreasing K test as described in the ASTM E647 documents. $da/dN= Cp\ (\Delta K)^{m}{}_{p}$

Region II represents the intermediate crack propagation zone where the length of the plastic zone ahead of the crack tip is long compared with the mean grain size, but much smaller than the crack length. The use of linear elastic fracture mechanics (LEFM) concepts is acceptable and the data follows a linear relationship between log da/dN and log Δ K. The crack growth rate is typically on the order of 10-6 to 10-3 mm/cycle, which corresponds to the majority of the test data results from ASTM E647.

$\mathbf{\Delta K=K_{max}(1-R)^{y}{}_{w}}$

Where Kmax = Δ K/(1-R),and equation reduces to $da/dN= Cw\ (\Delta K)^{m}{}_{w}$, which is equivalent to the Paris law with *Cp = Cw* and *mp = mw.*

Region III represents the fatigue crack growth at very high rates, da/dN > 103 mm/cycle due to rapid and unstable crack growth just prior to final failure. The da/dN versus ΔK curve becomes steep and asymptotically approaches the fracture toughness Kc for the material. The corresponding stress level is very high and causes a large plastic zone near the crack tip as compared with the specimen geometry:

$da/dN^{=}\ {}^{C}{}_{F}\ (\Delta K)^{m}{}_{y}/(1-R)Kc-\Delta K={}^{C}{}_{F}\ (\Delta K)^{m}{}_{y}/(1-R)(Kc-K_{max})$

Methodology

Fretting fatigue damage occurs in contacting components when they are subjected to oscillating loads and sliding movements at the same time. This phenomenon is schematically shown in figure 1. The fretting fatigue resistance of materials may be affected by many parameters such as contact pressure, axial stress, σ, friction between the pads and the specimen, pad geometry and sliding amplitude.

In this paper two numerical simulations are presented, namely: i. FE model for stress distribution and Estimating initial crack location, ii. New approach to FEA modeling of crack propagation.

Estimation of crack initiation location: Ansys parametric language design (APDL) code was used to analyze the specimen without crack. The finite element model of the test assembly is illustrated in figure 7. The model which is constructed on the basis of the schematic illustration of fretting fatigue shown in figure 6 consists of the specimen and the pads. The model consists of one set of four-node plane strain elements (PLAN82) for the specimen and another set for the contact pad. In addition, CONTA172 and TARGE 169 elements were used at interface (the interface surface between pad span and specimen). These contact elements allow pressure to be transferred between the contact pad and the specimen and avoid the pad penetrates into the specimen. The Augmented Lagrange method of friction was used with 0.5 as coefficient of friction. Note that only a quarter of the test configuration is considered due to double symmetry with respect to the X and Y axes. In addition to the boundary condition along the lines of symmetry, the specimen was constrained in Y direction at the bottom line nodes and right side of line of symmetry for specimen and both sides for contact pad constrained in X direction. As the figure 7 shows, a normal *P*/2 then an axial stress, σ is applied to the test assembly. The normal and tangential contact stress distributions are determined from the simulations. In order to have more accurate results, the finite element mesh was refined in the contact region. An acceptable element size was determined to be at least 9 μm x 9 μm in the refined contact zone from the convergence study. However, mesh size was much finer than this, which varied from case to case with one basic requirement that at least ten elements were present behind the contact edges.

