Optimization of Crank Shaft using FEA

[1]Mr. J. E. Pachpande, [2]Prof. Dheeraj Verma, [3]Prof. Nitin Kharche
[1]P. G. Student, [2]Asso. Prof. Mech Engg Deptt, [3]HOD Mech Engg.Deptt, Padm. Dr. V. B. Kolte C. O. E. Malkapur, India
pachpandej@gmail.com, dheeraj41@gmail.com, nitinkharche@gmail.com

***Abstract*-- The modal analysis of a 3-cylinder crankshaft of SWARAJ 855 XM tractor is being analyzed using finite element method. Crankshaft is a large volume production component with a complex geometry in the internal combustion engine. This converts the reciprocating displacement of the piston in to rotary motion of the crank. The crankshaft is subjected to shock and fatigue loads causing various stresses are induced in crankshaft so progressive fracture on crankshaft surface due to induced stresses. The crankshaft model was created by using PRO-E wildfire 4.0 for modeling the crank shaft. Then, the created model was import into ANSYS software for static structural analysis.**

The stresses induced on crankshaft was investigated through a systematic force analysis in ANSYS for different kind of two materials like forged steel, steel alloy CrMoV13 with same operating condition. In finite element analysis of different materials have done static structural analysis by putting the values of properties in ANSYS. In force analysis have calculated total deformations, maximum shear stress, equivalent (von-misses) stress, shear stress, maximum principle stress. The comparison of analysis results of two different materials will show the effect of stresses on different materials and to evaluate the most suitable material for manufacturing crankshaft which gives good performance and to avoid possible structural damages on the crankshaft surface.

***Keywords*- Finite Element Analysis, Pro-E, ANSYS, SWARAJ 855 XM tractor Crankshaft.**

I. INTRODUCTION

1.1 Crankshaft

Crankshaft is among the largest components in internal combustion engines. It is employed in different types of engines, from small one cylinder lawn-mowers to large multi-cylinder diesel engines. Crankshaft is one of the most critically loaded components and experiences cyclic loads in the form of bending and torsion during its service life. This review is performed for optimization of crankshaft by considering various materials used for manufacturing of crankshaft, various manufacturing process used for manufacturing of crankshaft and opportunities available for optimization by various geometric changes in shape of crankshaft. Amongst all materials used for manufacturing of crankshaft the best material is selected and is manufactured by method which is most suitable and will reduce the cost of production. So now the various optimization studies are made on crankshaft made by the selected material manufactured method considered.

1.2 Engine Specification Of Swaraj 855 XM

TABLE 1
Engine specification of SWARAJ 855 XM

Model	RB-33 XM+
HP Category	50-55 HP
Type	4 Stroke, Direct injection, Diesel engine
No. of Cylinders	3
Bore and Stroke	110 X 122 mm
Displacement	3480 cc
Rated Engine Speed	1800 rev/min
Air Cleaner	3- Stage Oil Bath Type
Cooling System	Water Cooled with No loss tank Oil Cooler for engine oil
Maximum combustion pressure	156.60bar

1.3 Design of crankshaft when the crank is at the dead centre

1.3.1 Design Of Crankpin:

We know that piston gas load (F_P),

Force on the Piston F_P = Area of the bore x P_{max}

$$\text{Force on Piston}(F_p) = \frac{\pi}{4} \times D^2 \times P_{max}$$
$$= \frac{\pi}{4} \times 110^2 \times 15.66$$
$$F_p = 147.60 \text{ KN}$$

Assume that the distance (b) between the bearing 1 and 2 is equal to twice the piston diameter (D).

$$b = 2D = 2 \times 110 = 220\text{mm}b_1 = b_2 = \frac{b}{2} = \frac{220}{2}$$
$$= 110\text{mm}$$

We know that due to piston gas load, there will be two horizontal reactions H_1 and H_2 at bearings 1 and 2 respectively, such that

$$H_1 = H_2 = \frac{F_p \times b_1}{b} = \frac{147.80 \times 110}{220} = 73.9\text{KN}$$

Bending moment at the centre of the crankpin

$$M_c = H_1 \times b_2 = 73.9 \times 110$$
$$m_c = 8129 \text{ KN-mm}$$

We also know that

$$M_c = \frac{\pi}{32} \times {d_c}^3 \times \sigma_b$$

Diameter of crankpin

$$d_c = 190.37 \text{ Say } 192\text{mm}$$

1.4 Design of crankshaft when the crank is at an angle of maximum twisting moment

1.4.1 Design of crankpin

Let d_c = Diameter of crankpin in mm.

