RESEARCHERID
THOMSON REUTERS
[ICAME-2018]
ISSN 2348 – 8034
Impact Factor- 5.070

GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES

DESIGN AND ANALYSIS OF AUTOMATIC LIFTING AND PLACING MACHINE

J.Anjaneyulu*[1], K. Srinivasa Rao[2] & Mr. K. Jeevan Reddy[3]
[*1]Asst. professor (Sr.), Dept. of Mech. Engg., Vasavi College of Engineering, Hyderabad
[2]K. Srinivasa Rao, Associate Professor, Dept. of Mech. Engg., Vasavi College of Engg., Hyderabad
[3]K. Jeevan Reddy, Asst. Professor, Dept. of Mech. Engg., SNIST, Ghatkesar, Hyderabad

ABSTRACT
In this paper, the automatic lifting and placing machine is analyzed for it's motion and stability. The mechanism motion characteristics are analyzed and the critical component of the machine input shaft is analyzed for stresses and displacements. However, this study is intended for introducing finite element methods from an application viewpoint. Many mechanical systems which have to work effectively under actual working conditions are required to model simulate and analyze for their design validation. The machine parts are modeled and assembled and motion analysis is carried out using ADAMS software. The shaft is analyzed for determining stresses and vibration characteristics are found by using ANSYS and the design evaluation is presented.

I. INTRODUCTION

Along the years, many unexpected failures of equipment and various machines have been occurred throughout the industrial world. A number of these failures have been due to poor design. However, it has been discovered that many failures have been caused by flaws in materials that lead to failure.

In most of the industrial and household machineries, one of the most vital factors to be considered is the speed on which the machinery runs. A good control over the speed assures effective functioning of the machine.

It is shown that the routine calculations reveal that particular failure, and probably many other similar failures, is not always simple. The change in shaft diameters or some uneven sections, which may be made, for example, by a lathe operator during manufacturing of the shaft may cause stress concentration and induce failure. Stress calculations involving stress concentration and fatigue calculations provide a powerful tool with which to arrive at a more complete analysis of failure. The underlying principle is that results of stress and fatigue calculations should be consistent with the proposed mode of failure.

Basically, the study was carried out in three parts: 1) Motion analysis of the Automatic lifting and placing mechanism2) Stress analysis of the shaft using Finite Element Method under given conditions, 3) Dynamic analysis of shaft. In fact, there is a strong correlation between them. Therefore, it will be very convenient to investigate each of them to arrive at the complete analysis of failure.

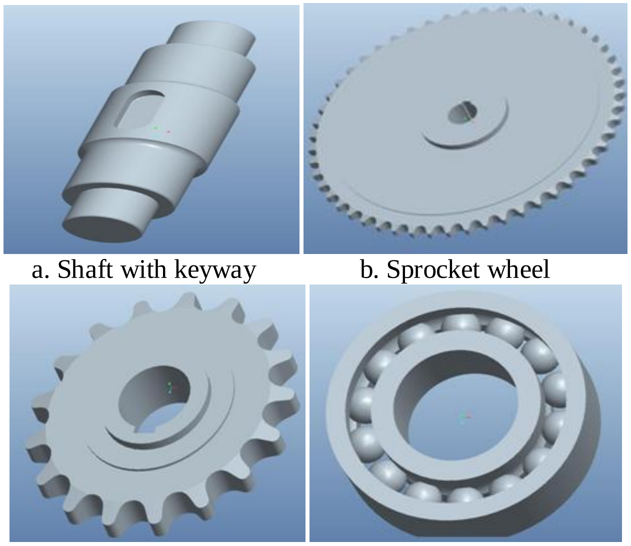
In the whole mechanism, it was observed that the unit fails when there is a sudden load or jerk occurs at the pick-up point or the drop point. This jerk in the unit causes the shaft assembly to undergo deformation and may lead to failure. There is also a situation where the unit must be operated without any resonance and must be operated at minimal amplitude of vibrations. Deepan Marudachalam, et.al [1] studied frequent failure of a shaft employed in a spinning machine. Failure occurred at the vicinity of change in cross section of the shaft where a relief groove is present. Bhumesh and Laukik [2] presented the static structural and fatigue analysis of single cylinder engine crank shaft. The main work was to model the crankshaft with dimensions and then simulate the crankshaft for static structural and fatigue analysis.R.A. Guzar and, S.V.Bhaskar [3] studied the shaft employed in an Inertia dynamometer rotated at 1000rpm. Considering the system, forces, torque acting on a shaft is used to calculate the stresses induced. Stress analysis also carried out by using FEA and the results are compared with the calculated values.