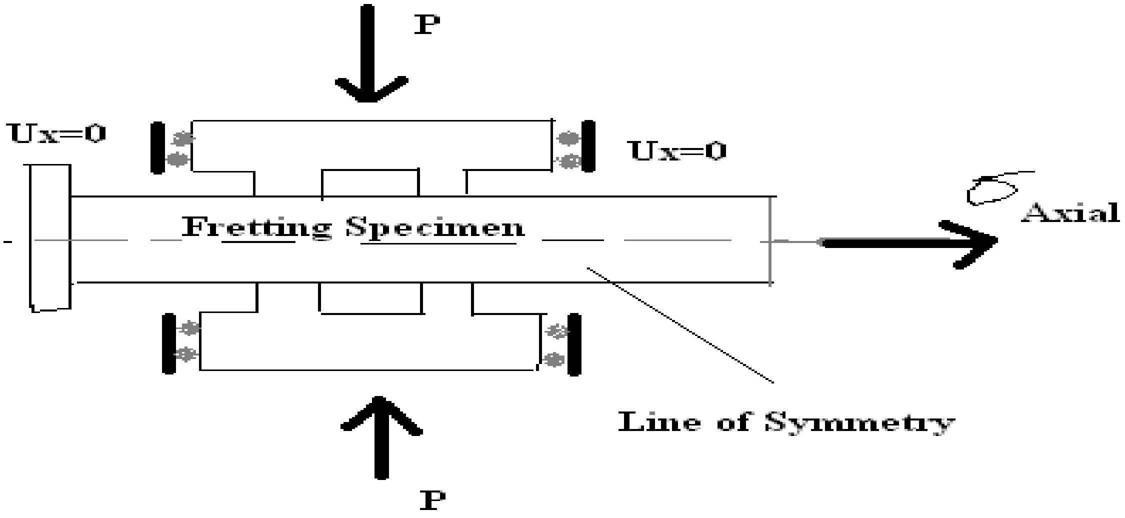

Figure-7
Schematic illustration of fretting fatigue

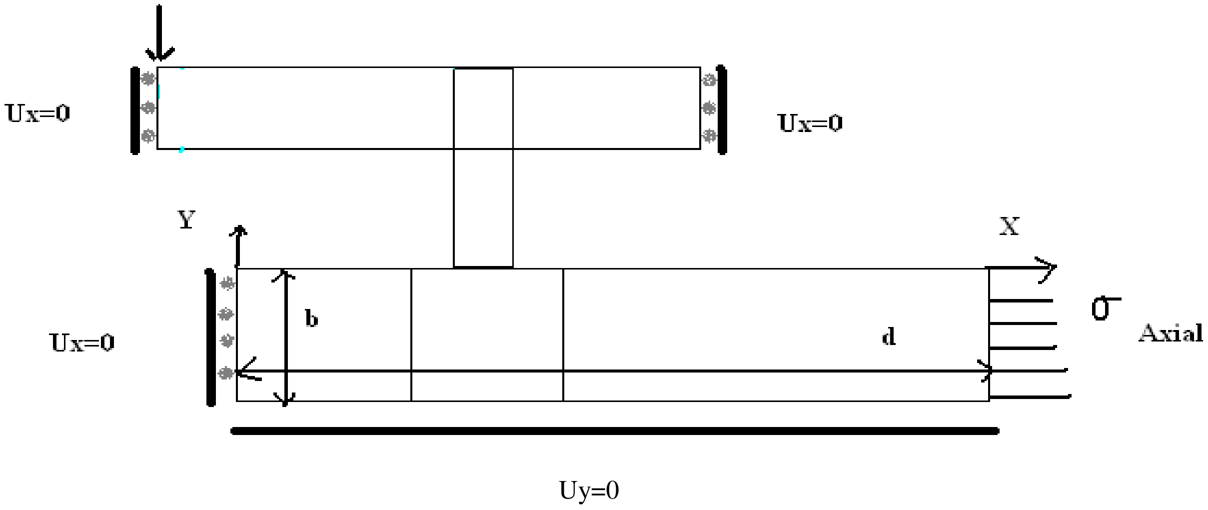

Figure-8
The schematic illustration of finite element model of fretting Fatigue assembly