We know that the bending moment at the centre of the crankpin,

$M_C = H_{R1} \times b_2 = 55.63 \times 110 = 6119.3$ KN-mm

$M_C = 6119.3$ KN-mm

Twisting moment on the crankpin,

$\mathrm{T_C} = \mathrm{H_{T1}} \times \mathrm{r} = 49.38 \times 61 = 3012.18$ KN-mm
$T_C = 3012.18$KN-mm
From this we have the equivalent twisting moment

$$\mathrm{T_e} = \sqrt{\mathrm{M_C}^2 + \mathrm{T_C}^2} = 6820.49 \text{ KN-mm}$$

$T_e = 6820.49$ KN-mm
We know that equivalent twisting moment (T_e)

$$\mathrm{T_e} = \frac{\pi}{16} \times (\mathrm{d_c})^3 \times \tau$$

Shear stress value is limited to 35 N/mm^2
$6820.49 = \frac{\pi}{16} \times (\mathrm{d_c})^3 \times 35$
$\mathrm{d_c} = 99.76$ mm
So $\mathrm{d_c} = 99.76$ mm

Since this value of crankpin diameter ($\mathrm{d_c}$= 99.76mm) is less than the when the crank is at top dead center already calculated value of crankpin dia. ($\mathrm{d_c}$= 192mm) therefore, we shall take, $\mathrm{d_c}$= 192mm
Design of crankpin against fatigue loading

According to distortion energy theory
The Equivalent (Von-Misses) stress induced in the crankpin is

$$\mathrm{M_{ev}} = \sqrt{(\mathrm{K_b} \times \mathrm{M_c})^2 + \frac{3}{4}(\mathrm{K_t} \times \mathrm{T_c})^2}$$

Here $\mathrm{K_b}$=combined shock and fatigue factor for bending (Take $\mathrm{K_b} = 2$)
$\mathrm{K_t}$= combined shock and fatigue factor for bending (Take $\mathrm{K_t} = 1.5$)

$$= \sqrt{(2 \times 61193)^2 + \frac{3}{4}(1.5 \times 3012.18)^2}$$

$$\mathrm{M_{ev}} = 12.85 \times 10^3 \text{KN-mm}$$

σ_v = Equivalent (Von-Misses) stress

$$\mathrm{M_{ev}} = \frac{\pi}{32} \times \mathrm{d_c}^3 \times \sigma_v$$

$$12.85 \times 10^3 = \frac{\pi}{32} \times 192^3 \times \sigma_v$$

$$\sigma_v = 18.49 \text{ N/mm}^2$$

τ =Shear stress

$$\mathrm{T_e} = \frac{\pi}{16} \times \mathrm{d_c}^3 \times \tau$$

$$6820.49 \times 10^3 = \frac{\pi}{16} \times 105^3 \times \tau$$

$$\tau = 4.90 \text{ N/mm}^2$$

σ_v= 18.49 N/mm^2 and also calculated shear stress on the shaft $\tau = 4.09$ N/mm^2

II. LITERATURE SURVEY

Evaluates and compares the fatigue performance of two competing manufacturing technologies for automotive crankshafts, namely forged steel and ductile cast iron. In this study a dynamic simulation was conducted on two crankshafts, cast iron and forged steel, from similar single cylinder four stroke engines. Finite element analysis was performed to obtain the variation of stress magnitude at critical locations.The optimization process includes geometry changes compatible with the current engine, fillet rolling and result in increased fatigue strength and reduced cost of the crankshaft, without changing connecting rod and engine block.[1]

Had studied on a forged steel crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations. The dynamic analysis resulted in the development of the load spectrum applied to the crankpin bearing. This load was then applied to the FE model and boundary conditions were applied according to the engine mounting conditions. Results obtained from the aforementioned analysis were then used in optimization of the forged steel crankshaft. Geometry, material, and manufacturing processes were optimized using different geometric constraints, manufacturing feasibility, and cost. The optimization process resulted in an 18% weight reduction, increased fatigue strength, and a reduced cost of the crankshaft.[2]

Studied the optimization options, their combination under a set of defined constraints and a comparison between the original forged steel crankshaft and the final optimized forged steel component. The main objective of this analysis was to optimize the weight and manufacturing cost of the forged steel crankshaft, which not only reduces the final production cost of the component, but also results in a lighter weight crankshaft which will increase the fuel efficiency of the engine. As the main objective of this analysis, it was attempted to reduce the weight and final cost of the component by changing the crankpin geometry, Increasing the oil hole diameter and fillet radius, increasing the oil hole depth and Changing the crank web geometry. [3]

