International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

# Modeling and Stress Analysis of Aerospace Bracket Using ANSYS And FRANC3D

**K.Kalyan Kumar [1], P.S.Srinivas[2] and D.Srinivasa Rao[3]**

[1]Velagapudi Rama Krishna Siddhartha College of Engineering ,Vijayawada-520007, A.P., India,

[2] Department of Mechanical Engineering, Velagapudi Rama Krishna Siddhartha College of Engineering ,Vijayawada-520007, A.P., India,
[3] Department of Mechanical Engineering, DMS SVH College of Engineering, Machilipatnam-521002, A.P., India,

## Abstract

Design of brackets used in aerospace engines needs various shapes and sizes for mounting bleed air ducting, starter air duct fuel lines and hydraulic lines engine core. These brackets when subjected to high cyclic fatigue loads cracks may be generated in them and make the bracket unstable during a mission. Early detection and monitoring of these cracks in these are needed before it completely breaks to avoid many engineering disasters. In this paper an attempt has been made to investigate the propagation of crack and its growth in a typical Ti-6AL-4V aero space bracket .Finite element program ANSYS is used to simulate crack and to estimate the stresses and stress intensity factor.FRANC3D is used for crack generation and its growth. The crack was investigated by selecting a specific type of crack in brackets as engineers found this type is justified. The yield strength of the material is compared against the von-misses stress near the crack tip .Thus monitoring of cracks and early detection of them can be carried out in air craft engine components.



**Keywords**:Aerospace Brackets, Condition Monitoring, Modeling,Titanium,ANSYS, FRANC3D

## Introduction

Aero engine designers design brackets in various shapes and sizes for mounting bleed air ducting, starter air duct, fuel lines and hydraulic lines to the engine core.

Many of the ducting supports used in Aero engine design have the shapes of L,T and Z to accommodate to multiple tubes in the tight space found in an aircraft. Generally the bracket thickness ranges from 0.125 to 0.5inch as brackets must be thin to reduce weight and cost and to serve its purpose in extreme environment. Any crack found in a bracket may cause the ducting to become unstable during a mission and thus induce high cyclic fatigue load on the overall major structures and shorten the structure life. Hence complete monitoring and early detection of cracks in these brackets is essential to replace broken brackets which mitigate the risk of damaging other components such as ducting. Monitoring of these cracks on brackets reveals the fatigue life of the component.

**Literature Review**

Nickel-based alloys such as Inconel 718 and Inconel 625 are widely used in aerospace industry for ducting and brackets. Honnorat [2]in his work proved only Titanium alloys could with stand the increasing demand for high strength per weight materials that needed for wide range of components even though Nickel based alloys such as Inconel 718 and Inconel 625 are widely used. Many aircrafts use Titanium due to their high tensile strength to density ratio, high corrosion resistance, fatigue resistance, high crack resistance and ability to with stand moderately high temperatures without creeping [4].About two thirds of all the Titanium metal Produced is used in air craft engines and frames. Among all the Titanium based alloys Ti-6-4 is by far most widely used according to Immarigeon et al.[3]. The behavior of these materials under aggressive environment as well as impact loads make them attractive for aero space applications. In these materials the most important concerns are the brittle inclusions, which are difficult to detect by non-destructive testing[2], can initiate cracks and an early failure of the structures. In this paper an attempt has been made to investigate the corner crack growth in a Titanium based alloy bracket. The work examines the stresses near the crack tip, computes the stress intensity factors and compares it against material toughness to determine the influence of the crack on the bracket by modeling and analyzing it.

## Theoretical Back Drop

Linear elastic material behavior is often assumed for the analysis of stress surrounding the crack tip for any homogeneous and isotropic material [5].The method of linear elastic fracture mechanics assumes the plastic region near crack tip is much smaller than the dimensions of the crack and structural member. This is very important concept and is called as small scale yielding for simplifying the stress analysis near crack tip. The surface displacement of crack is described by three modes. In mode I the crack surfaces move directly apart. Mode II is sliding or in plane shear mode where the crack surfaces slide over another in a direction perpendicular to the leading edge of the crack. Mode III is tearing and anti-plane shear mode where the crack surface move relative to one another and parallel to the leading edge of the crack. Within the scope of the theory of linear elasticity, a crack introduces a discontinuity in the elastic body such that the stresses tend to infinity as one approach the crack tip. The relation can be written in simplified form as $\sigma \approx \frac{K}{\sqrt{2\pi r}}$ ---------1.



