RESEARCH PAPER | Engineering | Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

Structural Integrity Analysis of Gas Turbine Rotor Component using Finite Element Analysis

KEYWORDS: Fir-tree, Hypermesh, Static and Modal analysis

DILIP A.B	Mr. SYED ZAMEER
M.TECH STUDENT, MECHANICAL ENGINEERING	ASSISTANT PROFESSOR, MECHANICAL ENGINEERING DEPT. GHOUSIA COLLEGE OF ENGINEERING, RAMANAGARA

ABSTRACT *Stress analysis plays important role in finding the static and dynamic stability of the aero structures. Turbines are the most important components in the aero engines. Stresses are generated in gas turbine rotor blades due to centrifugal force acting at the section of airfoil. In the present work, structural integrity of gas turbine blades are checked for 2 cusp and 3 cusps fir-tree contacts. Geometrical models are built using Catia software and later imported to Hypermesh for better quality mesh and results. The analysis results obtained from ANSYS for 3 cusp fir tree contact shows improved results with less radial displacement and stresses compared to 2 cusp fir tree contacts.*

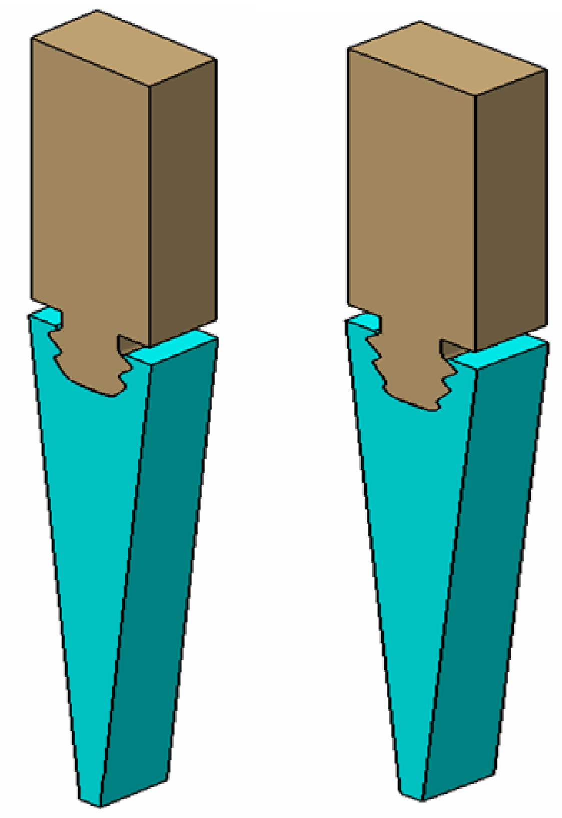
INTRODUCTION
Aero engine turbine discs have three critical regions for which lifetime certification is necessary the fir-tree rim region, the assembly holes or weld areas and the hub region. The safety of gas turbine engines has always been the main concern of aircraft certification authorities. One of the main factors concerned mechanical integrity of aero engine turbines is the interface region between the blade and the rotor disc. Stresses generated in this region are mainly produced by the centrifugal force resulting from the rotational speed of rotor mass of the blade, thermal stress, bending loads and torsion due to the gas pressure. A model based simulation process using FEM involves doing a sequence of steps. This sequence takes two canonical configurations depending on the environment which FEM is used.

The present study is focused on the study of stresses arising from the centrifugal loadings in a fir-tree joint using a 3D Finite Element model in the commercial code ANSYS R14.5. Aero engine designers are constantly faced with the challenge of establishing stress levels in these critical parts that will allow the use of suitable high strength heat resistant alloys operating in a safe thermo mechanical loading regime. At this stage, it is important to identify the pertinent parameters which influence the integrity of aero engine turbine Disc assemblies. These include (1) the strength, quasi static thermo mechanical strength, toughness and rupture strength of the different constituents of the assembly, (ii) the applied thermal and body forces associated with the temperature of the gases and the rotational speed of the disc, and (iii) residual stress state of the components/assembly. In this thesis, we focus our attention on the nonlinear contact nature of the turbine disc blade assembly.