Studied a static analysis was conducted on a cast iron crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations. Three dimensional model of the crankshaft was created in Pro-E software. The load was then applied to the FE model and boundary conditions were applied as per the mounting conditions of the engine in the ANSYS. Results obtained from the analysis were then used in optimization of the cast iron crankshaft. This requires the stress range not to exceed the magnitude of the stress range in the original crankshaft. The optimization process included geometry changes without changing connecting rod and engine block.[4]

Studied the problem occurred in single cylinder engine crank shaft. It consist of static structural and fatigue analysis of single cylinder engine crank shaft. It identifies and solves the problem by using the modeling and simulation techniques. The topic was chosen because of increasing interest in higher payloads, lower weight, higher efficiency and shorter load cycles in crankshaft. The main work was to model the crank shaft with dimensions and then simulate the crank shaft for static structural and fatigue analysis. The modeling software used is PRO-E wildfire 4.0 for modeling the crank shaft. The analysis software ANSYS will be used for structural and fatigue analysis of crank shaft for future work.[5]

This study describes the stress distribution of a forged steel crankshaft used in a single cylinder 4 stroke vertical engine by using commercial Finite Element Analysis (FEA) software ANSYS .The stress analysis results are significant to improve the component design at the early developing stage. Modal analysis was carried out to determine the natural frequencies of the crankshaft and the mode shapes were examined.[6]

III. FEA

The methodology developed is allowed to evaluate the most suitable material for manufacturing crankshaft which gives good performance and to avoid possible structural damages on crankshaft surface. The two types of methods are developed.

3.1 Computer Aided Design

In computer aided design the following steps is involved which is explained below

3.1.1 CAD Modeling

CAD technology is very important while designing any part.

Following are advantages of CAD technology:

- To increase the productivity of the designer
  - Helping designer to conceptualize the product.
  - Reducing time required to design and analyze.
- To improve the quality of the design
  - Allows the engineer to do a more complete engineering analysis and to consider a variety of design alternatives, therefore increasing quality.

3.1.3 3-D Modeling

The essential difference between pro/engineer and traditional cad systems is that models created in pro/engineer exist as three-dimensional solids. Other 3-d modelers represent only the surface boundaries of the model. Pro/engineer models the complete solid. This not only facilitates the creation of realistic geometry, but also allows for accurate model calculations, such as those for mass properties.

3.1.2 CAD/CAE soft wares are used

- PRO/E wildfire 4.0 - For 3D Component Design.
- Pro/Assembly - For Assembling Components
- ANSYS Workbench 11.0 - For FEM analysis

Pro/engineer is a parametric, feature based, solid modeling system. It is the only menu driven higher end software. Pro/engineer provides mechanical engineers with an approach to mechanical design automation based on solid modeling technology and the following features.

3.1.4 Parametric Design

Dimensions such as angle, distance, and diameter control pro/engineer model geometry. You can create relationships that allow parameters to be automatically calculated based on the value of other parameters. When you modify the dimensions, the entire model geometry can update according to the relations you created.

3.1.5 Feature-Based Modeling

You create models in pro/engineer by building features. These features have intelligence, in that they contain knowledge of their environment and adapt predictably to change. Each feature asks the user for specific information based on the feature type. For example, a hole has a diameter, depth, and placement, while a round has a radius and edges to round.

3.1.6 Associativity

Pro/engineer is a fully associative system. This means that a change in the design model anytime in the development process is propagated throughout the design, automatically updating all engineering deliverables, including assemblies, drawings, and manufacturing data. Associativity makes concurrent engineering possible by encouraging change, without penalty, at any point in the development cycle. This enables downstream functions to contribute their knowledge and expertise early in the development cycle.