ISSN 2348 – 8034
Impact Factor- 5.070

II. MODELLING OF THE MECHANISM

Basic description of the mechanism

The mechanism unit mainly consists of a shaft, sprocket wheel and pinions and lifting equipment. The complete drive system is located in the structure. Normally the shaft is carried and guided by radial (thrust) bearing. The shaft is driven by a motor/gearbox combination. The structure must not only be able to carry the corresponding weight of this large drive system but also the static and dynamic reaction forces and moments resulting from the operation process.



a. Shaft with keyway b. Sprocket wheel

c. Sprocket pinion d. Bearing

Fig.1. Part models of the mechanism

Assembly of the mechanism

The assembly model for the lifting mechanism is constructed based upon the required external dimensions. Each part is modelled according to the standard measurements and sub-assemblies were created. These sub-assemblies were assembled together to form the main assembly. Assembly constraints like mate/align, insert, point on line, slot and pin joints are used in the modelling of the main assembly. At first, one side of the mechanism which includes sprocket wheel and pinions, shafts, roller chain, supporting plates, bearings and bolts are assembled to form the sub-assembly. The chain is modelled by using the point on line and slot constraints. One closed chain link and an open chain link are assembled together and then each link is joined by one planar and two slot constraints forming a chain of roller links. First the closed link is imported and then open link is added to the closed link and so on until the chain completes forming one single chain of rollers following the curve path.

And it is then mirrored and called into the main assembly. The sub-assembly of four bar mechanism is then called into the mechanism and is constrained using pin joint. The tie rods are constrained to the fixed 4-bar plate by using pin joint. The round plate is constrained with two tie rods in the bearing by using pin joints at both sides. Two more tie rods are imported and constrained to the round plate. The holding plate is attached to the fixed 4-bar plate. A parallel shaft passing through the holding plate and supported by locking blocks in the roller chain is constrained in the bearings using pin joint. The structural frames are modelled and assembled in the main assembly. Then the chain block, locking block, guiding blocks, flange bearings and holding plate are assembled together for the support of lifting mechanism.

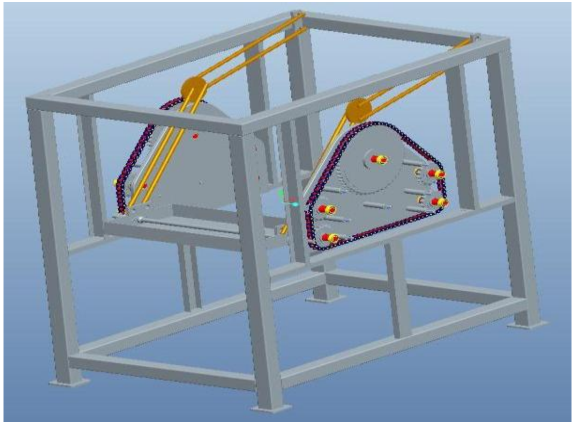


Fig.2. Isometric view of the Mechanism assembly

motion analysis

Multibody simulation (MBS) is a method of numerical simulation in which multibody systems are composed of various rigid or elastic bodies. Connections between the bodies can be modeled with kinematic constraints (such as joints) or force elements (such as spring dampers). Unilateral constraints and Coulomb friction can also be used to model frictional contacts between bodies. Multibody simulation is a useful tool for conducting motion analysis. It is often used during product development to evaluate characteristics of comfort, safety, and performance. For example, multibody simulation has been widely used since the 1990s as a component of automotive suspension design. It can also be used to study issues of biomechanics, with applications including sports medicine, osteopathy, and human-machine interaction.

