International Journal for Ignited Minds (IJIMIINDS)

Fatigue and Fracture Analysis of Spindle Index Assembly for Conveyor Applications Using Finite Element Analysis

Siddegowda Y P[a], Dr. A R Anwar Khan[b] & Mohammed Imran[c]

[a]*PG Scholar, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.*
[b]*Principal & HOD, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.*
[c]*Asst. Prof, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.*

ABSTRACT

This paper is presented on Fatigue and Fracture analysis of Spindle index Assembly for conveyor applications using FEA. The premature failure is predicted by virtual simulation techniques by checking through all possible combination of loads. In the present work, a failed spindle assembly used in the armature winding is analyzed to find the cause of failure. The results of both maximum and minimum loading conditions are stored for a node which is subjected to maximum stress conditions. An alternating stress is calculated from the Ansys module which shows higher value compared to the allowable fatigue limit of the member. Further the effect of fracture is analyzed in the problem. The growth of crack is introduced in the model by re-meshing and detaching the elements for required depth. The analysis is done for the same loads which shows increased alternating stress values and reduced number of cycles for given stress conditions.

Keywords *- Fracture, Spindle, Armature Winding, Fatigue, Fracture.*

1. INTRODUCTION

Stress concentrations plays important role in the structural safety of the engineering components. Stress concentration exists in the structure due to many factors like change of geometry suddenly, material combinations, holes, cut-outs, fillets, cracks, voids, no homogeneous nature of materials, welds , grooves, composite nature etc.

Shafts find wide usage in the industry for transferring the rotation from one point to another point. During the process of transfer of power, various members are mounted on the shaft. Based on the requirement, the shaft has various dimensions at different stages. The shaft may hallow or solid based on the stability required. The shafts may be uniform or stepped based on the desired transmission of the power. Due to all these factors, the shafts are subjected to various stress concentration factors. So design should consider all these factors.

1.1 Fatigue

Fatigue is the failure of the component under reversed or fluctuating or cyclic loading. This can be interpreted as the failure of the component under cyclic loads. Generally fatigue stress limit is smaller than normal design stress. So any component designed for fatigue should be design based on endurance limit instead of normal design stress.

1.2 Material Combinations: Bearing Problems

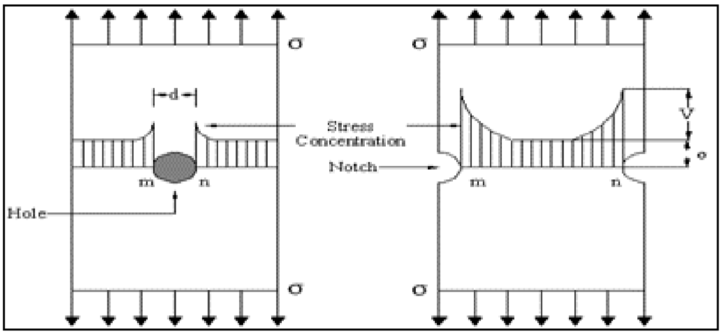


Fig 1: Stress Concentration due to Cutouts.

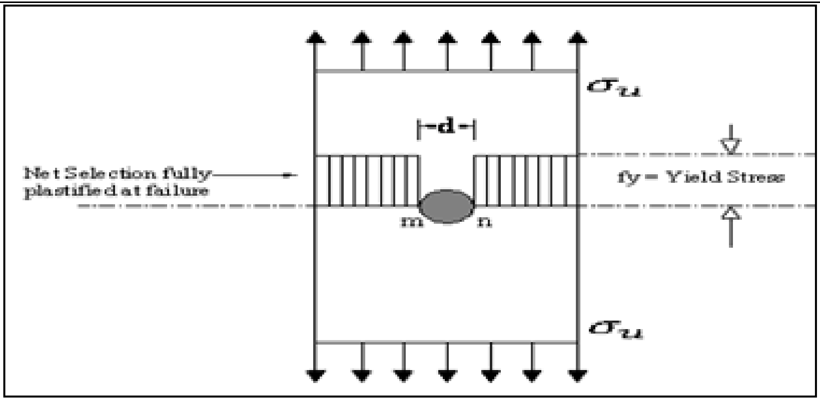


Fig 2: Stress reaching to Yield Stress at the Cut-out.