Results and Discussion

Finite element analysis of crack propagation: The fretting fatigue crack propagation part was thereafter conducted using the finite element code, FRANC2D/L[2]. For this purpose, a finite element model, the same geometry and boundary conditions as used in APDL code as shown in figure 7 including the contact interface was modeled.Then an edge crack was inserted in the model considering the contact to perform the crack growth analysis by FRANC2D/L code, which has capability for incremental crack growth using fracture mechanics principles as elaborated later. The model was considered a quarter of the test configuration the same as model that was used in APDL code. The mesh size in the FRANC2D/L model was validated by comparing its stress values with the original ANSYS results. A deviation of less than 1% was observed between the two finite element solutions. The contact was defined with a gap element using a new material at interface of pad and specimen, with coefficient of friction equal to 0.5. This means that in each step for crack propagation the effect of contact pad was considered and the stress intensity factors were updated after each crack increment. This proposed technique would solve most problems in previous studies where sub-modelling was used for crack propagation. The crack propagation analysis requires the length and orientation of the initial crack. The crack propagation path was represented by a curvilinear path consisting of, S, straight segments, as shown in figure 8. In the first step, an initial crack of length, (*l*0 =0.01mm), with an orientation, (1=45°) from the y-axis was introduced on the contact surface at end of sharp edge. This position is obtained from the ANSYS FE model and the initial length and orientation is observed from experimental results. Also previous experimental studies have shown that the crack in fretting fatigue tests always initiate at or very near to the sharp edge at this angle[3,4]. The analysis was then performed with a crack length increment of *Δl.* The incremented crack kinked at the tip of the initial crack at *(x2,y2)* to produce new crack at a slope of _2 in the second step of the analysis, and this process was continued. In FRANC2D/L, the mesh is modified in each incremental step using the Suhara-Fukuda algorithm[5]; the algorithm generates a mesh of triangular elements as shown in figure 12, and the new crack geometry is represented at each incremental step to reflect the current crack configuration. Along with other theories, the code uses the maximum tangential stress theory, proposed by Erdogan and Sih[6], to determine the crack growth direction. Where *KImax* and *KIImaxi* are the maximum stress intensity factors at the *i*th step corresponding to the two crack propagation modes, mode I and mode II, at the maximum load. In This way crack trajectory was developed incrementally for a given loading condition using maximum tensile stress criterion. Further, the crack growth rate was assumed to be governed by the mode I stress intensity factors, *KImax* and *KImin*[7]. The modified crack closure integral technique of Rybicki and Kanninen[8] was used to calculate these stress intensity factors. These values were then used with the sigmoid crack growth model to determine the crack propagation life, which was measured from experiments in previous studies[7,8,9]. Fretting fatigue crack grows with an angle of about 45◦ as compared with the surface. Therefore in this step a crack with length of *10*= 0.01 mm and angle of 45◦ were created. *Δ1*was considered 0.1 mm for crack propagation. Final failure occurs when the value of stress intensity factor, *KI* tends to its critical value, *KIC*.

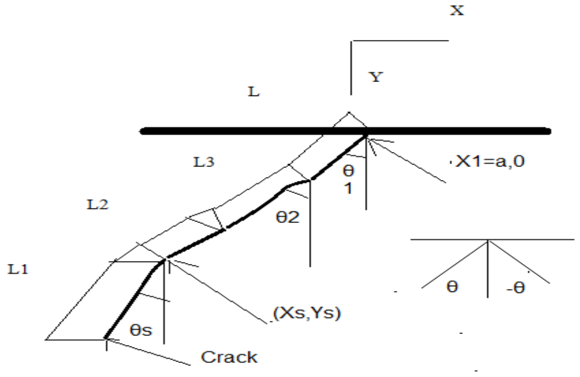

Figure-9
Trajectory of crack growth

FRANC2D/L CRACK GROWTH code was used in order to predict the crack growth life time curve. This program, realized by Domenico Quaranta[11], calculates crack growth life in generic 2D layered structures. The core of the program is Franc2D/L (based on Franc 2D, Copyright (C) Paul 'Wash'Wawrzynek and Tony Ingraffea)[2], which is used to extract stress intensity factors (SIF) history files for generic geometries and sets of loads. N40_F2DL_CG imports SIF history files and integrates the material da/dN equation (NASGRO model) for calculating crack growth for variable amplitude spectrum of loads. FRANC2D/L CRACK GROWTH code is able to show the amount of KI, KII and ΔK in every increment of crack propagation and it is designed to deal with Forman NASGRO material models (equation 2). The elements of the NASGRO crack growth rate equation were developed by Forman and Newman at NASA, and it has been implemented in FRANC2D/L CRACK GROWTH as follows[11]: $da/dN \,^{=}\, {}^{C}{}_{F}\, (\Delta K)^{m}{}_{y}/(1-R)Kc-\Delta K= {}^{C}{}_{F}\, (\Delta K)^{m}{}_{y}/(1-R)(Kc-K_{max})$

Where *C*, *n*, *p* and *q* are empirical constants, which are obtained by curve fitting the test data and *f*is the ratio of crack opening SIF to maximum SIF. The value of *f* is related to stress (load) ratio, flow stress and the plane stress/strain constraint factor. These values are provided by the FRANC2D/LCRACK GROWTH material database for each material. It is worth mentioning that _*K*th is not a simple threshold stress intensity range for long crack, but it includes the effect of short crack by involving 'intrinsic crack length', and the parameter *f.* Also C=6x10-10 and n=2.51 was used as fatigue ductility exponent[12], hence number of the cycles (dN) for a crack grow (da) in each increment, can be computed. Finally, the specimen fracture life was obtained.

