Journal of Manufacturing Engineering, 2007, Vol.2, Issue.2

REVIEW ON FINITE ELEMENT APPROACHES IN STRUCTURAL ANALYSIS OF AERO-ENGINE BLADE DISC ATTACHMENTS FOR FATIGUE LIFE ESTIMATION AND FAILURE ANALYSIS

K. Anandavel[1*], Raghu V. Prakash[2], Vivek Sanghi[3] and Kirit Patel[4]

[1] Senior Project Manager, Infotech Enterprises Limited, Bangalore.
[2]Associate Professor, Department of Mechanical Engineering, IIT Madras, Chennai.
[3] Deputy General Manager, Infotech Enterprises Limited, Bangalore.
[4] Senior Fellow and Director India Operations, Pratt & Whitney Canada

***E-mail:** anandavelk@infotechsw.com

ABSTRACT

Aero-engine rotors are critical components, as their failure during service has a direct consequence on aircraft flight safety. Failure critical regions such as blade-disc attachment, assembly holes, weld areas and hub are located in the compressor and turbine stages of the rotor assembly and it is essential to validate and certify their design life. Of the several failure critical regions, blade-disc attachment is considered as the most critical element in the fatigue analysis of a high performance aero-engine. Conventionally, a dovetail and fir tree attachment is used for securing the blades with the disc (both in the compressor and turbine stage), although integral blades and disk is used in some stages of compressor rotor. During the operation of aero engine, blade disc attachment endures frequent occurrence of high frequency, low amplitude stress due to vibratory stress. In addition to the above, variation of centrifugal stress, bending stress and thermal stress corresponding to various flight phases (such as take-off, cruise and landing) lead to high amplitude, low cyclic fatigue (LCF) loading. Analysis of state of stress in the blade-disc attachment is a challenging task as it involves unknown boundary conditions, varying levels of contact stresses and clearances at the interfaces of the blade disc assembly. Photo elastic method was used earlier for analyzing the stresses at the blade disc assembly. Finite element method is widely used in the recent times for the analysis of stresses and strains at the blade disc attachment, as well as to evaluate the dynamic characteristics of rotors leading to failure life prediction. This paper reviews the different approaches adopted during finite element analysis of the blade-disc attachment, their advantages and limitations and identifies the state of art. Based on the review, the following may be summarized; a)Modern finite element codes have matured to a stage where the interface conditions between blade root and disc with thermal mechanical loading at the blade disc attachment can be effectively handled starting from a simple analysis which assumes infinite coefficient of friction at the interface to complete modeling and assessment of the extent of interface motion. b) A three dimensional approach to model the problem, although two-dimensional representation may still play an important role in the design process by the way of reduced design and development cycle time. c) Sub-modeling (global-local) approaches to predict the high stress gradient near the contact edges of blade disc attachment. d) Integration of finite element analysis with metallurgical examination in failure analysis of aero engine rotors.

Keywords: *Fatigue, Failure analysis, Dovetail, Fir tree, sub modeling*

1. INTRODUCTION

Aero-engine rotors are critical components, as their failure during service has a direct consequence on aircraft flight safety. Failure critical regions such as blade-disc attachment, assembly holes, weld areas and hub are located in the compressor and turbine stages of the rotor assembly and it is essential to validate and certify their design life of the several failure critical regions, blade-disc attachment is considered as the most critical in the fatigue analysis of a high performance aero-engine. This point is substantiated in failure investigation studies [1-7], which have identified that crack initiation and its subsequent propagation due to fatigue loading as the primary reason for failure at the blade disc attachment. The cause for failure has been assessed to be due to the initiation of fatigue cracks under high amplitude, low cycle fatigue loading, and the subsequent growth during low amplitude, high cycle loading. The primary failures are observed both on the disc and blade at the attachment zone. Conventionally, a dovetail and fir-tree attachment is used for securing the blades with the disc (both in the compressor and turbine stage), although integral blade-disk unit is used in some stages of compressor rotor. Figure 1 presents the typical cracks that have occurred during service at the dovetail and fir-tree attachments.

