ANALYSES AND USE OF SOFTWARE ANSYS AND FRANC2D FOR FRACTURE MECHANICS

Júlio Toshio Mandai, juliomandai@gmail.com
Jorge Luiz de Almeida Ferreira, jorge@unb.br
University of Brasilia – UnB
Campus Universitário Darcy Ribeiro
Faculty of Technology, ENM – Mechanical Engineering Department, Bloco C
Brasilia – DF, 70910-900, Brazil

Abstract. *Fracture mechanics problems are difficult to be solved when the analytical solution is unknown, which is very frequent in most actual cases. In this way, finite element method modeling is seen like a powerful tool to contour this problems. Nowadays there are many softwares in market that solves fracture mechanics problems, however they are usually difficult to use and obtaining results can be not a obvious task. This work shows some procedures for solving bidimensional linear elastic fracture mechanics problems in two finite elements programs and it´s going to be made a educational proposal for using this softwares for a better understanding in Fracture Mechanics. The softwares have differents approachs for crack modelling. One is the famous Ansys, which is well know for its large utilization in industry, and the other is a free code distributed by Cornell University, called FRANC2D.*

Keywords*: Finite Element,Fracture Mechanics,Ansys, FRANC2D.*

1. INTRODUCTION

Engineering mechanical components generally operates under conditions of alternated loads that can be sufficiently severe in a point that fatigue resistance design be fundamental to it´s confiability, in a way the designer have to ensure a adequate fatigue life to the component. This mechanical components, in many cases, have cracks promoted by cyclic loadings. In this manner, it´s important to have a well understanding in fatigue phenomenon so that safer components can be designed.

Fracture mechanics has as primer objective predicts if a structure will fail or not by the presence of a crack. Crack analysis have to be performed starting from field stresses in the crack tip, which is quantified by stress intensity factor K_I. Thus, comparing K_I value with a peculiar characteristic of each material, called fracture toughness, it´s possible to predict if a crack member will fail or not when submitted by severe loads.

It´s well common a crack happen due to an unexpected over-loading in a structure no damaged. Usually this happens by a fail or a crack in the structure submitted by a cyclid loading or a normal service loading. In this way, the crack can growth starting from a fail or any stress concentration, decreasing the structure mechanical resistance until it´s collapse.

With large using of mechanical components under fatigue conditions, fracture mechanics is no longer just a problem of design engineers, who concerns about mechanical components life. Nowadays, fracture mechanics must be achieved by any engineering student, it´s a powerful tool for advanced study. Thus, this work presents a educational approach, showing some computational fracture mechanics tools as Ansys and FRANC2D.

2. CRACK GROWTH ANALYSIS CONCEPTS

Fatigue in materials subjected to repeated cyclic loading can be defined as a progressive failure due to crack initiation, crack propagation and the last one which is the instability stage. Crack growth theory shows that a crack propagates by very little increments in each load step, and this growth is consequence of plane slip and weakness of crack tip. Crack growth will be bigger as much as loading amplitude, in other words, crack growth per cycle Δa will be larger if the maximum stress in the cycle is higher and if the minimum stress is lower. Another influence factor in crack growth rate is the stress intensity factor K, and this influence is directly proportional to crack growth. Stress ratio cab be defined as $R = \sigma_{\min} / \sigma_{\max}$. Thus, we have that crack growth rate is a function of ΔK and R.

$$\frac{da}{dN} = f(\Delta K, R) \tag{1}$$

Under the action of cyclic loadings cracks can be initiated as a result of a plastic induced strain. Even if nominal stress is below the limit stress, in some regions stresses can be above yield material stress due to stress concentrations, as notches and discontinuities.

With the prime objective to obtain a relation between crack growth rate and ΔK, many tension tests were performed to obtain results as crack growth and variation in stress intensit factor ΔK. Paris (1962) was the first to figure out the relationship between this factors, which results in the following equation:

$$\frac{da}{dN} = C(\Delta K)^m \tag{2}$$

Where the coeficients C and m are material constants which are obtained experimentally. There are many other equations relating *da/dN* rate, which just add another terms to consider some particular characteristics. To simplicity, just the Paris equation will be used in this work to determine fatigue crack growth rate using Finite Elements Method (FEM). Crack propagation simulations is a incremental proceeding where a series of steps is repeated to model processing, where each simulation increment depends on previous computaded results.