The heart of any multibody simulation software program is the solver. The solver is a set of computation algorithms that solve equations of motion. Types of components that can be studied through multibody simulation range from electronic control systems to noise, vibration and harshness. Complex models such as engines are composed of individually designed components, e.g. pistons/crankshafts. The mobility of the mechanism is calculated as shown in Fig.3.

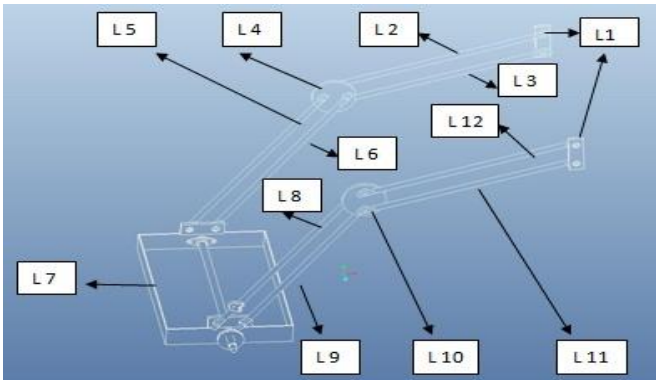
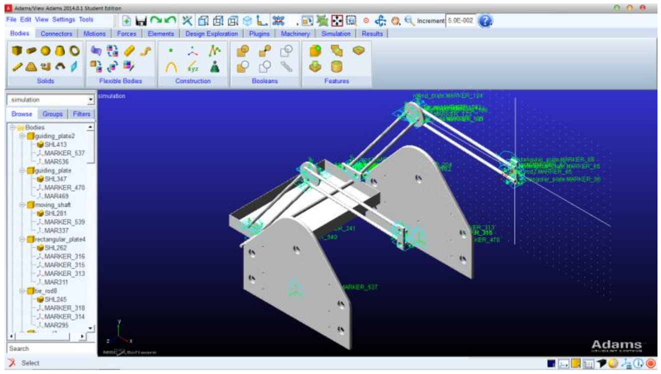


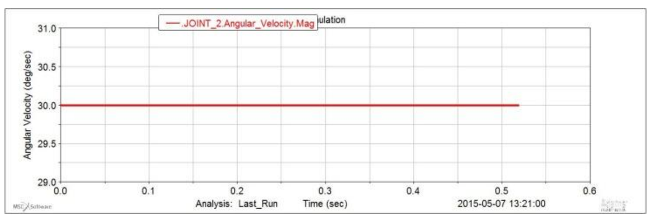
Fig.3. Numbers of links for mechanism.

Number of links, n = 12
Number of lower pairs, $p_1 = 16$
and number of higher pairs, $p_2 = 0$,
According to Grubler's criterion,
$F = 3(n-1)-2P_1-P_2$
$F = 3(12-1)-2(16)-(0)$
F = 1. Hence the mechanism requires one input only.

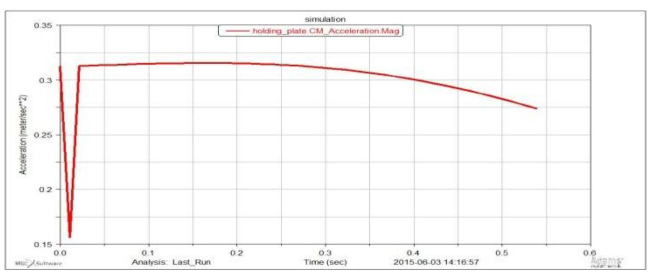
ADAMS software is used for simulation of the mechanism. The motion of the mechanism is analyzed and the plots are shown in Fig.4.



a. Assembly model in ADAMS



b. Input shaft with constant angular speed



c. Acceleration of the weight lifting plate
Fig.4. Results from ADAMS.

When the input shaft rotates at constant speed of 45 rpm, the output link holding plate on which the bottles are placed is subjected to maximum acceleration of 0.32 m/s^2.
Maximum inertia force F_i = 0.32*70 (kg-m/s^2)= 22.4 N.