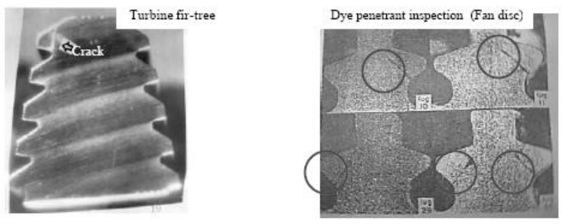


Figure 1- Location of crack initiation in fir –tree dovetail [4] [7].

During the operation of aero engine, blade-disc attachment endures frequent occurrence of high frequency, low amplitude stress due to vibratory stress. In addition to the above, variation of centrifugal stress, bending stress and thermal stress corresponding to various flight phases (such as take-off, cruise and landing) lead to high amplitude, low cyclic fatigue (LCF) loading. There is an acute variation in stress and displacement that occur at the edges of contacting surfaces in the blade-disc attachment zone. This contact edge effect can be significant when frictional contact forces are present and play an important role when evaluating the susceptibility of a component to fatigue due to surface damage. Analysis of state of stress in the blade-disc attachment is a challenging task as it involves unknown boundary conditions, varying levels of contact stresses and clearances at the interfaces of the blade disc assembly. Experimental techniques such as photo-elasticity, holography, brittle lacquer coating and strain gages have been used earlier, for estimating the stress concentrations at the blade disc assembly and optimizing the fillets in the attachment zone as presented in [7-10]. The inherent disadvantages associated with the experimental techniques are that of high cost and long lead-time. The modern competitive aero-engine market demands cost effective design and shorter design cycle time, which leads the manufacturer to take advantages of advancements in the numerical methods (Finite element method) and high performance computing hardware for the iterative design analysis. Finite element method is widely used in the recent times for the analysis of stresses and strains at the blade disc attachment, as well as to evaluate the dynamic characteristics of rotors leading to failure life prediction. This paper reviews the different approaches adopted during finite element analysis of the blade-disc attachment in stress analysis, their advantages and limitations and identifies the current practices and state of art.

2. REVIEW OF FINITE ELEMENT APPROACHES

A typical aero engine rotor assembly consists of a number of blades that are attached to the disk (shaft) by means of a dovetail or fir-tree attachment. During assembly of the blade to the disk, it is important to ensure that there is some clearance at the dovetail to insert the individual blades on to the disk. However, once the blade-disk assembly is loaded during normal engine operation, the individual blades exert a centrifugal component of force to the disk. As centrifugal component of force is directly proportional to the square of the rotor velocity, any change in the rotor speed affects the loading on the blade-disk assembly. If the blade-disk assembly has constant clearance at the time of assembly, the centrifugal forces would remain uniform at the disk. However, limitations in assembly and inspection procedures restrict our ability to ensure constant clearance for all the blades that are assembled to the disk. That causes additional variation in stresses due to contact and separation at the blade-disk attachment. To enhance fatigue life, normally the disk-blade components are subjected to shot peening to induce residual compressive stresses at the time of manufacture. During fatigue modeling, one has to consider the effect of residual stresses on alternating stresses. Further, variation in temperature due to combustion of gases can result in thermal loading of the engine components, in addition to the mechanical loading discussed above. All these make the fatigue analysis of disk-blade attachment to be a complicated exercise, both from the view point of analysis and experimentation. Failures observed on turbine rotor during service and developmental testing is frequently attributed to high stress gradients at the edges of contact between blade and disc. Accurate assessment of nature of contact stress and interfacial motion is imperative to assess the durability of blade-disc attachment, which is subjected fatigue damage. Accurate prediction of results at the blade-disc attachment depends on various technical aspects of FE modeling such as geometrical consideration (2D or 3D) for finite modeling, type of elements, element sizes, boundary conditions at the interface and on overall FE model, interface friction coefficient, numerical solution methods and results convergence. Various FE approaches and developments for prediction of stresses at the blade-disc attachment are discussed as below. Finite element approach and code was developed by S. K. Chan, et al. [11-12] to solve the problems of elastic contact bodies and applied to turbine blade fastenings to study the effect of clearance, friction and load on the stresses in turbine blade fastening. The code is based on displacement formulation using first order function and could handle plane stress and plane strain problems with triangular elements. Comparison of finite element results on blade-disc attachment with photoelastic results show the same overall trend but lacked detailed agreement at some location as shown in figures 2-4.

