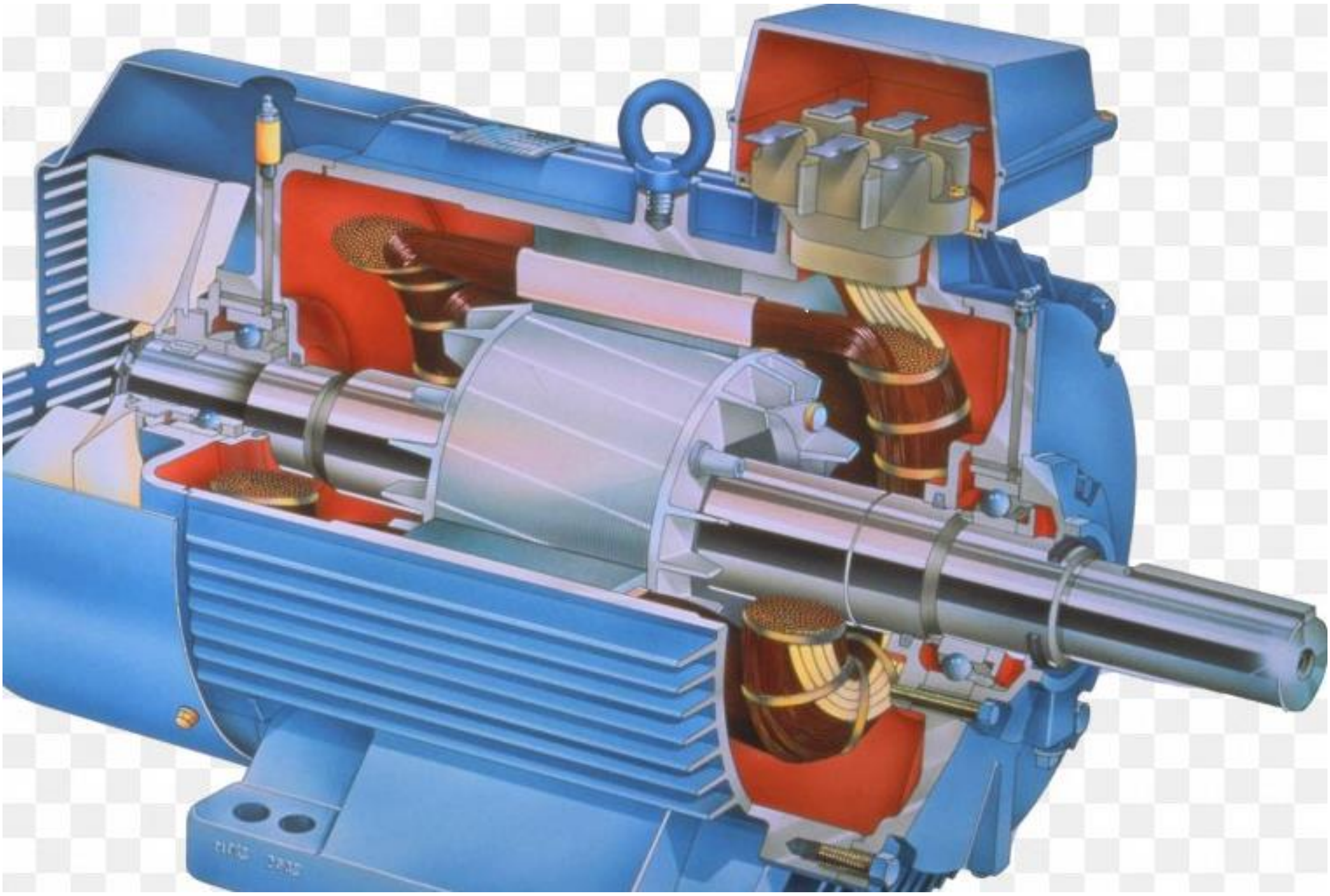
"STATIC AND DYNAMIC ANALYSIS FOR ROTOR SHAFT OF ELECTRIC MOTOR"



Author

Dr. Somnath G. Kolgiri

(ME & PhD Mechanical Engineering)

Associate Professor (HOD & CEO)

Department of Mechanical Engineering

S. B. Patil College of Engineering, Pune

Abstract

Shaft is common component used in variety of applications across industries. This is typical shaft used in motor which carried rotary components. Analytical static design is well defined procedure whereas analytical dynamic design of shaft for has not well established. In case of dynamic operations the behavior of shaft is different and stresses induced are multi-fold than static case. This project is sponsored by Laxmi Hydraulics Pvt. Ltd., Solapur (LhpL).

In this thesis a validation of static design is carried out along with FEA solution for static and dynamic case. Results are verified with the designs available in the company. Several recommendations were then made regarding possible solutions considering various types of loading conditions. Stresses, deflections and safety factors of the shafts are checked by commercial finite element analysis software (ANSYS 11.0). Results of finite element analysis software are within the limits as compare to analytical method.

First chapter discusses relevance and design methodology of electric motor.
Second chapter elaborates extensile literature survey regarding analysis of rotor shaft problems and finite element analysis aspects.
Third chapter records for problems in motor rotor shaft design Forth chapter records force analysis of motor rotor shaft and detail standard design procedure of combine twisting and bending moments of shafts.
Fifth chapter explain, finite element method approach for shaft design and model.
In sixth chapter contents background and company profile.
In seventh chapter, results obtained from commercial finite element analysis Software (ANSYS 11.0).

Keywords: Shaft, Static Loading, Dynamic Loading, CAD and FEA

Chapter 1

Introduction

1.1 Relevance :

Design methodology of electric motor

Electric motors are well known as 'Basic Prime movers' for Engineering Applications. Electric motors are available at different speed, sizes & output power. Due to its clean & noiseless operation, the electric motors are very popular drive. [1]

Basically, electric motors are classified as synchronous motor, A.C. induction motor & D.C. Motor. Among these, induction motors are used in large numbers. They are, therefore, standardized for various uses e.g. for mounting dimensions, for expected performance requirements etc. The induction motors run on 'induction principle'.

The principal components of electric motors are as shown in figure.1

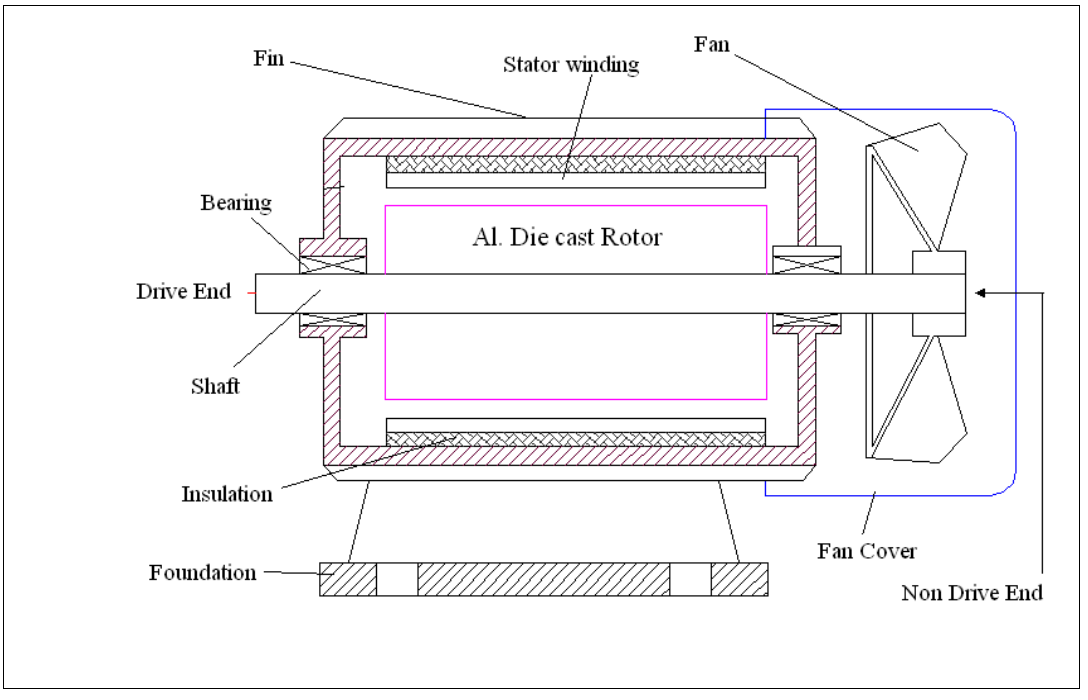


Figure.1 Principal Parts of the Electric Motor

Electric motors are essential in today's society. These devices power everything from small household appliances to large industrial processing equipment, and their potential applications continue to grow. The object of this study is the analysis of such an application. A high percentage of these motors are experiencing interference problems during running conditions, resulting in a high scrap rate for the manufacturer. There were two goals established for this study. The first goal was to model as accurately as practical the physical aspects of this unit and to determine the mechanical response of the system due to force inputs characteristic of those encountered during operating conditions. The second goal was to use the results of the study to make suggestions to the manufacturer regarding potential design modifications to eliminate the interference problem.

The motor unit is modeled using the finite element method (FEM). This method was required due to the lack of closed form solutions for the geometry of the motor components. It was initially known that the interference encountered during operation was a radial displacement of the rotor larger than the stator rotor air gap. It was felt that the ability to reproduce the magnitude of this interference for visual inspection with the software was of prime importance and much effort was put towards making this possible. The forcing functions used in predicting the component displacements were typical of those resulting from the air gap eccentricity that can occur in an electric motor. The nature of these forces was determined by other investigators. The true magnitudes of the forcing functions were not determined. Instead, the magnitude of unity was used for the analysis. This approach yielded a set of displacements that may be scaled accordingly.

The study is organized into several chapters. Chapter 2 presents the review of the technical literature for the study. Chapter 3 presents the background of the motor and the interference problem. Chapter 4 presents the modeling approach used for the FEM analysis, including all simplifications. Chapter 5 presents Finite Element Analysis procedure. Chapter 6 presents all Motor rotor shaft design: Case Study**.** Chapter 7 presents the results of the FEM analysis and analytic discussions**.** Chapter 8 presents all conclusions from the study and areas to be considered for future investigations.

1.2.1 Electrical Engineering orientation

Classification of A.C. Motors: [2]

1) According to principle of operation.

Synchronous motor.

i) Plain ii) super

Asynchronous motor.

(a) Induction motors.

Squirrel cage: single & double

Slip ring (external resistance)

(b) Commutator motors.

Series: single phase & universal.

Compensated: conductively & inductively.

Shunt: simple & compensated.

Repulsion: straight & compensated.

Repulsion starts induction.

Repulsion induction.

2) According to type of current.

i) Single phase ii) three phase

3) According to speed.

i) Constant speed ii) variable speed iii) adjustable speed

4) According to structural feature.

i) Open ii) enclosed iii) semi-enclosed

iv) Ventilated v) riveted frame eye

Induction Motor-General Principle:

As a general rule, conversion of electric power into mechanical power takes place in the rotating part of electric motor. The rotor does not receive electric power by conduction but by induction in exactly the same way as of a two winding transformer receives its power from the primary. That is why such motors are called as induction motors.

Construction of Induction Motor:

An induction motor consists of essentially two main parts

a) Stator

The stator of an induction motor to made up of number of stampings, which are slotted to receive the windings. The stator carries a 3-phase winding & is fed from 3-phase supply. It is wound by definite no. of poles. The no. of poles P, produced in the rotating field is p=2n, where n = number of stator of slots / poles/ phase. The exact number of poles is determined by the requirement of speed. Greater the number of poles, lesser the speed & vice versa.

b) Rotor

i) Squirrel case rotor

Almost 90% of induction motors are squirrel case rotor type, because this type of rotor has the simplest and most rugged construction. The rotor consists of cylindrical laminated core with parallel slots for carrying the rotor conducts which, it should be noted clearly, are not wires but consists of heavy bars of copper, aluminum or alloy. One bar is placed in each slot, rather the bars are inserted from the end of when semi closed slots are used.

ii) Phase – wound rotor

This type of rotor is provided with 3 phase, double layer, distributed winding consisting of coils as used in alternator. The rotor is wound for or many poles as number of stator poles and is always wound 3-phase even when the stator is wound two phases.

1.2.2. Mechanical Design orientation

Mechanical design consideration plays a most important role in the design of any induction motor. Many of them do not lend themselves to computer operation, but we have found the computer of considerable value in at least two areas of mechanical design problems.

A. Shaft and Bearing Calculations.

Radial load on the two bearings of a small induction motor plays an important part in selecting the proper size of ball bearing needed or in estimating the life of the bearing selected. An associated problem is to determine the combined shearing and bending forces acting on the shaft at various positions along its length so that its size may be made adequate at each of these places. All these related factors depend upon several things: horsepower developed and rpm, continuous and maximum; whether load is coupled or connected by means of a pinion, V-belt, or flat belt; elastic modulus and fatigue strength of the shaft material used; location and number of keyways in the shaft; weight of the rotor; unbalanced magnetic pull on the rotor; length between bearings; and length of extension. A computer program was developed to calculate all the desired factor listed above from the information given. This program, too, has been most useful.

B. Critical-speed Calculations

Critical speed is often a problem in high-speed machines, particularly when the length between bearings is great. Before the computer was obtained, the first critical speed was occasionally calculated by hand. The approach was based upon a graphical double integration of the classic expression

$$d2y /dx2 = M/EI$$

Where M= bending moment

E= modulus of elasticity of the shaft material

I= moment of inertia of the shaft section.

Essentially the process consisted of computing the moment of inertia for each section, drawing a curve of shaft moment along the shaft, and graphically performing the double integration indicated to obtain the shaft-deflection curve from which the critical speed was readily calculated. For a computer program, the most general case likely to be encountered was assumed. A typical shaft was assumed to be made up of 17 sections of different length and diameters, with a mass on each of these sections. For simpler problems, two or more sections are assumed to be identical, so that the program is useful for a wide range of rotors. This program has turned out to be most useful, and many more critical-speed calculations are now made than formerly, when they had to be done by hand.

1.3. Introduction to Rotor Shaft

Stresses in rotor shafts are complex, normally involving combinations of bending, torsion, and shear. Some stresses alternate or reverse in direction with each shaft revolution; others may be cyclically varying. Good design dictates holding maximum stress below the endurance or fatigue limit - the fairly low stress which can be sustained indefinitely through cycles of application.

Shock loading is encountered in motors for crushers and similar machines. That greatly increases shaft stress. It may be sudden load (in which two drive components are in contact before force is applied between them) or impact (in which one component moves through a distance before striking the other.)

Unlike strength, shaft stiffness (resistance to bending) is independent of steel composition. Shaft critical speed or resonance is a function of shaft deflection under the influence of gravity. Calculated shaft deflection, depends on rotor weight to be carried by the shaft, and upon the method chosen to allow for the non-uniform diameter of most shafts. In large machines, shaft stiffness is often enhanced by welding radial arms to the shaft, on which the rotor is mounted.

The shaft-rotor assembly will assume a deflection curve shape by bending under its own weight. Also depending upon the couplings used, the shaft may be subjected to unacceptably high bending stress.

When an existing motor is redesigned for higher horsepower, or is applied to a different type of load (particularly with a belt drive), a design analysis is necessary to ensure that increased stress does not risk shaft failure.

1.4. Function of Rotor Shaft

Provide an axis of rotation

Used to transmit power

Used to position / mount gears, pulleys, bearings etc.

Chapter 2

Literature Survey

Much information is available in the technical literature regarding dynamic interferences, which are commonly referred to as bumping, between the rotor and housing of Electric motor. However, this information is heavily biased towards such issues as the physics associated with the component contact and the effects of degradation problems such as excessively worn bearings and loose rotor blades. Since the system under study utilizes all new components it was decided that these findings were not relevant to this analysis.

Several references exist in the literature dating from the 1910's through the 1960's regarding air gap eccentricity in induction machines and one of its fundamental consequences unbalanced magnetic pull (u.m.p.). However, the theories behind many of these references are unclear and produce no definite conclusions.

In 1974, Rai [3] completed extensive experimental investigations on all aspects of air gap eccentricity in induction motors. These experiments examined both static and dynamic eccentricities on 2-pole motors featuring unslotted, wound and cage rotors. These experiments consisted of extensive measurements of eccentricity fields, air gap flux density, unbalanced magnetic pull, vibratory forces, losses and torque under various operating conditions from no- load to full-load and during starting. In this work it was determined that the following driving forces of different frequencies can occur in electrical machines Mechanical unbalance producing driving forces at the frequency of rotation. Inherent magnetic radial forces at twice line frequency. Static eccentricity producing twice line frequency vibratory driving forces. Defective rotor winding producing twice slip frequency vibration.

Dynamic eccentricity producing vibratory forces at rotational speed with a modulation at twice slip frequency. In addition, small vibratory forces at line frequency and twice line frequency can occur Small static eccentricities can produce vibratory forces at line frequency. Stator harmonics can produce radial forces at twice-line frequency. Certain combinations of harmonics cause vibrations at twice slip frequency.

It was reported that in general, a number of these potential driving forces might exist depending on the types of asymmetry present.

In 1989 Timar, Fazekas, Kiss, Miklos and Yang produced a text on noise and vibrations in electrical machines [4]. In the text it was stated that in single-phase asynchronous motors unbalanced magnetic pull can result in the production of a radial forces at angular frequencies of 2f1, 2sf1, 2(1-s)f1, 2(2-s)f1, (3-s)f1 and (1±s)f1 where f1 is the line frequency and s is the slip frequency of the rotor. Slip frequency is defined as the frequency of the voltages induced in the rotor due to relative motion between the stator and rotor fields.