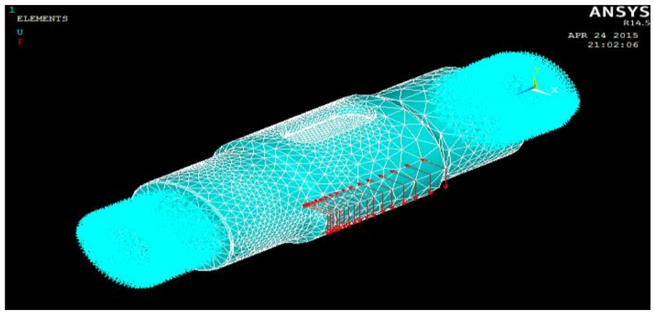
The maximum inertia force arises at pickup point is 22.4 N (to hold the bottles of total mass 70 kg).Hence Multibody kinematic simulation validates the mechanism assembly motion requirements.

Analysis using ANSYS
Static analysis.
The finite element analysis was performed on the input shaft of the mechanism by using ANSYS. element meshes were generated using the

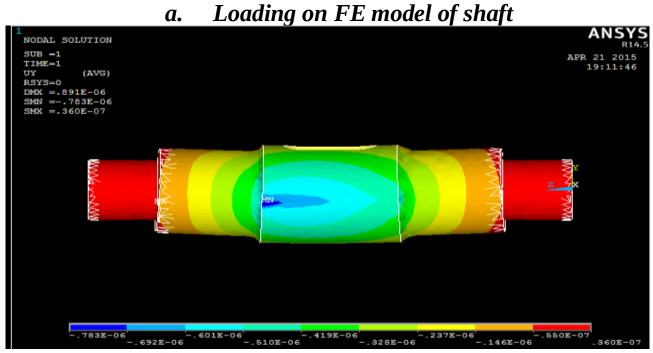
Element type	=	solid 187(tetrahedral 10 node element)
Material	=	En 24 steel
Young's modulus	=	2e11 N/m^2
Poisons ratio	=	0.3
Density	=	7700kg/m^3.

The loads are applied as both radial and tangential as shown in Fig.5 a. Also the gravity load due to rotation of the shaft is applied using angular velocity about Z-axis.The boundary conditions are applied at the bearing supports. The nodal displacements and stresses obtained are shown in Fig.5b and c respectively. As shown in figure 3.11, the maximum deflection is 0.891e-3 mm which is less than the allowable deflection. Hence the design is safe based on rigidity criteria.

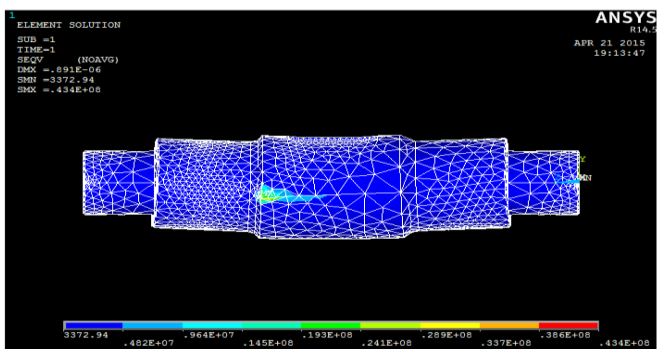
The maximum stress induced within shaft component is found to be 43.4 Mpa, which is very lower than that of the allowable stress of 465 Mpa. Hence the design is safe based on the strength criteria.



a. *Loading on FE model of shaft*



b. *Displacements plot*



c. *Von-Mises stress plot*

Fig.5. Results from ANSYS

Dynamic analysis.

i. Modal analysis.

When the loads are suddenly applied, or when the loads are of a variable nature, the mass and acceleration effects come into picture. If a solid body, such as an engineering structure, is deformed elastically and suddenly released, it tends to vibrate about its mean equilibrium position. This periodic motion due to the restoring strain energy is called free vibration. The number of cycles per unit time is called frequency. The maximum displacement from the

ISSN 2348 – 8034
Impact Factor- 5.070

equilibrium position is amplitude. The vibrations subside with time due to damping action. The undamped free vibration model of a structure gives the significant information about its dynamic behavior.