3.1.7 Part Modeling

- Starting out in part mode--describes how to start creating a part with pro/engineer.
- Sketcher--describes how to create sketches in a stand-alone sketcher mode.
- Datum--describes how to create datum features: datum planes, datum points, datum curves, datum axes, coordinates features, graphs, evaluate features.
- Sketching on a model--describes how to create 3-d sections in the process of feature creation.
- Feature creation basics--describes how to create extruded and revolved protrusions.
- Sweeps, blends, and advanced features--describes how to create sweeps, blends, and advanced features.
- Construction features--describes how to create construction features, such as holes, slots, and cuts.
- Rounds--describes how to add rounds to part geometry.
- Tweak features--describes how to create tweak features, such as draft, local push, and section dome.
- Creating surface features--describes how to create surface features.
- Creating advanced surface features--describes how to create advanced surface features.
- Working with quilts--describes operations that you can perform on quilts.
- Freeform manipulation--describes how to dynamically manipulate a surface of a part or quilt.
- Patterning features--describes how to pattern features.
- Copying Features--Describes how to create and place groups of features, and how to copy features.
- Regenerating the part--describes how to regenerate the part and resolve regeneration problems.

3.2 Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

1. Define the geometric domain of the problem.
2. Define the element type(s) to be used.

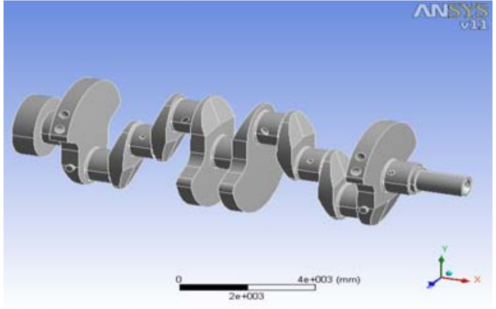

Fig. 1 IGES file of Crank Shaft for ANSYS

3. Define the geometric properties of the elements (length, area)
4. Define the element connectivity's (mesh the model).

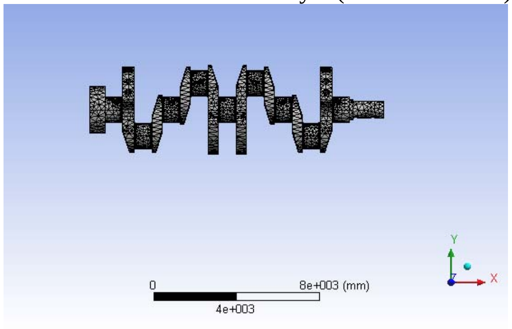

Fig. 2 Front View of the Meshed Element

5. Define the physical constraints (boundary conditions).
6. Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

3.3 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on gauss elimination, is commonly used.

In short in this phase a solver is used to solve the basic equation for the analysis type and to compute the results. This phase is taken care by the software programme. In the solution process, the solver goes through following steps to compute the solution for a steady state analysis.

- Formulate element matrices
- Assembly and triangularise the overall stiffness matrix.
- Calculate the solution by back substitution.
- Compute the stresses, displacements etc.

3.4 Post Processing

Analysis and evaluation of the solution results is referred to as post processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include:

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behavior.
- Produce color-coded temperature plots.

3.5 Finite Element Analysis Of Different Materials

In finite element analysis of different materials have done force analysis by putting the values of properties like young's modulus, Poisson's ratio, density, in ANSYS. In force analysis have calculated total deformations, maximum shear stress, equivalent stress, shear stress, maximum principle stress. The details analyses of different five materials with their values of properties are described below.