3. FINITE ELEMENT SIMULATION IN FRACTURE MECHANICS

The main aplication in linear elastic fracture mechanics (LEFM) using finite elements is knowing the stress field in crack tip. Though, to many geometries, we have in literature the analytical formulas to calculate stress intensity factor, in actual cases where we have geometry complexities and arbitrary loading using finite element methods is a essential issue.

Linear elastic fracture mechanics problems have some difficulties when using finite element method to modelling stress fields and strains in crack tip, since the crack tip mathematical singularity is not well represented in the model. This difficulty in modelling the stress field and strain in crack tip was first seen in the beggining of the development of finite elements applied to fracture mechanics (Chan, 1970).

Various numerical methods have been applied in fracture mechanics problems, among them we have finite difference method, finite element method and boundary elements. Where the last two of them were the most used to calculations in fracture mechanics. Due to difficulties in mathematical crack tip modelling, many efforts have been done to overcome this problem. A significant advance using finite elements in linear elastic fracture mechanics was done after the development, done simultaneously and independently by Henshell and Shaw (1975) and Barsoum (1976), in special finite elements, known as quarter-points.

Stress intensity factor K_I has a important meaning in linear elastic fracture mechanics and its calculation is one of primer objectives in this analysis. As showed before, in linear elastic fracture mechanics we have a mathematical singularity in term $1/\sqrt{r}$ to stress calculate in crack tip. This singuarity difficulties stress intensity factor magnitude determination.

Investigations done by Henshell & Shaw and Barsoum had a important contribute to the development of computational fracture mechanics, have in view from theirs works the modelling in crack tip singuarity can be done with special finite elements. This finite elements, called quarter points, become very popular by its simplicity and for providing very good results using meshes not so refined. Modelling in the crack tip was done by quadratic isoparametrics elements, which can be triangular or quadrilateral, where the intermediate node is moved by ¼ to the distance of crack tip. This proceeding introduces a new element singularity between modelling parametric cordinate space and cartesian space.

As in Ansys the stress intensity factor calculation is solved by CTOD (crack tip opening displacement) technique, will be done a brief explanation about this subject. CTOD is one of the most simple techniques and historically is one of the first to be used in finite element stress intensity factor acquisition (Chan 1970). The node point displacement in a mesh obtained by finite element method is directly replaced by a analytical expression to the crack tip. Generally this point is chosen to be a node where the displacement will be larger in the crack face, thus the displacement error will be least. The following figure show the configuration to this approach.

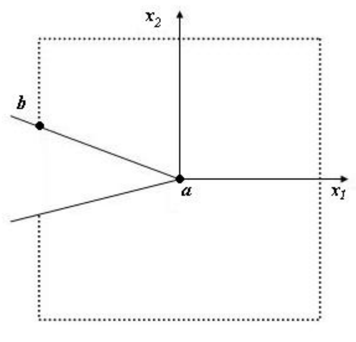

Figure 1. Correlate points distribution to CTOD calculation.

The analytical expressions used to stress intensity factor calculation in CTOD are:

$$K_I = \frac{\mu\sqrt{2\pi}\left(v_b - v_a\right)}{\sqrt{r}\left(2-2\upsilon\right)} \tag{3}$$

$$K_{II} = \frac{\mu\sqrt{2\pi}\left(u_b - u_a\right)}{\sqrt{r}\left(2-2\upsilon\right)} \tag{4}$$

$$K_{III} = \frac{\mu\sqrt{2\pi}\left(w_b - w_a\right)}{\sqrt{r}\left(2-2\upsilon\right)} \tag{5}$$

Where μ is the shear modulus, υ is the Poisson coeficient, r is the crack tip distance to the correlate point, and u_i, v_i e w_i are the displacements in x,y and z. This relation have the advantage to its simplicity and for effortless obtainment of three stress intensity factors. However, some care must be taken in chosing the correlate point to obtain a relative accurate result, and a good refined mesh in the crack tip region is necessary as well.

