International Journal of Engineering and Management Research, Vol.-2, Issue-6, December 2012
ISSN No.: 2250-0758
Pages: 26-31
www.ijemr.net

Finite Element Analysis and Optimization of Crankshaft Design

[1]Rinkle Garg, [2] Sunil Baghla
[1]Research Scholar [2]Assistant Professor,
Department of Mechanical Engineering, Yadavindra college of Engineering, Talwandi sabo (Bathinda)

ABSTRACT

In this study 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 ProE 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.

Keywords-Finite Element Analysis, ProE, ANSYS, Crankshaft

I. INTRODUCTION

It is a large component with a complex geometry in the engine, which converts the reciprocating displacement of the piston to a rotary motion with a four link mechanism. Crankshaft consists of the shaft parts which revolve in the main bearings, the crankpins to which the big ends of the connected rod are connected, the crank arms or webs (also called cheeks) which connect the crankpins and the shaft parts. The crankshaft main journals rotate in a set of supporting bearings (main bearings) causing the offset road journals to rotate in circular path around the main journal centers, the diameter of that path is the engine "stroke'. [1]

Crankshaft must be strong enough to take the downward force of the power stroke without excessive bending. So the reliability and life of the internal combustion engine depend on the strength of the crankshaft largely. And as the engine runs, the power impulses hit the crankshaft in one place and then another. The torsional vibration appears when a power impulse hits a crankpin toward the front of the engine and the power stroke ends. If not controlled, it can break the crankshaft [2].

II. FORCES IMPOSED ON A CRANKSHAFT

The crankpin is like a built in beam with a distributed load along its length that varies with crank position. Each web like a cantilever beam subjected to bending & twisting. Journals would be principally subjected to twisting.

1. Bending causes tensile and compressive stresses.
2. Twisting causes shear stress.
3. Due to shrinkage of the web onto the journals, compressive stresses are set up in journals & tensile hoop stresses in the webs. [1]

III. FORCE CALCULATION

The quadratic parabola load distribution equation along the crankpin axis is assumed by:

$$Qx = ax^2 + bx + c \qquad (1)$$

When x=±L, Qx=0; when x=0, Qx=Qmax, the above formula can be written as:

$$0 = aL^2 + bL^2 + c \qquad (2)$$

$$0 = aL^2 - bL^2 + c \qquad (3)$$

$$Qmax = c \qquad (4)$$

The a, b, c can be obtained by solving the (2), (3) and (4). The results are a=-Qmax/L2, b=0, c=Qmax. Accordingly, the load distribution along the crankpin axis can be obtained as follows:

$$Qx = Qmax(1-x^2/L^2) \qquad (5)$$

where x = -L.

The cosine load distribution equation along the radial direction within 120°is supposed by:

$$Q(x, \theta) = Qx * \cos k\theta \qquad (6)$$

where θ=-π/3-π/3. When θ=π/3, Q(x, θ)=0, the k can be calculated by (7):

$$\cos \pi/3k = 0 \Rightarrow k = \pi/2/ \pi /3 = 3/2 \qquad (7)$$

Accordingly, the load distribution along the crankpin radial direction within 120°can be written as follows:

$$Q(x, \theta) = Qx * \cos 3\,\theta /2$$
$$= Qx(1-x^2/L^2)*\cos 3/2\,\theta \qquad (8)$$

The total load applying on the crankpin neck surface can be calculated by the follow equation:

$$Fc = \int_{-L}^{L}\int_{-\pi/3}^{\pi/3} Q(x, \theta)*ds*dx$$
$$= \int_{-L}^{L}\int_{-\pi/3}^{\pi/3} Qmax(1-x^2/L^2)*\cos 3/2\ \theta* Rd\theta*dx$$
$$=16/9\ RL*Qmax \quad (9)$$

where ds=Rdθ, R is the crankpin radius. Qmax can be expressed in terms of (9) as the following expression,

$$Qmax = 9/16*Fc/RL \quad (10)$$

Rewriting (6) as:

$$Q(x, \theta) = 9Fc/16LR * (1-x^2/L^2)*\cos 3/2\ \theta \quad (11)$$

where Fc is the total load acting on the crankpin neck surface, x is crankpin load bearing length, x=-L-L [10,11]. The largest load was used in this paper when the crankpin working at the largest toque. Base on (11), equation loaded method was used to exert the load on the crankpin surface [2].