Experimental results: The experiments were conducted for stress ratio of *R*=0.1, frequency of 20 Hz at a constant normal force of *P*=1200 N at the contacts, and maximum working stress of 130, 180, 200, and 280 MPa. The stresses are the average tensile stress which are obtained from *P/A* in which *P* is the maximum tensile load monitored continuously by the axial load cell and *A* is the cross sectional area of the specimen. A comparison between the S-N curves obtained for pure fatigue and fretting fatigue tests is shown in figure. 9. It can be clearly observed that fretting fatigue can reduce the normal fatigue life considerably. The reduction is more significant for lower stresses which correspond to high cycle fatigue (HCF) tests. For high stresses corresponding to a low cycle fatigue (LCF) conditions, the reduction is not so important.

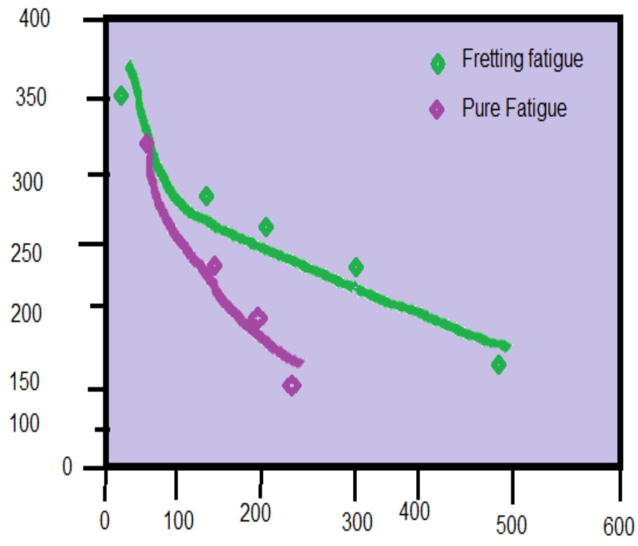

Figure-10
Maximum stress (MPa)/Fatigue Life (Cycles)

Validation of simulations: The simulations were validated by making a comparison between the variations of crack propagation (length) versus the number of cycles as predicted by numerical simulations in this work and the experimental measurements using replica. The results are given in table 1.As the results suggest a maximum of 24% difference is observed that in quite normal in fatigue context. Also there are a lot of factors that have effect on fretting fatigue crack propagation lifetime directly or indirectly that are not considered in this investigation, some parameters such as wear (i.e. removing material during the fretting fatigue cycles), environmental conditions, changing coefficient of friction due to presence of debris at contact interface, and a lot of factors that most of them decrease the fretting fatigue lifetime. So, due to authors believe this difference comes from a lot of variables that were not modelled, because the current software that is used for crack propagation does not have this capability.

Table-1
Comparing the experimental and numerical results for crack propagation life

Applied Stress(MPa)	Experimental cycles	Numerical cycles	Error
130	33910	44540	22%
140	-----	42750	-----
150	31840	38170	16.80%
160	----	35040	-----
170	----	33380	-----
180	23740	31584	24%

Conclusion

Numerical simulation of fretting fatigue was performed using Ansys parametric design language (APDL) and FRANC2D/L software. The former predicts the normal and tangential contact stress distributions and the latter calculates the number of cycles required for a known crack length. The calculation of cycles is based on Forman NASGRO equation when □*K* is computed by FRANC2D/Lcan predicts the cycles for a specific value of crack growth. The results indicate that the numerical simulations are capable of prediction the crack growth cycles and orientation accurately to some extent. The computed crack propagation lives were compared to the results of experimental study where total fatigue life was measured. The effects of contact geometries were determined on crack propagation behaviour. The following conclusions can be drawn based on this study: Maximum stress is created in contact region and stress values are higher near the sharp edge of contact which was the region of interest at which failure is expected, Von-Mises equivalent stress increases with the increase of pad width for both types of the pads considered in this investigation. However, for cylindrical on flat type contact the Von-Mises equivalent stress is significantly higher than that for flat on flat contacts and A comparison between the experimental and numerical results demonstrates a difference of about 24% in crack growth. It is observed that with increase of pad width for both flat and cylindrical cases, crack propagation life increased and these two parameters (pad width and crack propagation life) have a direct relation with each other.