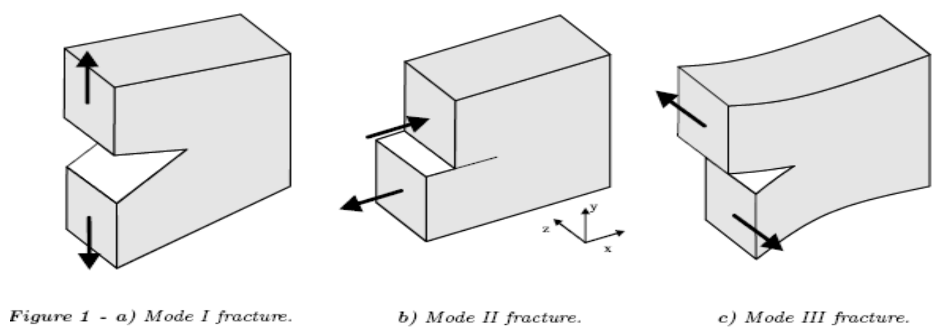


Fig.1 Different Modes of Fracture

Where $\sigma$ the stress, $K$ is the stress intensity factor and r is the size of the crack. The tensile stress in X and Y directions, and the shear stress in the X-Y plane can be calculated in terms of K and position can be written for Mode I as:

$$\sigma_x = \frac{K}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left(1+\sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right)$$

$$\sigma_y = \frac{K}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left(1-\sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right)$$

$$\sigma_{xy} = \frac{K}{\sqrt{2\pi r}}\left(\cos\frac{\theta}{2}\sin\frac{\theta}{2}\cos\frac{3\theta}{2}\right)$$

The fracture toughness can be defined in terms of the stress intensity factor K, but at a critical stress state as

$$K_{IC} = Y\sigma\sqrt{\pi a} \text{ --------------3.}$$

Where Y is a dimensionless parameter that depends on both specimen and crack geometry, $\sigma$ is applied stress and 'a' is crack length. Y factor is 1.0 for plate of infinite width and 1.1 for a plate of semi infinite width. When the thickness of the specimen is very large with respect to the crack length, the stress intensity factor for Mode I is often called as the plain strain Fracture Toughness. The plain strain fracture toughness for Mode I is $K_{IC}$ also a function of many other factors such as temperature, strain-rate and micro structure.



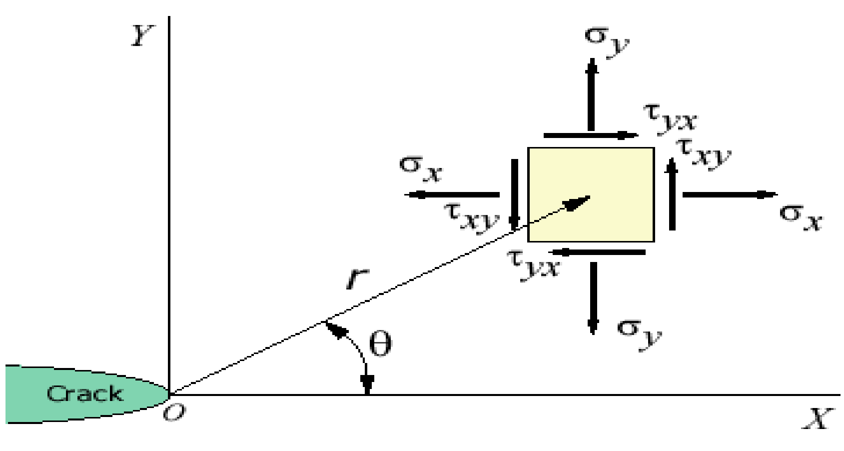


Fig.2 Fracture Toughness $K_{IC}$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

**Methodology**

In the present work a new bracket is modeled and crack analysis was done on the model to suit requirement. The dimensions of the bracket are 181.6mm x 65mm x 8mm. It has three holes and each of the diameter 8mm and a slot. Holes are located at a distance of 12mm from base and slot is located at a distance of 41mm from base. This bracket is used to hold fluid lines and electrical harness assemblies which connects engine to other components.

