Scientific Journal of Impact Factor (SJIF): 5.71
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 7, Issue 08, August -2020

Design, Optimization & Fatigue Analysis of Milling Machine Spindle.

Miss.PratibhaThorat (student),

Dept.: Mechanical Engineering, TKIET Warananagr

Abstract: *The spindle holds and drives the various cutting tools. It is a shaft, mounted on bearings supported by the column. The spindle is driven by an electric motor through a train of gears, all mounted within the column. In project describes the result of action and industrial tests conducted in order to decrease the dimensions as well as weight of milling machine spindle. Due to this weight and dimension reduction, whole design of machine becomes feasible. In earlier days there design of milling machine spindle was became very bulky due to this lot of investment we have to did. But due to redesign we have to optimize that problem and make one feasible design which reduces its dimensions and also weight as well as cost. On the other hand, computer simulation and design in Engineering has tremendous advances in the last few decades, due to the availability of powerful Finite Element Method (FEM). The use of Finite Element Analysis (FEA) for industrial applications is well accepted for more than four decades. Spindle is the main part used for transmission of speed for some distance and also uses that motion for removing unwanted material for carried out milling operation. Due to this that spindle always in rotating motion for that we have to calculate the stress at which that spindle runs at infinite cycle. Here we preferred finite element analysis for Fatigue failure. Above mentioned problems are planned for rectified by using re-design and optimization of dimensions and also check feasibility of our new design by using experimental and analytical method and compare both the designs and specification and give optimal solution.*

Keywords: *Spindle, shaft, Fatigue test, finite Element Analysis (FEA). SAE8620 material, Fatigue Test machine, etc .*

I. INTRODUCTION

Shaft, which has a circular cross-section, is a rotating machine element used in mechanical equipment and machines to transmit rotary motion and power. Bearings, flywheels, gears, clutches, and other machine elements are usually mounted on the shaft, and help in the power transmission process. A rotating axis of the machine is very important part. The shaft itself is named spindle, in shop-floor practice, the word often is used metonymy to refer to the entire rotary unit, including not only the shaft itself, but its bearings and anything attached to it. The spindle is the rotating shaft driven by the motor that holds the tool holder or within the case of an older over-arm horizontal mill the tool itself.

Following are the Examples of spindles:

- On a lathe (metal lathe or whether wood lathe).
- In rotating-cutter woodworking machinery, the spindle is the part on which shaped milling cutters are mounted for cutting features (such as curves, rebates and beads) into moldings' and similar millwork.
- Similarly, in rotating-cutter metalworking machine tools (such as drill presses and milling machines),

The spindle is the shaft to which the tool (such as milling cutter or a drill bit) is attached for example chuck.

Varieties of spindles include low-speed spindles, electric spindles, high speed spindles, machine tool spindles, grinding spindles, , and more.

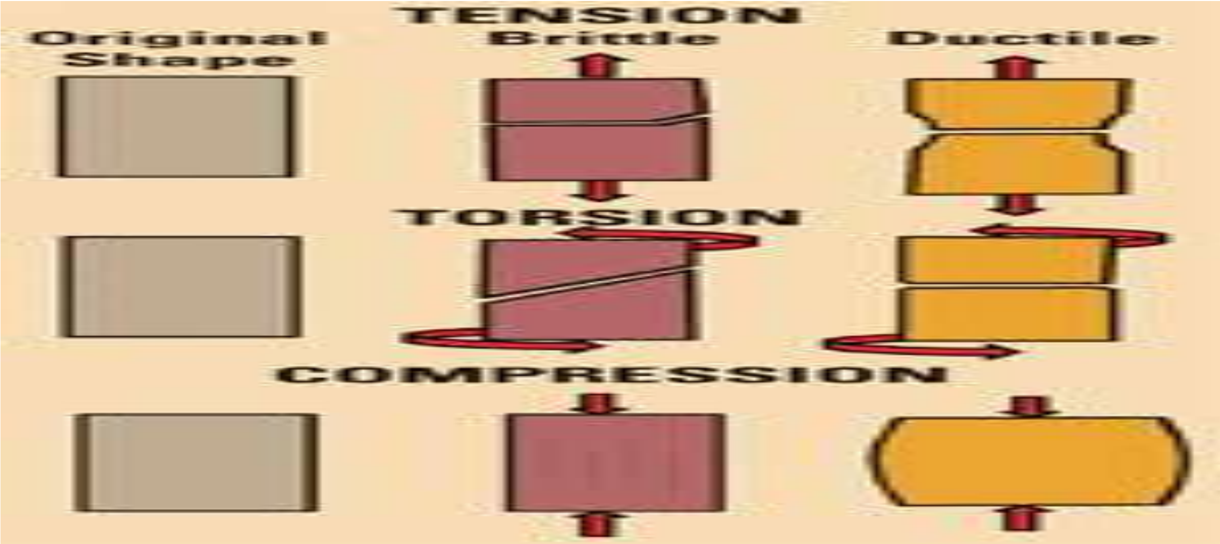
For most machines, shaft analysis should be relatively direct as a result of the shaft failure generally provides strong clues to find out the type and magnitude of forces and direction they acted on the shaft. The failed parts will tell exactly what happened.

There are only four basic failure mechanisms (i.e. corrosion, wear, overload and fatigue). The first two corrosion and wear almost never cause machine-shaft failures; on rare occasions they affected and leave clear evidence. From other two mechanisms fatigue is more common than overload failure.

Overload failures:

Overload failures are caused by forces that exceed yield strength or tensile strength of a material. As the appearance of an overload failure depends on whether the shaft material is brittle or ductile.As the industrial arena grows more sophisticated, it seems as though operations are confronting fewer and fewer broken machine shafts. When shaft breaks, however, there are almost always as many theories regarding the suspected culprits as there are people involved.