2. DEFINITION AND REQUIREMENT OF THE PROBLEM

2.1 Problem Definition

Geometrical Modeling and failure analysis of the spindle assembly using fatigue and fracture conditions is the main definition of the problem.

The objectives include

- Finding the Structural stress under given loading conditions
- Fatigue life estimation
- Estimation of Fracture Fatigue life with growth of crack.

2.2 Requirement of the Problem

The armature of 8 poles is to be winded by spindle indexing assembly. In the winding process, the spindle frequently changes its position between 90º or 45º or 180º based on the requirements. Due to which the system is subjected to fluctuating load or fatigue loads. The design should be based on fatigue to prevent any possible failure.

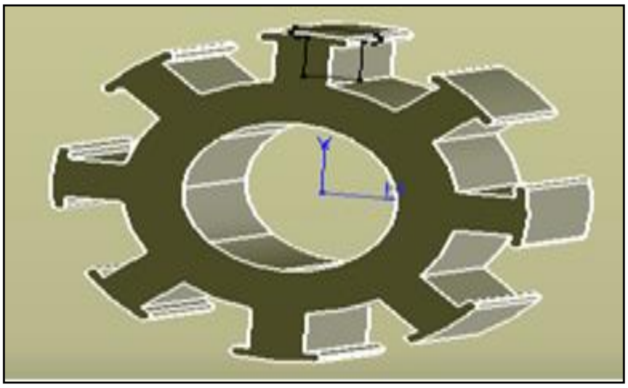


Fig 3: The 8 poles Armature.

3. GEOMETRY AND FE MODELING

3.1 Geometry Modeling

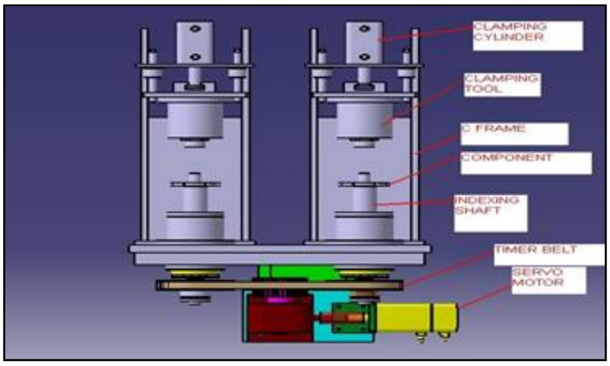


Fig 4: Complete representation of the Assembly.

The Fig 4, shows complete representation of the problem which shows all the components of assembly along with naming of the components. But in the analysis all the components will not be considered, but analysis will be limited to the Indexing Spindle assembly. Here power goes to indexing shaft through time belt which connected to a servo motor. The armature will be provided between clamping tool and the spindle. At a time two spindle indexing systems are used in the assembly.

3.2 FE Modeling

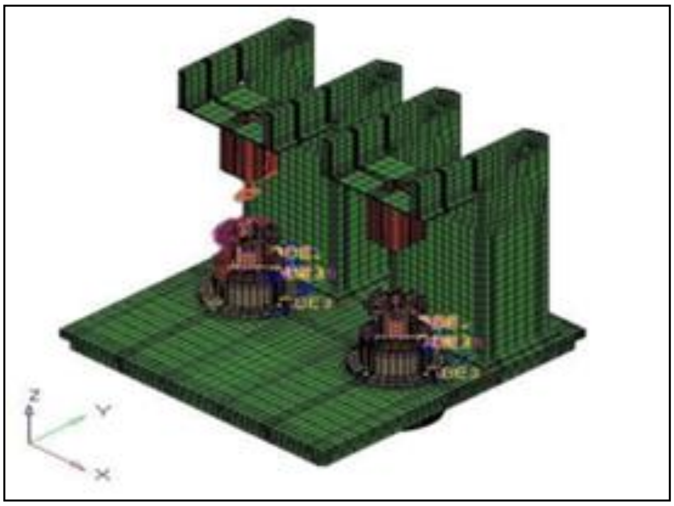


Fig 5: Assembly Mesh.

The Fig 5, shows assembly mesh of the structure. Initially all the components are hex meshed and later connected through either nodal merging or coupling constraints. RBE3 elements are created for load transfer. RBE3 is a special element for load transfer in the mechanical or structural components or assembly. Mesh is the life of finite element analysis without which analysis can't be done. The meshed assembly is exported to Ansys in 'inp' file format and imported to Ansys after switch off checking controls.