In 1994 Ozturk, Balikcioglu, Acikgoz and Bahadir produced a paper on the origins of electromagnetic vibrations in series fractional horsepower motors [5]. This paper summarized the works of Rai, Timar, and others with respect to unbalanced magnetic pull and the resultant forces.

In their research the authors used radial forces due to static eccentricity occurring at twice the voltage supply frequency and radial forces due to dynamic eccentricity occurring at rotational frequency. The concept of static and dynamic eccentricities will be discussed in Chapter 3.

The final area explored in the literature was the effect of laminations used in the construction of the stator and rotor. In 1979 Girgis and Verma [6] determined that the laminated construction of stators has virtually no effect on the location of natural frequencies. However, laminations cause a significant damping in amplitudes of vibration and thus must be taken into consideration when performing stator analysis.

Chapter 3

Motor Rotor Shaft Design Methodology

3.1 Problems in Motor rotor shaft design

3.1.1. Failure causes

In order to properly dimension the rotating system, the failure causes or modes have to be determined. Most failures are due to unpredictable factors and the cure is to create a robust design that can cope with the unknown. Predictable failures or lifetimes can be calculated although many parameters will vary and statistical considerations have to be taken into account.

Shaft

The shaft is designed to have an infinite life. This will be the case unless the shaft is overloaded or damaged.

Fatigue crack

The cause of a broken shaft is almost always fatigue. A crack starts at a stress concentration from a keyway or a sharp radius, or, in rare cases, from a material impurity. Flaws in the surface of a shaft, such as scratches, indents or corrosion, may also be the starting point of a fatigue crack. Loads that drive a crack are normally torsion loads from direct online starts or bending loads from the pulley or coupling end.

Plastic deformation

Plastic deformation can only occur in extreme load cases when debris is squeezed into a radial clearance that results in large deformations.

Defect shaft

Defect shafts that have passed the checks in the factory, checks for unbalance, as well as the check at test run are very rare. On site, however, it is important to prevent damage from corrosion, indents etc, by proper handling.

Joint

A poorly designed joint between the shaft and the rotor can be detrimental for the system both at mount (high mounting forces, impacts) and at run.

Loose connection

A loose connection can cause impacts that generate high loads and damage on the shaft

Misaligned connection

This is normally hard to achieve but with incorrect mounting it may occur.

Bearing

A roller bearing does not last forever, sooner or later fatigue or wear, or lubricant deterioration will ultimately destroy the bearings ability to function properly.

3.2. Highlight Design problems of Rotor shaft

Determine the rotational speed of the shaft, n (rpm).

Select the material from which the shaft will be made, and specify ultimate tensile strength and yield strength.

Apply a desired reliability for definition of reliability factor, CR.

Apply a design factor, N.

Propose the general form of the geometry for the shaft, considering how each element on the shaft will be held in position axially and how power transmission from each element to the shaft is to take place.

Specify the location of bearings to support the shaft. The reactions on bearings supporting radial loads are assumed to act at the midpoint of the bearings.

Determine the design of the power-transmitting components which will be mounted on the shaft, and specify the required location of each device.

Determine the power to be transmitted by the shaft.

Determine the magnitude of torque at point of the shaft where the power-transmitting element is mounted.

Determine the forces exerted on the shaft

Preparing a torque diagram.

Resolve the radial forces into components in perpendicular directions, vertically and horizontally.

Solve for the reactions on all support bearings in each plane.

Produce the complete shearing force and bending moment diagrams to determine the distribution of bending moments in the shaft.

Analyze each critical point of the shaft to determine the minimum acceptable diameter of the shaft at that point in order to ensure safety under the loading at that point.

If a vertical shearing force V is the only significant loading present, this equation should be used to compute the required diameter for a shaft.

Displacement

Even if a shaft can cope with rather large deflections, other components such as seals etc may fail if the deflections or angular deformations are too large.

Stresses

The static and dynamic stresses of the shaft are needed to design the shaft accurately with respect to plastic deformation and fatigue Stresses in rotor shafts are complex, normally involving combinations of bending, torsion, and shear. Some stresses alternate or reverse in direction with each shaft revolution; others may be cyclically varying. Good design dictates holding maximum stress below the endurance or fatigue limit - the fairly low stress which can be sustained indefinitely through cycles of application.

Unlike strength, shaft stiffness (resistance to bending) is independent of steel composition. Shaft critical speed or resonance is a function of shaft deflection under the influence of gravity. Calculated shaft deflection, depends on rotor weight to be carried by the shaft, and upon the method chosen to allow for the non-uniform diameter of most shafts.

In large machines, shaft stiffness is often enhanced by welding radial arms to the shaft, on which the rotor is mounted.

The shaft-rotor assembly will assume a deflection curve shape by bending under its own weight. Also depending upon the couplings used, the shaft may be subjected to unacceptably high bending stress.

When an existing motor is redesigned for higher horsepower, or is applied to a different type of load (particularly with a belt drive), a design analysis is necessary to ensure that increased stress does not risk shaft failure.

3.3. Motor rotor shaft design methodology suggested

Richard L.Nailen, EA [7] Engineering Editor has emphasized in his article that the forces, torques, and bending moments that are created in the shaft during operation. In the process of transmitting power at a given rotational speed, the shaft is inherently subjected to a torsional moment, or torque. Thus, torsional shear stress is developed in the shaft. Also, a shaft usually carries power-transmitting components, such as gears, belt sheaves, or chain sprockets, which exert forces on the shaft in the transverse direction (perpendicular to its axis). These transverse forces cause bending moments to be developed in the shaft, requiring analysis of the stress due to bending. In fact, most shafts must be analyzed for combined stress.

Because of the simultaneous occurrence of torsional shear stresses and normal stresses due to bending, the stress analysis of a shaft virtually always involves the use of a combined stress approach. The recommended approach for shaft design and analysis is the distortion energy theory of failure. Vertical shear stresses and direct normal stresses due to axial loads also occur at times, but they typically have such a small effect that they can be neglected. On portions of shaft where no bending or torsion occurs, such stresses may be dominant.

3.4. Interference problem and initial assumptions

As stated earlier the interference experienced during operation occurs between the stator and rotor of the motor. Currently each motor must be tested for this problem as it leaves the assembly line, and approximately thirty-five percent fail.

At the beginning of this analysis the concept of unbalanced magnetic pull was not known to the author. The initial assumption was that the 1750 rpm (29.2 Hz) frequency of the torque applied to the rotor was close to an eigenvalue of one of the assemblies in the system, thus causing random lateral rotor displacements. Rotor shaft, rotor, and upper and lower support collars. If the actual amount of interference between the components could be accurately determined using finite element analysis, appropriate recommendations for modifications could be made. It was felt that the most likely cure would be to increase the air gap between the stator and the rotor. There would be a practical limit to an air gap increase, however, as this increase has a negative effect on motor output.

3.5Air gap asymmetries and the resulting forces

The discussion of air gap asymmetries and unbalanced magnetic pull presented in this section is a synopsis of the works of Rai [3] and Timar [4].

The main task in creating electromechanical energy conversion by an asynchronous or induction machine is to produce a rotating sinusoidal magnetic field with a pole number 2p in the air gap. A magnetic flux is induced in the air gap due to current flowing through the conductors in the slots.

These slots break up the uniformity of the air gap. Thus, the gap will change periodically as the relative position of the stator and rotor changes. The total magnetizing current of an induction machine is supplied from the stator (primary winding), and therefore the air gap is designed to be as small as possible. However, even the smallest deviation of the air gap from the shape of a cylindrical sleeve of uniform thickness will result in considerable relative distortion of air gap pretence.

Deviation from a uniform air gap is also highly possible due to tolerances required in the manufacturing process. This non-uniform air gap can result from one or more of the following manufacturing flaws:

1) Imperfect centering of the rotor in the stator bore,

2) Non-circular stator bore,

3) Non-circular rotor and

4) Bent shaft.

The first two flaws produce a stationary or static eccentricity in the air gap of an electric machine, while the last two flaws produce a rotating or dynamic eccentricity.

Regardless of whether the air gap eccentricity is static, dynamic or both, the resulting gap will be at a maximum at one point thus producing a minimum flux and a minimum at one point thus producing a maximum flux. A non-uniform flux can result in the following adverse effects:

Flux harmonics

Unbalanced magnetic pull

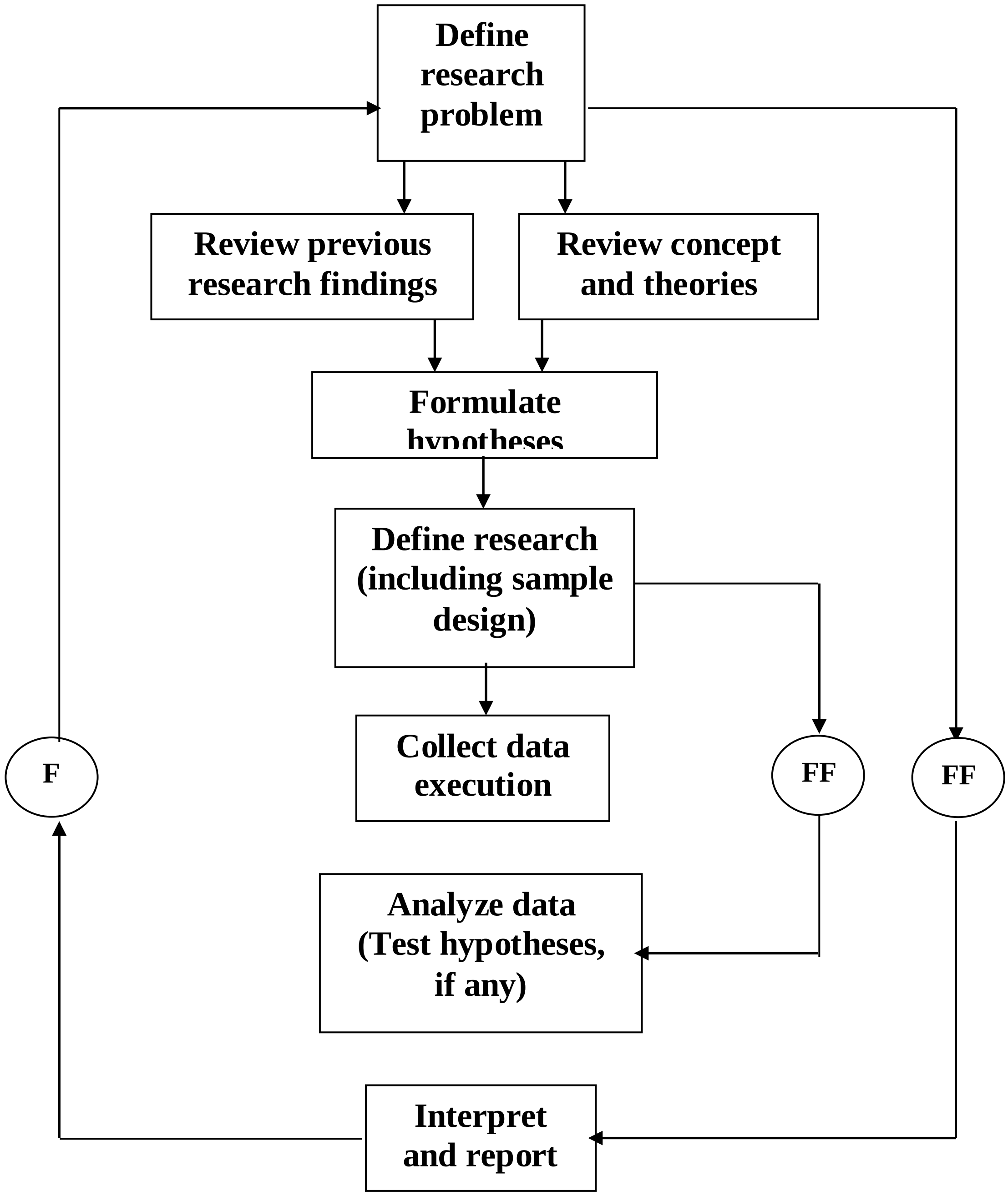
Noise

Harmonics in the rotor and stator currents

Variation in losses and irregular distribution of heat around the stator bore as a result of 1) and 4)

Variation in torque as a result of 1) and 4), both during starting and steady state conditions.

3.6. Route map of solution



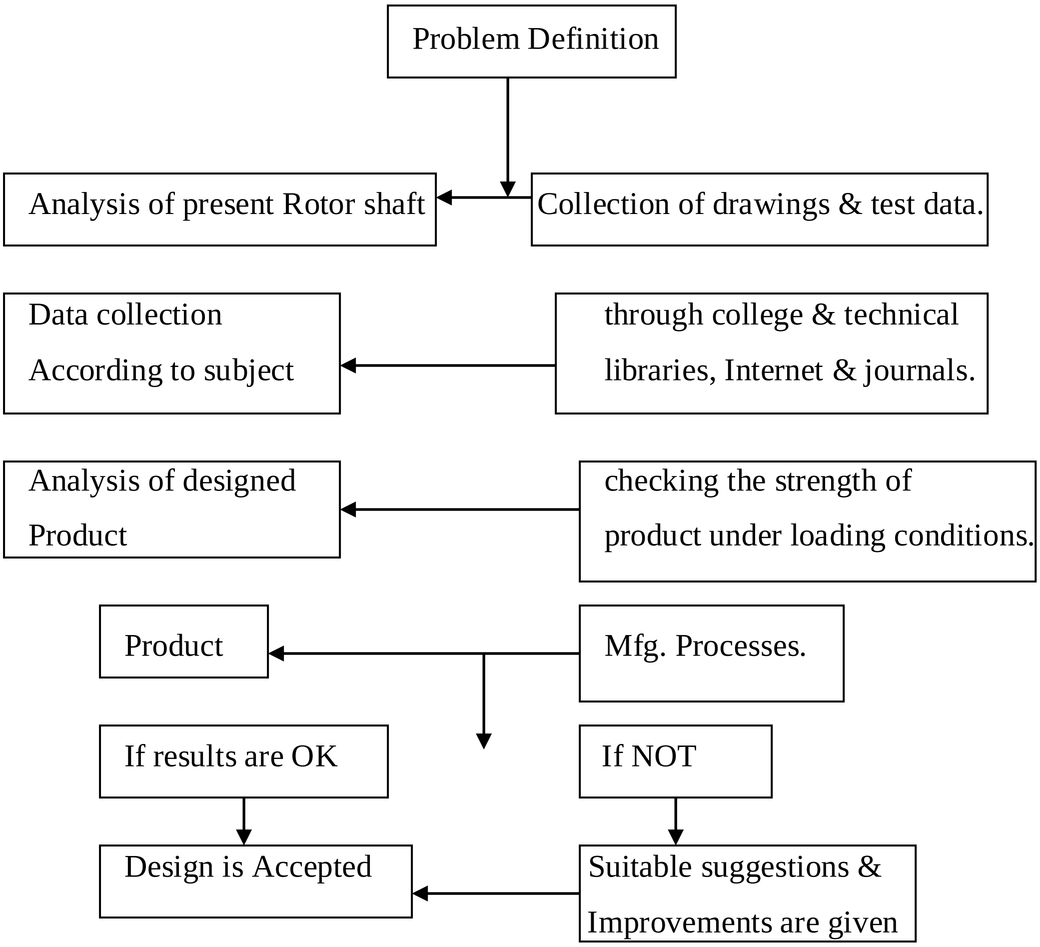


Figure. 2 Research work flow chart

Chapter 4

Force Analysis of Motor Rotor Shaft

4.1 Introduction

Stresses in steel shafts are complex, normally involving combinations of bending, torsion, and shear. Some stresses alternate or reverse in direction with each shaft revolution; others may be cyclically varying. Good design dictates holding maximum stress below the endurance or fatigue limit - the fairly low stress which can be sustained indefinitely through cycles of application.

Shock loading is encountered in motors crushers and similar machines. That greatly increases shaft stress. It may be sudden load (in which two drive components are in contact before force is applied between them) or impact (in which one component moves through a distance before striking the other.)

Unlike strength, shaft stiffness (resistance to bending) is independent of steel composition. Shaft critical speed or resonance is a function of shaft deflection under the influence of gravity. Calculated shaft deflection depends on rotor/shaft weight to be applied to the shaft, and upon the method chosen to allow for the non-uniform diameter of most shafts.

In large machines, shaft stiffness is often enhanced by welding radial arms to the shaft, on which the rotor is mounted. The welding process must be carefully controlled, from preheat to stress relief, to minimize the fatigue cracking in service.

The any shaft-rotor assembly will assume a deflection curve shape by bending under its own weight. In some large drive trains, containing several coupled machines, the various support bearings may have to be located and shaped so that the full length of shafting is free to follow its natural curvature. Otherwise, depending upon the couplings used some shaft areas may be subjected to unacceptably high bending stress.

When an existing motor is redesigned for higher horsepower, or is applied to a different type of load (particularly with a belt drive), a design analysis is necessary to ensure that increased stress does not risk shaft failure.

4.2. Force due to air gap asymmetries and unbalanced Magnetic pull

The discussion of air gap asymmetries and unbalanced magnetic pull presented in this section is a synopsis of the works of Rai [3] and Timar [4].

The main task in creating electromechanical energy conversion by an asynchronous or induction machine is to produce a rotating sinusoidal magnetic field with a pole number 2p in the air gap. A magnetic flux is induced in the air gap due to current flowing through the conductors in the slots.