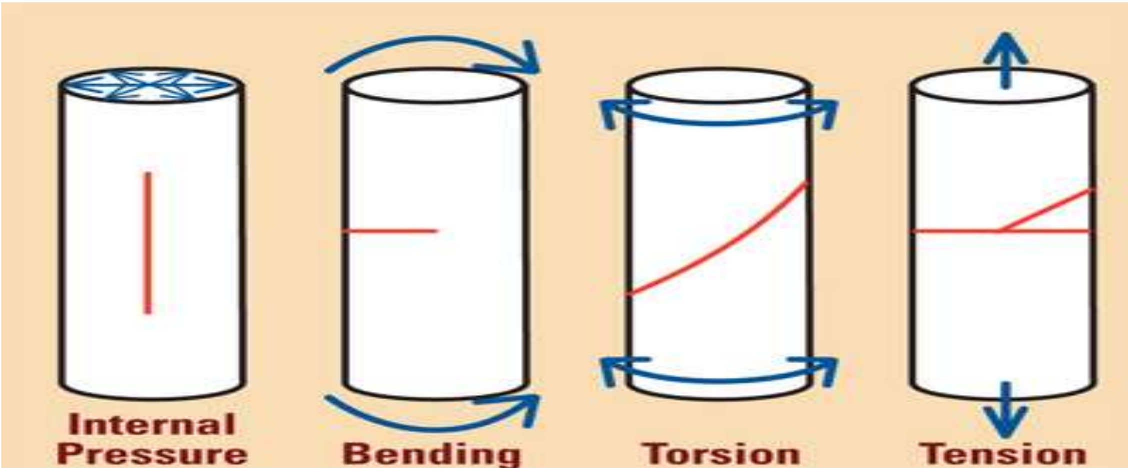
Fig. 1 The appearance of an overload failure depends on whether the shaft material is brittle or ductile.Whether associated with motors, pumps or any other types of industrial machinery shaft failure analysis is frequently revoked, often being perceived as difficult and expensive. For most machine shafts, however, analysis should be relatively straightforward. That's as a result of the failure generally provides strong clues to the type and magnitude of forces on the shaft and therefore the direction they acted in: The failed parts will tell exactly what happene



(Figure1: An overload failure Appearance)

1.1Fatigue failures:

Fatigue is caused by cyclical stresses and the forces that cause fatigue failures aresubstantially less than those that would cause plastic deformation. Confusing the situation even further is the fact that corrosion will reduce the fatigue strength of a material. The amount of reduction is dependent on both the severity of the corrosion and the number of stress cycles.
Once they are visible to the naked eye, cracks always grow perpendicular to the plane of maximum stress. Figure 2 shows the fracture planes caused by four common fatigue forces. Because the section properties will change as the crack grows, it's crucial for the analyst to look carefully at the point where the failure starts to determine the direction of the forces. For example, while it is common for torsional fatigue forces to initiate a failure, the majority of the crack propagation could be in tension. That's because the shaft has been weakened and the torsional resonant frequency has changed.
The condition or roughness of the fracture surface is one of the most important points to look at in analyzing a failure because of the difference between overload failures and fatigue failures. With overload failures crack travels at a constant rate and the surface is uniformly rough. Fatigue-induced cracks however travel across the fracture face at ever-increasing speeds. As a result, the typical fatigue fracture face is relatively smooth near the origin(s) and ends in a comparatively rough final fracture



(Figure2: Fracture planes caused by fatigue forces)

II. EXPERIMENTAL FATIGUE LIFE ESTIMATION:

According to research point of view this test is vital among all. The test is carried out on rotating beam fatigue testing machine. The fatigue testing machine which is used that shown in fig 4.1.

Procedure

A fatigue testing machine is used to test the fatigue properties at various mean stresses. A standard specimen is clamped in two chuck at the end. One end point is fixed and another one is loaded and give motion and apply load on the component equally by adding leaver which join both end and apply load. According to this region of rotating beam between both end is subjected to pure bending with constant loading at its free end under influence of constant load. Specimen is rotated by drive through motor around longitudinal axis due to this rotation under loading condition at component break. Study crack initiation phenomenon due to this method.

Working Principle:
Design of the machine is based on the rotating beam principle. The specimen is a simple beam symmetrically loaded at two points. When rotated one half revolution, the stresses in the fibres originally below the neutral axis are reversed from tension to compression and vice versa. Upon completing the revolution, the stresses are again reversed so that during one revolution the test specimen passes through a complete cycle of flexural stress (tension and compression).

(Figure 3.: Experimental Set)

For calculation of application of load for un notched specimens according to this testing machine reversed bending formula is applicable. Formula taken over here is:

$M / I = \sigma / Y$

Where, $M = W \times g \times L$,

M= Moment of inertia in N-m,

g = Acceleration due to gravity in m/sec2 = 9.81 m/sec2,

W = Weight or load in kg,

L = Total length up to the application of load = 230mm,

σ = Yield stress in MPa,

I = d4/64 in mm4,

Y = d/2 in mm.

d= gauge diameter in mm= 8mm

(Figure 4. Specimen rod of Material SAE8620)

2.1 EXPERIMENTAL RESULTS

Sequence no	Stress (Mpa)	Cycle no (N)
1.	314	31,940
2.	295	79,419
3.	245	55,825
4.	197	1,31,063
5.	157	2,99,040

Table 1: Experimental Results

The benchmark for establishing the behaviour of engineering materials under dynamic/fatigue loading is the "S-N" diagram. Here, "S"corresponds to the stress level and "N" to the number of cycles. Due to the uncertainties involved in the materials' behaviour and characteristics, a large number of specimens are tested at different stress levels for generating the "S - log N" diagram.
Ideally, the main objective in such tests is two-fold. First, to establish (for a given material), up to what stress levels the material will enjoy an infinite life (Endurance Limit); and second, to correlate the number of cycles at different stress levels that a material will be able to go through before failure.