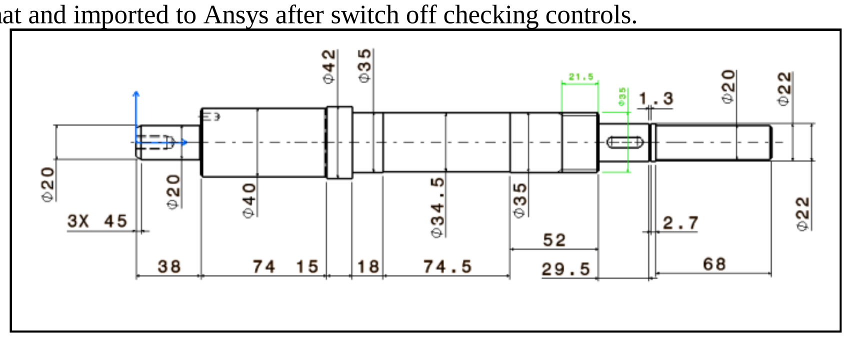


Fig 6: Dimensional view of the Spindle Shaft

The Fig 6, shows dimensional view of the spindle shaft. Minimum diameter is 20mm and maximum diameter is 42 mm as shown in the figure. Catia drafting module is used to represent the dimensions of the problem.

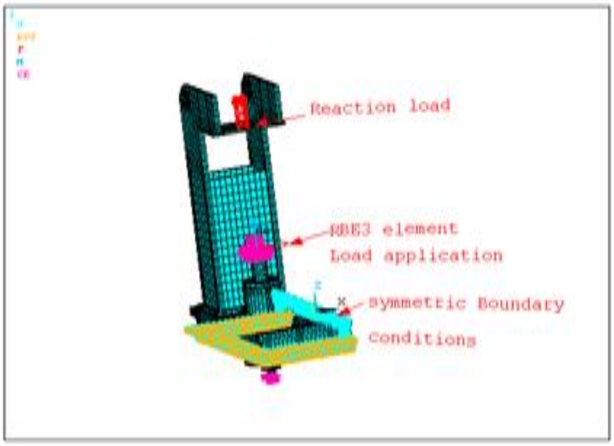


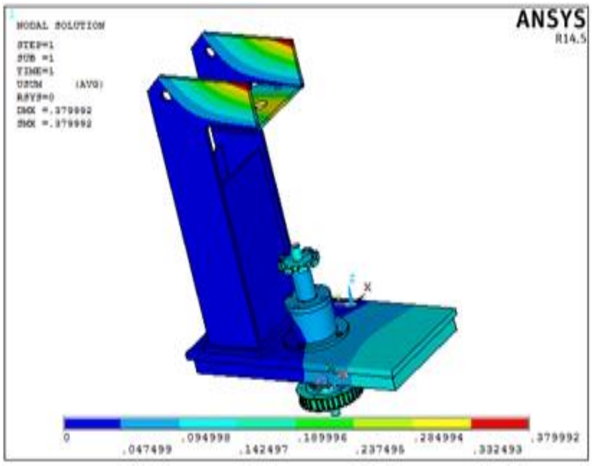
Fig 7: Boundary conditions of the Problem

The Fig 7, shows applied boundary conditions on the problem. The load is applied through RBE3 element. Symmetrical boundary conditions are applied to simulate other half geometry in the problem. The load passing through clamping cylinder is applied as a reaction load. A torque load of $120x10^3$ N-mm is applied through RBE3 element. A reaction load of extent of 2000N is applied as shown in the figure. Displacements are called essential boundary conditions and forces are called non-essential boundary conditions for execution of the problem.

4. RESULTS AND DISCUSSION

Analysis is carried out for the fatigue loading conditions. Due to indexing loads, the maximum and minimum loads are having large variation. Maximum load variation is considered for the fatigue analysis. A complete load reversal is considered for fatigue analysis to find the safety of the members.