Modal and harmonic analyses are performed to estimate the natural frequencies and amplitudes of forced vibration respectively, thereby preventing the resonance condition.

Resonance is occurred when the frequency of external force is same as that of the natural vibrations. At this state of resonance, angular speed is equal to angular frequency.

I.e. $\omega = \omega_n$

Where,

ω = angular speed

ω_n = angular frequency

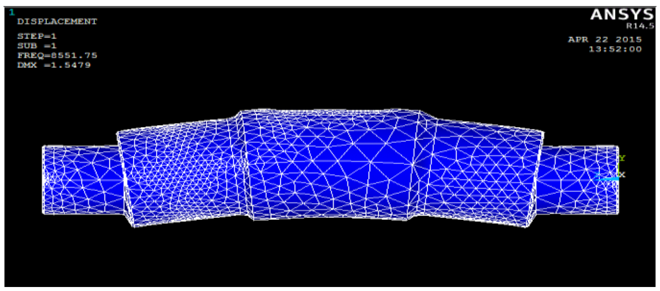
Since $\omega = 2\pi N/60$, $\omega n = 2\pi f$;

The value of applied frequency **f** corresponds to **N =** 45 rpm.

i.e. $2\pi N/60 = 2\pi f$

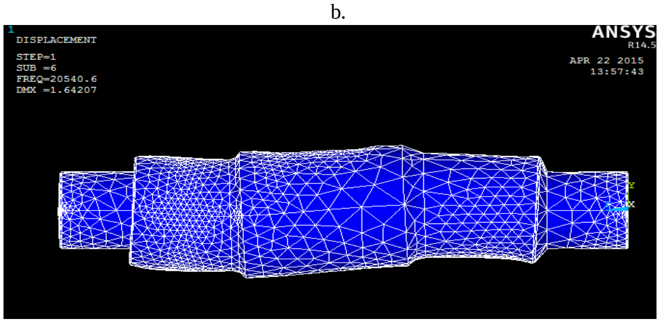
Hence, frequency f= 45/60 = 0.75 Hz.

Modal analysis is carried out to estimate the natural frequencies and corresponding mode shapes. Modes are extracted and shown for the first 3 modes in Fig.6.

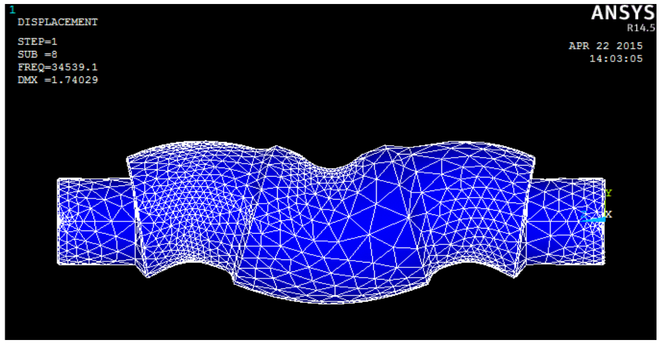


a. Mode-I

b.



c. .Mode-II



d. Mode-III

Fig.6. Mode shapes for first three modes

ii. Harmonic analysis.

Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system. Harmonic response analysis is used to predict the sustained dynamic behavior of the structure, thus verifying whether or not structure will successfully overcome Resonance, fatigue, and other harmful effects of forced vibration.

Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidal (*harmonically*) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. Harmonic response analysis is a linear analysis. Any nonlinearity, such as plasticity and contact (gap) elements, will be ignored, even if they are defined.

Harmonic analysis results are in the form of a graph commonly known as FRF (frequency response function). FRF for this analysis is shown in Fig.7. This FRF is specific to the node at the trailing edge of the tip of the shaft, which showed the most aggressive behavior to cyclic loading. Other nodes were also analyzed, as they showed similar response but value of displacement was lesser compare to selected node, so shown FRF is associated to tip trailing edge node. Fig 4.11 shows the variation of deformation with respect to operating frequency.