4. USE OF FRANC2D

FRANC2D/L software was born from the NASIP (Nasa Airframe Structural Integrity Program) program by NASA, which was a program developed to increase the understanding in structural integrity problems in many areas. This NASA program had the mission to develop computational engineering tools which were available for engineers in research centers and in industry. This tools shouldn´t have to solve any engineering problem, but each tool should be focused in specific problems to solve. Thus, FRANC2D/L (FRacture ANalysis Code-2D with Layers) code was developed to be a specific fracture mechanics software. This software was originally developed by Cornell University

The FRANC2D code was originated in response to necessity in modelling actual behavior of crack propagation using finite elements, even knowing the limitations in modelling this kind of problems. The author code decide for a new approach to criate his finite element algorithm, resulting in a new crack propagation analysis. The re-meshing operation in crack tip area is well defined and the problem geometry is preserved. This code can be downloaded in internet together with the software CASCA for free. FRANC2D analysis is made in two parts, firstly the geometry and mesh have to be defined in CASCA. In the second part in FRANC2D we determine the boundary conditions, problem characteristics, stress analysis, input crack singularities, propagate the crack and conclude the problem.

Four finite element kinds can be used in CASCA, quadrilateral with eight nodes (Q8), quadrilateral with four nodes (Q4), triangular with six nodes (T6) and triangular with three nodes (T3). In the case of using some of this elements, mesh can be structured or not. For boundary conditions definition the proceeding is the same used in any finite element code, and once calculated the problem, a crack can be inserted to make the crack propagation analysis. When the crack is inserted, the program cleans up the elements near the crack way, inserts the quarter points elements, and finally remeshes the previously region to solve the new problem.

The methods used to calculate the stress intensity factor in FRANC2D are: Integral J, calculated by equivalent domain integral method, and Energy Release Rate calculated by crack closing method. A graphical analysis can be done from the Paris model to determination of crack propagation rate, where a plot with crack lenght versus life cycle number is generated. In FRANC2D user´s guide, as quoted in references there is a step-by-step model showing how to use the program, since the crack geometry in CASCA until crack propagation in FRANC2D.

In the following diagram, is presented a schematically approach used in the program. Due to the requirement of input a arbitrary crack increment to crack propagation analysis, is necessary to calculate externally the crack propagation rate *da/dN.* A small algorithm was written to calculate the increment using the Paris equation, this increment is referred to the stress intensity factor calculated in the previous step. Thus, for each step is calculated a increment with the stress intensity factor from the earlier step, in this manner, the user can input the increment value in FRANC2D to perform the crack propagation analysis.

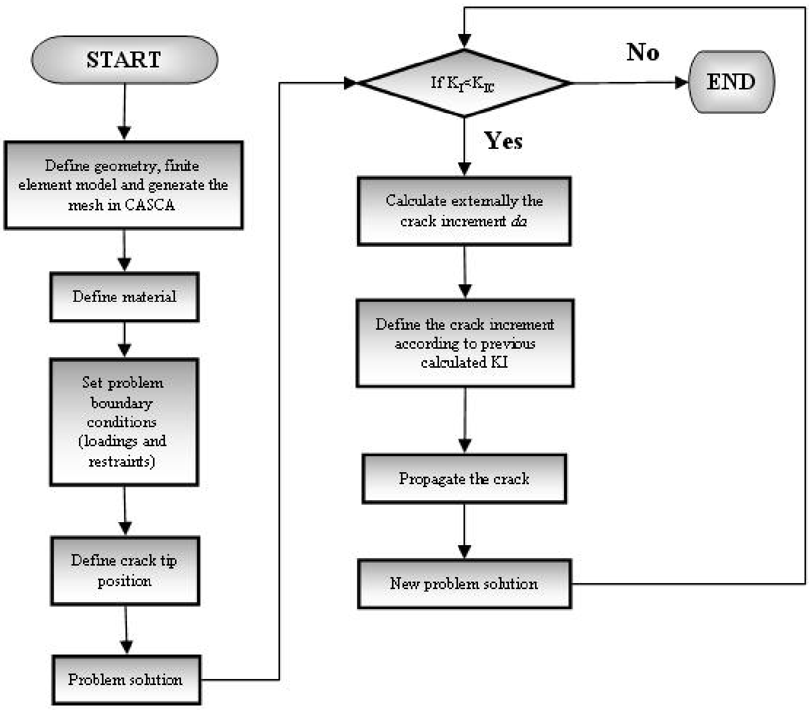

Figure 2. Diagram of crack propagation approach in FRANC2D.