These slots break up the uniformity of the air gap. Thus, the gap will change periodically as the relative position of the stator and rotor changes. The total magnetizing current of an induction machine is supplied from the stator (primary winding), and therefore the air gap is designed to be as small as possible. However, even the smallest deviation of the air gap from the shape of a cylindrical sleeve of uniform thickness will result in considerable relative distortion of air gap presence.

Deviation from a uniform air gap is also highly possible due to tolerances required in the manufacturing process. This non-uniform air gap can result from one or more of the following manufacturing flaws:

1) Imperfect centering of the rotor in the stator bore,

2) Non-circular stator bore,

3) Non-circular rotor and

4) Bent shaft.

The first two flaws produce a stationary or static eccentricity in the air gap of an electric machine, while the last two flaws produce a rotating or dynamic eccentricity.

Regardless of whether the air gap eccentricity is static, dynamic or both, the resulting gap will be at a maximum at one point thus producing a minimum flux and a minimum at one point thus producing a maximum flux. A non-uniform flux can result in the following adverse effects:

Flux harmonics

Unbalanced magnetic pull

Noise

Harmonics in the rotor and stator currents variation in losses and irregular distribution of heat around the stator bore as a result of 1) and 4)

Variation in torque as a result of 1) and 4), both during starting and steady state conditions.

The effect considered for this study is unbalanced magnetic pull.

As stated, the presence of a magnetic flux in the air gap of an electric machine produces an attractive force between the stator and rotor in the direction of the flux. In an induction machine the sinusoidal flux distribution revolves in the air gap producing a wave of magnetic pull rotating at twice the frequency of the fundamental field. In a system with a uniform air gap no resultant force occurs due to symmetry. When eccentricity is present, however, upon energizing of the stator equal but opposite unilateral magnetic forces acting on the stator and rotor are produced that always act in such a manner as to increase eccentricity. This phenomenon is known as unbalanced magnetic pull (u.m.p.). A graphical representation of the peripheral distribution of unbalanced magnetic pull on the rotor of a rotating electrical machine from Timar [4] is shown in Figure 2. This figure clearly shows the minimum and maximum magnetic field strengths (g) and the resulting radial forces. It is obvious that all force terms perpendicular to the eccentricity (e) cancel. This leaves the force terms parallel to the eccentricity (e), which sum to produce a resultant radial force (F) resultant. An equal but opposite force situation occurs with the stator.

Due to the sinusoidal distribution of flux around the air gap unbalanced magnetic pull due to static eccentricity is generally treated as a harmonic, vibratory, radial force between the rotor and stator for a given operating condition. The frequency of this harmonic force is twice voltage line supply frequency for the following reason. As the magnetic field distribution revolves in the air gap of an induction machine, it passes a given point twice during each cycle of voltage change, once as a flux entering the stator and once as a flux entering the rotor. Since a force of attraction between the stator and rotor occurs regardless of the direction of the flux, the harmonic input must occur at twice the frequency of the applied voltage.

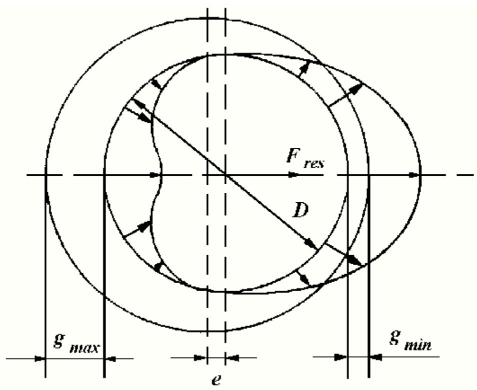


Figure. 3 Peripheral distribution of unbalanced magnetic pull

Unbalanced magnetic pull can also arise in electrical machines due to the presence of dynamic eccentricity. This form of unbalanced magnetic pull differs in that it rotates at the rotational speed of the rotor, and thus the effects are similar to those encountered with a mechanical unbalance. This situation can be referred to as magnetic unbalance.

The magnitude and distribution of the forces produced due to unbalanced magnetic pull are dependent upon the distribution of magnetic flux in the air gap. These magnitudes are typically determined by using the Maxwell stress tensor method in conjunction with the finite element method to extract the magnetic flux density in the air gap for a series of static rotor positions. The accurate determination of the flux density is a complicated and crucial step in a complete analysis of an electric motor. Since this phase of the analysis requires a much more extensive electrical engineering background than possessed by the author, it was decided to set the magnitude of the u.m.p. based forces to unity. This approach results in a set of displacements that may be scaled accordingly upon accurate solution of the flux distribution problem.

Ozturk [5] summarized that for analysis purposes the equal but opposite vibratory forces due to unbalanced magnetic pull act upon the stator and rotor of a rotating electric machine. These forces occur at twice the electrical supply frequency for static eccentricity and at the rotor rotational frequency for dynamic eccentricity. If the natural frequency of any part or combination of parts of the mechanical system lies near these forcing frequencies serious vibration problems can result.

Timar [4] defined the relationship between the amplitude of the tangential forces (torque producing forces) and radial forces (i.e., u.m.p. forces) in the air gap as

$$F_{tangential} = \frac{P \quad g}{R} F_{Radial}$$

Where

p = number of pole pairs

= order of the rotor space harmonics

dg = geometrical air gap

R = outer radius of the rotor.

The order of the rotor space harmonics for a single-phase AC machine is defined as

$= g'S2 \pm 2g+1$

Where

$g = 0, \pm 1, \pm 2\ldots$

g = negative g

S2 = number of rotor slots

Normally the ratio of tangential to radial force is considerably less than one. Therefore, it is obvious that the radial force magnitude should be much larger than the tangential.

In summary the presence of eccentricity in the air gap of a rotating electric machine causes equal but opposite harmonic forces of large magnitude at two different forcing frequencies to act upon the stator and rotor at the point of minimum flux density.

4.3 Loads:

Static Loading

Radial

Tangential

Axial (Thrust)

Dynamic (Cyclic) Loads

Fully reversed

Repeated

Fluctuating

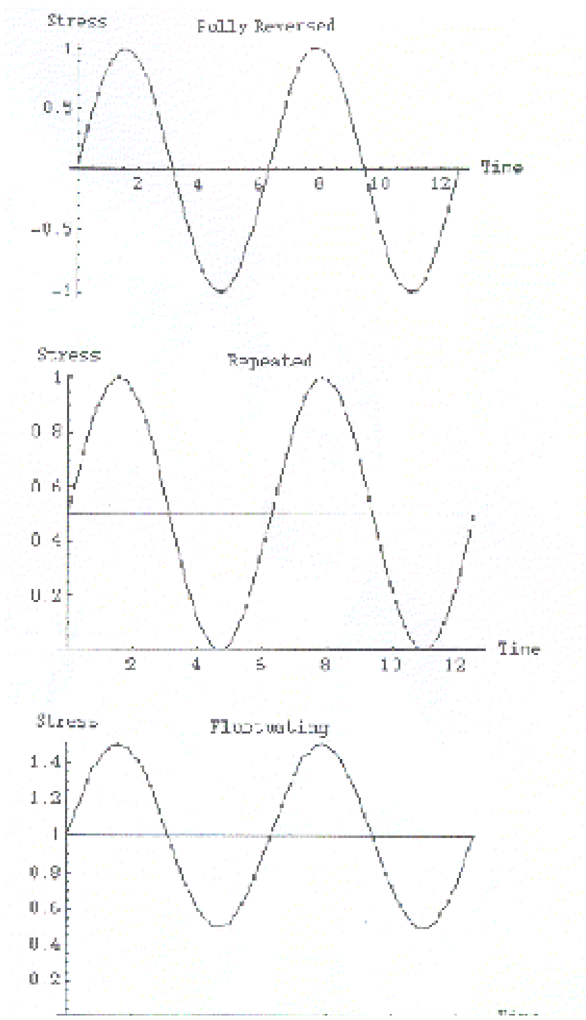


Figure. 4 Stresses

4.4 Stresses:

- Stress due to Axial Loading

$$\sigma_x = \frac{4F}{\pi d^2}$$

- Stress due to Bending

$$\sigma_x = \frac{My}{I}$$

$$\sigma_{max} = \frac{32M}{\pi d^3}$$

- Stress due to Torsion

$$\tau_{xy} = \frac{16T}{\pi d^3}$$

4.5 Stress Concentrations:

Stress caused by a sudden change in form

Fillets (on shoulders)

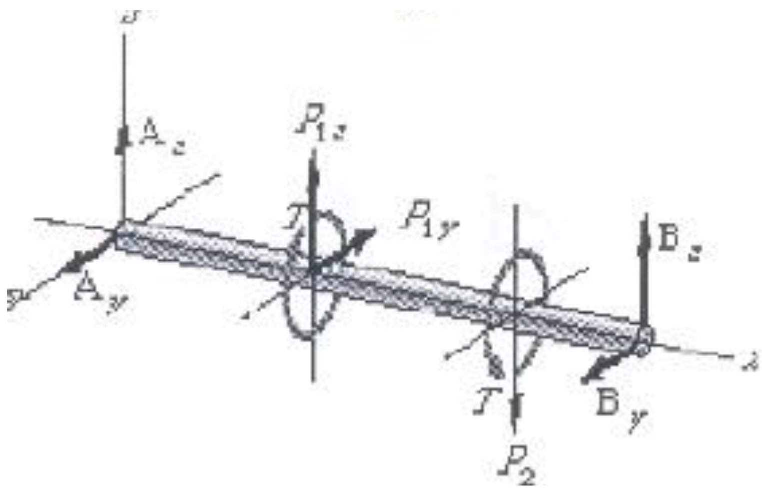
Holes (for pins)

Grooves (for snap rings)

Stress concentrations in torsion arise when the geometry is interrupted. For example, Hibbeler (1997) provides stress concentration factors for shoulder fillets in shafts. Other concentrators are caused by keyways (Boresi and Sidebottom, 1985; Machinery's Handbook, 1996), grooves and drill-through holes (Roark and Young, 1975; Machinery's Handbook, 1996). When applying formulas from these sources, it is important to note the definition of the nominal or reference stress

And the stress concentration factor itself.

$$K = \frac{\text{highest value of stress on "feature"}}{\text{Nominal stress on minimum cross section}}$$



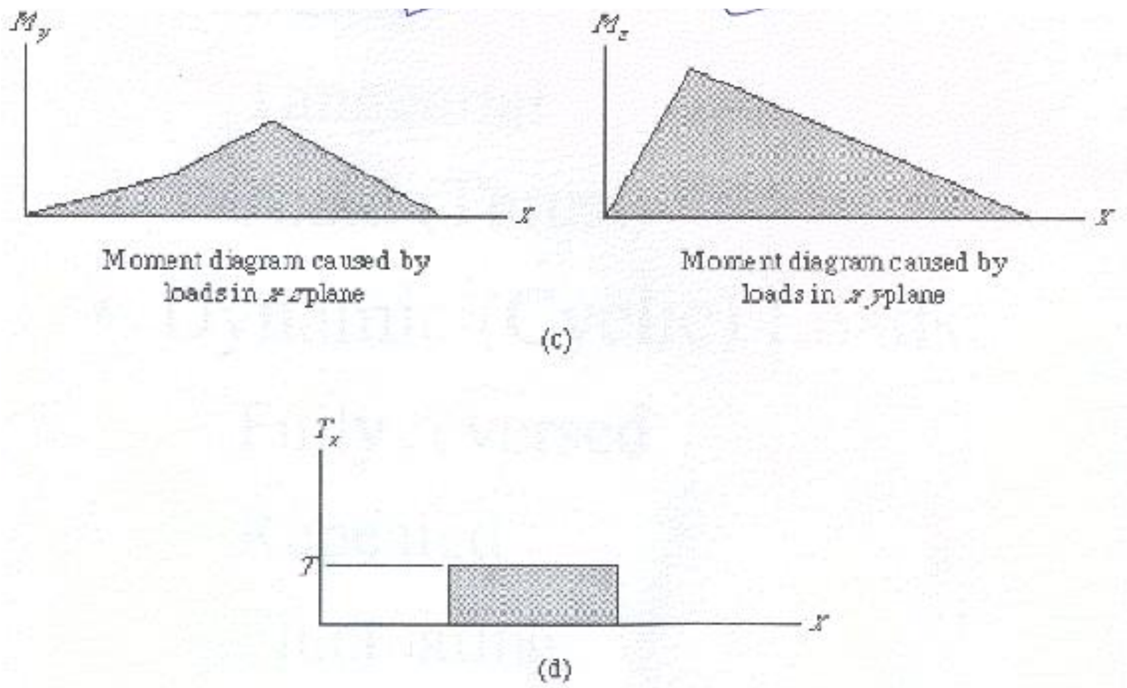


Figure. 5 Stress concentrations

Chapter 5

Finite Element Analysis

History: -

The finite element analysis was first developed in 1943 by Richard Courant, who used the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 established a broader definition of numerical analysis. Development of the finite element method in structural mechanics is usually based on an energy principle such as the virtual work principle or the minimum total potential energy principle.

Application:-

In its applications, the object or system is represented by a geometrically similar model consisting of multiple, linked, simplified representations of discrete regions i.e., finite elements on an unstructured grid. Equations of equilibrium, in conjunction with applicable physical considerations such as compatibility and constitutive relations, are applied to each element, and a system of simultaneous equations is constructed. The system of equations is solved for unknown values using the techniques of linear algebra or non-linear numerical schemes, as appropriate. While being an approximate method, the accuracy of the FEA method can be improved by refining the mesh in the model using more elements and nodes. A common use of FEA is for the determination of stresses and displacements in mechanical objects and systems. However, it is also routinely used in the analysis of many other types of problems, including those in heat transfer, solid state diffusion and reactions with moving boundaries, fluid dynamics, and electromagnetism. FEA is able to handle complex systems that defy closed-form analytical solutions.

Finite element analysis:-

Computer-aided engineering (CAE) is the application of computer software in engineering to evaluate components and assemblies. It encompasses simulation, validation, and optimization of products and manufacturing tools. The primary application of CAE, used in civil, mechanical, aerospace, and electronic engineering, takes the form of FEA alongside computer-aided design (CAD).

In general, there are three phases in any computer-aided engineering task:

Pre-processing – defining the finite element model and environmental factors to be applied to it

Analysis solver – solution of finite element model

Post-processing of results using visualization tools

Pre-processing:-

The first step in using FEA, pre-processing, is constructing a finite element model of the structure to be analyzed. The input of a topological description of the structure's geometric features is required in most FEA packages. This can be in either 1D, 2D, or 3D form, modeled by line, shape, or surface representation, respectively, although nowadays 3D models are predominantly used. The primary objective of the model is to realistically replicate the important parameters and features of the real model. The simplest mechanism to achieve modeling similarity in structural analysis is to utilize pre-existing digital blueprints, design files, CAD models, and/or data by importing that into an FEA environment. Once the finite element geometric model has been created, a meshing procedure is used to define and break up the model into small elements. In general, a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. FEA packages use node numbers to serve as an identification tool in viewing solutions in structures such as deflections. Elements are bounded by sets of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by mesh numbers, which allow references to be made to corresponding deflections or stresses at specific model locations.

Analysis (computation of solution):-

The next stage of the FEA process is analysis. The FEM conducts a series of computational procedures involving applied forces, and the properties of the elements which produce a model solution. Such a structural analysis allows the determination of effects such as deformations, strains, and stresses which are caused by applied structural loads such as force, pressure and gravity.

Post processing (Visualization):-

These results can then be studied using visualization tools within FEA environment to view and to fully identify the implications of the analysis. Numerical and graphical tools allow the precise location of data such as stresses and deflections to be identified.

Applications of FEA to the mechanical engineering industry:

A variety of specializations under the umbrella of the mechanical engineering discipline such as aeronautical, biomechanical, and automotive industries all commonly utilize the benefits of integrated FEA in the design and development of their products. Several modern FEA packages include specific components such as thermal, electromagnetic, fluid, and structural working environments. In a structural simulation FEA helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs. FEA allows detailed visualization of where structures bend or twist, and indicate the distribution of stresses and displacements. FEA software provides a wide range of simulation options with regards to controlling the complexity of both the modeling and the analysis of a system. Similarly, the desired level of accuracy required and the associated computational time requirements can be managed simultaneously to cater for most engineering applications. FEA allows entire designs to be constructed, refined, and optimized before the design is actually manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The introduction of FEA has substantially decreased the time taken to get products from concept to the production line. It is primarily through improved initial prototype designs using FEA that the testing and development stages have been accelerated. In summary, the benefits of FEA include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue.

Computer-aided design and finite element analysis in industry:

The ability to model a structural system in 3D can provide a powerful and accurate analysis of almost any structure. 3D models, in general, can be produced using a range of common computer-aided design packages. Models have the tendency to range largely in both complexity and in file format, depending on 3D model creation software and the complexity of the model's geometry. FEA is a growing industry in product design, analysis, and development in engineering. The trend of utilizing FEA as an engineering tool is growing rapidly. The advancement in computer processing power, FEA, and modeling software has allowed the continued integration of FEA in the engineering fields of product design and development. In the past, there have been many issues restricting the performance and ultimately the acceptance and utilization of FEA in conjunction with CAD in the product design and development stages. The gaps in compatibility between CAD file formats and FEA software limited the extent to which companies could easily design and test their products using the CAD and FEA combination, respectively. Typically, engineers would use specialist CAD and modeling software in the design of the product and then wish to export that design into a FEA package to test. However, those engineers who depended on data exchange through custom translators or exchange standards such as IGES or STEP cite occasional reliability problems causing unsuccessful exchange of geometry. Thus, the creation of many models external to FEA environments was considered to be problematic in the success of FEA.The current trend in FEA software & industry in engineering has been the increasing demand for integration between solid modeling and FEA analysis. During product design and development engineers require automatic updating of their latest models between CAD and FEA environments. There is still a need to improve the link between CAD and FEA, making them technically closer together. However, the demand for unitary CAD-FEA integration coupled with the improved computer and software developments has introduced a more robust and collaborative trend where compatibility problems are beginning to be eliminated. Designers are now beginning to introduce computer simulations capable of using pre-existing CAD files, without the need to modify and re-create models to suit FEA environments.