PROPOSED WORK: Analyzing the fir-tree type turbine blade disc attachment for stresses under inertia conditions. Since attachment is very important to keep the assembly intact, prior information of the nature of stress and deformations helps in better design. So the main objectives include

- Bladed disc fir attachment modeling.
- Analysis in the nonlinear domain due to moving boundaries.
- Application of inverse technique for solving the problem.
- Comparison of stress nature between 2 and 3 cusps of contact.
- Checking for dynamic stability of the models.

Fig 1: Proposed Model 1 (2 cusp) & Model 2 (3 cusps)

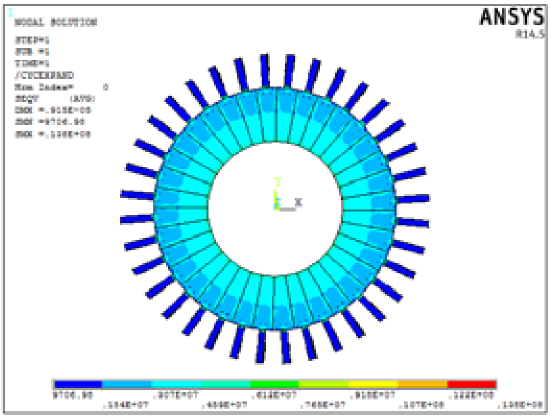


Geometrical models are built for two configurations. One with two cusps considered as proposed model 1 and other with three cusps as proposed model 2. The concentration is given to find the effect of cusps on geometrical strength of the fir tree joint. The figure 1 shows the geometrical modeling which has done using Catia 3D modeling software.

Volume : 4 | Issue : 7 | July 2014 | ISSN - 2249-555X

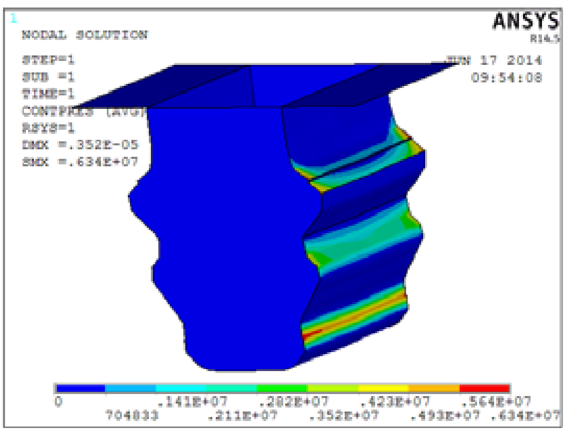
RESULTS:

Fig 2: Vonmisses Stress Plot



The figure 2 shows tress development for proposed model 1 (2 cusp). Maximum stress is 13.8 Mpa due to elastic material behavior. Non linear geometrical contacts are defined with Newton-Raphson iteration technique for solving the problem.

Fig 3: Contact Pressure Plot



The figure 3 shows uniform contact pressure generation between the members. Maximum contact pressure generation is 6.3 Mpa it is represented in red colour.

Proposed Models	Radial Displacement (microns)	Radial Stress (Mpa)	Hoop Stress (Mpa)	Vonmises Stress (Mpa)	Contact Pressure (Mpa)
1	2.62	7.8	12.7	13.8	5.51
2	0.98	6.95	6.77	10.2	6.3

The table shows results for proposed model 1 (2 cusp) and model 2 (3 cusps). The results shows that proposed model 2 has less radial, hoop and vonmisses stress compared to proposed model 1. The radial displacement is also less for proposed model 2 compared to model 1. But the contact pressure development is lightly increases for model 2 compared to model 1. This can be attributed lesser cross sectional area of resistance at the bottom of proposed model 2.