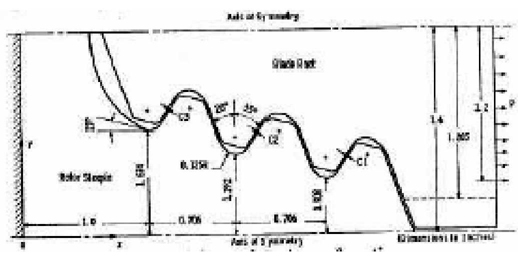


Figure 2- Fir-tree root fastening configuration [12]

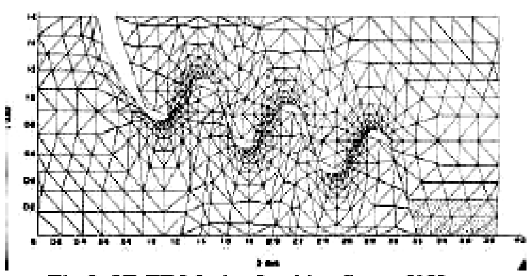


Figure 3 – 2D FE Mesh of turbine fir-tree [12]

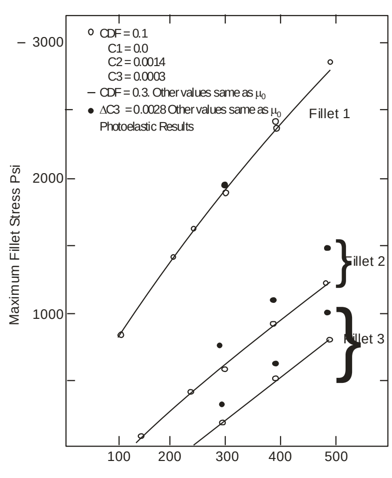


Figure 4 - blade root fillet stress [12]

The study results concluded that impact of change in clearance is high while the effect of change in friction is low on the maximum fillet radius, for the load range used in the analysis. Theoretical procedure was established by G.D. Singh, *et al.* [13-14] for the evaluation of load sharing on the fir-tree lobes and suggested an approach to treat the firtree root as an assembly of lobes. The approach focuses the 2D (plane stress) finite element analysis to individual lobes. Expressions were derived using the regression techniques, from the deflection results obtained from F E analysis and geometrical parameter, to represent the stiffness characteristics of the lobes. The maximum stress at the fillets can be obtained by treating each lobe as a cantilever and using appropriate stress concentration factor from the load estimation using the above approach. This approach does not reveal the interface phenomenon at blade disc contact zone. A practical approach to the design of a disk-blade attachment, combining the accuracy of three-dimensional analysis with the economy of two-dimensional analysis, was established by R.G. Alderson, *et al.* [15]. Two-dimensional analysis was recommended in the process of expediting redesign configurations and identifying the optimum configuration of design and three-dimensional analysis for accurate prediction of stress variation along the thickness on the final design configuration. At the side surfaces of segment, cyclic symmetric boundary condition was used. Free motion between disc and blade in the plane of contact surface and zero relative motion normal to the surfaces are the interface boundary conditions used in this study. The paper also recognized the general temptation among the stress analyst and its inadequacies to use very high friction, because of high contact stresses, by defining nodes at the disc and blade contact surface to have identical displacements. The study emphasize on the care required in obtaining the realistic boundary conditions and the need for program development for effective handling of friction at the interfaces. The study indicated that the plane stress assumption and imprecise representation of broach geometry do not produce highly accurate stress prediction, although it gives adequate trends. An iterative approach, which assumes infinite coefficient of friction between two was referred by P.H.B. Boddignton et al. [16]. The solution is performed using an iterative process, starting from the initial assumption that the interface is a continuous solid. At each stage in the iteration, the calculated distribution of normal forces along the interface is examined. If any regions are in tension, it is concluded that the two bodies would undergo localized separation and the interface is disconnected at these points. The analysis is then repeated with the new boundary conditions until a solution is reached where interface control regions are in compression only. The application of iterative approach in the analysis of dovetail joint does not give accurate stress distribution and fail to model or assess the extend of surface motion. Finite element code was developed [16] to model the behavior of friction coefficient in a real assembly and limited its application to the model which experiences small displacement and the interface is well defined. Simplified specimen configuration shown in figure 5 , representing dovetail interface, was analysed and tested using Ti-IMI 829 by C. Ruiz, *et al*. [17].