2.2. S-N Curve for SAE 8620 Steel

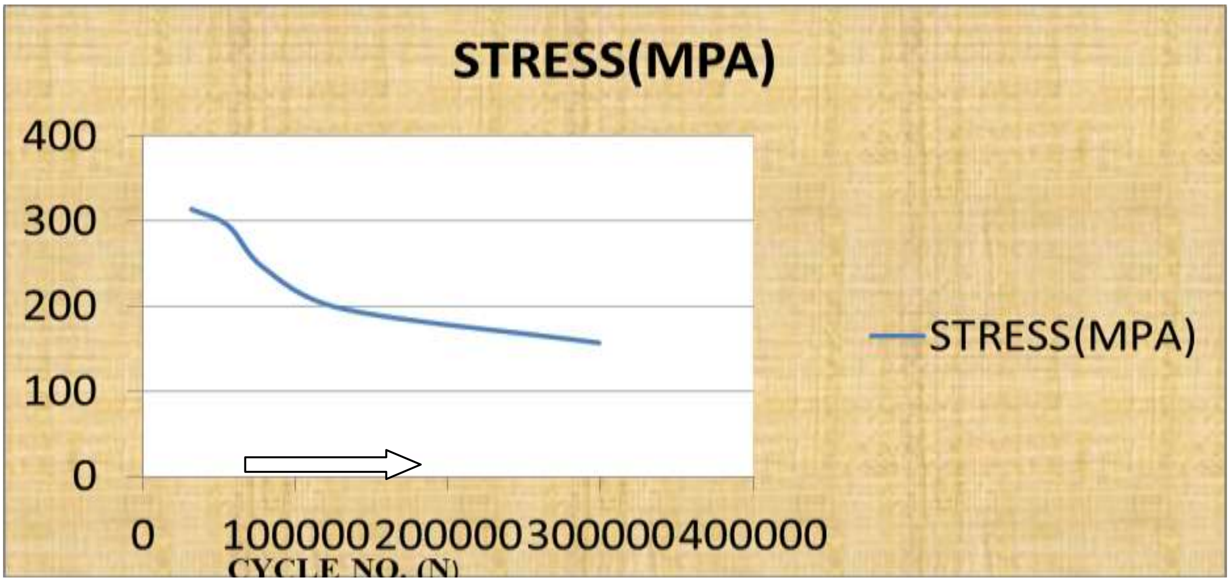


Figure 5.: S-N Curve for SAE 8620 Steel

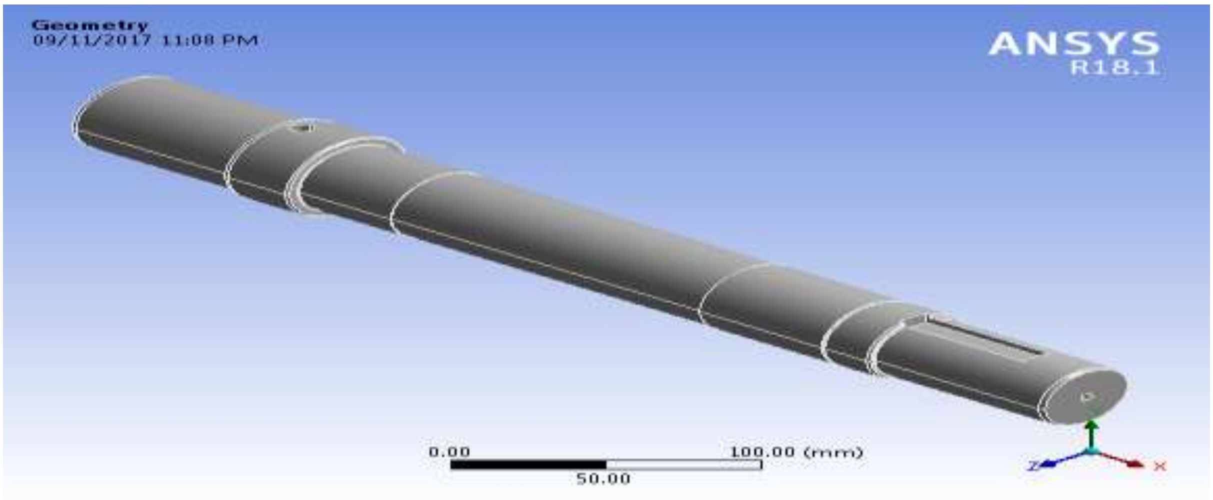
III. FEA ANALYSIS:-

FEA analysis of femur bone is carried out as per the following procedure,

1. Development of spindle CAD Model
2. Meshing Model of spindle
3. Material Properties of spindle
4. Loading and Boundary Conditions
5. FEA Results For Stress Analysis Of spindle

3.1. DEVELOPMENT OF SPINDLE CAD MODEL

To carry out FEM analysis of any component, the solid model of the same is essential. It is also called body in white. So the solid model of spindle is required and this can be done by using CATIA. 2-D model and 3-D model both are essential. CAD Model. CATIA model i.e. 3-D model of material SAE8620 shown in figure 6. for study.



(Figure 6.:3D model of spindle)

3.1.1. MESHING MODEL OF SPINDLE

After creating model, for further Finite element analysis (FEA), surface mesh is generated by using ANSYS for spindle.
ANSYS is a pre-processor which mesh the given components.A complete set of geometry editing tools helps to efficiently prepare CAD models for the meshing process. Meshing algorithms for shell and solid elements provide full level of control, or can be used in automatic mode