Current FEA trends in industry:

Dynamic modeling:-

There is increasing demand for dynamic FEA modeling in the heavy vehicle industry. Many heavy vehicle companies are moving away from traditional static analysis and are employing dynamic simulation software. Dynamic simulation involves applying FEA in a more realistic sense to take into account the complicated effects of analyzing multiple components and assemblies with real properties.

Modeling assemblies:-

Dynamic simulation, used in conjunction with assembly modeling, introduces the need to fasten together components of different materials and geometries. Therefore, CAE tools should have comprehensive capabilities to easily and reliably model connectors, including joints that allow relative motion between components, rivets, and welds. Typical MSS models are composed of rigid bodies (wheels, axles, frame, engine, cab, and trailer) connected by idealized joints and force elements. Joints and links may be modeled as rigid links, springs, or dampers in order to simulate the dynamic characteristics of real truck components. Force transfer across assembly components through connectors makes them susceptible to high stresses. It is simpler and easier to idealize connectors as rigid links in these systems. This idealization provides a basic study of assembly behavior in terms of understanding system characteristics; engineers must model joining parameters like fasteners accurately when performing stress analysis to determine how failures might take place. "Representing connectors as rigid links assumes that connectors transfer loads across components without deforming and undergoing stress themselves. This unrealistic idealization yields incorrect predicted stresses in the regions local to the connectors, the exact locations where part failures will most likely initiate. Understandably, the detailed inclusion of every connection point and/or mechanism in an assembly is impractical to model. Therefore, improved representations of fasteners that are simple to use yet reliable should be investigated for use on a case-by-case basis.

Current modeling techniques in industry:-

Engineers at Leyland Trucks currently model their trucks using specialist dynamic FEA software. Each model contains a flexible body and chassis, springs, roll bars, axles, cab and engine suspension, the steering mechanism, and any frequency-dependent components such as rubber mounts. Extra details such as brakes and out-of-balance engine forces can be included on an "as-needed" basis.

Dynamic FEA simulation enables a variety of maneuvers to be accurately tested. Tests such as steady-state cornering, roll-over testing, lane changing, J-turns, vibration analysis, collisions, and straight-line braking can all be conducted accurately using dynamic FEA. Non-linear and time-varying loads allow engineers to perform advanced realistic FEA, enabling them to locate critical operating conditions and determine performance characteristics.

As a result of the improved dynamic testing capabilities, engineers are able to determine the ultimate performance characteristics of the vehicle's design without having to take physical risks. As a result of dynamic FEA, the need for expensive destructive testing has been lessened substantially.

FEA and the truck industry:

The truck industry is becoming more like other industries such as the automotive industry with regard to FEA's involvement in the design process. However, due to the unique need by truck manufacturers to provide a variety of different body configurations, it is unlikely that trucks will ever move to the unitary streamlined FEA integration process, as seen in the automotive industry. The design process hasn't reached the required level of maturity where it can be simulation-driven. Furthermore, the traditional design philosophy of the tried-and-tested truck designing techniques taking precedence is still held strong across the truck industry. Although the industry remains far from adopting a complete bottom-up FEA integrated design process, FEA is fast increasing its role in product design and development in the truck industry.

5.1. Finite Element Model Verification

While finite element analysis is a powerful tool, its successful application is highly dependent upon the analyst's understanding of the finite element method and the associated pitfalls. Therefore, it is always necessary to validate a finite element model against some other standard. In many cases it is possible to use closed form solutions on simplified geometry for validation. In other cases no form of closed solution may be similar to the actual system. In these instances the only analytical recourse is to perform FEA with different models of the same system to assure that convergence has been reached. The compressor assembly under analysis for this project required both such approaches. After examining all of the components involved it was felt that validation of the rotor-shaft assembly and the stator assembly would be sufficient.

5.2. Rotor Shaft Assembly Model Verification

When viewing the cantilever rotor shaft assembly it can be hypothesized that the motor body is analogous to the fixed base of a cantilever beam. Virtually all displacements will occur in the rotor shaft, the upper support collar and the rotor shaft section above the motor body.

The combined length of the upper support collar and exposed rotor shaft is greater than the outside diameters of both the upper support collar and the rotor shaft. Thus, the assembly may be approximated as a cantilever beam with an additional end mass as shown.

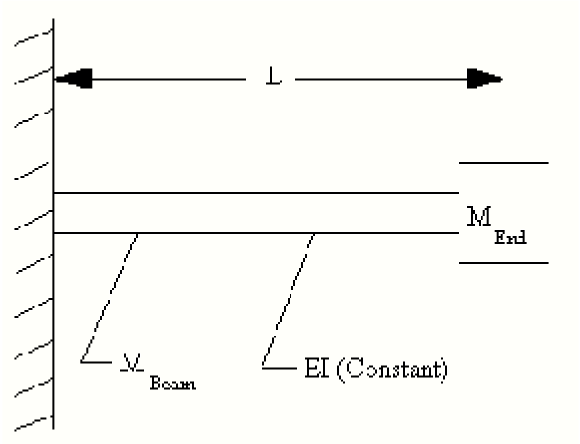


Figure. 6 Cantilever beam

From Rao [8] it is known that this is easily treated as the following single degree of freedom spring-mass system with the following equivalent stiffness K and mass M values.

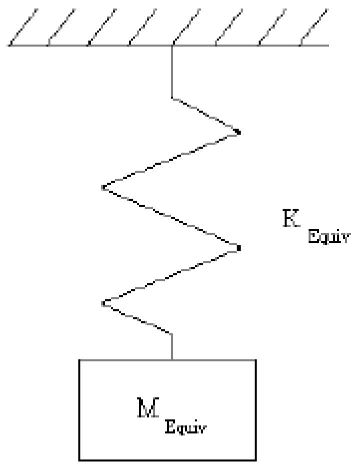


Figure. 7 Spring-mass system

$$K_{equiv} = \frac{3EI}{L^3}$$

$$K_{equiv} = 0.23M_{BEAM} + M_{END}$$

E = Modulus of Elasticity

I = Area Moment of Inertia

L = length of beam

The problem with this approach is that the section modulus EI for the beam must remain constant.

The upper support collar introduces a change in I for the system, thus violating this requirement.

The decision was made to analyze the beam using I values for the upper support collar I1, the rotor shaft I2 and an equivalent I value weighted for I1 and I2.

From Thomson [9] the first natural frequency of a cantilever beam may be found using the following relationship.

$$_1 = 3.52\sqrt{\frac{EI}{M_{BEAM}L^3}}$$

From Cheng [10] it is known that the static deflection of a cantilever beam due to an in-plane force perpendicular to the axial direction as shown may be found through the following relationships.

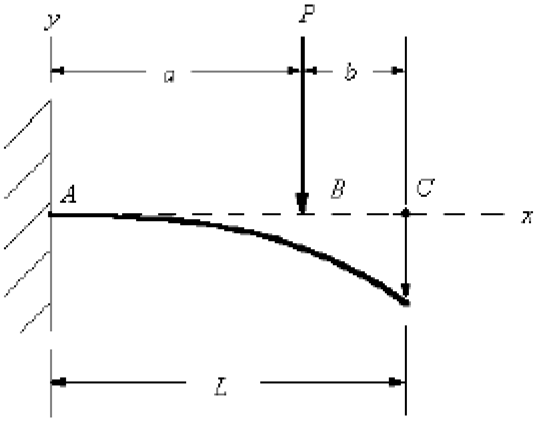


Figure. 8 Static deflection of a cantilever beam

$$y_{AB} = \frac{Px^2}{6EI}(3a-x)$$

$$y_{BC} = \frac{Pa^2}{6EI}(3x-a)$$

Where

y = displacement

P = applied load

Chapter 6

Motor Rotor Shaft Design: Case Study

6.1. Background

Electric motors are well known as 'Basic Prime movers' for Engineering Applications. Electric motors are available at different speed, sizes & output power. Due to its clean & noiseless operation, the electric motors are very popular.

Stresses in steel shafts are complex, normally involving combinations of bending, torsion, and shear. Some stresses alternate or reverse in direction with each shaft revolution; others may be cyclically varying. Good design dictates holding maximum stress below the endurance or fatigue limit - the fairly low stress which can be sustained indefinitely through cycles of application. Shock loading is encountered in motors for crushers and similar machines. That greatly increases shaft stress. It may be sudden load (in which two drive components are in contact before force is applied between them) or impact (in which one component moves through a distance before striking the other.)

Unlike strength, shaft stiffness (resistance to bending) is independent of steel composition. Shaft critical speed or resonance is a function of shaft deflection under the influence of gravity. Calculated shaft deflection, depends on rotor weight to be carried by the shaft, and upon the method chosen to allow for the non-uniform diameter of most shafts. In large machines, shaft stiffness is often enhanced by welding radial arms to the shaft, on which the rotor is mounted.

The shaft-rotor assembly will assume a deflection curve shape by bending under its own weight. Also depending upon the couplings used, the shaft may be subjected to unacceptably high bending stress.

When an existing motor is redesigned for higher horsepower, or is applied to a different type of load (particularly with a belt drive), a design analysis is necessary to ensure that increased stress does not risk shaft failure.

6.2. Company profile

Laxmi Hydraulics Pvt. Ltd. is established in year 1981. The company is engaged in manufacturing of Induction Motors, Monoblock Pumps, Jet Centrifugal Combination Pumps and Submersible Monoblock Pumps. Its products are suitable for various applications in Machine Tools, Textiles, Air conditioning, Dairy equipments, Crane duty, Pollution control equipment, Adequate, Agriculture, Coal mines, Refineries, Pharmaceuticals, Chemicals, Fertilizer, Petroleum, Industries etc.

At present LHP is one of the widely accepted brands in India in the manufacturing of motors & pumps. LHP is an ISO 9001 – 2002 company. It is one of the most disciplined organizations. The company follows EXP & other management techniques such as 6 sigma, 5S, TPM & KANBAN. Adopting modern machines, manufacturing processes & capable manpower, LHP has maintained that in process rejection below 0.2% & has achieved 95% customer satisfaction index [11]. At present LHP has strength of more than 220 employees. It has it's own factory premises of more than 3350 sq. meters at Hotgi road, MIDC, Solapur. LHP's turnover is more than Rs. 400 million and an annual growth rate of more than 51%. Numbers of customers are increasing at the rate of 30% per annum. In process rejection is below 0.5%. Customer complaints below 0.4%. Company has manufactured & supplied more than 1,000,000 motors [12].

Quality of product is achieved by a team of highly dedicated experienced and trained persons and well defined and documented procedures. The secret of LHP's success lies in fulfilling customers changing expectations of accepting challenges for continuous improvement in product and services. LHP believes in adopting latest technology and trends to meet individual customer requirements.

There are three shifts in LHP

The objectives & mission of company are –

Driven by commitment.

To provide quality motors of the highest reliability, at the most competitive prices deliver them in time.

Never say 'No' to the customers.

To deliver what we commit.

Present standards for fan & electric motor manufacture:

For electric motor:

The induction motors are manufactured as per Indian standard IS:325 and equivalent UK standard BS 5000. [11]

As per the standard the temperature rise for different class of insulation is given below.

Permissible Temperature rise[12].

Class of insulation	Maximum permissible Temperature limit c°	Maximum permissible temperature rise for windings at ambient Temperature in c°			
		40	45	50	60
B	130	80	75	70	60
F	155	105	100	95	85
H	180	125	120	115	105

Following figure shows the standard set up for motor manufacturers. The various dimensions are shown over the figure.

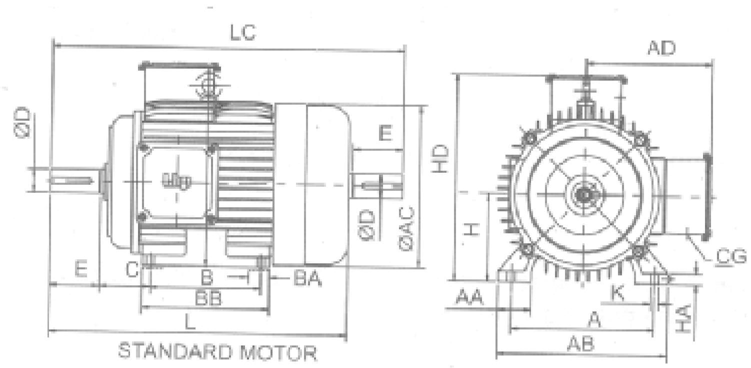


Figure. 9 Standard setup for motor manufacturer

H = 132 mm

A = 216 mm

B = 178 mm

C = 89 mm

AA = 50 mm

AB = 256 mm

AC = 258 mm

AD = 195 mm

BB = 205 mm

ϕD = 38 k6 mm

E = 80 mm

HA = 16 mm

HD = 310 mm

L = 492 mm

6.3. Motor rotor shaft design methodology

This overview of the theory for design of power transmission shafts is meant to supplement that given in your textbook. It covers power transmission, stress concentrations and design of round shafts. We can model shafts rotating at constant angular velocity using static's because we know from dynamics that when angular acceleration equals zero, ΣT = 0. Knowing the speed of rotation, we can convert between torque T and power P using the equations given in the text, namely

$$T = \frac{P}{\omega}, \quad \omega = 2\pi f, \quad \therefore T = \frac{P}{2\pi f}$$

Where power is measured as torque per unit time and w and f are the angular velocities usually given in radians per second and revolutions per second, respectively. We deliberately cast these equations to solve for torque because this is useful in stress analysis. Furthermore, f is the cyclic frequency which given per second is known as Hertz (Hz) or cycles per second and given per minute is known as RPM or revolutions per minute. The former is commonly associated with Watts or N×m/s and the latter with horsepower (hp), a U.S. Customary Unit, in which case,

$$T(\text{ft.lb}) = \left(\frac{550\,\text{ft . lb}}{\text{hp.s}}\right) X \left(\frac{60\,\text{s}}{\text{min}}\right) X \left(\frac{HP}{2\pi\, RPM}\right) \approx 5250 \left(\frac{HP}{RPM}\right)$$

Here torque is specifically calculated in ft×lb units.

(Note: To distinguish between like variables and units, we employ upper case italic for variables, lower case standard for units.)

In design applications we typically need to specify shaft dimensions such as a diameter and fillets. The design is determined with respect to allowable stress (strength) and sometimes twists. For pure torque loading, it is convenient to rewrite the stress-from-torque expression as

$$\iota = \frac{T\,r}{J} = \frac{T\,r}{\pi r^4/2} = \frac{2T}{\pi r^3} = \frac{16T}{\pi d^3} \Rightarrow d = \left(\frac{16T}{\pi\,\iota}\right)^{1/3}$$

Where r and d denote the radius and diameter of the shaft, respectively. Moreover in the absence of other factors, we would replace the shear stress by an allowable value found using a factor of safety together with material test data. (See the problems below.)

If a complication such as a stress concentrator exists, then a design problem may become iterative if initially we do not have the shaft geometry from which to determine the stress concentration factor. Then the design algorithm is: 1) solve the problem for dimensions, 2) use the stress concentration factor to check if allowables are violated, and 3) repeat until optimal dimensions which satisfy all requirements are found. In the case of power take-off applications which typically involve belts or chains wrapped around sheaves or gears, bending as well as torsion occurs. This is treated by Hibbeler (1997) in asection devoted to shaft design. Interestingly, Machinery's Handbook (1996, p. 280) recommends for allowable stresses 60% of σ_Y for axial and 30% of σ_Y for shear where σ_Y is the yield stress in tension. This amounts to factors of safety FS (assume that yield in shear is 60% tensile yield) of

$$FS_{axial}\ \sigma_Y / 0.6\sigma_Y = 1.7 \quad FS_{shear} = 0.6\ \sigma_Y / 0.3\sigma_Y = 2.0$$

Which are rather high values indicating shaft design is viewed very conservatively. An official standard is not cited.

6.4. Force Analysis of Motor Rotor Shaft

Design of shaft:

Static case: Analytical solution:

Given:

Motor Capacity = 200 Hp

Motor Speed (N) = 1500 r.p.m.

$\omega = 2\pi N / 60 = 157$ rad/sec.

$$\text{Torque (T)} = \frac{\text{Power}}{\text{Speed}} = \frac{200 \ \text{x}\ 6600}{157}$$

T = 8407.6 in lb.