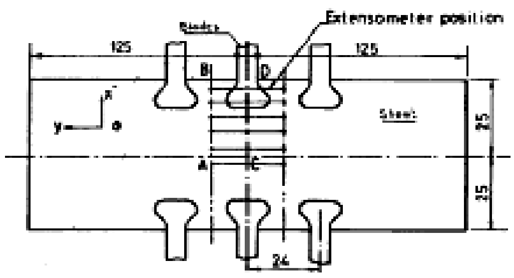


Figure 5-Geometric configuration of test specimen[17]

The FE code [16] was used to study the dovetail interface stresses using 2D plan-stress finite element model with symmetric boundary conditions as shown in figure 6.

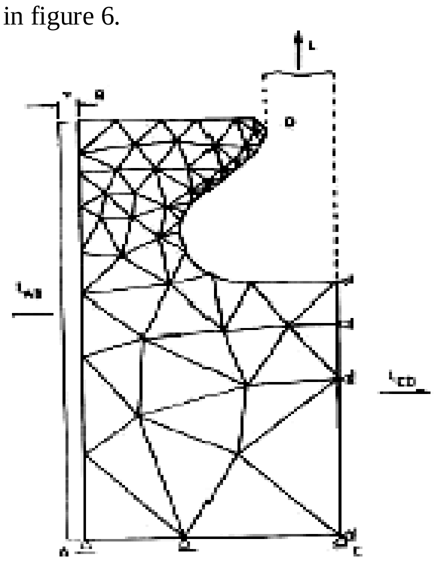


Figure 6- 2D FE model of test specimen [17]

The stress variation along the interface profiles is shown in figure 7, which indicates the acute stress gradients at the contact edge locations, for different friction coefficient.

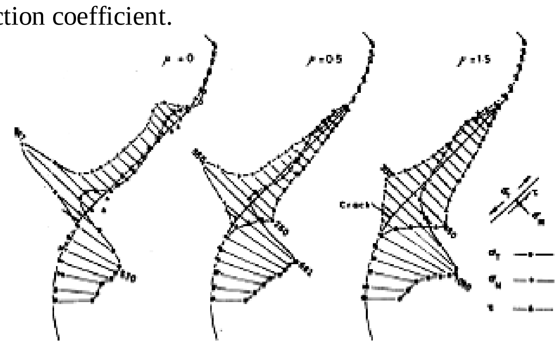


Figure 7- Stress variation along the specimen dovetail interface [17]

The element size effect study identifies the necessity for substantial mesh refinement where the stress gradients are the highest and rapid changes in contact conditions are found. Specimen inspection during fatigue testing indicates the crack at the contact edges at the bottom of dovetail. Application of probabilistic approach for estimation of the effect of dimensional tolerance in the blade-disc attachment was suggested by Murari P. Singh [18]. Finite element code (ZAMO) was developed and implemented by G. Zboinski [19] for frictional contact analysis of large three-dimensional (3D) contact analysis of linear elasticity. The code utilizes incremental description of the problem, displacement formulation and an iterative procedure to solve non-linear equilibrium equations at each incremental step. The usefulness of the program for technological problem is supported with the application on real turbo machinery blade attachment calculations. The effect of critical geometric features of dovetail attachment and interface conditions such as inner radius, outer radius, flank angle, flank length and friction coefficient at the blade disc interface was studied by S. A. Meguid, *et al*. [20]. The study further focused on the process of simulating crack initiation and subsequent propagation in regions exhibiting maximum stress concentration. Two-dimensional (plane stress) model using four-noded quadrilateral element and interface element at the contact interfaces was adapted for the midsection of the blade disc attachment. The interface elements allowed the modeling of gap and friction conditions at the interface. Finite element model of a sector is shown in figure 8.