IV. LITERATURE REVIEW

Solanki et al. [1] presented literature review on crankshaft design and optimization. The materials, manufacturing process, failure analysis, design consideration etc were reviewed. The design of the crankshaft considers the dynamic loading and the optimization can lead to a shaft diameter satisfying the requirements of the automobile specifications with cost and size effectiveness. They concluded that crack grows faster on the free surface while the central part of the crack front becomes straighter. Fatigue is the dominant mechanism of failure of the crankshaft. Residual imbalances along the length of the crankshafts are crucial to performance.

Meng et al. [2] discussed the stress analysis and modal analysis of a 4 cylinder crankshaft. FEM software ANSYS was used to analyze the vibration modal and distortion and stress status of crank throw. The relationship between frequency and the vibration modal was explained by the modal analysis of crankshaft. This provides a valuable theoretical foundation for the optimization and improvement of engine design. Maximum deformation appears at the center of the crankpin neck surface. The maximum stress appears at the fillet between the crankshaft journal and crank cheeks, and near the central point journal. The crankshaft deformation was mainly bending deformation was mainly bending deformation under the lower frequency. Maximum deformation was located at the link between main bearing journal and crankpin and crank cheeks. So, the area prone to appear the bending fatigue crack.

Montazersadgh and Fatemi [5] choose forged steel and a cast iron crankshaft of a single cylinder four stroke engine. Both crankshafts were digitized using a CMM machine. Load analysis was performed and verification of results by ADAMS modeling of the engine. At the next step, geometry and manufacturing cost optimization was performed. Considering torsional load in the overall dynamic loading conditions has no effect on von mises stress at the critically stressed location. Experimental stress and FEA results showed close agreement, within 7% difference. Critical locations on the crankshaft are all located on the fillet areas because of high stress gradients in these locations. Geometry optimization results in 18% weight reduction of the forged steel. Fillet rolling induces compressive residual stress in the fillet areas, which results in 165% increase in fatigue strength of the crankshaft.

Gongzhi et al. [6] carried out dynamic strength analysis of crankshaft for marine diesel engine. The finite element models of crankshaft, bearing, piston and connecting rod was created in ANSYS and simplified by sub structure technique. The result files of reduced models were introduced into EXCITE software to create multi-body dynamics calculation of crankshaft in one working cycle. Compared with single crankshaft strength analysis, non linear multi-body dynamics method is closer to the actual boundary conditions of the crankshaft load. Combining AVL-EXCITE multi-body dynamics and ANSYS finite element method analyzed the dynamic characteristics of marine crankshaft. This shows that under normal operating conditions, the maximum stress of crankshaft occurs on journal fillet.

Yingkui and Zhibo [7] established three dimensional model of a diesel engine crankshaft by using Pro E software. Using ANSYS analysis tool, the finite element analysis for the crankshaft was conducted under extreme operation conditions and stress distribution of the crankshaft was presented. The crank stress change model and the crank stress biggest hazard point were found by using finite element analysis, and the improvement method for the crankshaft structure design was given. This shows that the high stress region mainly concentrates in the Knuckles of the crank arm & the main journal, and the crank arm and the connecting rod journal, which is the area most easily broken.

Balamurugan et al. [8] done computer aided modeling and optimization analysis of crankshaft to evaluate and compare the fatigue performance. They compared two crankshafts, cast iron and forged steel, from a single cylinder four stroke engines. Finite element analysis was performed to obtain the variation of stress magnitude at critical locations. The dynamic analysis was done analytically and was verified by simulation in ANSYS. The optimization process included geometry changes compatible with the current engine, fillet rolling and results in increased fatigue strength and reduced cost of the crankshaft, without changing connecting rod and engine block. Analysis results in testing the crankshaft under static load containing stresses and deformation. Forged iron crankshaft is able to withstand the static load, it is concluded that there is no objection from strength point of view also, in the process of replacing the cast iron crankshaft by forged steel crankshaft.