σ_{all} = 58 to 68 kgf/mm^2

ρ = 7500kgf/m^3

Self weight = 95 kg

Rotor weight = 216 kg

Fan weight : 5 kg

Material : Carbon steel 40C8

Min diameter of diagram : d min = 80

Solution:

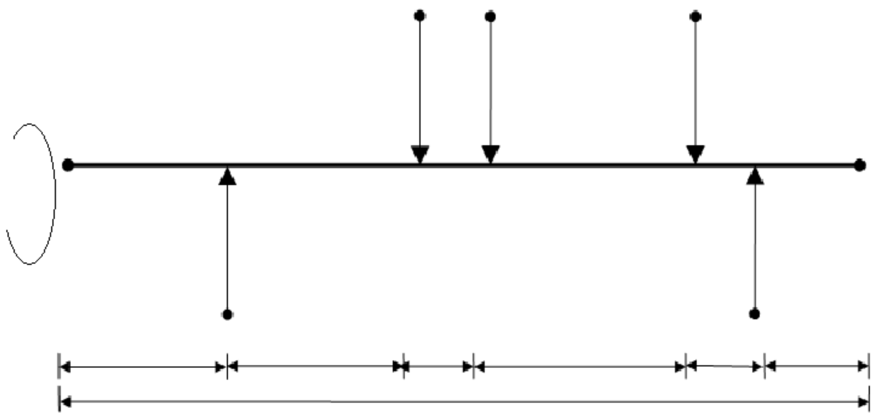


Figure. 10 Force diagram of shaft

FBD of shaft:

T = 8407.6 x 25.4 x 0.4536 x 9.81

= 950271.7342 N-mm

Rotor weight = 216 x 9.81 = 2118.96 N

Fan weight = 5 x 9.81 = 49.05 N

Self weight = 95 x 9.81 = 931.95 N

To calculate reaction forces:

931.95 + 2118.96 + 49.05 = $R_B + R_F$

$R_B + R_F = 3099.96$

Taking moment @ R_B

931.95 x (402-42) + 2118.96 x (402) + 49.05 x (402 + 435)

R_F x (402 + 435 + 76)

$\Rightarrow R_F = 1345.440$ N

$\Rightarrow R_B = 1754.52$ N

To plot BM:

$BM_A = 0$

$BM_B = 0$

BM_C = (402 –42) x 1754.52 = 631627.2 N-mm

BM_D = 1754.52 x (402) – 931.95 x 42 = 666175.14 N-mm

BM_E = 1754.52 (402 + 435) – 931.95 x (435 +42) – 2118.96 x 435

= 102245.49

$BM_F = 0$

$BM_G = 0$.

FBD in SI units:

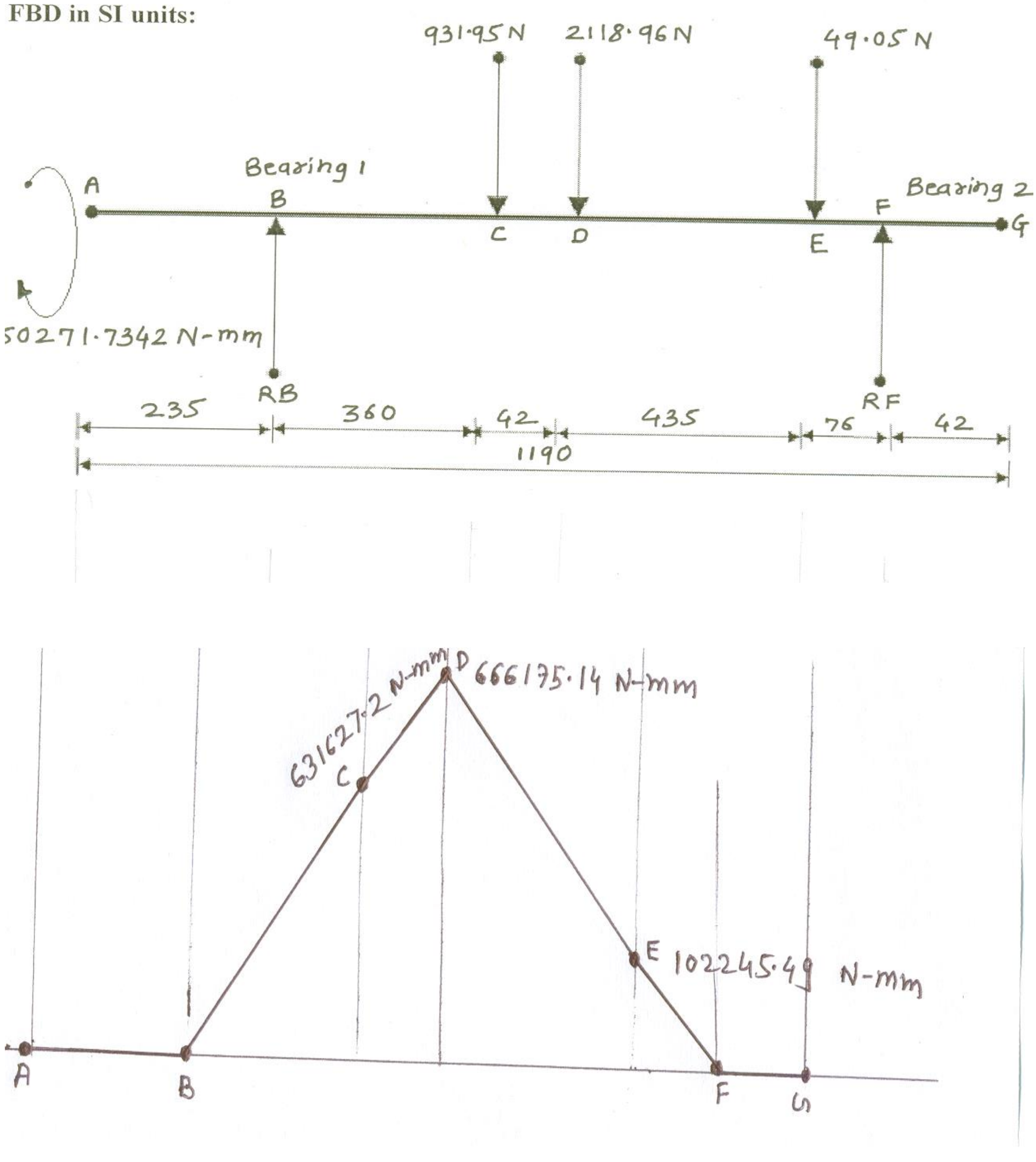


Figure. 11 Force bending diagram of shaft

From BMD: BM_{max} = 666175.14 N-mm

To calculate equivalent torque: (T_e):

$$T_e = \sqrt{(kb.mb)^2 + (kt + mt)^2}$$

M_t = 950271.7342 N.mm

M_b = 666175.14 N-mm

For given application and loading:

K_t = 1.0 k_b = 1.5

$$\Rightarrow T_e = \sqrt{(1.0 \text{ x } 950271.7342)^2 + (1.5 \text{ x } 666175.14)^2}$$

= 1378964.224 N-mm

Using torsion theory:

$$\tau_{max} = \frac{16\ T_e}{\pi d^3}$$

There are key way on the shaft.

$$\tau_{max} = 0.75 \quad \frac{16 \text{ x } 1378964.224}{\pi 80^3}$$

τ_{max} = 10.28 N/mm^2

Induced

For given material : 40C8

But

S_{ut} = 580 N/mm^2 S_{yt} = 380 N/mm^2

Assuming FOS = 2

$\tau_{allowable}$ = 380/2 = 190 N/mm^2

$\tau_{allowable}$ = 190/2 = 95 N/mm^2

$\tau_{maximum}$ induced < < $\tau_{allowable}$

Hence shaft is safe for given loading condition.

Deflection of Shaft :

Using Macauly's method –

The bending moment at any section distance X from the B is given by –

$$EI \frac{d^2y}{dx^2} = 931.95\ X \mid -931.95\ (X-360) \mid -2118.96\ (X-402) \mid -49.05\ (X-837)$$

Integrating, we get

$$EI\frac{dy}{dx} = 465.97\,X^2 + C_1 \mid -465.97\,(X-360)^2 \mid -1059.48\,(X-402)^2 \mid -24.52(X-837)^2$$

Integrating again, we get

$EIY = 155.32\ X^3 + C_1X + C_2 \mid -155.32\ (X-360)^3 \mid -353.16\ (X-402)^3 \mid -8.17\ (X-837)^3$

At

$X = 0,$

$Y = 0\ \therefore C_2 = 0$

At

$X = 913$

$Y = 0$

$$\therefore C_1 = \frac{155.32 \times (553)^3 - 155.32 \times (913)^3 + 353.16 \times (511)^3 + 8.17 \times (76)^3}{913}$$

$\therefore C_1 = 49066344.09$

Hence the deflection at any section is given by

$EIY = 155.32X^3 - 49066344.09X \mid -155.32(X-360)^3 \mid -353.16(X-402)^3 \mid -8.17\ (X-837)^3$

The maximum deflection is given by a point D (rotor).

$Y_D = Y_{maximum}$

To find the maximum deflection at D. Put X = 402mm in the deflection equation.

By analyticaly deflection is-

$Y_{maximum} = 0.212$ (In rotor Region)

6.5 Finite Element Approach.

The FEA is a numerical procedure for analyzing structures of complicated shapes, which otherwise would be difficult by classical analytical methods. Analytical solution is a mathematical expression that gives values of desired unknown quantity at any location in a body or a structure and it is valid for an infinite number of locations in body or structure. But analytical solutions can be obtained only for simple engineering problems. It is extremely difficult and many a times impossible to obtain exact analytical mathematical solutions for complex engg. problems. In such cases FEM is used which gives approximate solution.

Description of Method:

FEA works on the principle of divide and rule, that is, it transforms a physical system having infinite unknowns into small finite elements having finite number of unknowns. The unknowns are called degrees of freedom. Degree of freedom is the number of independent coordinates which must be specified to define all displacements. Instead of solving the problem for the entire body in one operation, the solutions are formulated for each constituent unit and combined to obtain the solution for the entire body i.e. going from part to whole Thus in FEM, body or structure is divided into finite number of smaller units known as elements. This process of dividing the body into finite number of elements is known as discretization. The assemblage of elements then represents original body or structure. Physical problem involves an actual structure subjected to certain loads and constraints. The material properties and the governing relationships are considered over the elements and expressed in terms of unknown values at element corners. An assembly process duly considering the loading and constraints, results in a set of equations. Solution of these equations did by FEM gives us the approximate behavior of the continuum.

The elements are considered to be interconnected at finite number of joints which are known as nodes or nodal points. The finite element method has become a powerful tool for the numerical solution of wide range of engineering problems. Application range from deformation and stress analysis of automotive, aircraft, building, and bridge.

Structures to field analysis of heat flux, fluid flow, magnetic flux, seepage and other flow problems. Thus we can define FEA precisely as- "Finite element analysis is a mathematical representation of a physical system comprising of a part / assembly (model), material properties, and applicable boundary conditions {collectively referred to as preprocessing}, the solution of that mathematical representation {solving}, and the study of the results of that solution {post processing}."

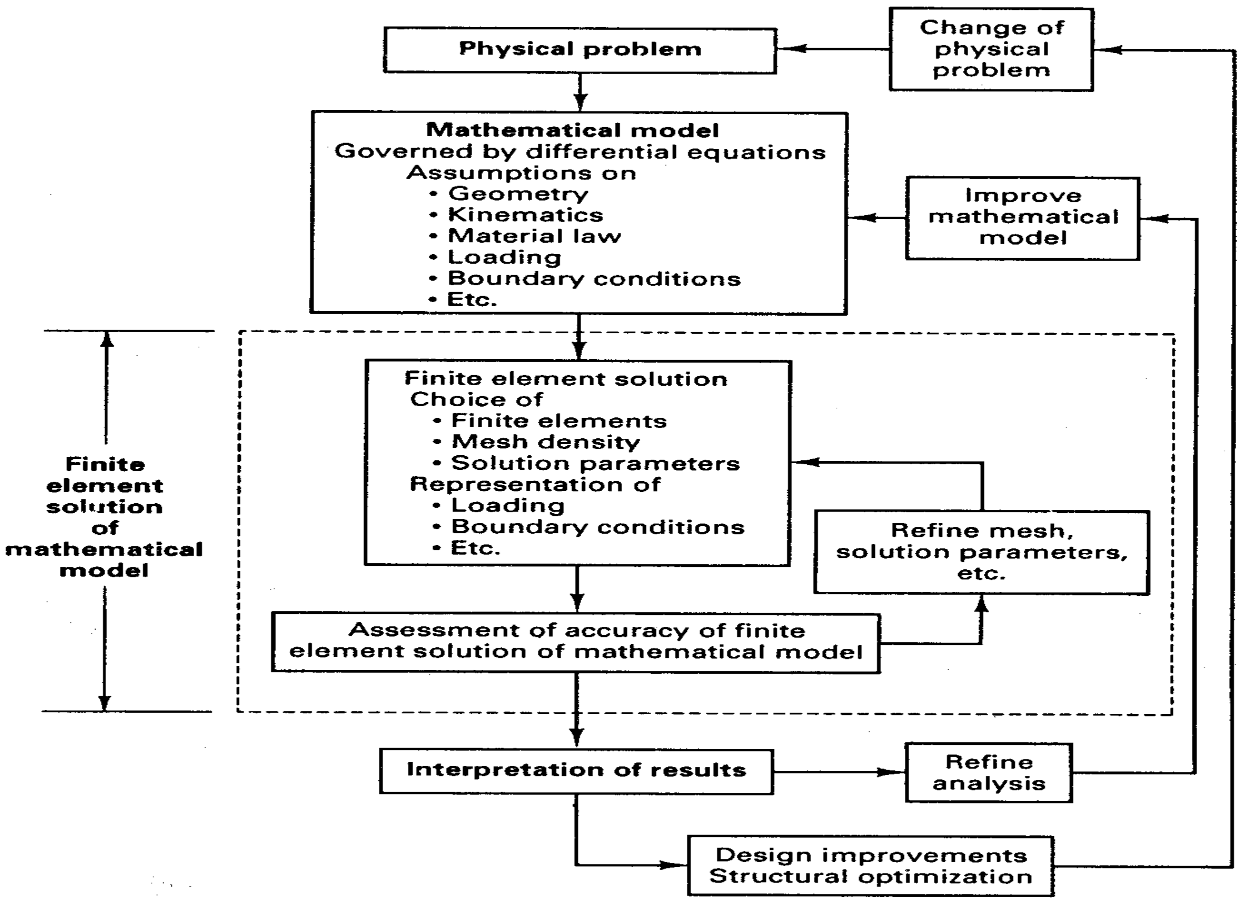


Figure. 12 Finite element process

Steps in finite element method and ansys package:

The basic steps for analyzing an engineering problem by finite element method are as follows-

Preprocessing phase:-

1. Create and discretize the solution domain into finite elements that is subdivide the problem into nodes and elements. Continuum is the body or structure or solid being analyzed. Discretization is the process of dividing the body into finite no. of elements. The elements may have different sizes. The choice of element type, its shape and refinement are required to be decided before discretization. The success of discretization which is also known as meshing lies in how closely the discredited continuum represents the actual continuum. Element can be a 1D element like a rod element, combination of 1D and 2D elements, 2D elements like 3node triangular element, 4node quadrilateral element,6 node triangular element,8 node quadrilateral element etc. and also 3D elements like tetrahedron, rectangular prism, prism etc..

2. Assume a shape function to represent the physical behavior of an element; that is an approximate continuous function is assumed to represent the solution of an element. Shape functions maps the values of degrees of freedom from nodes to points within the element. This is important as values of degrees of freedom are calculated only at the nodes. Displacement model or shape functions the gives values at any point within the element. From displacements, strains and stresses within element are to be determined. If shape functions are not truly representative values of these derived quantities would be compromised.
3. Develop equations for an element. Now the element stiffness matrices are formulated for all the elements which relate the applied actions to the degrees of freedom in the element. Element stiffness matrix, which depends upon the material and geometry of element can be formulated by -1.direct stiffness method or 2. Energy method. Element stiffness matrix is given by -

$$\{f\} = [k]\{u\}$$

Where f = nodal forces, k = stiffness of each element and u = displacement of each node. Here [k] is the element stiffness matrix.

4. Assemble the elements to present the entire problem. Construct the global stiffness matrix. Element stiffness matrices are assembled to form the global stiffness matrix for entire body. Relation between global stiffness matrix [K] global load vector {F} and global nodal displacement vector {U} is expressed as a set of simultaneous algebraic equations.

$$[K]\{U\} = \{F\}$$

. Apply boundary conditions, initial conditions, and loading.

Specified boundary conditions are incorporated in equilibrium equations using one of the following approaches namely the elimination approach or the penalty approach.

Processing or solution phase:-

Solve a set of linear or non linear algebraic equations simultaneously to obtain nodal results such as displacement temperature etc.

After including the specified boundary conditions the modified equations are solved for unknown values like nodal displacement by using various methods like gauss elimination, cholesky’s factorization, gauss sidle, Jacobins iterations frontal techniques etc. In solving equations, advantage is taken of banded property of global stiffness matrix.

Post processing phase:-

Results generated are presented in graphical form or in a tabular form as described by the user. It plots various stress, displacement, strains etc. Computation of the element strains and stresses from the nodal displacements.

Components of strains at any point within element are given by-

$$\varepsilon = [B]\{Un\}$$

Similarly component of stresses at point within element are given by

$$\sigma = [D]\{\varepsilon\}$$

Where [B] is element strain nodal displacement matrix

[D] Is element stress strain matrix.

Having understood the fundamental concepts of finite element modeling it is necessary to use general purpose finite element software such as ANSYS effectively. Use of computer and computer software is inevitable because in finite element methods the amount of data to be handled is dependent upon the number of elements into which the body is divided and also it is necessary that accuracy is maintained. ANSYS is a comprehensive general purpose finite element computer program that contains over 100,000 lines of codes. It is a powerful and impressive engineering tool that can be used to solve variety of problems including engineering problems.