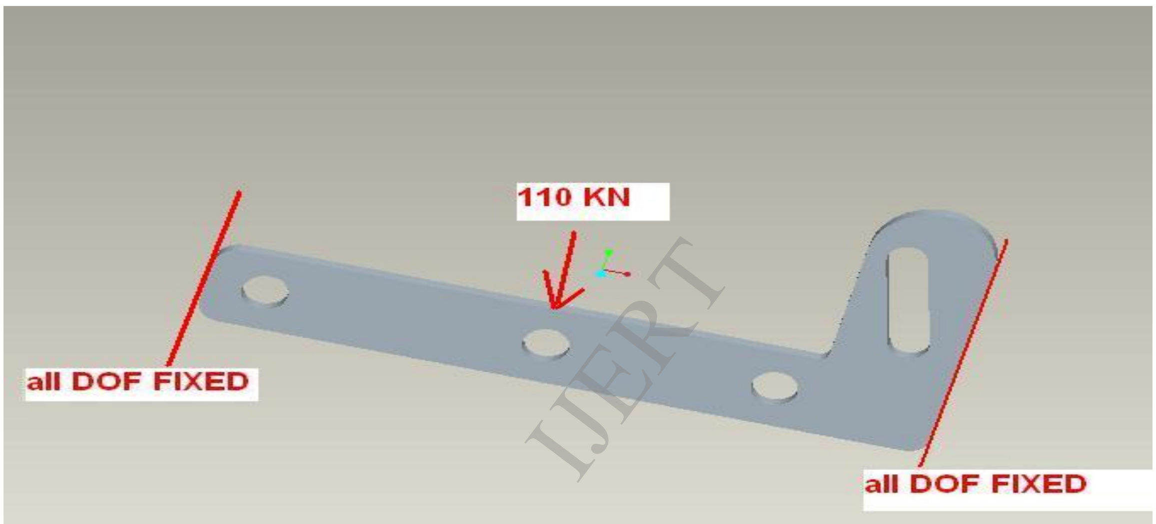


Fig.3 The Undamaged Bracket and Boundary Conditions.

The first step of the analysis was to perform finite element elastic stress analysis on the undamaged bracket to determine the stress distribution across the entire bracket and identify the weakest point or high stress regions. The bracket was meshed in the ANSYS environment using 20-noded brick elements (Solid95) since this is the only type of element compatible with FRANC3D. Boundary conditions were applied to the bracket as shown in Figure 1 and a single load step static stress analysis was performed on the bracket using ANSYS. Once the calculation was completed, the ANSYS postprocessor was used to identify the high stress region for

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

introduction of the initial crack. Finally, ANSYS wrote a database DB file (.cdb), which was subsequently used as input to FRANC3D for crack growth analysis.