3.5.1 Forged steel

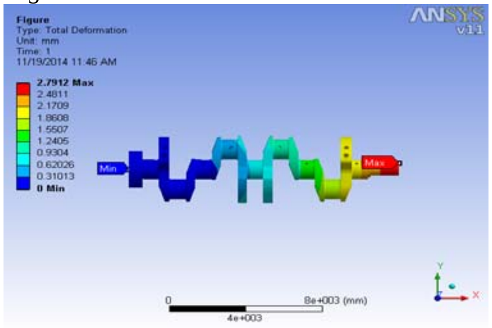

Fig. 3 Total Deformation induced in Forged Steel

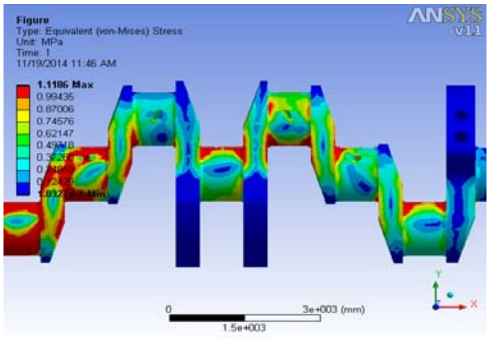

Fig. 4 Equivalent Stress induced in Forged Steel

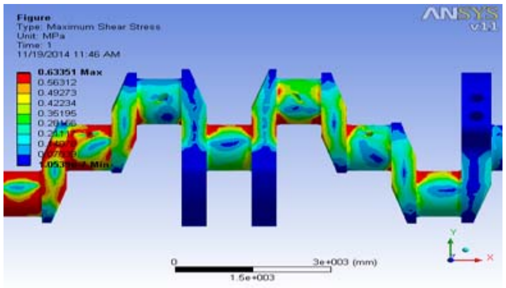

Fig. 5 Maximum shear stress induced in Forged Steel

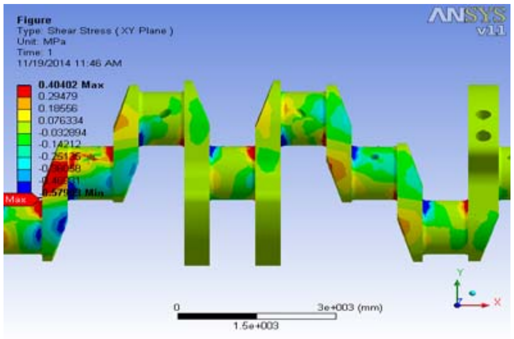

Fig. 6 Shear Stress induced in Forged Steel

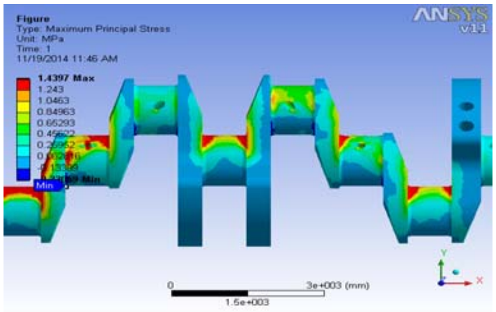

Fig. 7 Maximum Principle Stress Induced in Forged Steel

3.5.2 Steel Alloy (CrMoV13)

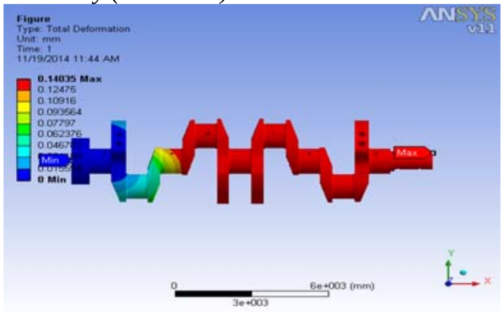

Fig. 8 Total Deformation induced in Steel Alloy (CrMoV13)

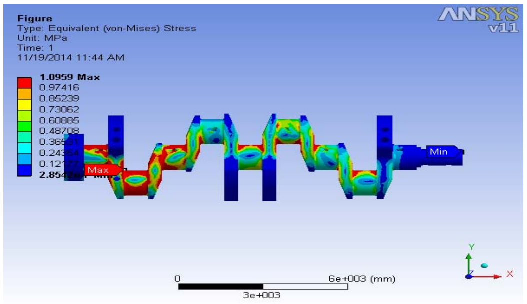

Fig. 9 Equivalent Stress induced in Steel Alloy (CrMoV13)

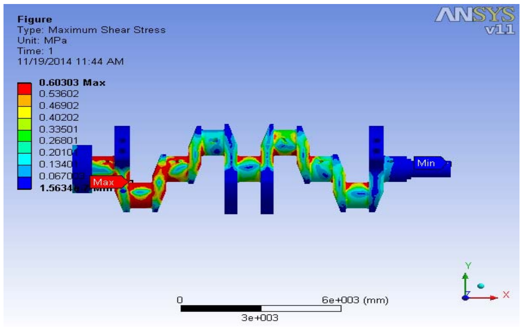

Fig. 10 Maximum shear stress induced in Steel Alloy (CrMoV13)

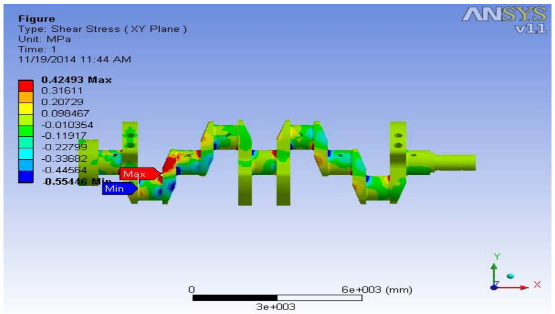

Fig. 11 Shear Stress induced in Steel Alloy (CrMoV13)