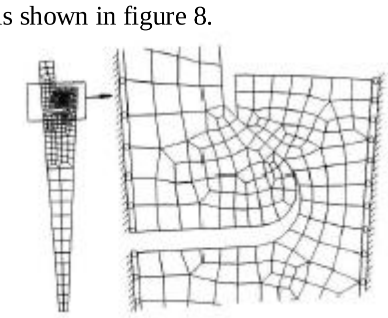


Figure 8 -2D FE model of dove tail attachment [20]

Preprocessing was carried out using commercial package SDRC-IDEAS and ABACUS code was used for solving the model. The maximum equivalent stress occurs at a position just below the lower contact of blade and these matches with photo elastic experimental observation. The crack initiation and subsequent propagation were simulated in finite element analysis with the initial crack of 0.5 mm length as shown in figure 9. The initial crack was allowed to grow in the stress concentration region with its direction of propagation being dictated by the maximum minimum principle stress criterion.

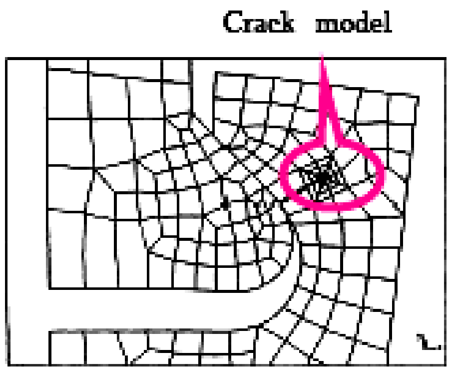


Figure 9-FE modeling with crack in dovetail contact [20]

Subsequently, the newly developed maximum minimum stress trajectories were evaluated using finite element method. A further increment of crack growth was then given in the new direction and the whole process was repeated several times to track the propagation of these cracks. Comparative study, between two-dimensional and three-dimensional nonlinear finite element analysis on dovetail joints, of P.Papanikos, *et al*. [21] indicates two dimensional analysis underestimates the maximum equivalent stress along the interface up to 40%. The study emphasize limitations of two-dimensional analysis in predicting the stress variation across the thickness of realistic disc, assessing the effect of skew angle in the stress field and its implication on the safety margin of disc assembly.

The models used ANSYS finite element code with eight-noded quadrilateral and six-noded triangular plane stress elements for two dimensional analysis and twenty-noded hexahedral and ten noded tetrahedral elements for three-dimensional modeling. Interface conditions are simulated using node-to-segment contact elements. The study with the same analysis approach on fir-tree attachment in turbine disc brought out similar conclusion by S. A. Meguid, *et al.* [22]. With sub modeling approach, a sub-region within the original global configuration and centered on the stress concentrator of interest is analyzed by itself, with the consequent of reduction in computation. Evaluation of sub modeling on series of test problems and dovetail attachment as shown in figure 10 was carried out by N.G.Cormier, *et al.* [23] and G.B. Sinclair, *et al.* [24]. The model uses ANSYS two dimensional element (PLANE 42), point-to-surface contact element (CONTAC 48) at the interface. The challenges stem from the high stress gradients near the edges of contact and from the nonlinearities attending conforming contact with friction.