Jian et al. [9] analyzed three dimensional model of 380 diesel engine crankshaft. They used ProE and ANSYS as FEA tools. First of all, the 380 diesel engine entity crankshaft model was created by Pro E software. Next, the model was imported to ANSYS software. Material properties, constraints boundary conditions and mechanical boundary conditions of the 380 diesel engine crankshaft were determined. Finally, the strain and the stress figures of the 380 diesel crankshaft were calculated combined with maximum stress point and dangerous area. This article checked the crankshaft's static strength and fatigue evaluations. That provided theoretical foundation for the optimization and improvement of engine design. The maximum deformation occurs in the end of the second cylinder balance weight.

Bin et al. [10] investigated the vibration model of 480 diesel crankshaft and the stress analysis of crankpin. Three dimensional models of 480 diesel engine crankshaft and crankpin were created through Pro E software. Finite element analysis software ANSYS was used to analyze the vibration model and the distortion and the stress status of the crankpin. This explains the relationship between the frequency and the vibration model. Stress analysis of crankpin provides the maximum deformation and maximum stress point. The crankshaft deformation was mainly bending deformation under lower frequency. The maximum deformation was located at the link between main bearing journal and crankpin and crank cheeks. So the bending crack was prone to appear at this area. The maximum deformation appears the bottom of crank cheek. The maximum stress appears at the transition radius of crank cheek and connecting rod journal.

Ling et al. [11] conducted fatigue life prediction modeling and residue life assessment based on statistics of historical working state (SHWS) at crankshaft using fatigue damage accumulation (FDA) theory. Dynamic response of typical operation performance is analyzed with software ANSYS of finite element analysis, and high stress zone was found. Than via rain flow cycle counting for stress time obtained, together with SHOP, fatigue load spectra of key parts are compiled. Finally, FDA model is built up with nominal stress method and residue life based on SHWS is predicted for crankshafts of diesel engine. It was found that main shaft journal of crankshaft near the power output side and connected rod journal have relatively high stress, and FDA of the same time is relatively larger.

Zhou et al. [12] performed dynamic simulation analysis of the diesel engine crankshaft by means of three dimensional FEA. By calculating each neck's loads, simulating the displacement boundary and mechanical conditions of the overall crankshaft, the stress and deformation is studied, simultaneously; the fatigue strength and the frequency bands of crankshaft under alternating loads were analyzed. The stress concentration mainly occurred in the fillet of spindle neck and the stress of the crankpin fillet is also relatively large. The crankshaft free modal analysis can obtain the 1 to 6 order natural frequencies. From the natural frequencies values, it is known that the chance of crankshaft resonant is unlike.

Xiaoping et al. [13] sets the automatic solving model which integrates the parameters geometric module, finite element analysis module, numerical calculation module. This model was based on the multi-disciplinary collaborative design optimization platform, combining the finite element analysis, variable structure parameters of the design and the design of experiment. Then the sample matrix is setup by latin hypercube sampling method. The sensitivity analysis, the main effect analysis and the interaction analysis of the key structural parameters on crankshaft fatigue strength were accomplished. Results show that the crankshaft fillet radius has the greatest influence on the crankshaft strength and the crank radius has less influence on the crankshaft strength.

Guangming and Zhengfeng [14] performed study on torsional stiffness of engine crankshaft. Modified Ker Wilson formula and Carter formula were employed to calculate torsional stiffness of engine crankthrow in the case of different thickness and width of both sides of crankthrow. Furthermore, the finite element modals of crank and free part of crankshaft linked to torsional dynamic models were developed. Then the intrinsic torsional vibration frequency of crank and free part of crankshaft are carried out by finite element method. According to mechanics of materials and empirical formula, a theoretical calculation formula of crank torsional stiffness is proposed in the conditions of the thickness and width on both sides of crankthrow are different. By calculation and comparative analysis of an real crankshaft example to verify the finite element analysis method and finite element model is feasible.

V. LOADING AND BOUNDARY CONDITIONS

Crankshaft is a constraint with a ball bearing from one side and with a journal on the other side. The ball bearing is press fit to the crankshaft and does not allow the crankshaft to have any motion other than rotation about its main axis. Since only 180 degrees of the bearing surfaces facing the load direction constraint the motion of the crankshaft, this constraint is defined as a fixed semicircular surface as wide as ball bearing width. The other side of the crankshaft is journal bearing. Therefore this side was modeled as a semicircular edge facing the load at the bottom of the fillet radius fixed in a plane perpendicular to the central axis and free to move along central axis direction.