References

1. Grewal B.S., Higher Engineering Mathematics, Khanna Publication **(2012)**

2. Daniel Swenson, Mark James A crack propagation simulator for plane layered structures, *Version 1.4 User's Guide* FRANC2D/L, Kansas State University **(1998)**

3. Hoeppner D.W., Chandrasekaran V. and Elliot C.B. Eds., Fretting-fatigue. Current Technologies and Practices, ASTM STP 1367, American Society for Testing and Materials, West Conshohocken, PA, **(2000)**

4. Cortez R., Mall S. and Calcaterra J.R., Fretting Fatigue: Current Technologies and Practices, ASTM STP 1367, Hoeppner D.W., Chandrasekaran V., Elliot C.B., Eds, American Society Testing and Materials, Philadelphia, 183-198 **(2000)**

5. Haritos G., Nicholas T., Painter G., 18th symposium Fracture mechanics (Philadelphia:ASTM) 206-20 **(1988)**

6. Gallagher Jea et al. Improved high-cycle fatigue (HCF) life prediction (Wright- Patterson Air Force Base Ohio) **(2001)**

7. R. Hojjati Talemi and Soori M., Experimental investigation of fatigue, Sustainable Construction and design (Islamic Azad university Iran) **(2011)**

8. Forman R.G. and Mettu S.R., Frac. Mech. 22nd Symp.**1** (Philadelphia: ASTM) **(1992)**

9. Wang Q.Y., Pidaparti. Fatigue crack growth in adhesively bonded composite-metal double-lap joints, *International Journal Composite Structure,* **57**, 109-115 **(2002)**

10. Sih G.C., Some Basic Problems in Fracture Mechanics and New Concepts, *Eng. Fracture. Mech*, **5**, 365 **(1973)**

11. Owen D., Stress intensity factors for cracks in a plate containing a hole and in a spinning disc, *Int. J. Fract*, **4,** 471-476, **(1973)**

12. Newman J., An improved method of collocation for the stress analysis of cracked plates with various shaped boundaries, NASA TN, 6376, 1-45 **(1971)**

13. Murakami Y., A method of stress intensity actor calculation for the crack emanating from an arbitrarily shaped hole or the crack in the vicinity of an arbitrarily shaped hole, *Trans Jap. Soc. Mech Engineering*, **44**, 423-32 **(1978)**

14. Bowie O.L., Analysis of an infinite plate containing radial cracks originating at the boundary of an internal circular hole, *Math. Phys*, **35**, 60-71 **(1956)**

15. Yan X., Cracks emanating from circular whole or square hole in rectangular plate in tension, *Eng. Fracture Mech*., **73,** 1743-1754 **(2007)**

16. Laurencin J., Delette G. and Dupeux M., An estimation of ceramic fracture at singularities by a statistical approach, *J. Eur. Ceramic Soc*, **28,** 1-13 **(2007)**

17. Kutuka M.A, Atmacab N. and Guzelbey I.H., Explicit formulation of SIF using neural networks for opening mode of fracture, *Int. J. Eng.Struct*, **29**, 2080-2086 **(2007)**

18. Abdul-Aziz Y., Abou-bekr N. and Hamouine A., Numerical modeling of the crack tip singularity, *Int. J. Mater. Sic,* **(2007)**

19. Aour B., Rahmani O. and Nait-Abdelaziz B., A coupled FEM/BEM approach and its accuracy for solving crack problems in fracture mechanics, *Int. J. Solids Struct*, **44,** 2523-2539 **(2007)**

20. Stanislav S. and Zdenek K., Two parameter fracture mechanics: Fatigue crack behavior under mixed mode conditions, *Eng. Fracture Mech*, **75**, 857-865, **(2008)**

21. Gustavo V.G., Jaime P. and Manuel E., KI evaluation by the displacement extrapolation technique, *Eng. Fract. Mech*, **66**, 243-255 **(2000)**

22. Pathak Sunil, Turbo charging and Oil Techniques in Light Motor Vehicles, *Research Journal of Recent Sciences*, **1**, 60-65, **(2012)**

23. Zienkiewicz O., Taylor R. and Zhu J., The Finite Element Method: Its Basis and Fundamental, 6th edition Baker and Taylor Books, Oxford, ISBN: **10,** 752 **(2005)**

24. Löhner R, Automatic unstructured grid generators, Finite Element Analysis **25**, 111-134 **(1997)**

25. Chang R., Static finite element stress intensity factors for annular cracks, *J. Non destruct.Evaluat*, **2**,119-124 **(1981)**

26. Shahani A. and Tabatabaei S., Computation of mixed mode stress intensity factors in a four point bend specimen, *Applied Math*, **32**, 1281-1288 **(2008)**