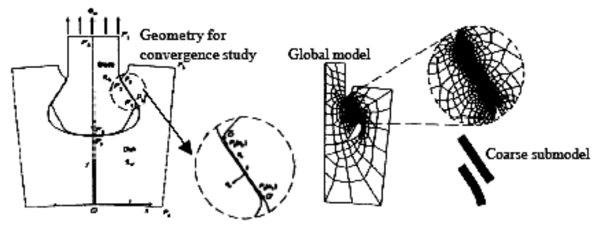


Figure 10 - Sub modeling of dovetail [23]

To meet these challenges with a finite element analysis, refined grids are needed with mesh sizes near the edge of the contact of the order of one percent of local radius of curvature there. Effect of mesh refinement on convergence of peak stress near the edges of dovetail attachment is shown in figure11.

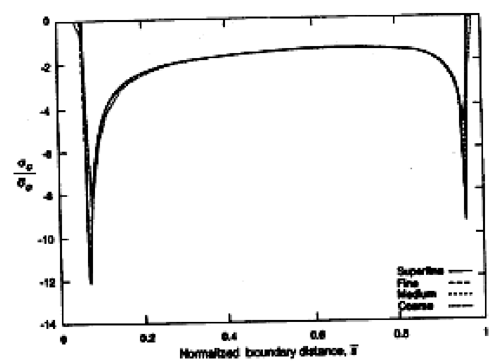


Figure 11- Convergence of peak stress-Entire contact region [23]

Table.1 lists and compares the different level of mesh refinement in global and sub model with the corresponding peak stress estimation and computation time for solving the model.

Table 1: Element numbers, runtime and normalised peak stress for different mesh grid [23]

Finite Element Grid	Number of spatial element	Number of contact elements	CPU time (min)	Peak normalised contact stress
Coarse global	915	1023	5	-8.6
Medium global	2604	4228	21	-9.4
Fine global	8317	5853	66	-11.4
Superfine global	28559	23383	614	-12.1
Coarse submodel	768	1587	1	-
Medium submodel	3072	6332	5	-12.9
Fine submodel	12288	25170	25	-13.2

The successful implementation of sub modelling in the real life application rests with the control of boundary condition error introduced on the interface between the global region and the sub model. The study [23] suggested a procedure for control of boundary condition error is as follows. First, the global analysis is conducted on a series of three or more systematically refined grids. Second, displacements on the interface of the region selected for sub modelling are checked for convergence. Third, if converging, a first submodel is run using displacements from the ultimate global grid and then from the penultimate global grid.

Fourth, if there is no change in the peak stress to within the level of accuracy sought for this stress with the two boundary conditions, the displacement boundary conditions are deemed to have converged sufficiently. The detailed study provides confidence for application of sub modelling procedure in the blade disc attachment problem, which can provide mesh of sufficient resolution in return for moderate computational effort, for accurate prediction of stress near contact edge. R. Rajasekaran, *et al*.[25] study on application of submodelling on Cattaneo- Mindlin problem indicates the Lagrange formulation predicts the correct slip-stick behaviour while penalty formulation incorrectly predicts the partial slip as full sliding at the contact.

3. DISCUSSIONS

The review on finite element approaches indicates the use of two-dimensional analysis at the blade disc attachment with infinite friction coefficient till 1970s, research effort on development of two-dimensional finite element code including contact interfacial motion and its use in 1980s and 1990s, and prevailing use of commercial codes with contact simulation in 2D and 3D analysis since 1990s. With the recent advancements in the computing power and improved capabilities in finite element approaches to simulate the interface boundary conditions, aero-engine industries currently practices two dimensional analysis during initial design iteration phases and three-dimensional analysis on the final design configurations of blade disc attachment for cost effective, and reduced design and development cycle time. The finite element analysis is applied for stress analysis of blade disc attachment in the design analysis, and failure analysis during development phase and in service. The commercial finite element codes have been widely used in recent times to simulate the interface boundary conditions and analysis of blade disc attachment. It is generally accepted that the crack initiation at the blade disc attachment depends mainly on the contact stress that occurs in the vicinity of highly localized stress concentrations caused by the frictional forces between the surfaces of blade and disc in contact. The finite element analysis, in the context of blade disc attachment, is consistently used for optimization of geometric parameters and predicting the contact stresses to estimate the fatigue life of rotors. Numerous design sensitive studies were performed to examine the effects of design tolerances and boundary conditions imposed on the blade. Blade disc attachment is a critical location susceptible to failure during the service life of aero engine, due to various uncertain factors involved in the design. The safety regulation agencies involve in the failure analysis to ascertain the root causes for failure and to ensure remedial action to address the root causes, by the way of incorporating stringent measures in certification procedure. Finite element analyses are now integral part of failure analysis of rotor blade attachment along with metallurgical examination. Finite element analysis are used to carry out sensitive study with different speeds, manufacturing damages, friction coefficient, etc. to help to identify the root causes of failures. Three-dimensional FE analyses are used to identify the primary failure location on the fir-tree as shown in figures 12-13.