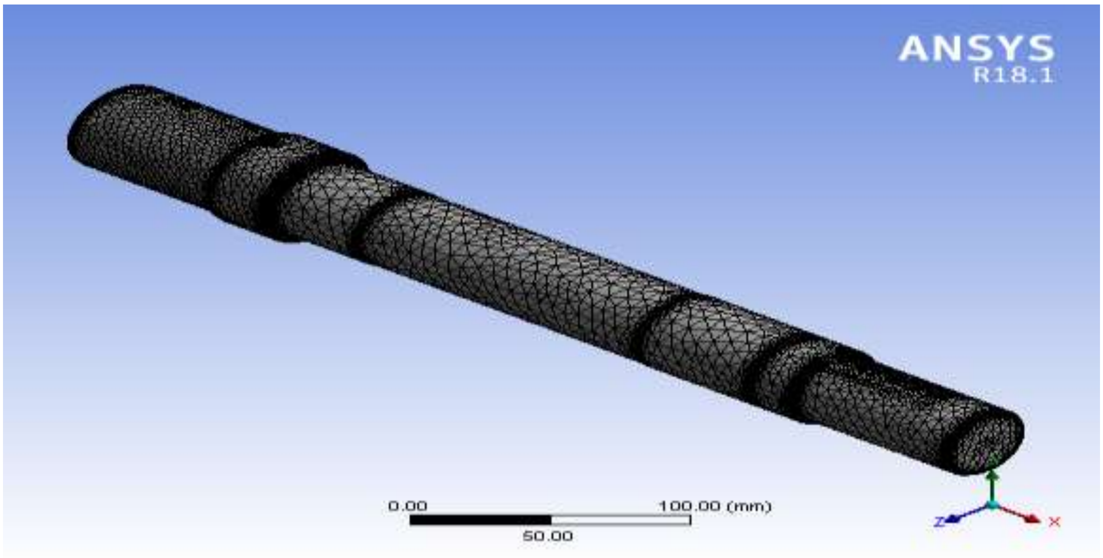


Figure 7. MESHING MODEL OF SPINDLE

3.1.2. MATERIAL PROPERTIES OF SAE8620

Sr. No.	**Properties**	**Value**
1.	Youngs Modulus	2x105 MPa
2.	Tensile strength	650x880 MPa
3.	Elongation	8-20%
4.	Fatigue	275MPa
5.	Yield Strength	350-550MPa
6.	Thermal Expansion	10 e-6/K
7.	Specific Heat	460 J/Kg.K
8.	Melting Temperature	15000C

Table No. 2 : MATERIAL PROPERTIES OF SAE8620

3.2. LOADING & BOUNDARY CONDITIONS

Spindle is solid and inflexible. The Finite element model of spindle with mesh was imported in ANSYS. An eccentric and concentrate loads of 11154 N, 7405 N, 3748 N,392.4 N and moment of 7000 N are applied at the point on spindle at the distance and at one end there moment is applied which found from theoretical analysis and actual conditions. These are shown in figure 8 Also applying material properties i.e. modulus of elasticity, density etc. After all this done. Solve option in ANSYS starts. The ANSYS will start automatically calculation of stresses, deflection.

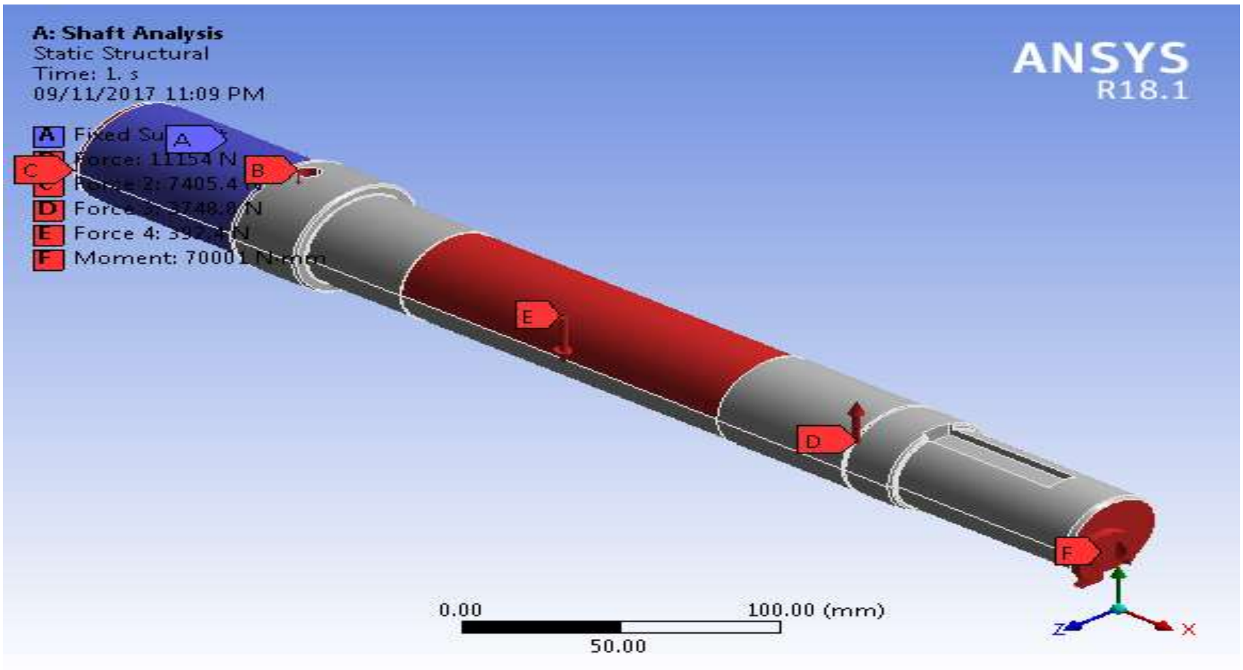


Figure 8: Axial load and constraint on spindle

In summary, we followed following procedure to carry out FEM stress analysis:

1) Go to file, select import and then geometry of component.
2) Select 2D component, click on the auto mesh then click on the displayed from that select element size definition in which also select tetrahedral as a mesh type.
3) Select one component to make close volume from all.
4) Click on the ruled, select nodes and then click on the create button to make that closed volume component.
5) Repeat 3 & 4 nos. procedure for remaining components and after that close the window.
6) After that just go to the 3D meshing tool and select tetra mesh in which click on element displayed and then click on mesh.
7) Next go to analysis and define constraints.
8)Again go to analysis then select forces, select the nodes over which load is applied and then put the value of load .Load value is given with the +/- sign, sign shows direction of load.
9) Click on 3D, element type, select solid as name and define solid 92.
10) From the current window click on the material icon and fill the values for density (kg/m3) and young's modulus (MPa).
11) Go to utility menu then select component manager and give element type and material selection.
12) Go to file menu, export and then select solver deck from that export all (Suppose name of the file is femur)
13) Now come to ANSYS.
14) Open file which saved in hyper mesh.
15) Click on the file menu and then select read input from.
16) Click on the plot and select elements.
17) Go to plt ctrl from that symbols and select all applied properties.
18) Go to solve menu.
19) Go to general post proc, plot results then counter plot and then click on the nodal solution.
20) Repeat procedure from 9 to 19 for next remaining loads.