Mesh Generation:

Introduction:

In order to carry out a finite element analysis, the model we are using must be divided into a number of small pieces known as finite elements. Since the model is divided into a number of discrete parts, FEA can be described as a discretization technique. In simple terms, a mathematical net or "mesh" is required to carry out a finite element analysis. If the system under investigation is 1D in nature, we may use line elements to represent our geometry and to carry out our analysis. If the problem can be described in two dimensions, then a 2D mesh is required. Correspondingly, if the problem is complex and a 3D representation of the continuum is required, then we use a 3D mesh.[13]

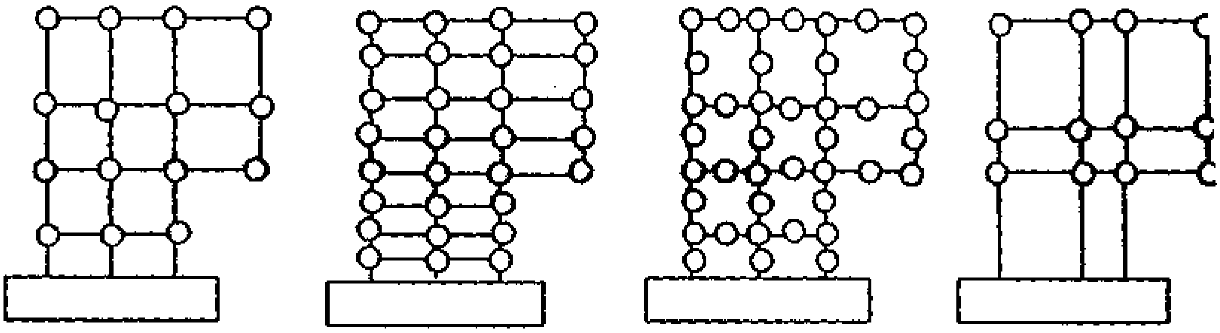
Mesh refinement:

After generation of coarse mesh, it is refined as per the geometry and critical sections of the model. It can be refined in three different ways as follows [13]

H-refinement: Here clement size is changed (decreased) without changing the element type.

P-refinement: Here element type is changed (to higher order) without changing element size.

R-refinement: Here existing nodes are moved without changing the element type and size.



Original mesh Uniform h-refinement Uniform p-refinement r-refinement

Figure.13 Different ways of mesh refinement

A p – refinement converges to the solution faster than h – refinement. Fig. 14 shows the above-discussed ways of mesh refinement; ‘h’ refinement is used near the fillet area and ‘r’ refinement used at other locations.

Mesh transition:

Mesh transition occurs when refined mesh interfaces with coarse mesh. It connects different types of elements. One common method of performing a transition is to use an intermediate belt of different elements as shown in the Fig. 15

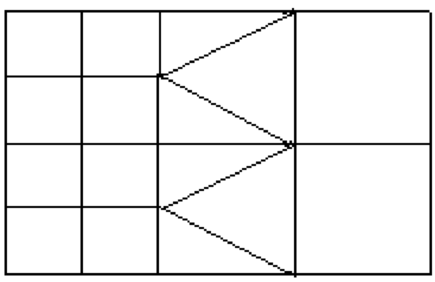


Figure. 14 Mesh Transition

Area Meshing:

Area elements can be triangular or quadrilateral in shape. The selection of the element shape and order is based on considerations relating to the complexity of the geometry and the nature of the problem being modeled. Membrane elements don't have any thickness. As a consequence they have no bending stiffness; loads can only be carried in the element plane. Plate & shell elements are used to model thin walled regions in 3D space. The plate element is formulated around plate theory, which assumes that the load is carried via bending. Shell elements are used to model shells, where there is combination of flexure & membrane action. Plate elements are considered applicable where the out of plane distortion is little more than the plate thickness. There are also special elements, which facilitate accurate modeling of thick plates. If the deflection is greater than the plate thickness, membrane action should be considered, and so shell elements should be used. Shell element nodes have five degrees of freedom; the missing is the in-plane rotational freedom (sometimes referred to as the drilling freedom). Solid elements come in different varieties. Axisymmetric elements are used to describe the cross-section of an axially symmetric part. Plane strain elements are used to describe section of long objects (such as a shaft or wall cross-section). The strain in the out-of-plane direction is taken to be zero, reflecting the assumption that the strain is in one Plane stress elements are used to describe sections of thin objects (such as a wrench). The stress in the out-of-plane direction is taken to be zero, reflecting the assumption that the stress is in one plane.

The two dimensional elements are shown in fig .16. below.

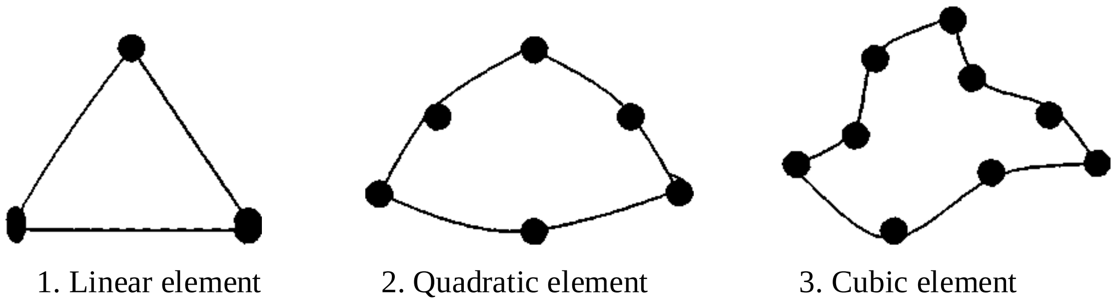
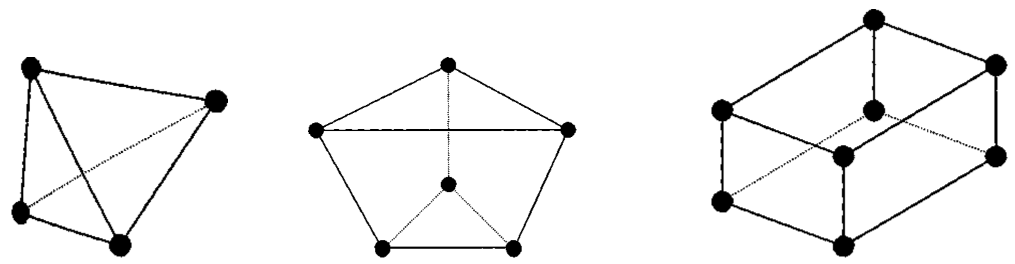


Figure. 15 Two Dimensional Elements

Volume meshing:

3D elements take the form of cubes called hexahedrons (hexes), 3D triangles called tetrahedrons and 3D wedges known as tetrahedrons. Decisions on element selection hinge on understanding the role of the element shape and order of interpolation.

Modeling with 3D-Elements is the most flexible approach. These types of elements are used for thick structures that have neither a constant cross section nor an axis of symmetry. Solid modeling will nearly always make analysis preparation easier. Meshing and solving can take a long time; particularly if the structure is thin-walled (large numbers of elements are required to produce a mesh). The three dimensional element are shown in fig. 18. below.



1. Tetrahedral 2. Prismatic 3. Hexahedral

Figure. 16 Three dimensional elements

Boundary conditions:

The finite element method is applicable for wide range of boundary value problems in engineering. Boundary conditions are nothing but applied load & constraints that component experiences under actual working condition. We know that solution sought in the region of the body while values of certain variables are prescribed on boundary of region. The values of variable prescribed on boundary of region are boundary condition.

There are two types of boundary conditions:

1. Geometric or essential boundary conditions:

These are also called as kinematics boundary conditions. Geometric & essential boundary conditions include displacements & slope.

2. Force or natural boundary conditions:

These are also called as static boundary condition. This includes prescribed forces and moments.

Let's have look on boundary conditions by example.

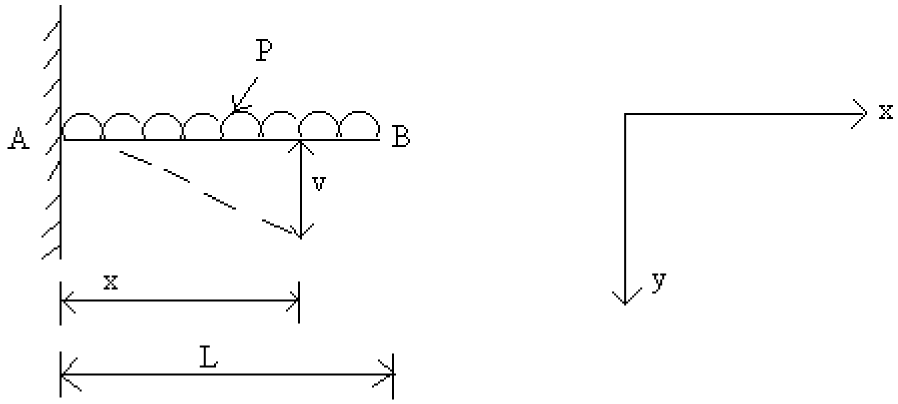


Figure. 17 Cantilever beam subjected to uniformly distributed load.

P = UDL acting on beam, N/m

v = Vertical deflection of beam @ distance x, m

L = Length of beam, m

E = Modulus of elasticity of cantilever beam N/m2

The differential equation governing vertical deflection 'V' of the beam is

$$P = E.I.\frac{d^4v}{dx^4}$$

The solution to above equation must satisfy boundary conditions at B & C.

i) Geometric boundary conditions at A

$$x = 0, \quad \therefore \quad v = 0, \quad \therefore \quad \frac{dv}{dx} = 0$$

ii) Force boundary conditions at B

x = L

$$\text{Shear force} = E.I.\frac{d^3v}{dx^3} = 0$$

$$\text{Bending moment} = E.I.\frac{d^2r}{dx^2} = 0$$

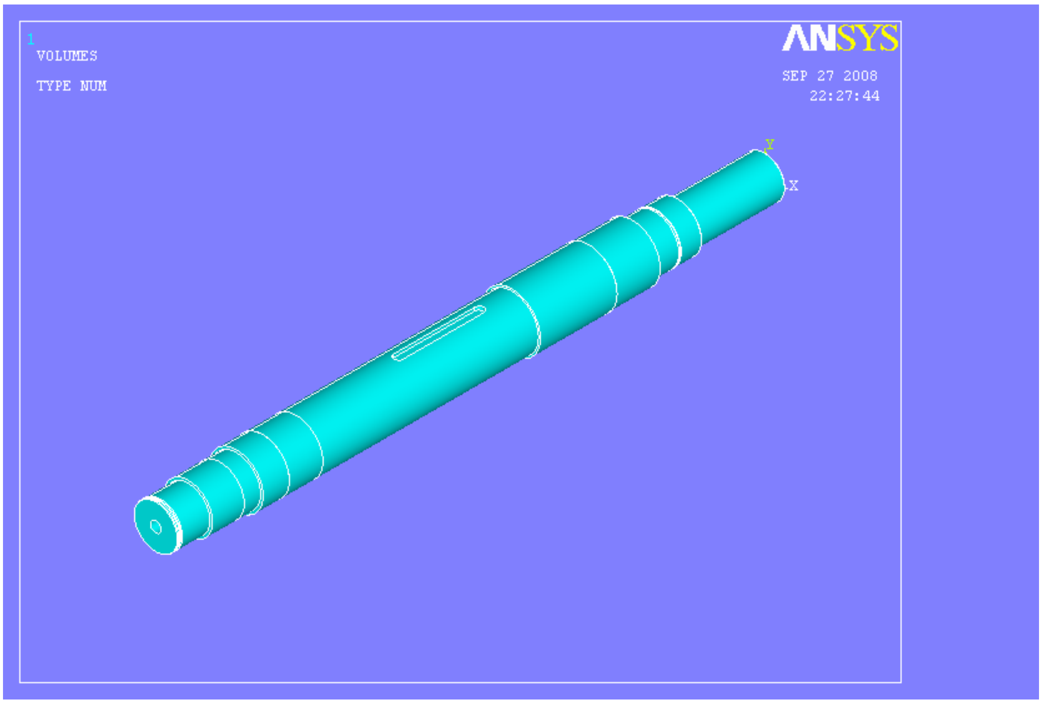
Chapter 7

Results And Discussions

7.1 Static Analysis of Shaft

CAD Model :

CAD model resembling the actual shaft is modeled in CATIA and then exported as IGES. This IGES is imported in ANSYS 11 and used for further FEA analysis. Following figure shows the CAD model after importing in ANSYS.



Mesh Generation :

After successful import of CAD model, elements were generated with SOLID 187 element of ANSYS.SOLID 187 is 3-D 10-Node Tetrahedral Structural Solid as shown in figure.

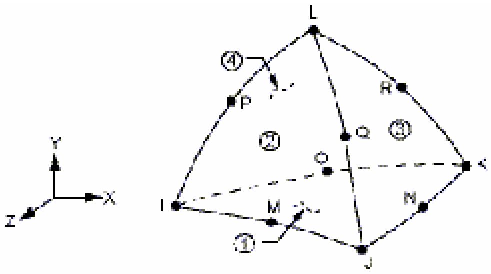


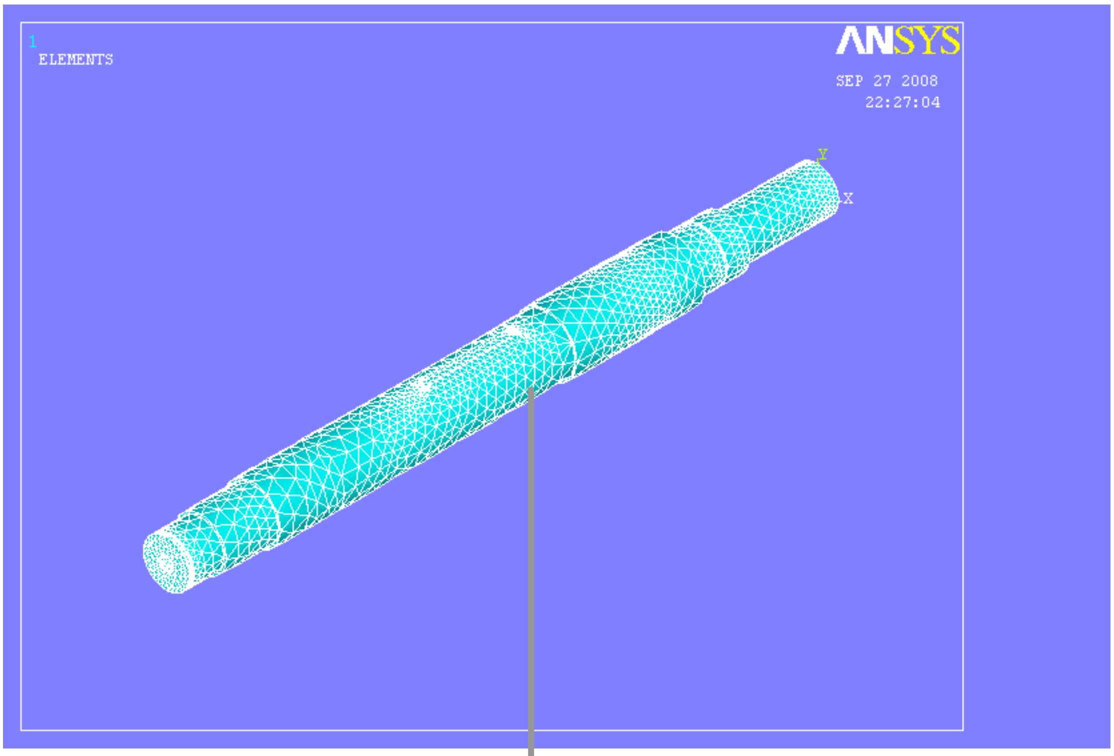
Figure. 18 Mesh generation

SOLID187 element is a higher order element. It has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

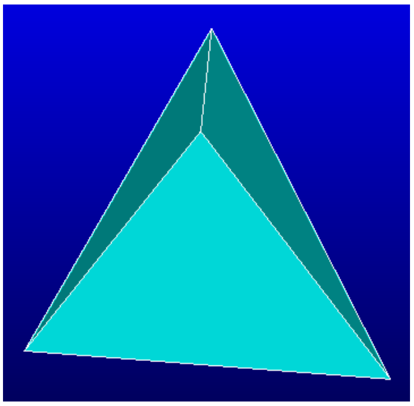
Number of Elements in Model: 52770

Number of Nodes in Model: 77235

Following figure shows meshing of shaft.