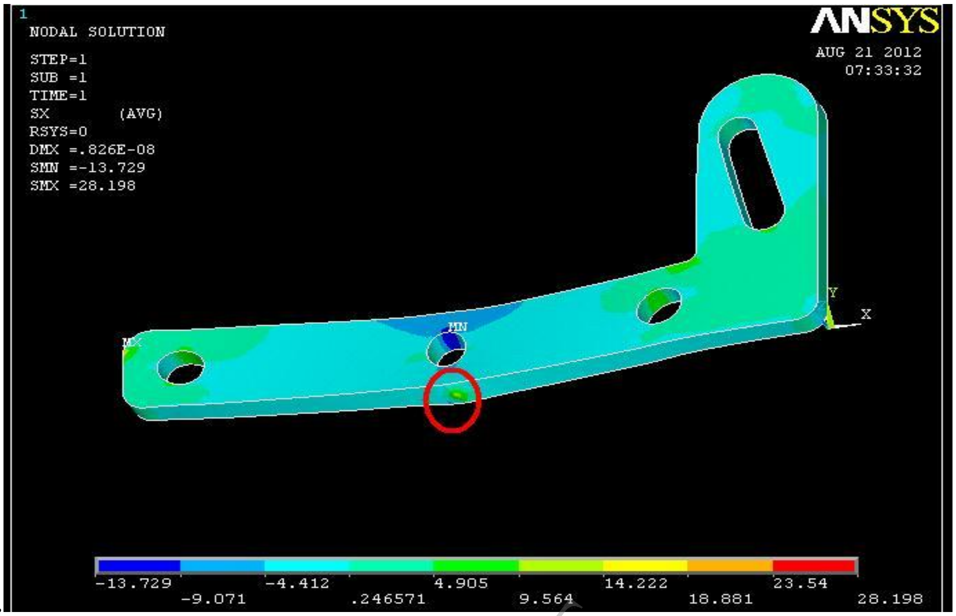


Fig.4 Stress Distribution of the undamaged bracket (Von Mises)



Step two of the analysis consists of introducing an initial crack in the structure by creating the basic geometry of the crack. After opening in FRANC3D, the .cdb file produced by ANSYS in step one, an initial crack was inserted in the structure by creating a half-millimeter radius circular region, and positioning it at the weakest point identified in step one, see in Figure 4. The stress singularity at the top left corner of the bracket was due to point load effect, so its influence on the structure was not real and was not considered. One quarter of the circular crack was inserted near the middle hole to simulate the corner crack. After that, FRANC3D automatically generated finer elements locally around the inserted Crack. To ensure the new node pattern was not predetermined by the old mesh, FRANC3D erased the old elements around the crack before inserted new elements of the proper shape and number. Subsequently, FRANC3D filled the area between the new elements and the unmodified portion of the mesh with transition elements. Finally, the FRANC3D GUI was used to create a new text file (.macro) readable by ANSYS for subsequent stress analysis.

Step three consists of performing elastic stress analysis of the cracked bracket produced by FRANC3D using ANSYS. The macro file contains all the procedures for running the load step analysis and post-processing the results.

Step four of the analysis consists of using FRANC3D to compute the stress intensity factor and to further extend the crack based on the node displacements at the crack tip obtained from the results file (.dtp). The direction of crack propagation is determined by a "propagation angle criterion" and the stress-intensity factors are calculated. The crack is then extended and a rosette of quarter-point singular elements are placed around the new crack-tip. In each iteration, the authors used the default value for the crack tip size. Then the program computed the stress-intensity factor based on the stress result (.dtp) and the new crack length. The new crack length is calculated based on a polynomial series, where the variable of the series is the displacement of the node at the crack tip [7]. In this study, the authors repeated steps three and four twice to examine the partial propagation of the crack.



**Results**

The stress field around the crack is shown in fig.5. The initial and propagated cracks in FRANC3D are shown in fig.6-7.The variation of stress intensity factors for Mode I type of crack for both initial crack and propagated crack are shown in fig.8-9. Analysis of all these results helps us in monitoring and deciding the component sustainability.