3.3. FEA RESULTS FOR STRESS ANALAYSIS OF MILLING MACHINE SPINDLE:

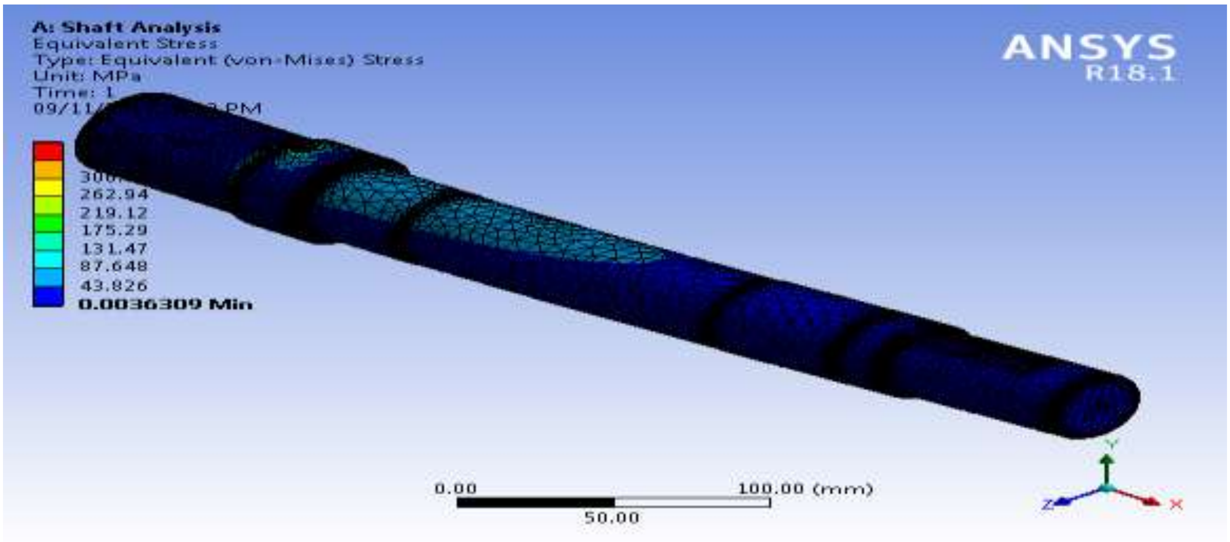


Figure 9. Finite element Analysis of SAE8620 Material for Equivalent stress

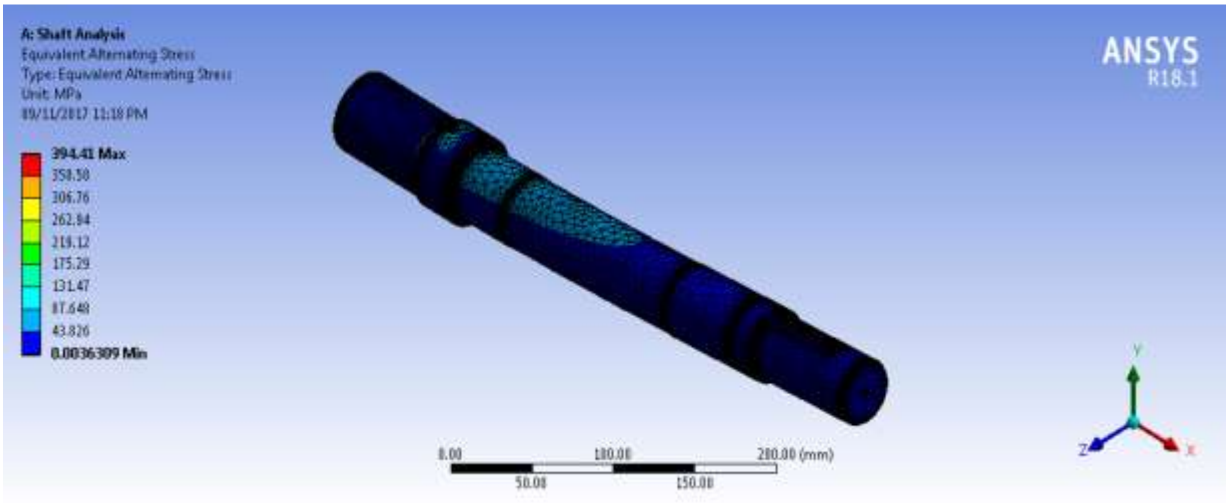


Figure 10. Stress Distribution of Fatigue Equivalent Alternative Force