Under forced vibrating conditions, a graph was obtained between amplitude and frequencies as in the Fig.7 which indicates that the amplitude of vibration between the frequency ranges of 34539.2 Hz to 42865.5 Hz reaches the peak value. It means the system must be avoided working under the above range of frequencies, as amplitude of forced vibrations is very high thus causing resonance.

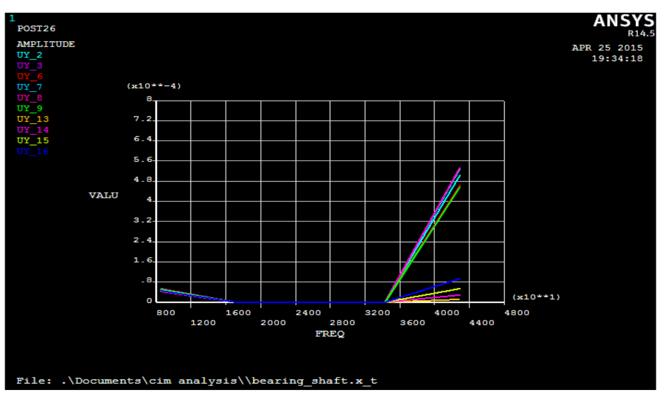


Fig.7. Frequency responses for a force of 350 N

III. CONCLUSION

Based on the analysis performed using multibody simulation, it is concluded that the mechanism with the dimensions considered can perform the required motion satisfactorily.

Shaft component with En-24 material satisfies the requirements for operating conditions of the system.

From modal and Harmonic analyses, it is observed that the first natural frequency is far higher than maximum operating frequency, this gives clear indication that shaft is safe against resonance phenomenon. The harmonic response within specified range is also acceptable, as maximum value of displacement is far lesser than static displacement.

This work can be extended by considering flexibility of the material of the shaft and inertia effect to estimate the stresses during motion of the mechanism. The fatigue analysis can be performed to estimate the life of the shaft in the mechanism.

REFERENCES

1. *Deepan Marudachalam M.G, K.Kanthavel, Krishnaraj, "Optimization of shaft design under fatigue loading using Goodman method", International Journal of Scientific & Engineering Research Volume 2, Issue 8, August-2011.*
2. *Bhumesh J. Bagde, Laukik P. Raut, "Finite Element Structural and Fatigue Analysis of Single Cylinder Engine Crank Shaft", International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 7, July – 2013*
3. *R.A. GUZAR, S.V.BHASKAR, "Shaft Design under Fatigue Loading By Using Modified Goodman Method", International Journal of Engineering Research and Applications,Vol. 3, Issue 4, Jul-Aug 2013, pp.1061-1066..*
4. *Prof. S. G. Kolgiri, Prof. Sudarshan D Martande, Prof. Nitin S Motgi, "Stress Analysis For Rotor Shaft Of Electric Motor" International Journal Of Application Or Innovation In Engineering & Management,Volume 2, Issue 7, July 2013.*
5. *S.P.Timeshenko And J.N.Goodier", Theory Of Elasticity", 3rd Ed., Mc Graw-Hill, New York, 1970.*
6. *Richard G. Budynas And J.Keith Nisbett, "Mechanical Engineering Design", 9th Edition, Mc Graw-Hill, New York, 2011.*
7. *Robert Cook, Concept and Application of Finite Element Analysis, John Wiley, 1089.*
8. *. Se´Bastien Berger, Olivier Bonneau, Jean Fre´Ne,"Influence Of Axial Thrust Bearing On The Dynamic Behavior Of An Elastic Shaft: Coupling Between The Axial Dynamic Behavior And The Bending Vibrations Of A Flexible Shaft".*
9. *J. Wittenburg, Dynamics Of Multibody Systems, Berlin, Springer (2008)M. Young, The Techincal Writers Handbook. Mill Valley, Ca: University Science, 1989.*
10. *J. García De Jalón, E. Bayo, Kinematic And Dynamic Simulation Of Multibody Systems - The Real-Time Challenge, Springer-Verlag, New York (1994)*