Stress field around initial crack is 56 GPa

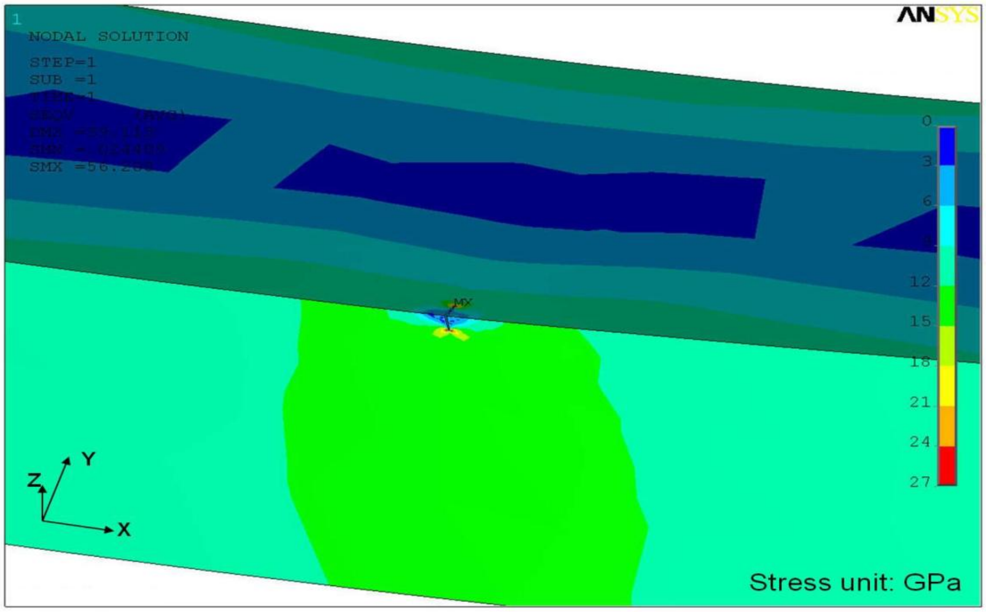


Fig.5  Stress Field around the Crack.