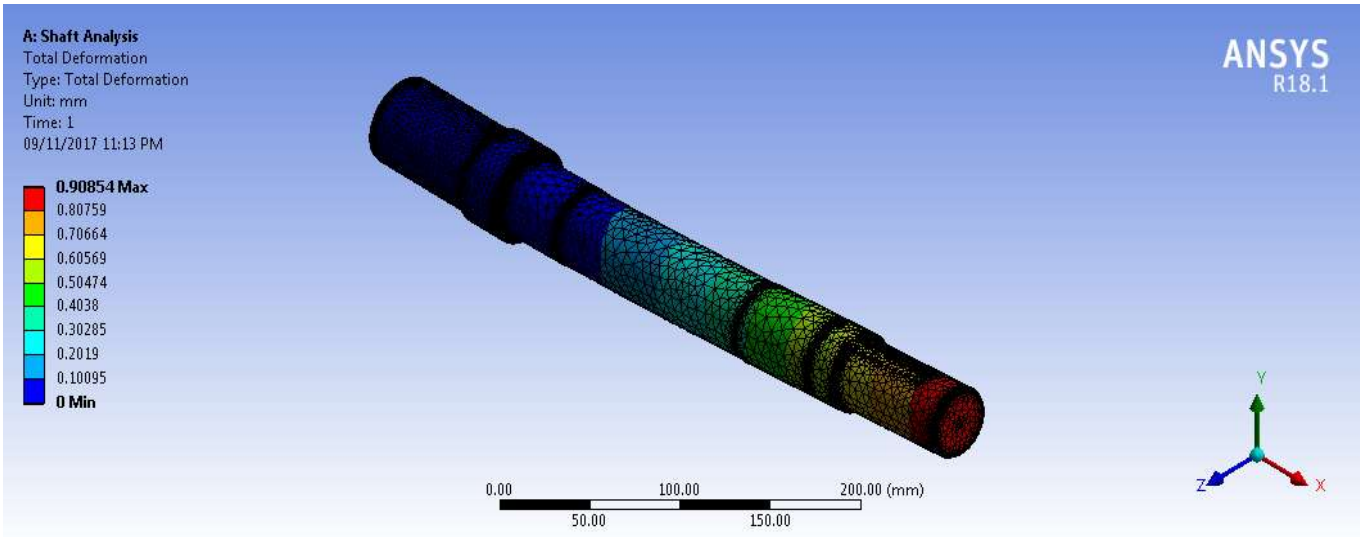


Figure 11. Finite Element Analysis for total deformation

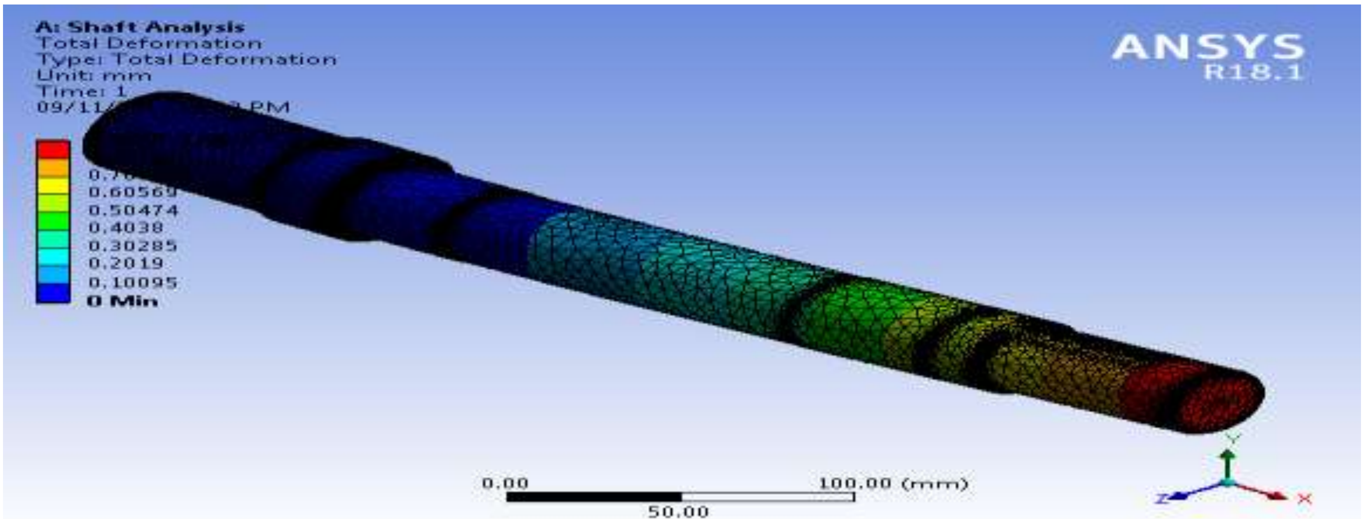


Figure 12 Finite Element Analysis for total deformation

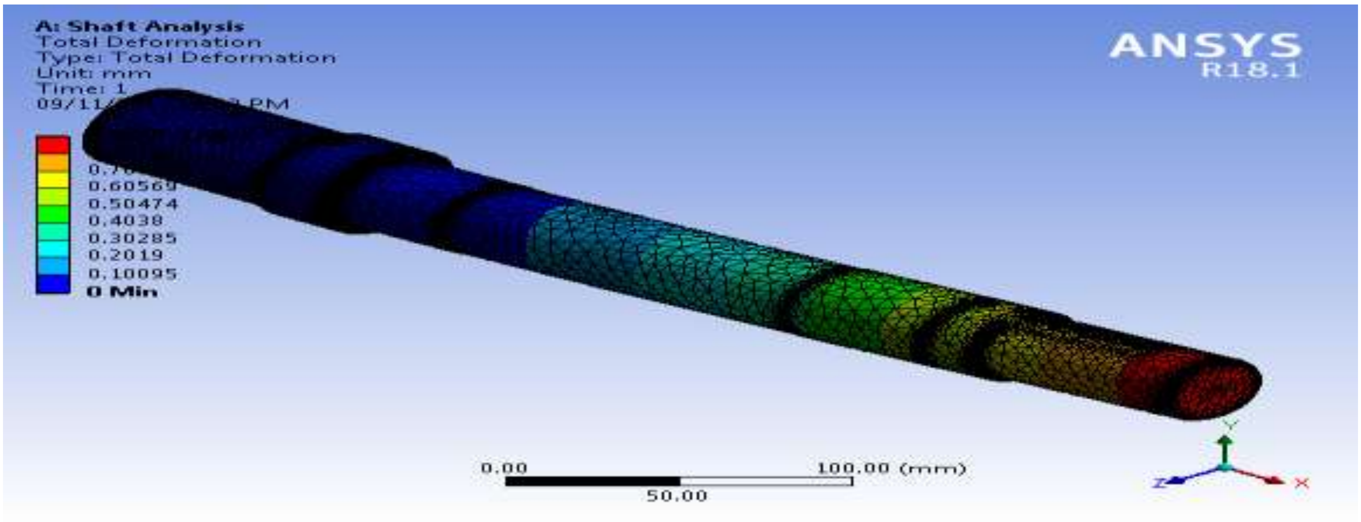