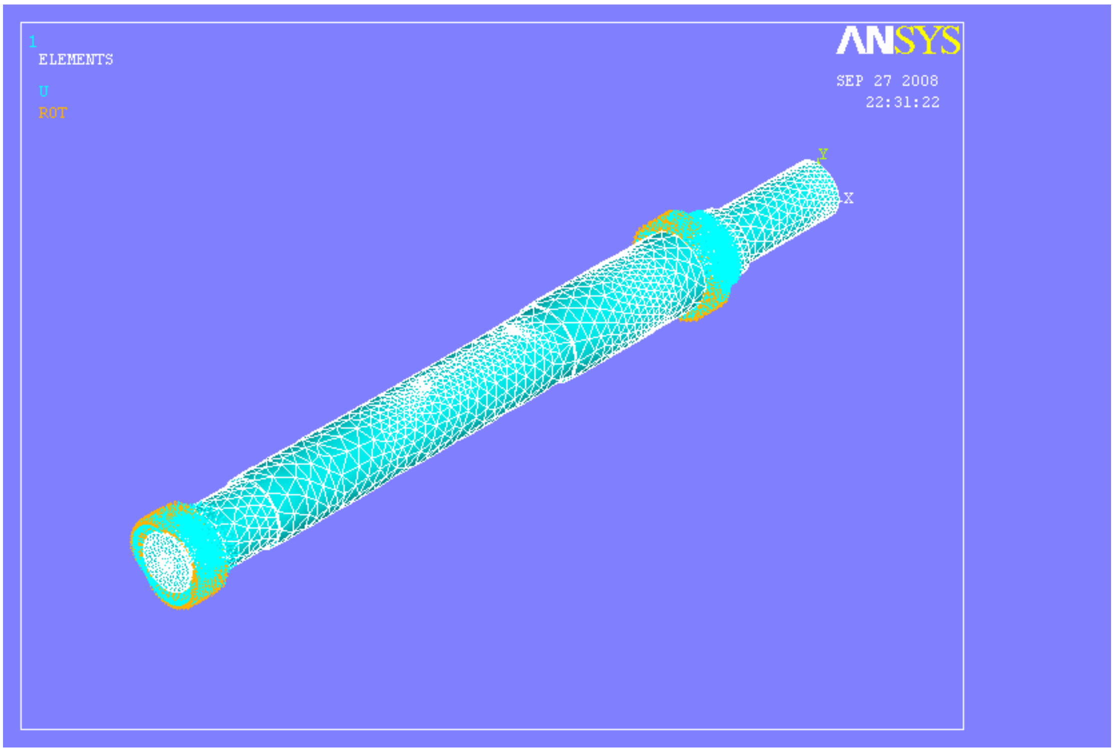
Typical ANSYS element looks as below:



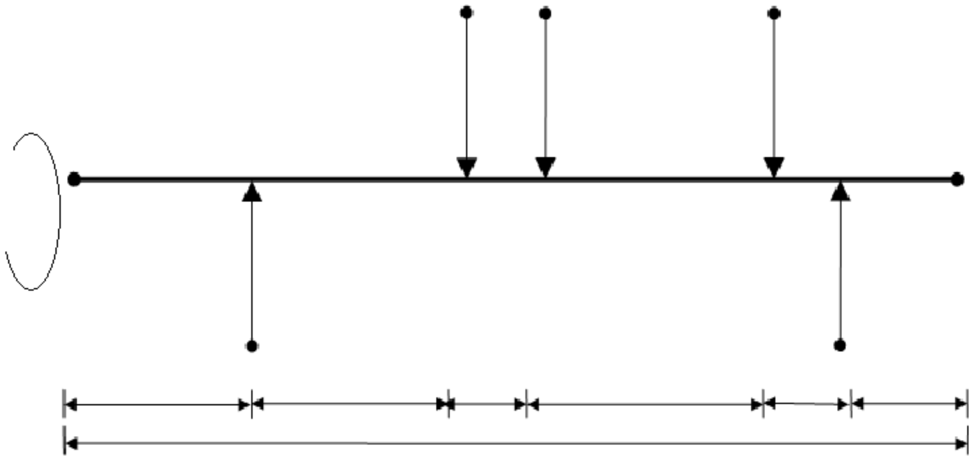
Loads and Boundary Conditions:

Following loads were applied on Shaft (See figure below and FBD):

Weight of Rotor:	2118.96 N
Weight of Fan:	49.05 N
Self Weight:	932.95 N (automatically considered by ANSYS)
Torque:	950271.7342 N-mm



Application of point load is avoided to avoid stress singularity. All point loads shown in FBD were converted to pressure by dividing force by area on which those are acting. Apart form these load, two boundary conditions were applied at bearing locations.



Material Properties:

Material: 40C8 Steel

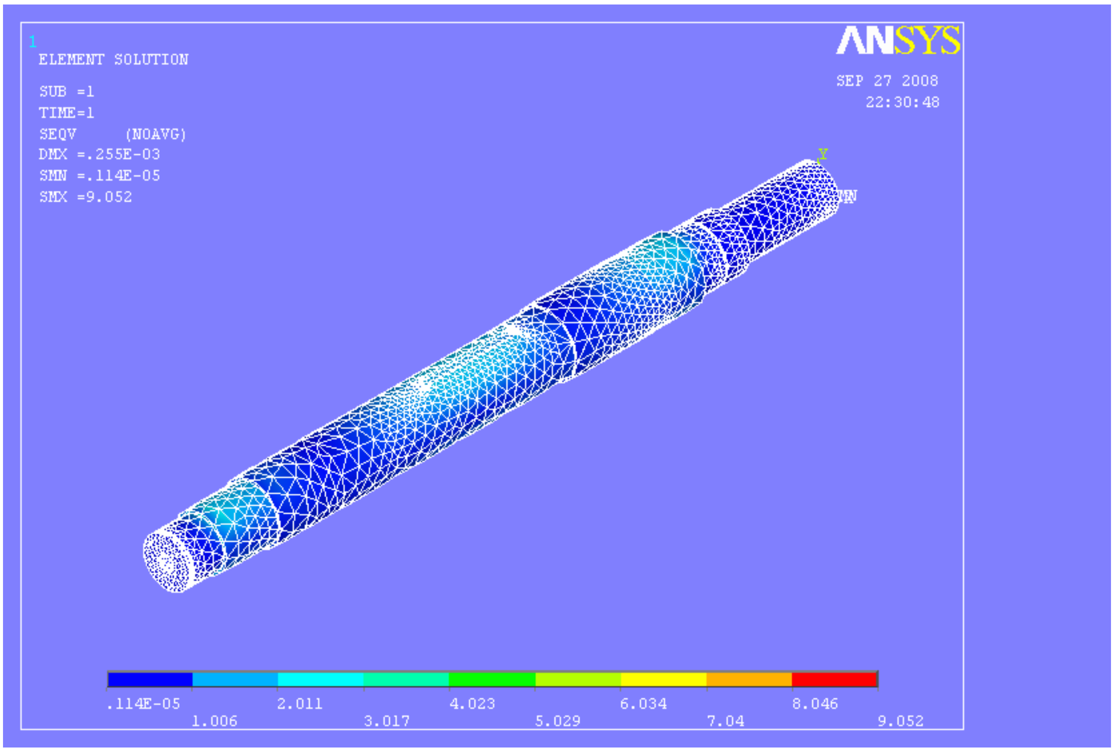
Young's Modulus of Elasticity: 210 GPA

Poisson's Ratio: 0.3

Density: 7500 Kgf/m3

Results:

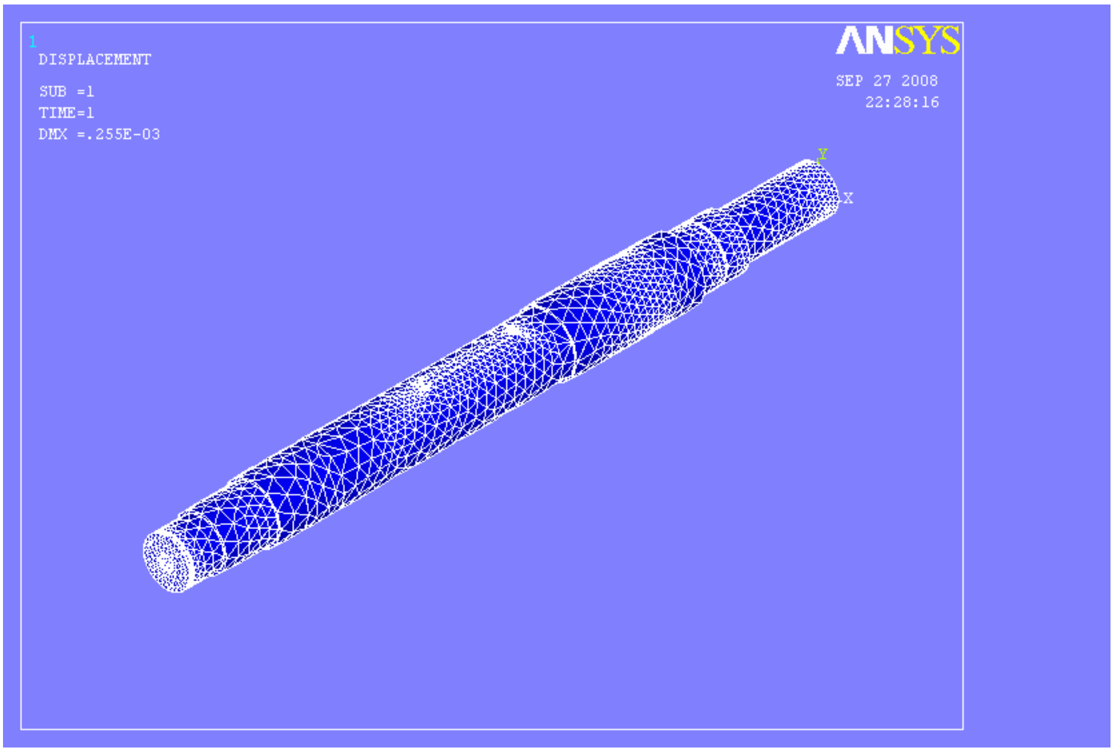
Induced Stresses in shaft are as shown below:



From figure it can be seen that 9.052 N/mm^2 is maximum stress induced in the shaft due to loads and boundary conditions.

Analytically calculated stress is 10.28 N/mm^2 which is well agreement with ANSYS results for static case.

Induced shaft deflection in the static loading case is as shown below:



Comparison between Analytical & FEA Results :

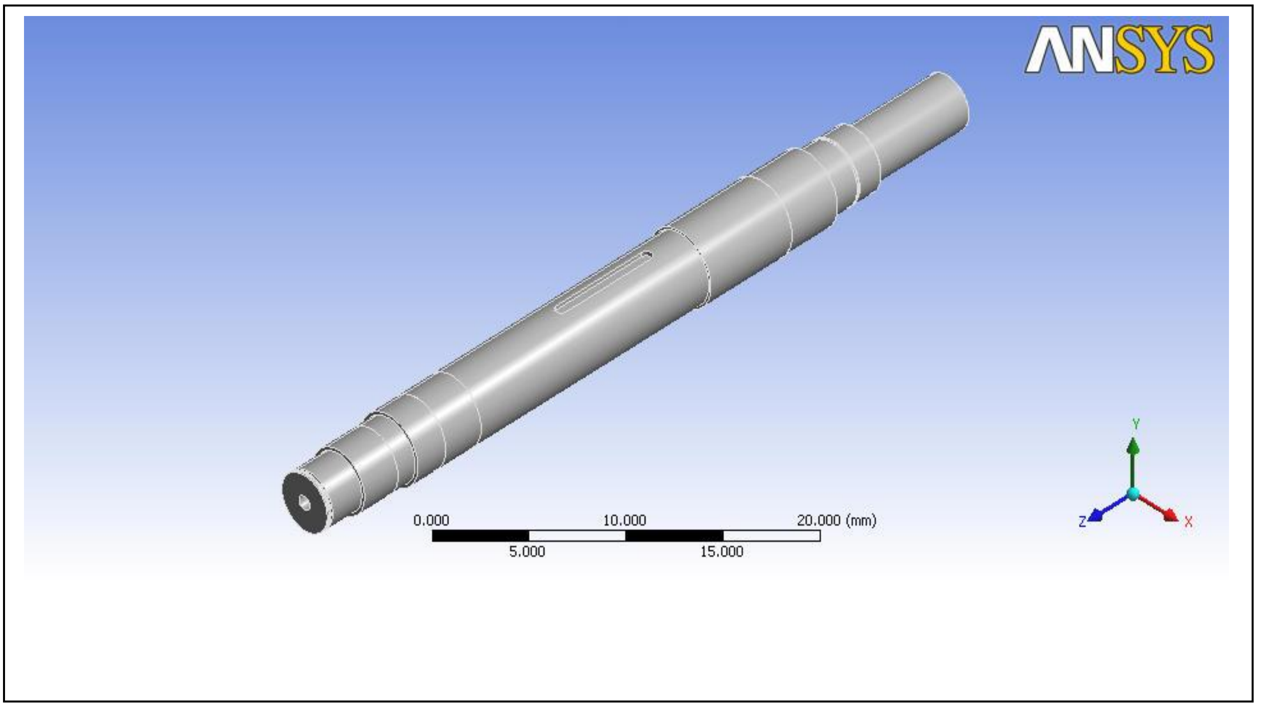
Item	Analytical	FEA	Error %
Combined Stresses (τ)	10.28	9.052	11.94
Deflection (Y)	0.212	0.255	10.52

7.2 **Dynamic Analysis of Shaft**

A dynamic analysis is one in which the results of a modal analysis are used with a known spectrum to calculate displacements and stresses in the model.

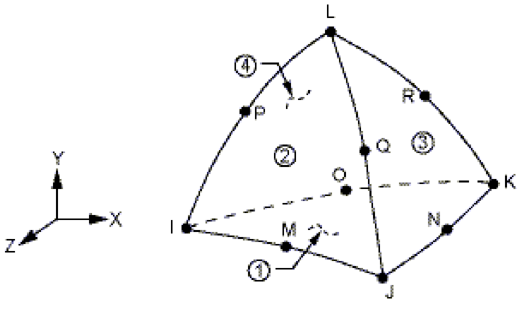
CAD Model:

CAD model resembling the actual shaft is modeled in CATIA and then exported as IGES. This IGES is imported in ANSYS Workbench 11 and used for further FEA analysis. Following figure shows the CAD model after importing in ANSYS.



Mesh Generation:

After successful import of CAD model, elements were generated with tetrahedral (SOLID) elements of ANSYS. Tetrahedral Structural Solid as shown in following figure:

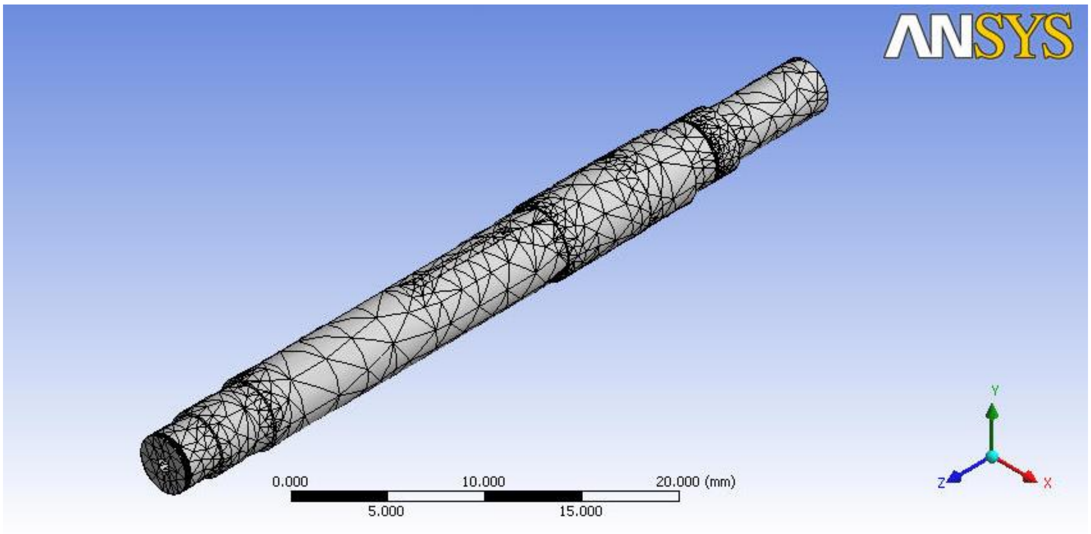


It is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

Number of Elements in Model: 6182

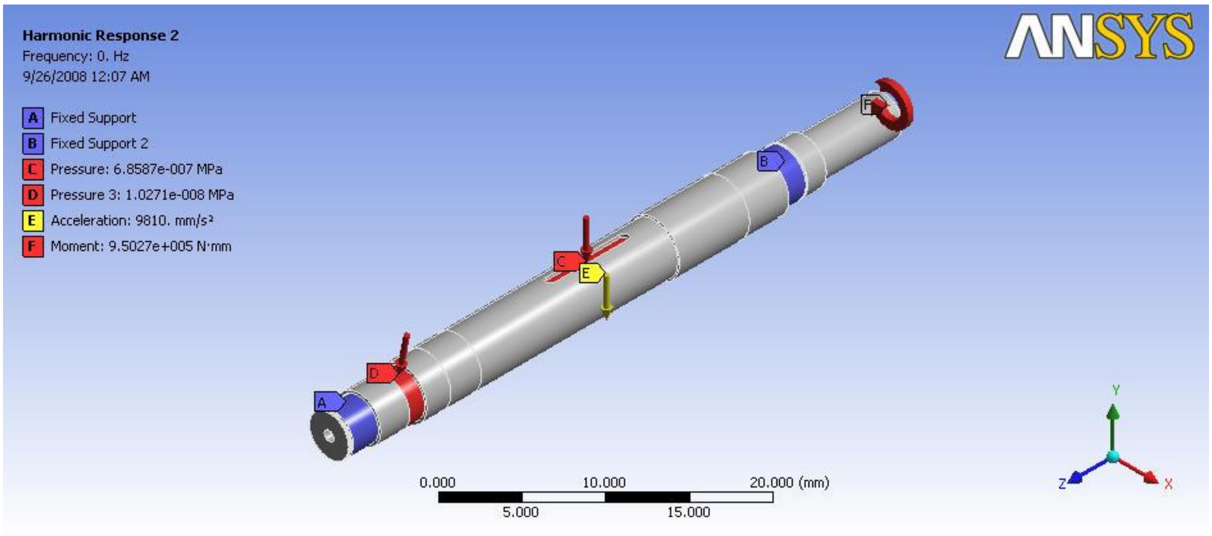
Number of Nodes in Model: 11545

Following figure shows meshing of shaft.