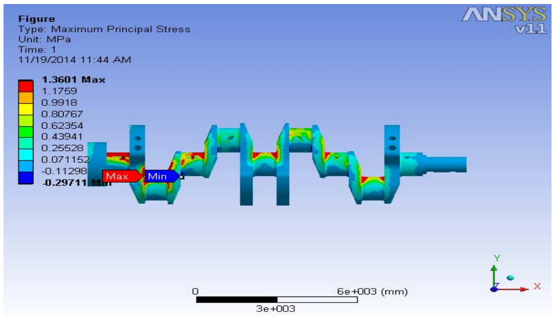

Fig. 12 Maximum Principle Stress Induced in Steel Alloy (CrMoV13)

IV. RESULTS

The results obtained from the above Finite Element Analysis of different materials are collected and make it in tabulated form. These Result tables are shown in below.The results value of total deformation, maximum shear atress,shear stress, equivalent stress, maximum shear stress are compared with each other and find less value to determine suitable material for crankshaft. .

TABLE 2

Results for Forged Steel and Steel Alloy CrMoV13

	Forged Steel	**Steel Alloy CrMoV13**
Young's Modulus (E) MPa	221000	467800
Poisson's Ratio (ν)	0.3	0.21
Density (ρ) kg/m^3	7833	7313
Total Deformation (mm)	2.7912	1.3595
Equivalent Stress (MPa)	10.884	10.663
Maximum Shear Stress (MPa)	6.164	5.8674
Shear Stress (MPa)	3.931	4.1344
Max Principle Stress (MPa)	14.009	13.233

V. CONCLUSION

In this paper the crankshaft model was created by PRO-E wildfire 4.0 for modeling the crank shaft. Then, the created model was import into ANSYS software for static structural analysis.

The analysis of the crank shaft will be done using two different materials. These materials are forged steel, steel alloy CrMoV13. The comparison of analysis results of two materials will show the effect of stresses on two different materials and this will help to select suitable material.

So the Steel Alloy CrMoV13 is suitable material for manufacturing the crankshaft. FEA Results equivalent (Von-Misses) stress, shear stress nearly by matches with the theoretical calculation for validation of model. So we can say that FEA is a good tool to reduce time consuming theoretical Work and to reduce costly experimental work.

ACKNOWLEDGMENT

It's give me a great pleasure to express my deep gratitude to guide **Prof. Dheeraj Verma** for his valuable support and help for time to time during work. I am also very thankful to our principal sir, **Dr. A. W. Kharche** who has provided us such facilities and training to ensure us a bright future finally yet importantly.

REFERENCES

[1] C.M.Balamurugan, R.Krishnaraj, Dr.M.Sakthivel, *"Computer Aided Modeling and Optimization of Crankshaft"*, 'International Journal of Scientific & Engineering Research', ISSN 2229-5518, Issue 8, vol.2, (2011).pp.1-6.

[2] Farzin H. Montazersadgh, *"Optimization of a Forged Steel Crankshaft Subject to Dynamic Loading"*, 'SAE International', Paper No 2008-01-0432, (2008).

[3] Kartik D. Kothari, Ramdevsinh L. Jhala, *"Optimization For Cost And Weight of A Forged Steel Crankshaft Using Geometry Variables"*,ASME,(2009),pp 13.1-13.7.

[4] Rinkle Garg, Sunil Baghla, *"Finite Element Analysis And Optimization Of Crankshaft Design"*, 'International Journal Of Engg.and Management Research',ISSN:2250-0758, vol-2, Issue-6,(2012),pp:26-31.

[5] Bhumesh J. Bagde, Laukik P. Raut, *"Finite Element Analysis Of Single Cylinder Engine crankshaft"*, 'International Journal of Advances in Engineering & Technology',ISSN: 2231-1963,vol. 6, Issue 2,(2013), pp. 981-986.

[6] Rajasekhar Jagathani, D.V.Nooka Raju, I.Ajit Kumar, *"Design and Analysis of a Crankshaft"*, 'International Journal Of Innovative Research in Engineering Science and Technology',ISSN: 2320-981X,vol.1, Issue 6,(2013), pp.8-11.

[7] F.S. Silva,*" Analysis of a vehicle crankshaft failure"*, 'International journal of engineering research and applications', science direct', Engineering Failure Analysis 10 (2003) , pp.605-616.

[8] H. Bayrakc¸eken, S. Tasgetiren, F. Aksoy, *"Failures of single cylinder diesel engines crank shafts"*, 'science direct'', Engineering Failure Analysis 14 (2007), pp. 725-730.