5. USE OF ANSYS IN FRACTURE MECHANICS

The Ansys finite element software has a specific tool to calculate stress intensity factor (command KCALC), and this K_I computation is done from the Crack Tip Opening Displacement Method. However, despite its versatility and wide using in industry, the Ansys program don´t have a proceeding to compute fatigue crack propagation. In this way, will be presented in this section a automated way to fatigue crack propagation in two dimensional geometries using Ansys.

Fracture mechanics problems resolution in Ansys can be accomplished using linear elastic theory and elasto plastic theory analysis. The most important region in fracture mechanics model is the region which includes the crack tip, in this case this is the differentiation parameter between a simple stress analysis and a analysis envolving stress intensity factor in Ansys. This modelling is made using the command KSCON, where the main parameters to be defined are the keypoint where the crack tip will be stated, the radius and number of elements in the singularity region of crack tip.

The modelling can be done using the symmetry advantadge (when it´s the case), where the user can use symmetry with only one-half of the model utilizing the proper symmetric or assymetric boundary conditions. In the process of obtaining K_I it´s necessary to specify a path containing three nodes, from the crack tip node to the external node which defines the crack. This path is necessary to the CTOD computation, as explained before, we should have the displacements nodes representing the crack.

Once accomplished this analysis, the stress intensity factor value can be taken from the post processing part of the program, POST1, where the parameters used in command KCALC are input.

5.1. Crack Propagation Methodology in Ansys

A crack propagation methodology was implemented in Ansys from a group of input files written in APDL language (Ansys Parametric Design Language), which is a language used in Ansys to automate common tasks or to write a model in terms of different variables. APDL language also comprises many other features, as repeating commands, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations.

In this manner, for implementation were created three input files, where the first loads the two others, one performing the stress intensity factor computation and the other performing stable crack growth in accordance with K_I previously calculated. In other words, firstly a simple stress intensity factor analysis is performanced in the pre-established crack, and afterwards this value is compared with crictical fracture toughness value to the material K_{IC}, so that in accordance with the imposed condition the crack will propagate or not.

In each new crack propagation step, the crack growth rate *da/dn* is computed in accordance with the earlier K_I. This crack growth rate is computed using Paris equation, which was well explained before. Thus, the next crack propagation step is performed with the new *da/dn* calculate in the previous step, and this cycle repeats until the end of the loop command. The crack moves in accordance with *da/dn* growth rate, and this growth happens with the total sum of previously *da/dn* calculated.

In this approach the crack tip is determined by a keypoint and by a boundary condition restraining the crack line. Therefore, a initial idea to make the crack move was just moving the keypoint to the new position (in terms of *da/dn*) and redefine the boundary condition. However, Ansys has a hierarchy to the geometry definition, a solid model has to be defined in terms of keypoints, lines, areas and volumes. In this case, where we will have just areas, the crack tip moving have to modify the whole area defining the geometry.

In this manner, to use the chosen approach some changens were performed. Fistly was necessary to clean up the finite element mesh, delete all the area, remove the boundary line condition and delete the line which encloses the crack tip keypoint, to finally delete the keypoint and re-create in the new position, as well as the other components that defines the crack and geometry. Having the keypoint in a new position, all the others usual proceedings are performed, which are defining crack tip characteristics, re-defining boundary conditions, create the new mesh and run the program again.

This proceeding is contained inside a loop with "n" steps, using the *DO command, where each propagation step corresponds a new cycle. And crack growth condition is perfomed by *IF command, where the crack will propagate in the case of K_I being lesser than the correspondent K_{IC}. This condition corresponds to the stable crack propagation phase, in the case of K_I greater than K_{IC} the crack will propagate in a catastrophic way. That is, this proceeding will be repeated many times until the crack reaches a dimension where the computed stress intensity factor reaches a maximum stable condition to crack propagation. The results obtained with stress intensity factor are printed in a apart output text file using the command *VWRITE. In the following diagram is presented a brief approach with the necessary steps used to perform the crack propagation analysis in Ansys.