Figure 13. Finite Element Analysis for total deformation

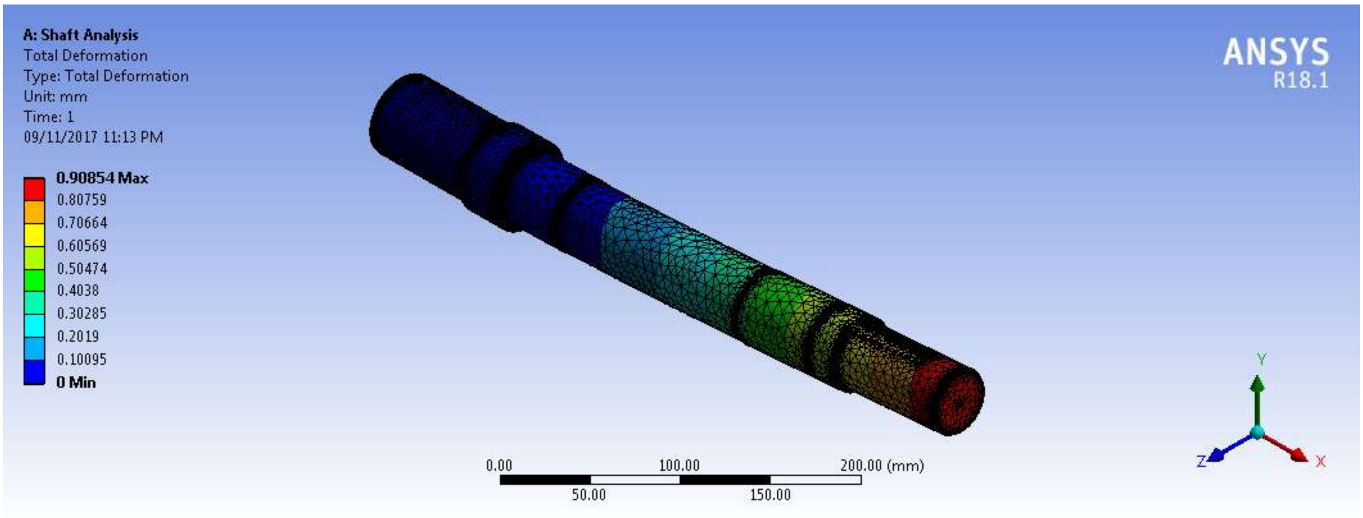


Figure 14. Finite Element Analysis for total deformation

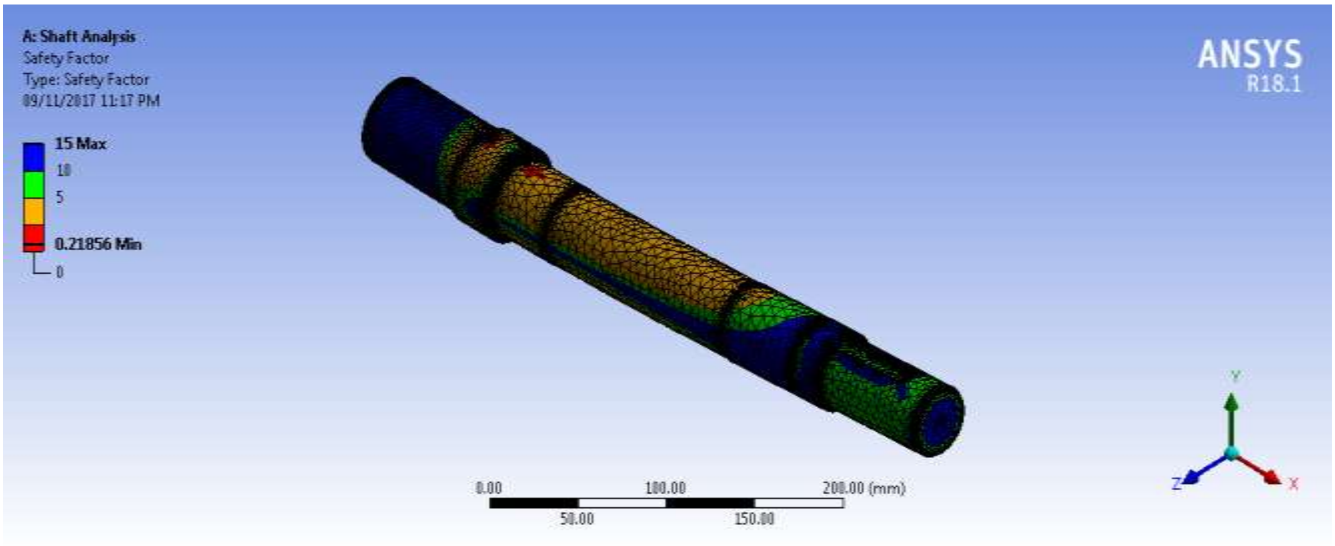


Figure 15. Finite Element Analysis for total deformation and Fatigue Safety Factor