The distribution of load over the connecting rod bearing is uniform pressure on 120 degree of contact area. Since the crankshaft is in interaction with the connecting rod, the same loading distribution will be transmitted to the crankshaft. In this study a pressure of 3.5 MPa is applied at the crankpin at top dead center position of piston.

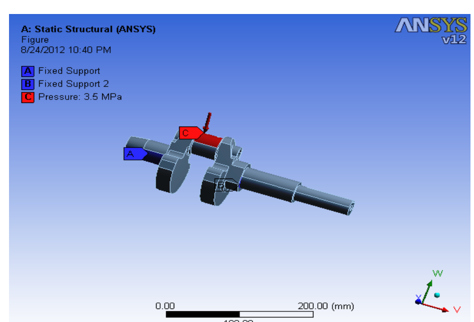

Figure 1: Loading and boundary conditions

VI. GEOMETRY OPTIMIZATION

In order to achieve the objectives various changes in the initial design of the crankshaft were done and they were analyzed among them two cases showed the most effective results.

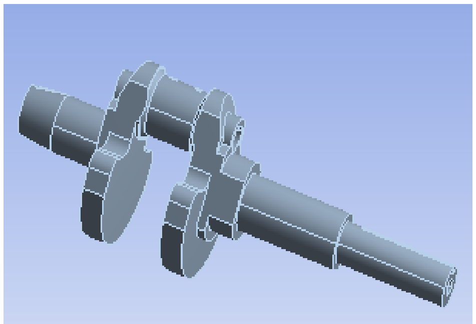

Figure 2: Modified design1

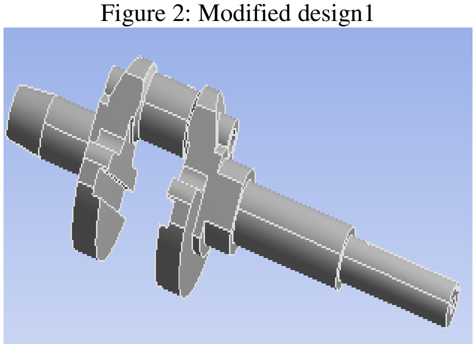

Figure 3: Modified design2

In design 1 web thickness is decreased and the hole depth is increased. After doing modification volume of the modified design 1 is 4.4908 e+005 mm^3 and the weight of the modified design 1 is 3.2334 kg. In design2 hole depth at journal bearing is increased and material is removed on the web. Volume of the new design after modification is 4.5973 e+005 mm^3 and the weight of the modified design 2 is 3.3101 kg.

After designing the load and boundary conditions were assigned to the new model. After applying the loading and boundary conditions, equivalent stress, equivalent strain, shear stress and total deformation diagrams are obtained. And then the various stress, strain and deformation of the different designs are then compared, analyzed and the best results give the final optimized design.

VII. COMAPRISION OF RESULTS

After applying loading and boundary conditions results from ansys were obtained and compiled in table 1.

Table 1: Results obtained from ANSYS

	Initial design	**Modified design 1**	**Modified design 2**
Total Deformation (mm)			
Min	0.	0.	0.
Max	3.8075e-003	3.192e-003	3.3603e-003
Equivalent Stress(MPa)			
Min	3.4289e-009	5.2014e-008	1.5781e-008
Max	7.7865	5.7567	6.4028
Shear Stress(MPa)			
Min	-4.2463	-3.0443	-3.3576
Max	2.8397	2.1781	2.6943
Equivalent Elastic Strain			
Min	3.1172e-014	4.7285e-013	1.4346e-013
Min	7.0787e-005	5.2334e-005	5.8207e-005
Volume (mm3)			
	5.0372e+005	4.4908e+005	4.5973e+005
Weight (kg)			
	3.6268	3.2334	3.3101