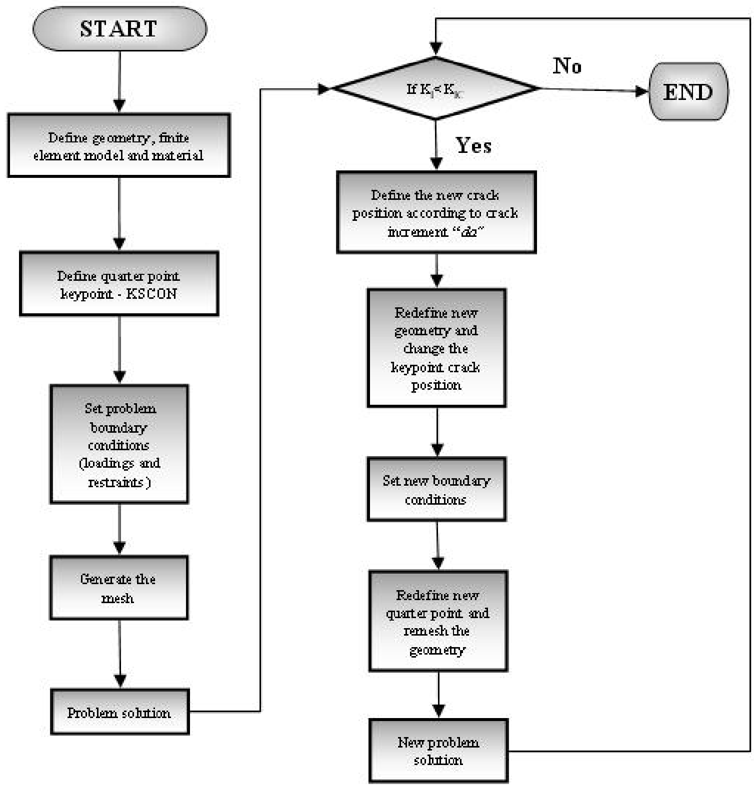

Figure 3. Diagram of crack propagation approach in Ansys.

6. CASE STUDY

As case study was considered a finite width plate with small sharp but central notch. The specimen is subjected to a cyclic stress of constant amplitude. This geometry was chosen to attend the most simple case to a initial crack propagation analysis, and this will be useful to Ansys APDL code validation. Fracture toughness value K_{IC} and material constants C and m are collected in literature (Downling 1998), where $C = 2{,}71 \cdot 10^{-8}$, $m = 3{,}7$ and $K_{IC} = 29MPa\sqrt{m}$. Figure bellow shows up the study geometry with its values and characteristics.

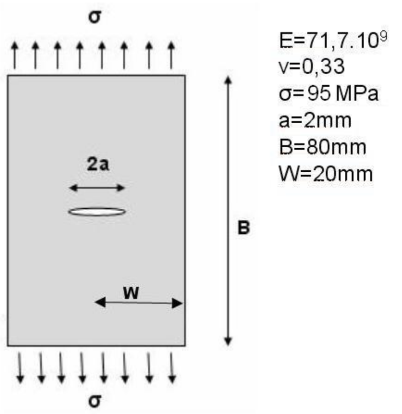

Figure 4. Geometric schematic view used in case study.

The simulations were performed considering a loading cyclic amplitude with $\sigma_{max}=95MPa$ as maximum stress and a minimum stress, σ_{min} equal to zero, in other words, $R=0$. To a Ansys simulation were inputted forty propagation steps in the program loop, however the program stops running by thirty seven steps. The program stops before because K_I value was in the edge to reach K_{IC} value. FRANC2D simulation obey the same steps propagation number, but this was determined by the user, since any stop criterion was performed to stop the crack.

In both programs 8 node quadrilateral elements were used in plane strain condition. Element PLANE82 for Ansys case and Bilinear 4side for FRANC2D. In Ansys were used just one fourth of the geometry with the respective boundary conditions. In FRANC2D were used the same boundary conditions, but the simmetry relation is not the same. FRANC2D requires both sides of geometry which will be separated by the crack growht to represents the crack tip, this is just a construtive consideration. In the following figure this boundary condition explanation becomes easier to understand.