4.1 Maximum Loading Conditions



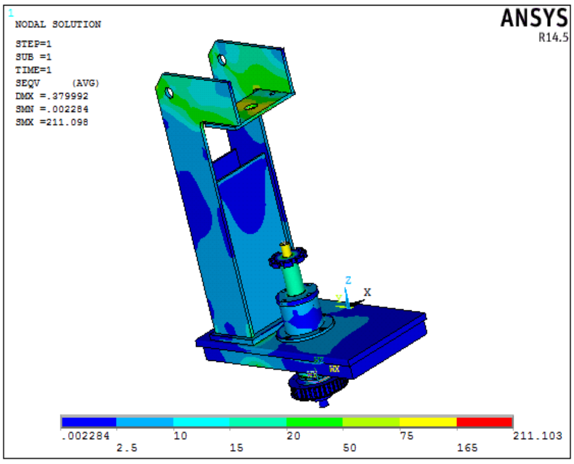
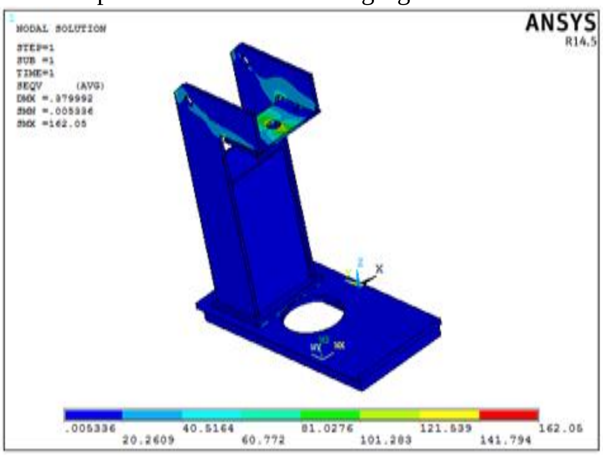


Fig 8: Displacement Plot of Assembly

Fig 9: Von-Mises Stress Plot.

The Fig 9, shows deformation of around 0.379mm due to the given loading conditions. Maximum deformation is taking place on the top plate. This can be mainly attributed to the cantilever effect of the problem for which the deformation is maximum at the unsupported end.

The Fig 10, shows maximum stress of 211.103MPa as shown in the Fig for the symmetric assembly. The status bar shows variation of stress from 0.002284MPa to 211.103MPa. The color code shows variation of stress in the problem. The blue color region has minimum stress and red color has maximum stress. Von-Mises stress can be defined as the stress corresponding to the stored or distortion energy. This theory is widely accepted in the engineering stress industry for predicting the failure of the ductile components. In the present problem, all the components are ductile only. So von-Mises theory is applied in this analysis. The individual component results are represented in the following figures.



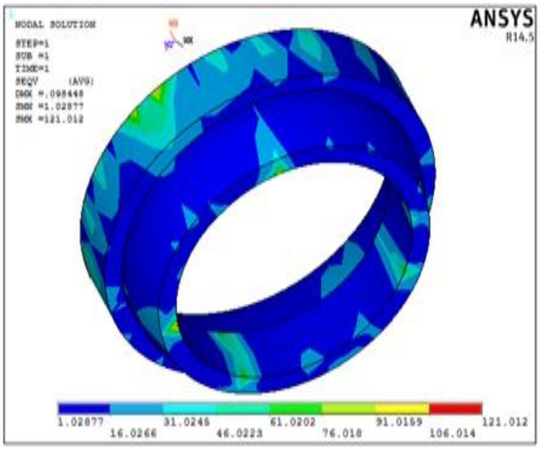


Fig 10: von-Mises Stresses in Base Plate.

Fig 11: von-Mises Stresses in Gland

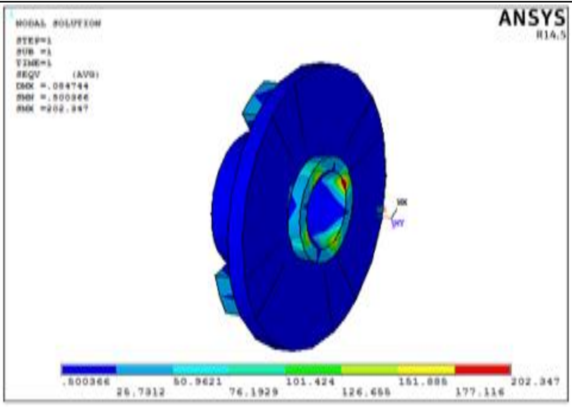


Fig 12: Braking Element 2.