Following loads were applied on Shaft (See FBD):

Weight of Rotor:	2118.96 N
Weight of Fan:	49.05 N
Self Weight:	932.95 N (automatically considered by ANSYS)
Torque:	950271.7342 N-mm
Speed:	1500 rpm



Application of point load is avoided to avoid stress singularity. All point loads shown in FBD were converted to pressure by dividing force by area on which those are acting. Apart form these load, two boundary conditions were applied at bearing locations.

Material Properties:

Material: 40C8 Steel

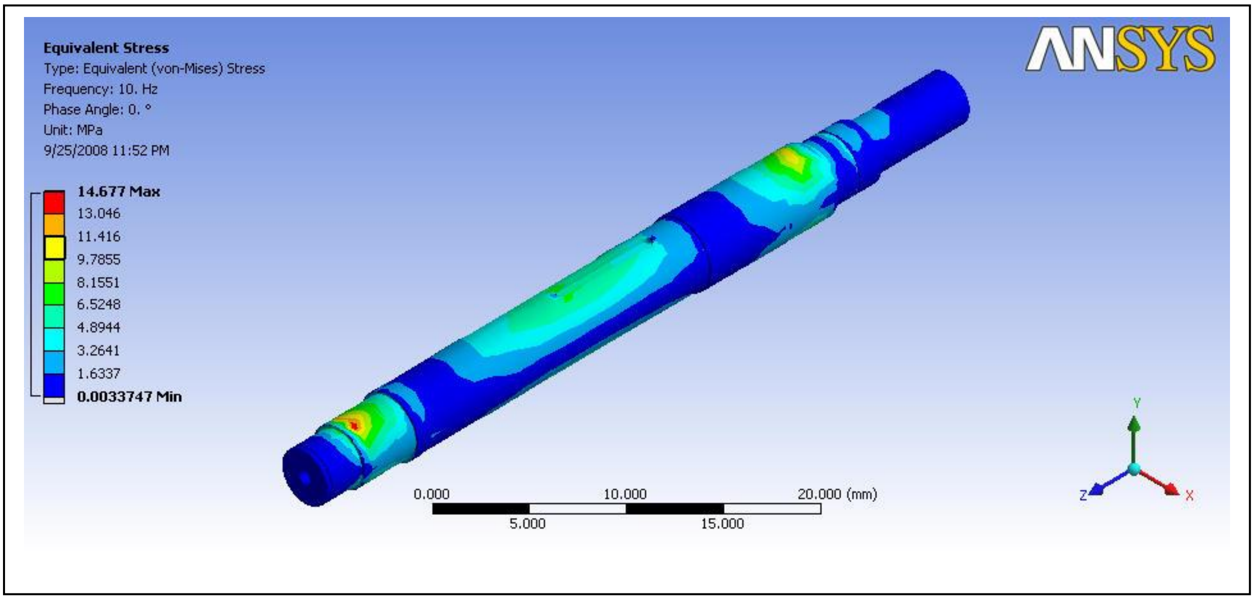
Youngs Modulus of Elasticity: 210 GPA

Poisson's Ratio: 0.3

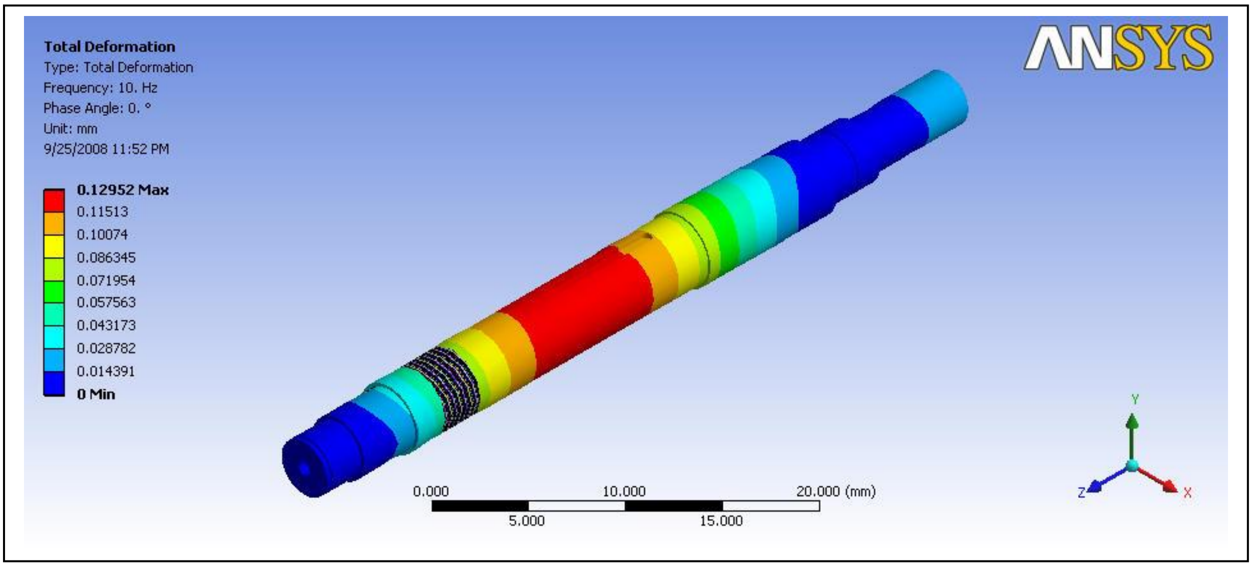
Density: 7500 Kgf/m^3

Results:

Induced stresses due to dynamic effect in the shaft are as shown below:



Deflection due to dynamic effects in the shaft is as shown below:



Chapter 8

Conclusion

Following conclusions are drawn from this study

The motor rotor shaft is designed on the basis static and dynamic loadings. The design is validated analytically for static case and pattern is observed for dynamic case. As expected in dynamic loading stresses induced in shaft are higher than static case. The stresses induced in the shaft are validated by FEA using ANSYS software for both the cases.

The maximum bending moment in shaft is near right hand bearing support, at rotor location. The maximum equivalent stress is in keyway and bearing support area in both the cases.

Problems in ANSYS were solved using SOLID elements. Moreover, Finite element analysis is also carried out to find deflection in the shaft, which is within allowable limit for both cases.

Scope for Further Work

In future work can be expand in following direction Modal analysis can be carried out to determine vibration characteristics such as natural frequency and node shape. The alternate material or composite material can be used to enhance the low weight and heavy load sustained capacity. Thermal analysis of the shaft by using finite element analysis.

REFERENCES

1. *V.K. Mehta,* "Electrical Technology", Khanna Publication, 2002, p.p. 10-11.
2. *B.L. Thereja,* "A Textbook of Electric Technology", Vol. – II, S. Chand Publication, Delhi, 2001, p.p. 44
3. *Rai, G.B.* 1974. Air Gap Eccentricity in Induction Motors. Thesis. ERA Technology Ltd. Leatherhead, Surrey, England.
4. *Timar, P.L.,* A. Fazekas, J. Kiss, A. Miklos and S.J. Yang. 1989. Noise and Vibration of Electrical Machines, Elsevier Science Publishers, Amsterdam, the Netherlands and Akademiai Kiado, Budapest, Hungary
5. *Ozturk, C.O., A. Balikcioglu, H. Acikgoz and A. Bahadir.* 1994. Origins of the Electromagnetic Vibrations in Series Fractional Horsepower Motors. In ISMA Proceedings Vol. 19 – Tools for Noise and Vibration Analysis, 1103-1114.
6. *Girgis, R.S. and S.P. Verma.* 1979. Resonant Frequencies and Vibration Behaviour of Stators of Electrical Machines as Effected by Teeth, Windings, Frame and Laminations. IEEE Proceedings, 1446-1454.
7. *Richard L. Nailen,* "Electric Apparatus," The Magazine of Electric & Electronic Applications, Sept. 2006.
8. *Rao, S.*S. 1990. Mechanical Vibrations. 2nd ed. Addison-Wesley Publishing Company, Reading, MA USA
9. *Thomson, W.T.* 1993. Theory of Vibrations with Applications. 4th ed. Prentice Hall, Englewood Cliffs, NJ USA.
10. *Cheng, F.-H.* 1986. Applied Strength of Materials. Macmillan Publishing Company, New York, NY USA.
11. *LhpL Solapur*, "Technical Database", 2006, p.p. 4
12. *LhpL Solapur*, "Technical Manual", AC Motors, p. Induction p. 7
13. *Kaulus-Jurgan* Bathe, Finite Element Procedures in Engineering Analysis, Prentice- Hall India,1990 .
14. *Swokowski, E.W.* 1991. Calculus. 5th ed. PWS-KENT Publishing Company, Boston, MAUSA.
15. *Ramamrutham*, 1992, "Steel Table", Edition, Dhanpnt Rai & Sons, Delhi.

16. *Ramani, A.* 1993. Finite Element Modeling of a Refrigeration Compressor for Noise Prediction Applications. Thesis. Virginia Polytechnic Institute and State University, Mechanical Engineering Department, Blacksburg, VA.
17. *Knight, C.E.* 1993. The Finite Element Method in Mechanical Design. PWS-KENT Publishing Company, Boston, MA USA.
18. *Dr. P.C. Sharma and Aggarawal,* A text Book of Machine Design, Katson Publication,1996.
19. *M.F. Spotts,* "Design of Machine Element", Prentice Hall International, 1998.
20. *Rao Prakash D.S.,* "Strength of materials", University Press Ltd., p.p. 142-170.S.
21. "Design Data Book", Faculty of Mechanical Engineers, P.S.G. College of Technology, Coimbutore, DPV Printers, Coimbutore.
22. *R.S. Khurmi and J.K. Gupta*, "Machine Design", Eurasia Publishing House (Pvt.) Ltd., New Delhi.
23. *Shigley, J.E. and L.D. Mitchell.* 1983. Mechanical Engineering Design. 4th ed. McGraw-Hill Book Company, New York, NY USA.
24. *Robert Cook*, Concept and Application of Finite Element Analysis, John Wiley,1089.
25. *A.P. Belegundu and T.R. Chandrapatla,* Introduction to Finite Element Engineering, Prentice-Hall India,2002.
26. *S.P. Patil,* Mechanical System Design, Jaico Publishing House,2004
27. *P. Seshu*, Text Book of Finite Element Analysis, Preentice-Hall India, 2004.

APPENDIX-1

PLANE 42 element descriptions:

PLANE42 is used for 2-D modeling of solid structures. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. The element is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities (Fig.4.5).

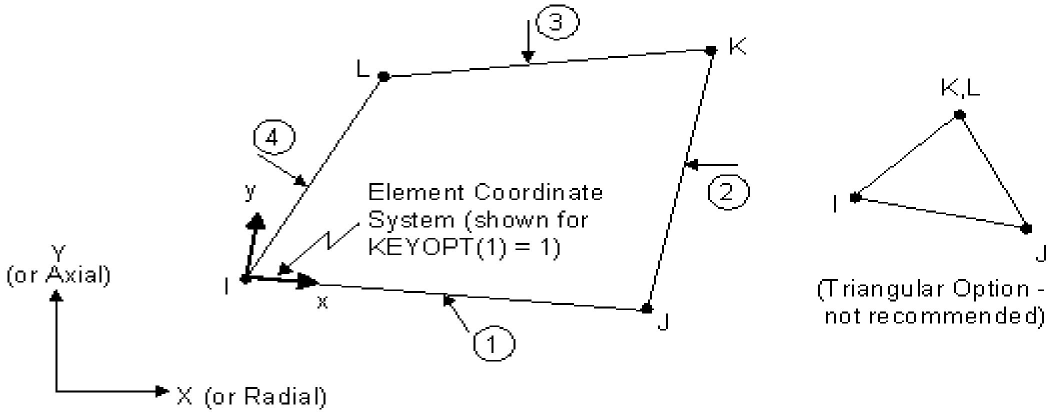


Figure. 17: Plane 42 element

Input Data:

The geometry, node locations, and the coordinate system for this element are shown in Plane42. The element-input data includes four nodes, a thickness (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

Element loads are described in Node and Element loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on PLANE 42 .Positive pressures act into the element. Temperatures and fluencies may be input as element body loads at the nodes. The node I temperature T (I) defaults to TUNIF. If all other temperatures are unspecified, they default to T (I). For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occur for fluency except that zero is used instead of TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT (3) = 3) and on a full 360° basis for an axisymmetric analysis. KEYOPT (2) is used to include or suppress the extra displacement shapes.

KEYOPT (5) and KEYOPT (6) provide various element printout options.

KEYOPT (9) = 1 is used to read initial stress data from a user subroutine.

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using SOLCONTROL, INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetrical matrix is needed for pressure load stiffness effects, use NROPT, UNSYM.

APPENDIX-2

SOLID187

3-D 10-Node Tetrahedral Structural Solid

MP ME ST <> <> PR <> <> <> PP ED

SOLID187 Element Description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. See SOLID187 in the ANSYS, Inc. Theory Reference for more details about this element.

Figure 187.1 SOLID187 Geometry

SOLID187 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 187.1: "SOLID187 Geometry".

In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Linear Material Properties.

Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 187.1: "SOLID187 Geometry". Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T (I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems, you can use ESYS to orient the material properties and strain/stress output. Use RSYS to choose output that follows the material coordinate system or the global coordinate system. For the case of hyper elastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(6) = 1 or 2 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations in the **ANSYS Elements Reference.**

You can apply an initial stress state to this element through the ISTRESS or ISFILE command. For more information, see Initial Stress Loading in the ANSYS Basic Analysis Guide. Alternately, you can set KEYOPT (10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the Guide to ANSYS User Programmable Features. The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use NROPT, UNSYM. The next table summarizes the element input. Element Input gives a general description of element input.

SOLID187 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX, THSY, THSZ),

PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),

DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

Face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features

Plasticity
Hyperelasticity
Viscoelasticity
Viscoplasticity
Creep
Stress stiffening
Large deflection
Large strain
Initial stress import
Automatic selection of element technology
Birth and death
Supports the following types of data tables associated with the TB command: AHYPER, ANEL, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, RATE, CREEP, HYPER, PRONY, SHIFT, CAST, SMA, ELASTIC, SDAMP, PLASTIC, and USER.

Note

See the ANSYS, Inc. Theory Reference for details on the material models.

See Automatic Selection of Element Technologies and ETCONTROL for more information on selection of element technologies.

KEYOPT (6)

Element formulation:

0 --

Use pure displacement formulation (default)

1 --

Use mixed formulation; hydrostatic pressure is constant in an element (recommended for hyperelastic materials)

2 --

Use mixed formulation, hydrostatic pressure is interpolated linearly in an element (recommended for nearly incompressible elastoplastic materials)

KEYOPT (10)

User-defined initial stress:

0 --

No user subroutine to provide initial stresses (default).

1 --

Read initial stress data from user subroutine USTRESS (see the Guide to ANSYS User Programmable Features for user written subroutines)

SOLID187 Output Data

The solution output associated with the element is in two forms:

Nodal displacements included in the overall nodal solution

Additional element output as shown in Table 187.1: "SOLID187 Element Output Definitions"

Several items are illustrated in Figure 187.2: "SOLID187 Stress Output". The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate system and are available for any face (KEYOPT (6)). The coordinate system for face JIK is shown in Figure 187.2: "SOLID187 Stress Output".

The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in Element Solution are met. A general description of solution output is given in The Item and Sequence Number Table. See the ANSYS Basic Analysis Guide for ways to view results.

Figure 187.2 SOLID187 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [ETABLE, ESOL]. The O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is always available, a number refers to a table footnote that describes when the item is conditionally available, and a - indicates that the item is not available.

SOLID187 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [6]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH: EQV	Equivalent thermal strains [6]	1	1
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1

Name	Definition	O	R
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND: ELASTIC, PLASTIC, CREEP	Strain energy density	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

Nonlinear solution, output only if the element has a nonlinear material

Output only if element has a thermal load

Available only at centroid as a *GET item.

Available only if OUTRES, LOCI is used.

Available only if the USERMAT subroutine and TB, STATE are used.

The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (MP, PRXY); for plastic and creep this value is set at 0.5.

For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL. SOLID187 Item and Sequence Numbers” lists output available through ETABLE using the Sequence Number method. See The General Postprocessor (POST1) in the ANSYS Basic Analysis Guide and The Item and Sequence Number Table in this manual for more information. The following notation is used in Table 187.2: "SOLID187 Item and Sequence Numbers": Name Output quantity as defined in Table 187.1: "SOLID187 Element Output Definitions" Item Predetermined Item label for ETABLE command

I, J, R

Sequence number for data at nodes I, J, R

SOLID187 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-

See Surface Solution in this manual for the item and sequence numbers for surface output for ETABLE.

SOLID187 Assumptions and Restrictions

The element must not have a zero volume.

Elements may be numbered either as shown in Figure 187.1: "SOLID187 Geometry" or may have node L below the I, J, K plane.

An edge with a removed midside node implies that the displacement varies linearly, rather than parabolic ally, along that edge. See Quadratic Elements (Midside Nodes) in the ANSYS Modeling and Meshing Guide for information about using midside nodes.

When mixed formulation is used (KEYOPT(6) = 1 or 2), no midside nodes can be missed.

If you use the mixed formulation (KEYOPT(6) = 1 or 2), you must use either the sparse solver (default) or the frontal solver.

Stress stiffening is always included in geometrically nonlinear analyses (NLGEOM, ON). It is ignored in geometrically linear analyses (NLGEOM, OFF) when specified by SSTIF, ON. Priestess effects can be activated by the PSTRES command.

SOLID187 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

Annexure