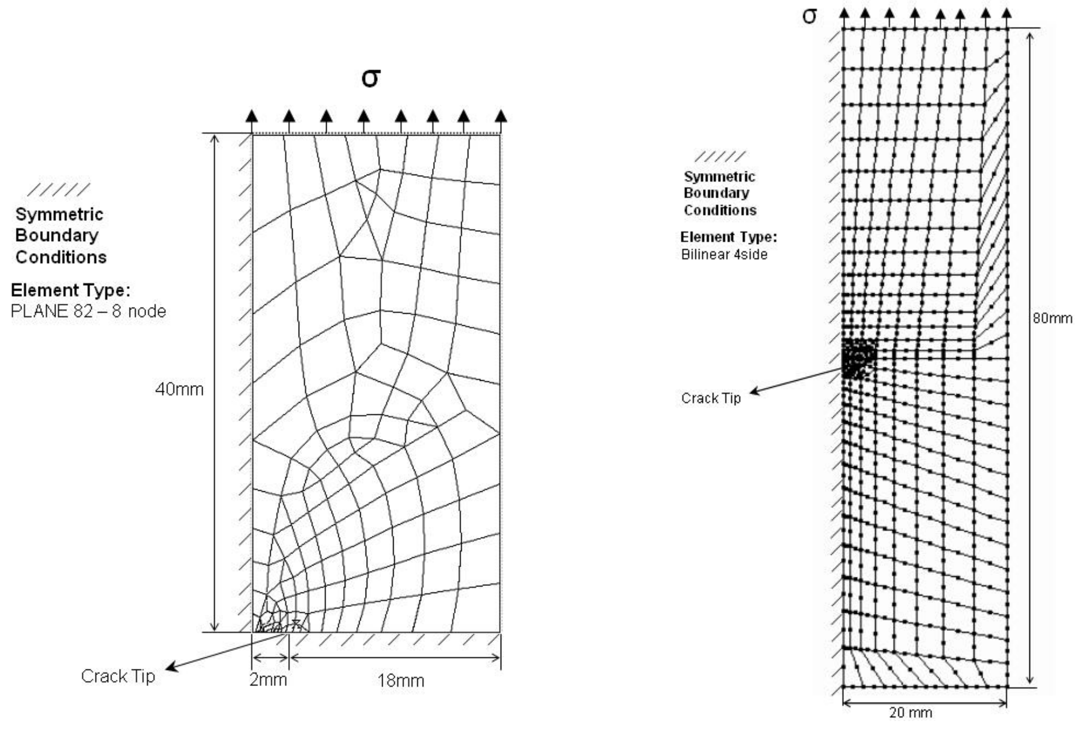

Figure 5. Boundary conditions in Ansys and FRANC2D.

The obtained crack progress and stress intensity factor K_I data are plotted in the figure below, and this graph will be useful to compare Ansys and FRANC2D results with analytical results.

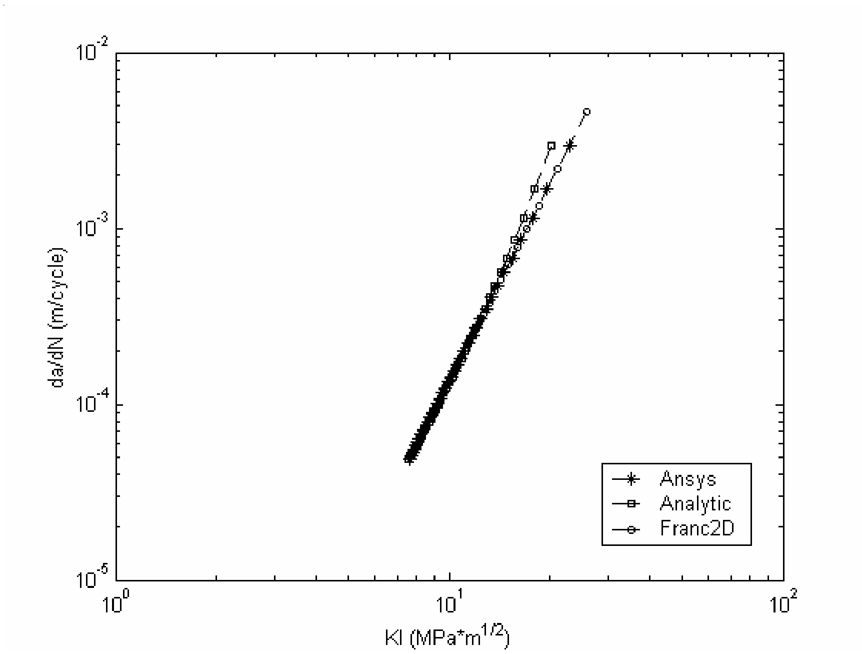

Figure 6. Diagram of crack propagation rate versus stress intensity factor.