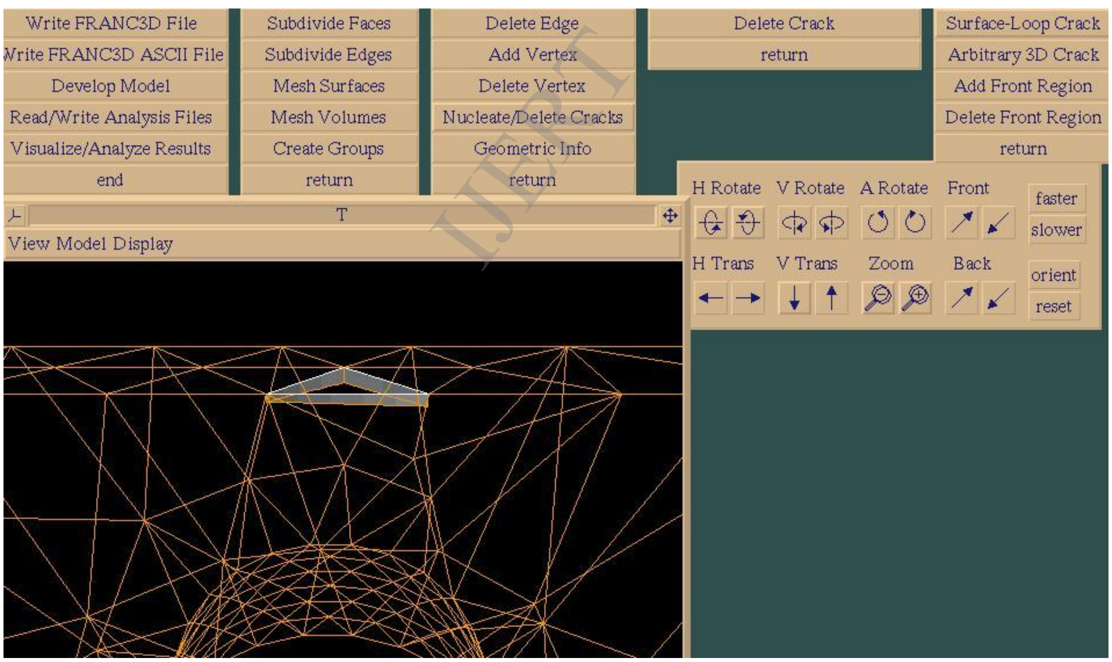


Fig.6 Initial crack in FRANC3D

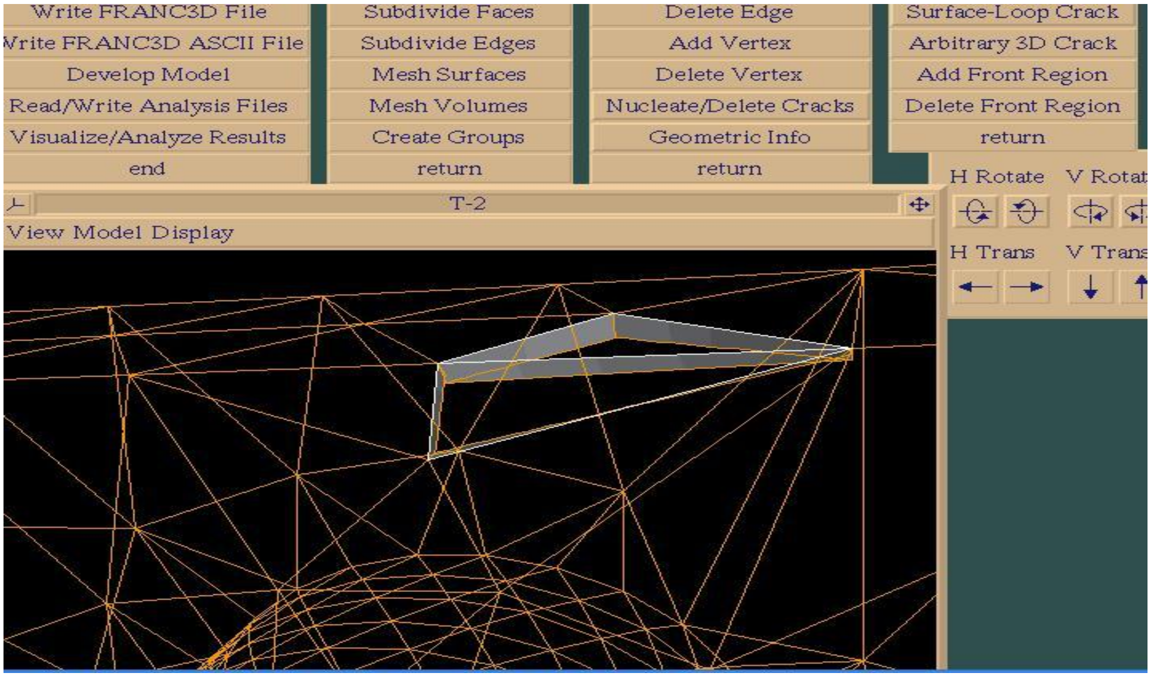


Fig.7 Propagated Crack in FRANC3D

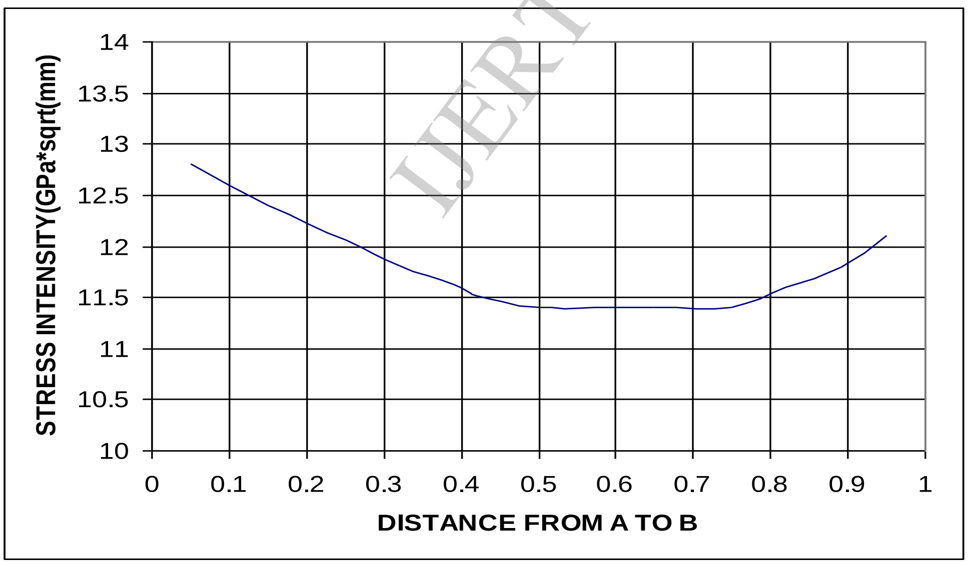




Fig.8 Mode I stress-intensity factors at initial Crack from one end to other end

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

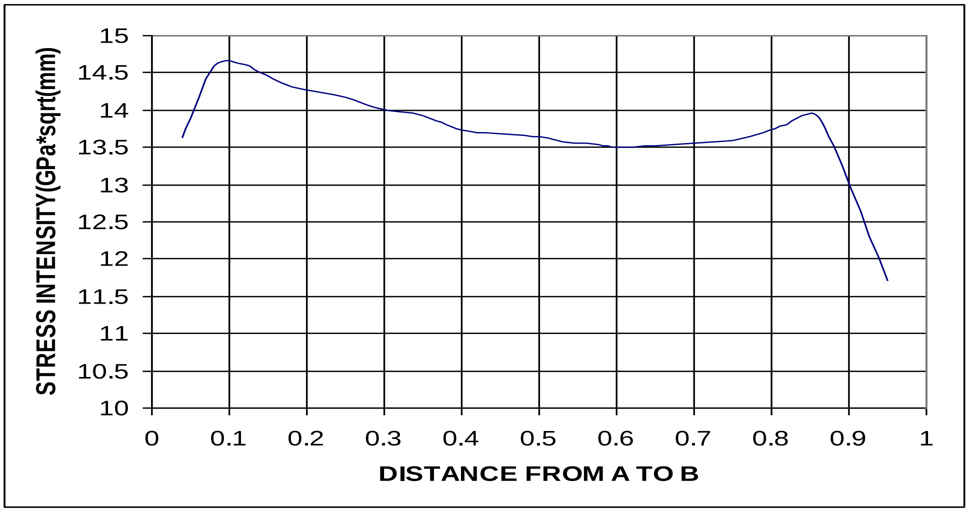


Fig.9 Mode I stress-intensity factors for propagated crack from one end to other end.

## Conclusions

In this work an attempt has been made to find the process of crack propagation and stress distribution in a typical Ti-6Al-4V aerospace bracket by using ANSYS and FRANC3D.This type of analysis is more economic and time saving phenomenon and can be used to monitor the cracks in various components of aerospace structures and components. The first step of the analysis consisted of using ANSYS to perform elastic stress analysis on an uncracked bracket to identify the high stress regions. In step two, the uncracked model was imported to FRANC3D and an initial crack of simple geometry was introduced and several ANSYS files were created with crack. Step three of the analysis consists of using ANSYS to perform elastic stress analysis of the previously cracked bracket produced by FRANC3D. For the model of cracks, the results show that the Von Mises elastic static stress is above the yield strength for the two load