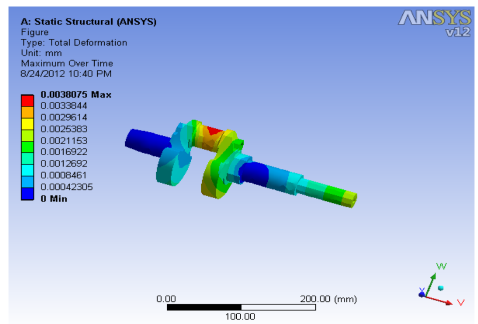

Figure 4: Total Deformation of Initial Design

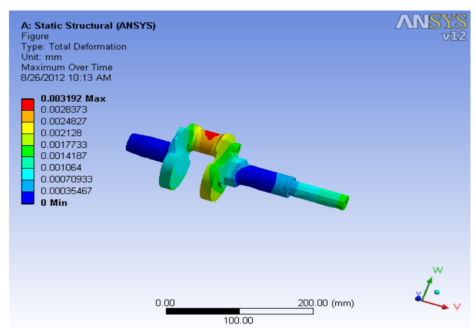

Figure 5: Total Deformation of Modified Design 1

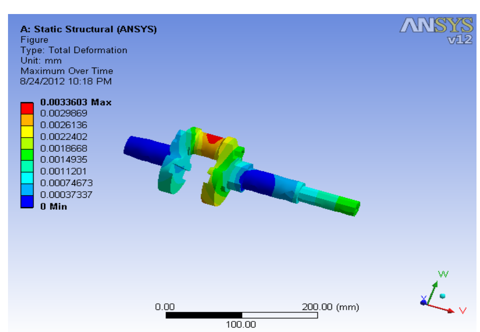

Figure 6: Total Deformation of Modified Design 2

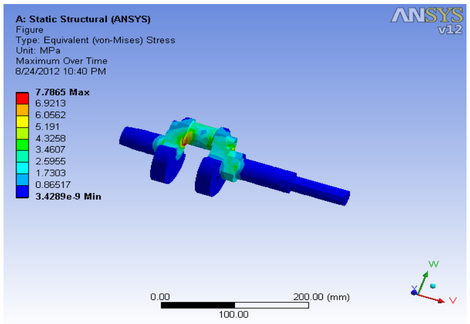

Figure 7: Equivalent Stress of Initial Design

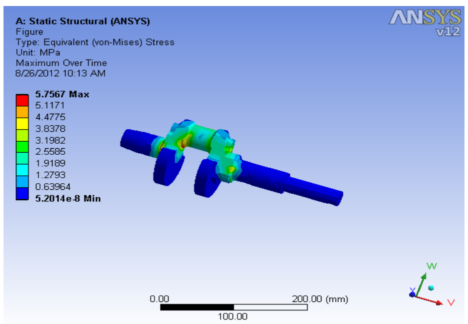

Figure 8: Equivalent Stress of Modified Design 1

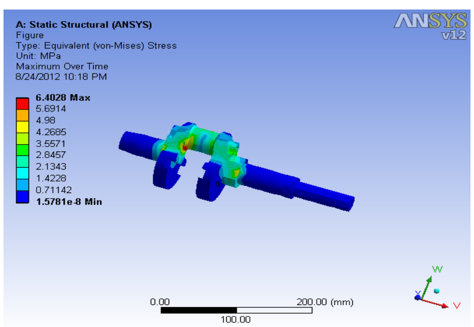

Figure 9: Equivalent Stress of Modified Design 2

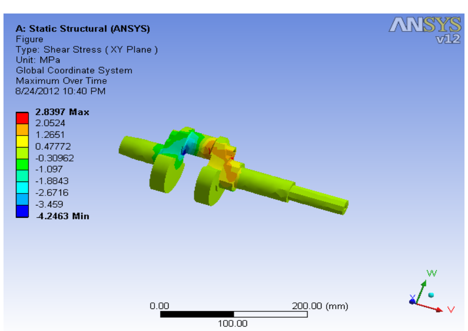

Figure 10: Shear Stress of Initial Design

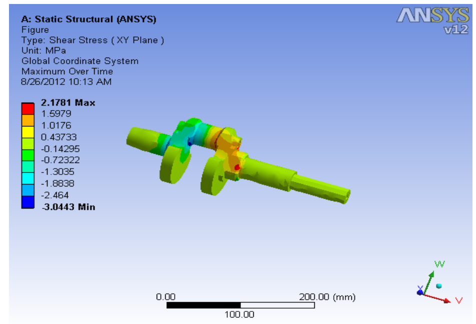

Figure 11: Shear Stress of Modified Design 1

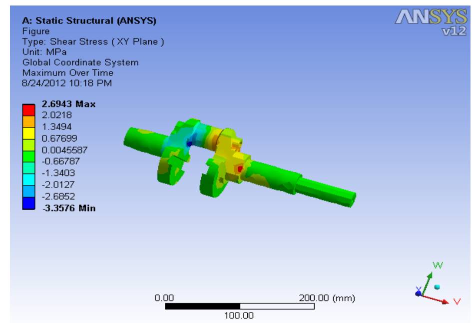

Figure 12: Shear Stress of Modified Design 2

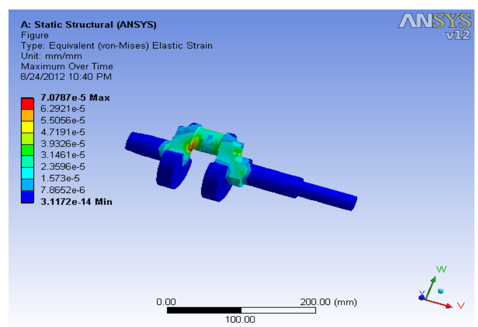

Figure 13: Equivalent Strain of Initial Design

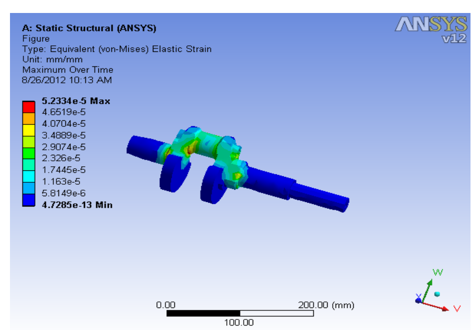

Figure 14: Equivalent Strain of Modified Design 1

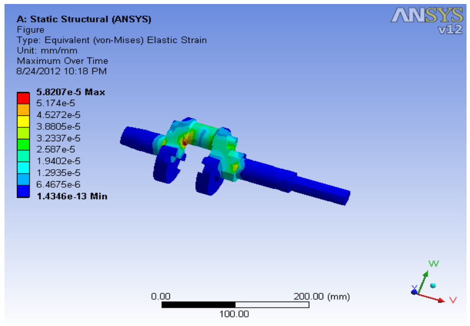

Figure 15: Equivalent Strain of Modified Design 2

VIII. CONCLUSION

Finite Element analysis of the single cylinder cast iron crankshaft has been done using FEA tool ANSYS Workbench. From the results obtained from FE analysis, many discussions have been made. The results obtained are well in agreement with the similar available existing results. The model presented here, is well safe and under permissible limit of stresses.

1. Results show the improvement in the strength of the crankshaft as the maximum limits of stresses, total deformation and strain is reduced.