Obtained results in the graph are equivalent to the stability crack propagation área, that is, this is exaclty the validity region for Paris equation, and the result is a straight line as expected. The analytic curve separates a little to the Ansys and FRANC2D curves in the final part of the process, this happens because the geometric factor from the analytical equation trends to differ in proportion to the crack lenght increasing. Ansys and FRANC2D curves maintain practically in the same line along the crack growth path.

Fatigue crack growth process can be defined in three stages. The first is the crack initiation stage, with slow crack growth in the order of $da/dN \leq 10^{-6} mm/cycle$. In first stage the crack will begin in a free crack geometry when the K_I value reaches a threshold fracture thoughness value K_{th} (in this case the value is $K_{th} \leq 7 Mpa\sqrt{m}$). The second stage is the controled crack propagation region, showed in the figure above, where the K_I valeu must varies between K_{th} and K_{IC}. Crack growth rate *da/dN* for the second stage varied between 10^{-5} and 10^{-3} *mm/cycle.* The third stage is the instable crack propagation, where the complete crack colapse occurs, which is not represented in the graph

7. CONCLUSION

Using of finite element codes to complex engineering problems solution is widely spread in many fields. A good understand about the operation code and the problem to be solved is very important to obtaining good results. Thus, this job is presented to contribute with the understanding in fracture mechanics solutions using finite element programs in a way to help engineering students or interested researchers in the area to use fracture mechanics programs. The continuity in researching and studying the correctly finite element utilization with applications in fracture mechanics is inserted in a research project performed by GAMMA (Advanced Mechanics of Materials Group) in University of Brasilia. This research project aims the indentification of fatigue crack conditions in materials used to construct hydraulic turbine blades, supported by Eletronorte.

8. REFERENCES

Ansys User's Manual. Theory Manual. Ansys 10.1.

Broek, David. The Practical use of Fracture Mechanics. Kluwer Academic Publishers. 1989..

Broek, David. Elementary Engineering Fracture Mechanics. 3 ed, Martinus Nijhoff Publishers.1982

Callister, W. D. Jr. Ciência e Engenharia de Materiais: Uma Introdução. LTC, 5ª Ed. 2000.

Downling, Norman E. Mechanical Behavior of Materials: engineering methods for deformation, fracture and fatigue / Norman E. Downling. – 2nd ed, 1998.

Ingraffea, A. R. "Computational Fracture Mechanics," Encyclopedia of Computational Mechanics, John Wiley and Sons , to appear (2004).

Miranda, A.C.O. Propagação de trincas por fadiga em geometrias 2D complexas sob cargas cíclicas variáveis. PUC – Rio de Janeiro. Departamento de Engenharia. Civil, 2003.

M.S. Alam, M.A. Wahab. Finite element modeling of fatigue crack growth in curved-welded joint using interface elements. SID, vol. 1, no. 3, pp.171-184,2005

Newman, J. C. Jr., Advances in Finite Element Modeling of Fatigue Crack Growth and Fracture. Fatigue '02: The Eight International Fatigue Congress, Stockholm, Sweden, June 2-7, 2002.

Paris, P. C., and Erdogan, F., A Critical Analysis of Crack Propagation Laws. Journal of basic Engineering, pp. 528-534,1963.

Paris, P. C., The growth of fatigue cracks due to variations in load, Ph.D. Thesis, Lehigh University (1962).

R.S Barsoum, On the use of isoparametric finite elements in linear fracture mechanics. International Journal of numerical of Methods in Engineering, 1976, 10, 25-37.

R.S. Barsoum, Triangular quarter-point elements as elastic and perfectly-plastic crack tip elements. International Journal of numerical of Methods in Engineering, 1977, 11, 85-98.

R.D. Henshell and K.G. Shaw, Crack Tip finite elements are unnecessary. International Journal of numerical of Methods in Engineering, 1975, 9, 495-507.

S.K. Chan, I.S. Tuba and W.K. Wilson, On the finite element method in linear fracture mechanics. Eng Fracture Mech, 1970, 2, 1-17.

Swenson, D., and James M. FRANC2D/L: A Crack Propagation Simulator for Plane Layered Structures. Version 1.4 User's Guide. Kansas State University.

9. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.