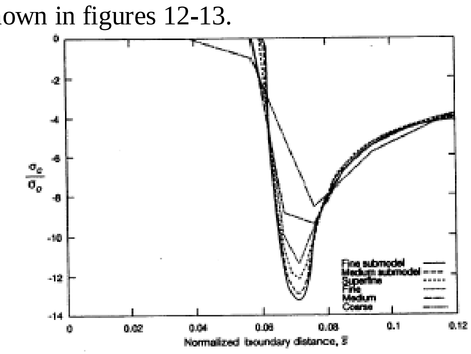
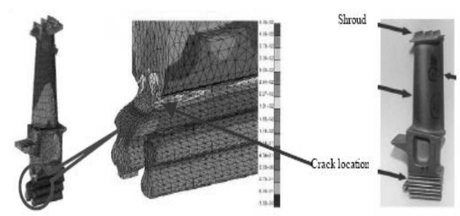


Figure 12- Convergence of peak stress – sub-modeling region [23]



(a) (b)

Figure 13 (a) - 3D FE analysis in failure analysis **(b)-** Blade failure at fir tree.

4. CONCLUSION

Based on the literature review, the conclusions are made as below:

- Modern finite element codes have matured to a stage where the interface conditions between blade root and disc with thermal mechanical loading at the blade disc attachment can be effectively handled starting from a simple analysis which assumes infinite coefficient of friction at the interface to complete modeling and assessment of the extent of interface motion.
- A three dimensional approach to model the problem, although two-dimensional representation may still play an important role in the design process by the way of reduced design and development cycle time.
- Sub-modeling (global-local) approaches to predict the high stress gradient near the contact edges of blade disc attachment.

- Integration of finite element analysis with metallurgical examination in failure analysis of aero engine rotors.

Scope for potential improvement in finite element modeling of crack initiation and growth in the blade disc attachment for damage tolerance analysis and application of fracture mechanics.