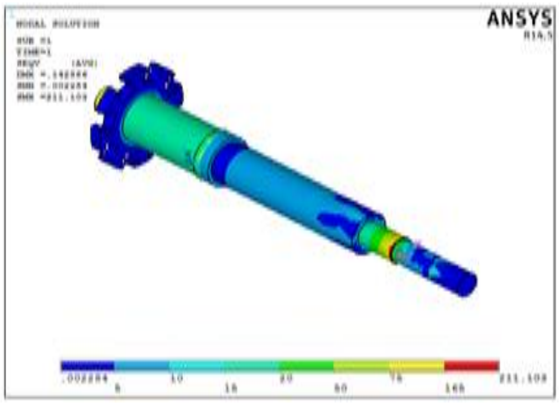


Fig 13: Stress in the Spindle with Armature.

4.2 Fatigue Life Estimation

Fatigue estimation is carried out by selecting the node corresponding to the maximum stress. The stress values for both load cases will be obtained at the same node find the alternating stress and fatigue life of the assembly.

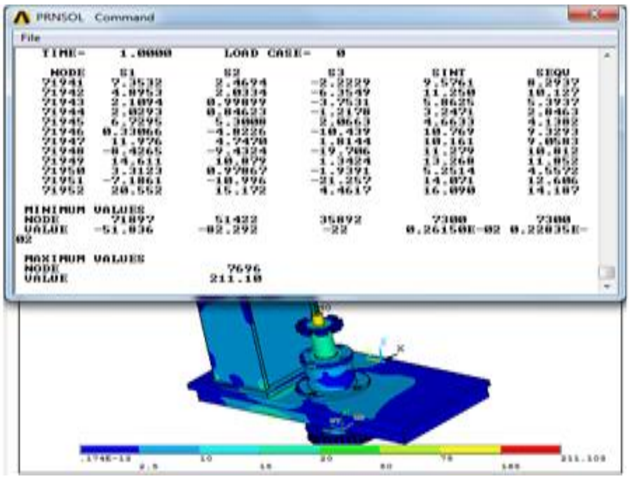
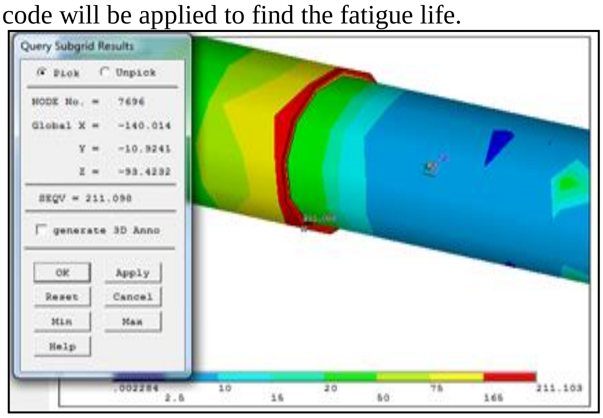


Fig 14: Node Number for Maximum Stress.

For fatigue analysis location is important to apply fatigue life theory. So maximum stress node is listed through nodal solution which is shown in above figure. The node number corresponding to maximum stress is 7696. In the fatigue analysis, results are stored at the node corresponding to maximum stresses and later ansys code will be applied to find the fatigue life.



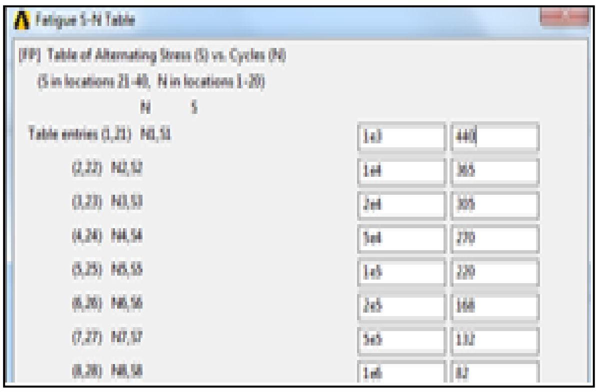


Fig 15: Query Results for Node at Max Stress.

Fig 16: Fatigue Life Corresponding to 16MnCr5.

Ansys query can be used to obtain the node corresponding to the maximum stress condition which is shown in the figure.

The Fig 16, shows fatigue life applied through Fatigue module of Ansys. Generally permanent life is considered for the structure corresponding to one million cycles. 82MPa is the stress corresponding to the fatigue life of the component.