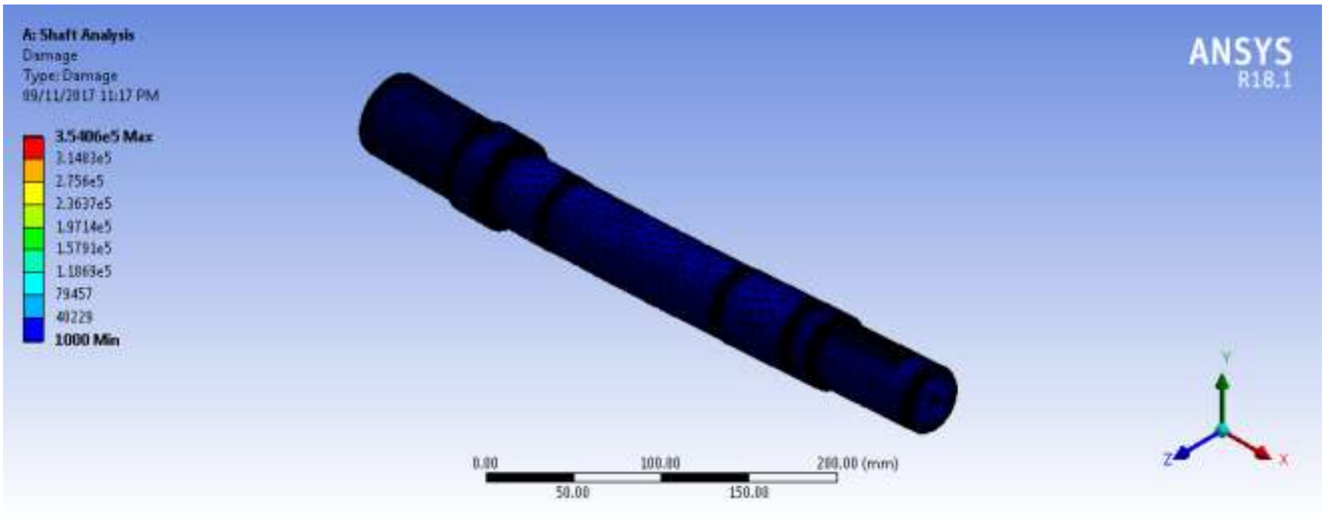


Figure 16: Finite Element Analysis for Fatigue Damage

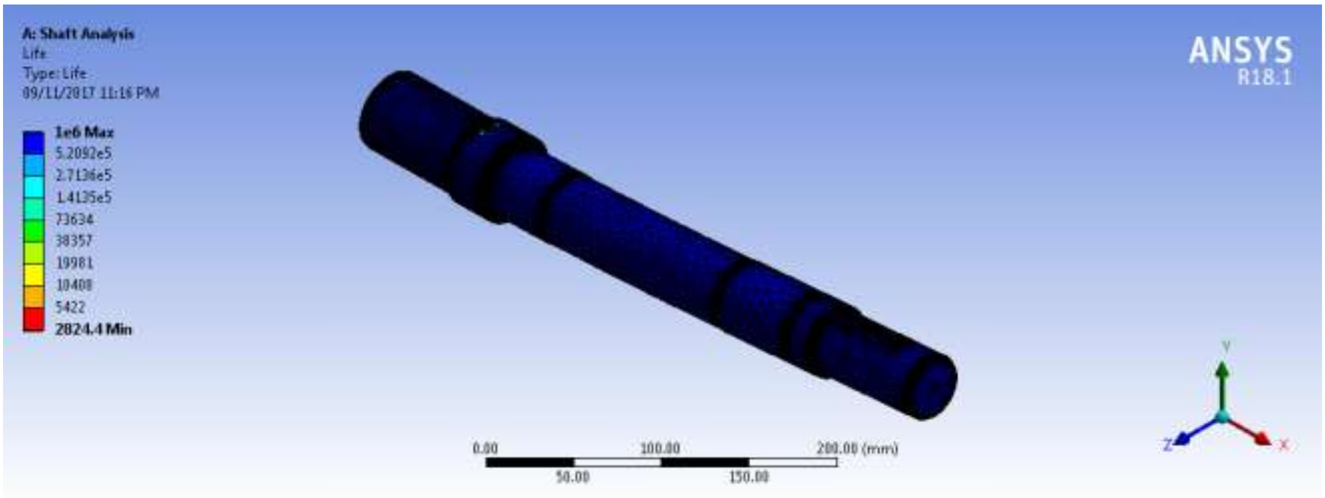


Figure 17: Finite Element Analysis for Fatigue Life

RESULTS AND CONCLUSION

RESULT:

EXPERIMENTAL LIFE CYCLE FOR SAE8620 MATERIAL	299040 CYCLES
FROM FINITE ELEMENT ANALYSIS LIFE CYCLE FOR SAE8620 MATERIAL	271360 CYCLES
PERCENTAGE ERROR	9.25%

Table 3.Result Interpretation

CONCLUSION

In this project worked on life cycle of SAE8620 material and get life cycle values by experimental is 2990440 cycle and from finite element analysis life cycles are 271360 cycles. From that result, there is 9.25% error is obtained which is permissible.
That whole project is carried out for comparison of en8 material with SAE8620 material. Whole properties of en8 material are taken from standard specification and that properties are compare with properties of SAE8620 material which are finalized by carried out experimental and theoretical with analytical method like finite element analysis.
In this whole project by work find out life cycle of material SAE8620 are 299040 cycles and as per standard data life cycle for EN8 material are 220260. As per comparison fatigue life of SAE8620 is little bit more than EN8 material.
From the all over work of project minimize the spindle dimension, as data provided by MAHARASHTRA ENGINEERING SYSTEM dimensions of EN8 spindle is 66mm diameter average and from this project work that dimension is reduced up to 64mm by changing material to SAE8620. Which result in to reduction in weight of spindle which effect on its lifespan….

REFERENCES

[1] Tushar R Deshmukh. , Prof/Mrs Jaya G Broke , "Design and Manufacturing of Spindle of Universal Milling Head", International Journal for Research in Applied Science & Engineering Technology (IJRASET), ISSN: 2321-9653, Volume 5 Issue II, February 2017
[2] P. S. Valsange, M. L. Kulkarni, "Design and analysis of spindle head for micro milling machine" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), p-ISSN: 2320-334X, Volume 11, Issue 3 Ver. VII, PP 98-109, May- Jun. 2014.
[3] R. A. Gujar, S. V. Bhaskar (2013). "Shaft Design under Fatigue Loading By Using Modified Goodman Method", International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 3, Issue 4, Jul-Aug 2013, pp.1061-1066.
[4] A. Göksenli , I.B. Eryürek (2009). "Failure analysis of an elevator drive shaft", Engineering Failure Analysis 16 (2009) 1011–1019.
[5] DeepanMarudachalam M.G, K.Kanthavel, R.Krishnaraj "Optimization of shaft design under fatigue loading using Goodman method", International Journal of Scientific & Engineering Research Volume 2, Issue 8, August-2011, 2229-5518.
[6] Sandeepgujaran andshivajigholap "fatigue analysis of drive shaft", international journal of research in aeronautical and mechanical Engineering Issn (online): 2321-3051.
[7] JaroslavMackerle "Finite-element analysis and simulation of machining: a bibliography (1976–1996)", Journal of Materials Processing Technology 86 (1999) 17–44.
[8] Practical Finite Element Analysis- N. S. Gokhale, S. S. Deshpande
[9] Introduction to Finite Elements in Engineering- Ashok. D. Belegundu.
[10] V.B. Bhandari "Design of Machine Elements", McGraw Hill Education, 3rd edition sixteenth reprint 2014.