CONCLUSIONS:

The contacts are defined between the blade and rotor interface using entity sets created in the Hypermesh and Ansys contact manager. Targe170 and contac174 elements are defined across the interface. Two cusp analysis shows higher stresses and deformation due to insufficient contact region to take the load. Three cusp analysis shows better results and uniform stress generation when compared with two cusp analysis. The stresses, radial displacement and vonmisses stresses are increasing with increased speed. The stress and contact pressure development is having parabolic proportion to the speed. Modal analysis is carried out to find the dynamic nature of the system. The results shows greater difference of natural frequency with three cusps compared to the two cusp contacts. The analysis is carried in the nonlinear domain due to moving boundaries. Inverse technique is used for solving the problem. The results are captured for vonmisses stress and contact pressure. Contact pressure shows regions of higher contact and indicates possible reasons of wear out.

FUTURE SCOPE:

- Thermal effects can be considered to find the contact stress nature.
- Optimization of the members can be carried out based on the stress nature.
- Composite usage can be checked for strength estimation.
- Topology optimization can be carried out.
- Vibrational analysis can be carried out for harmonic and transient loads.

REFERENCE

[1] G. D. Singh and S. Rawtani, "Fir Tree Fastening Of Turbo Machinery Blades –I International Journal of Mechanical Science Vol 24, No 6, 1952, Pages 344-354. | [2] Cheng-Hung Huang and Tao-Yen Hsiung "An Inverse Design Problem of Estimating | Optimal Shape of Cooling Passages in Turbine Blades", International Journal of Heat | and Mass Transfer 42 (1999) 4304-4319. | [3] Wenbin Song a, Andy Keane, Janet Rees, Atul Bhaskar, Steven Bagnall, "Turbine Blade Fir-Tree Root Design Optimisation Using Intelligent CAD and Finite Element Analysis",Computers and Structures 50 (2002) 1553–1564 | [4] Jianfu Hou, Bryon J. Wicks, Ross A. "Antoniou an Investigation of Fatigue Failures of Turbine Blades in a Turbine Engine by Mechanical Analysis", Engineering Failure Analysis 9 (2002) 201–211 | [5] Amr M.S. El-Hefny, Mustafa Arafa, A.R. Ragab and S.M. EL Raghy "Stress Analysis of A Turbine Rotor Using Finite Element Modeling"Production Engineering &Design For Development, PEDD4, Cairo, February, 2006 | [6] Allen J.R.Ericson, "Nastran Analysis of a Turbine Blade and Comparison with Test and Field Data", ASME-GT-44. | [7] H. D. Conway and K. A. Farnham, "The Contact Stress Problem for Indented Strips and Slabs Under Conditions of Partial Slipping," Journal of International Engineering Science, Vol. 5, 1964 pages 145-154. | [8] S.A. Meguid, P.S. Kanth, A. Czekanski "Finite Element Analysis of Fir-Tree Region in Turbine Discs" Finite Elements in Analysis and Design 35 (2000) 305}314 | [9] Sinclair, G. B., Cormier, N. G., Griffin, J. H., and Meda, G., "Contact Stresses in Dovetail Attachments: Finite Element Modeling," J. Eng. Gas Turbines Power, 2002, 124, pp.152-191 | [10] Gwo-Chung Tsai "Rotating Vibration Behavior of the Turbine Blades with Different | Groups of Blades", Journal of Sound and Vibration, 2004, Pages 544-545. | | [11] Rajasekaran R, Nowell D, "On the Finite Element Analysis of Contacting Bodies using Sub Modeling". J Strain Analysis; 2005, 40(2); pp.95-106. | [12] A. G. Hernried and Wei-Ming bian "a Finite Element Approach for Determining the Frequencies and Dynamic Response of Twisted, Nonuniform Rotating Blades with Small or no Precone", Printed in Great Britain, Computers & Structures, VoL 45, No.5, pp. 925-933, 1993. | [13] A. Zmitrowicz, "A Note on Natural Vibrations of Turbine Blade Assemblies With Non-Continuous Shroud Rings", Journal of Sound and Vibration, Vol.192, pp. 521-533, 1996 | [14] Hong Hee Yoo, Jung Hun Park and Janghyun Park "Vibration Analysis of Rotating Pre-Twisted Blades" Computers & Structures 49, 2001 | [15] István Bagi and Gábor Vörös "Dynamic Analysis of A Turbine Blade Using The Finite Element Method" Budapest, 2002, pp 595-595 |