Load Case1	Sxx	Syy	Szz	Sxy	Syz	Sxz
1	2.610	-3.689	14.012	75.546	4.579	95.108
2	2.600	-3.624	13.878	-50.455	4.549	-63.436

Table 1: Stress Corresponding to Node 7696 for Maximum and Minimum Loads.

The table shows values for stress components for the given loading conditions. Generally in the global Cartesian coordinate system, stress at a point is represented by 6 stress components. To predict the failure other stresses will be calculated from these values. Either von-Mises stress or principal stress are calculated from these values.

- ***Fatigue Estimation from the Ansys fatigue module:***

Alternating stress developed: 202.51MPa
Cycles used/Allowed: 1e6/0.1237e6cycles
Cumulative Fatigue usage: 8.08263 cycles
Above data shows failure of the component for fatigue. The spindle is failing under fatigue as the alternating stress is exceeding the allowable fatigue value of 82MPa. So major mode of failure for the spindle assembly is fatigue. It shows the life of the component is only for 123700 cycles instead of 1000000 cycles. The cumulative usage factor shows failure of the component. Generally cumulative usage factor value is more than one; it indicates failure of the either assembly or component. Similarly safe fatigue shows if usage factor is less than one.

4.2 5% crack Generation:

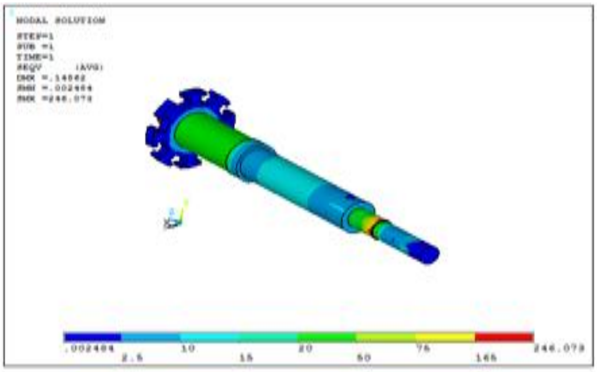


Fig 17: Stress for 120000N-mm Torque

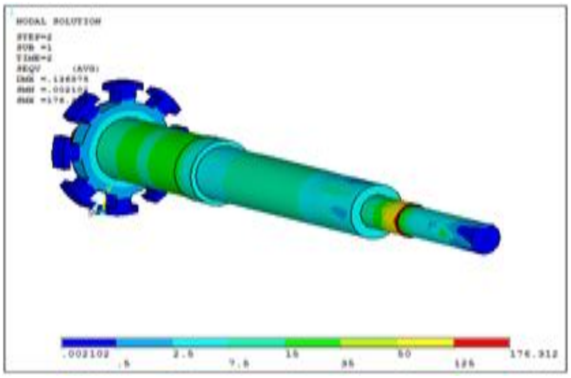


Fig 18: Stress with the Crack for 90000N-mm Torque.

The Fig 17 shows maximum stress generated in the spindle assembly with 5 % crack at the groove region. Maximum stress development of 246MPa is higher than the allowable stress of 232MPa for the spindle material and so the component is not safe for the given loading conditions.

The Fig 18, shows stress corresponding to the minimum loading case on the spindle assembly. Maximum stress of 176.31MPa can be observed from the status bar.

- ***Fatigue Estimation from the Ansys fatigue module for 5% crack:***

  Alternating stress developed: 243.02MPa
  Cycles used/Allowed: 1e6/0.7141e5cycles
  Cumulative Fatigue usage: 14.00446cycles
  Similarly other two analysis are carried out and the results are presented as follows.

Percentage of Crack	Alternating Stress, (MPa)	Fatigue Life (number of Cycles)
-	202	123700
5	243	71410
10	282	32400
15	398	11250

Table 2: Fatigue Life Variation with Given Loading Conditions.

The table shows fatigue life and alternating stress estimations for the problem for the given loading conditions. Alternating stress value is increasing along with the increase of crack growth. So if crack growth is allowed to grow, the life of the structure will reduce.