cycles considered in this work. The Mode I stress intensity factors for the cracked model are below the material's fracture toughness. Therefore, it appears that the bracket can tolerate small corner cracks in the structure. The analysis procedure thus can be used to continuously monitor the brackets and to take appropriate decision on time of replacement of such brackets.

ISSN: 2278-0181
Vol. 1 Issue 8, October - 2012

**REFERENCES**

[1] FRANC3D & ANSYS Tutorial. September 2010.

http://www.fracanalysis.com/Franc3D Documentation.

[2] Honnorat, Yves. "Issues and breakthrough in manufacture of turbo engine titanium parts." Materials Science and Engineering, A213, 1996: 115-123.

[3] J.P.Immarigeon, R.T.Holt, A.K. Koul, L.W.Zhao, Wallace and J.C. Beddoes ,

Lightweight Materials for Aircraft Applications, NRC Institute for Aerospace Research, 1995.

[4] Titanium. http://en.wikipedia.org/wiki/Titanium (accessed January 12, 2011).

[5] Barsom, M. John , Rolfe and T. Stanley Fracture and Fatigue Control in Structures:

Application of Fracture Mechanics, Philadelphia, 1999.

[6] C.P.Paris, and G.C Sih, “Stress Analysis of Cracks,” in Fracture Toughness Testing and Its

Applications, ASTM STP 381, American Society for Testing and Materials, Philadelphia, 1965.

[7] B.J.Carter, P.A. Wawrzynek, and A.R.Ingraffea, Automated 3D Crack Growth

Simulation, Cornell Fracture Group, 2003.