5. REFERENCES

1. Lucjan Witeck, 2006, "*Failure analysis of turbine disc of an aero engine"*, Engineering failure analysis, Vol.13, pp. 9-17.
2. Jianfu Hou, Bryon J. Wicks and Ross A. Antoniou, 2006, "*An investigation of fatigue failures of turbine blades in a gas turbine engine by mechanical analysis"*, Engineering Failure Analysis, Vol. 9, pp. 201-211,
3. N.S. Xi, P.D. Zhong, H.Q. Huang, H. Yan and C.H. Tao, 2002, *" Failure investigation of blade and disc in first stage compressor"*, Engineering Failure Analysis, Vol. 7, pp. 385-392,.
4. M.L. Xie, P.D. Zhong, N.S. Xi, Y. Zhang and C.H. Tao, 2000., "*Analysis of fracture failure of fir-tree serrations of stage II turbine disks"*, Engineering Failure Analysis, Vol. 7, pp. 249-260.
5. V.N. Shlyannikov, B.V. Itchenko and N.V. Stepanov, 2001, *"Fracture analysis of turbine disks and computational experimental background of the operational decisions*", Engineering Failure Analysis, Vol. 8, pp. 461-475
6. Autralian Transport Safety Board, Examination of a failed fan blade, Technical Analysis Report No: 8/01.
7. V. J. Parks and R.J. Sanford, 1978, *"Photoelastic and holographic analysis of a turbine-engine component", Experimental* Mechanics, pp. 328-333.
8. J. Durelli, J.W. Dally and W.F. Riley, 1958 *"Stress and strength studies on turbine blade attachments*", Proceedings of Society for Experimental Stress Analysis, Vol. XVI, No. 1, pp. 171-186, 1958.
9. A.J. Durelli, and K. Rajaiah, 1980, November, "*Lighter and Stronger"*, Experimental Mechanics, pp.369-379.
10. B. Kenny, E.A. Patterson, M. Said and K.S.S. Aradhya, 1991, "*Contact stress distribution in a turbine disc dovetail type joint – A comparison of photoelastic finite element results"*, Strain, pp. 21-24,.
11. S.K. Chan and I.S. Tuba, 1971, *"A finite element method for contact problems of solid bodies –Part I. Theory and validation"*, Int. J. Mech. Science, Vol. 13, pp. 615-625, 1971.
12. S.K. Chan and I. S. Tuba, 1971. *"A finite element method for contact problems of solid bodies –Part II. Application toturbine blade fastenings"*, Int. J. Mech. Science, Vol. 13, pp. 627-639, 1971.
13. G.D. Singh and S. Rawtani, 1982, " *Fir tree fastening of turbomachinery blade-I Deflection analysis"*, Int. J. Mech. Science, Vol. 24, No. 6, pp. 377-384, 1982.
14. G.D. Singh and S. Rawtani, 1982 *"Fir tree fastening of turbomachinery blade-II Step load analysis*", Int. J. Mech. *Science*, Vol.24, No.6, pp. 385-391,1982
15. R.G. Alderson, M.A. Tani, D.J. Tree and R.J. Hills, 1978, "*Three-dimensional optimization of a gas turbine disc and blade attachment"*, J. Aircraft, Vol. 13, No. 12, pp. 994 -999.
16. P.H.B. Boddington, K.C. Chen and C. Ruiz, 1985, " *The numerical analysis of dovetail joint"s*, J. Computers & Structures, Vol. 20, No. 4, pp. 731-735,.
17. C. Ruiz, P.H.B. Boddington and K.C. Chen, 1984, *"An investigation of fatigue and fretting in a dovetail joint"*, Experimental Mechanics, vol. 24, pp. 208-217.
18. Marari P. Singh, "*Probabilistic estimation of the effect of dimensional tolerance for turbine/compressor blade attachment"*. Dresser-Rabd, Wellsville, NY, USA.
19. G. Zboinski, 1993, *"Finite element computer program for incremental analysis of large three-dimensional frictional contact problems of linear elasticity"*, J. Computers and Structures, Vol. 6, No. 4, pp. 679-687.
20. S.A. Meguid, M.H. Refaat and P. Papanikos, 1996., *"Theoretical and experimental studies of structural integrity of dovetail joints in aero engine discs"*, J. Materials Processing Technology, Vol. 56, pp. 668-677.
21. P. Papanikos, S. A. Meguid and Z. Stjepanovic, 1998. *"Three dimensional nonlinear finite element analysis of dovetail joints in aero engines discs"*, Finite Elements in Analysis and Design, Vol. 29, pp. 173-186
22. S.A. Meguid, P.S. Kanth and A. Czekanski, 1998. *"Finite element analysis of fir-tree region in turbine discs"*,Finite Elements in Analysis and Design, vol. 35, pp. 305-317.
23. N.G. Cormier, B.S. Smallwood, G.B. Sinclair and G. Meda, 1999, "*Aggressive submodelling of stress concentrations*", Int. J. Numer. Meth. Engng., Vol. 46, pp. 889-909,
24. G.B. Sinclair, N.G. Cormier, J. H. Griffin and G. Meda, 2002, *"Contact stresses in dovetail attachments: Finite element modeling"*, J. Engng. Gas Turbine Power, Vol. 124, pp. 182-189.
25. R. Rajasekaran and D. Nowell, , *2005 " On the finite element analysis of contacting bodies using submodelling"*, J. Strain Analysis, vol.40, No.2