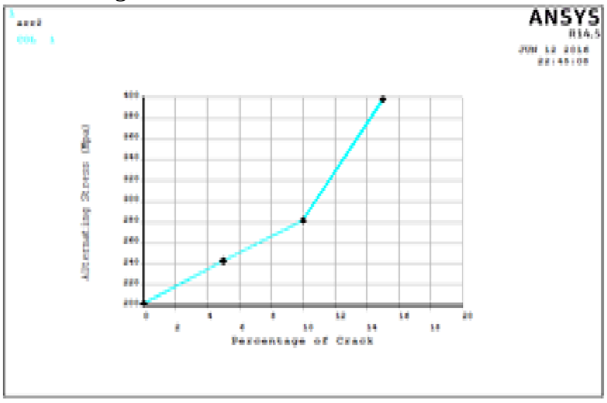


Fig 19: Percentage of Crack vs. Alternating Stress.

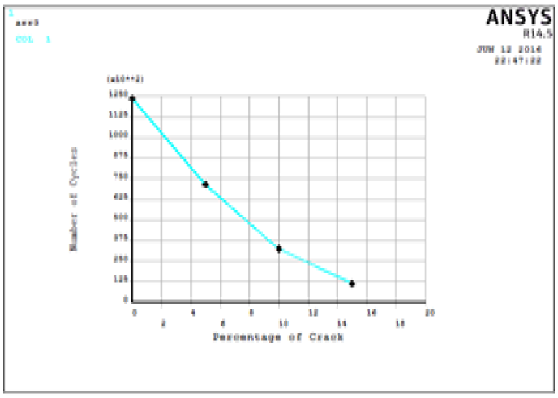


Fig 20: Percentage of Crack vs. Number of Cycles.

The Fig 19, shows increasing alternating stress with the increase of percentage of increase of crack growth. This can be understood from the fact that the resistance or moment of inertia of the section will reduce with the percentage increase of crack growth.

The Fig 20, shows increased stress with growth of crack. This can be attributed to reduced resisting cross section of the spindle for the given load transfer. Stress will increase once resisting cross section reduces. Similarly with the increase in the stress, safe fatigue cycles will reduce.

5. CONCLUSION

An indexing spindle assembly used in the coil winding applications is analyzed for structural failure using finite element analysis. The prime cause is analyzed through fatigue analysis.

- The connected mesh is analyzed for fatigue load conditions. Two load cases (maximum and minimum loads) are represented and analyzed for structural safety conditions. The individual component results are presented for von-Mises stress condition.
- The von-Mises stress for both the load conditions shows safety of the assembly and individual components for maximum and minimum loads. But higher stresses are observed near the groove where diameter is less. This can be mainly attributed to lesser moment of inertia along with stress concentration effects.
- But Fatigue analysis shows higher alternating stress development under two load cases. The alternating stress value is much higher compared to the allowable fatigue limit stress of 82MPa. So the spindle assembly will not come for one million cycles corresponding to the permanent life. So failure of the spindle assembly is mainly attributed to higher alternating stress due to load reversals.
- Further analysis with crack growth shows increased alternating stress and reduced life time. This can be mainly attributed to the reduced resistance due to growth of crack and also stress concentration effects.
- All the results are represented with required pictorial views.

REFERENCES

1. Rankine W.J, "*On the centrifugal force of rotating shafts*", Engineer, Vol.27,1869, p.249
2. Dunkerley S, "*On the Whirling of Vibration of Shafts*", Phil, Trans. Roy, Soc, Series, A. Vol. 185,1894, p. 279
3. Jeffcott, H.H., "*The lateral vibration of loaded shafts in the neighbourhood of a whirling speed – The effect of want of balance*", Phil.mag, series 6, Vol37,1919, p.304
4. J.A. McGeehan and P.R. Ryason "*Preventing Catastrophic Camshaft Lobe Failures in Low Emission Diesel Engines*", Chevron Products CoMPany. SAE Paper, 2000, P 16-19.
5. F.S. Silva, "*Analysis of a vehicle crankshaft failure*".Volume10, 2003, P605–616.
6. CharnontMoolwan, "*Failure Analysis of two High Gear box Shaft*", I-SEEC, Elsevier publications, 2012.
7. Zhi-wei Yu AE Xiao-lei Xu, "*Failure Analysis on Fractured Diesel Engine Camshafts*", ASM International 2008.
8. M.A. Mohiuddin,Y.A. Khulief, "*Modal characteristics of rotors using a coinical shaft finite element*", Computer Methods Application Mechanical Engineering, Vol- 115, 1994, P 125-144.