2. The weight of the crankshaft is also reduced by 3934g. Thereby, reduces the inertia force.
3. As the weight of the crankshaft is decreased this will decrease the cost of the crankshaft and increase the engine performance.

REFERENCES

[1]. A. Solanki, K. Tamboli, M. J. Zinjuwadia, (2011), "Crankshaft Design and Optimization- A Review" National Conference on Recent Trends in Engineering & Technology.

[2]. J. Meng, Y. Liu, R. Liu, (2011), "Finite Element analysis of 4-Cylinder Diesel Crankshaft" I.J. Image, Graphics and Signal Processing, vol 5, pp 22-29

[3]. http://www.epi-eng.com (02-08-12 at 10:44 am)

[4]. www.google.com (02-08-12 at 01:14 am)

[5]. F. H. Montazersadgh, A. Fatemi, (2007), "Project Report on Stress Analysis and Optimization of Crankshafts Subject to Dynamic Loading" The University of Toledo

[6]. Y. Gongzhi, Y. Hongliang, D. Shulin, (2011), "Crankshaft Dynamic Strength Analysis for Marine Diesel Engine" Third International Conference on Measuring Technology and Mechatronics Automation, pp 795-799

[7]. G. Yingkui, Z. Zhibo, (2011), "Strength Analysis of Diesel Engine Crankshaft Based on PRO/E and ANSYS" Third International Conference on Measuring Technology and Mechatronics Automation, pp 362-364

[8]. C. M. Balamurugan, R. Krishnaraj, M. Sakthivel, K. Kanthavel, D. Marudachalam, R. Palani, (2011), "Computer Aided Modeling and Optimization of Crankshaft" International Journal of Scientific & Engineering Research, vol 2 , issue 8.

[9]. M. Jian, L. Yong-qi, L. Rui-xiang, Z. Bin, (2010), "Intension Analysis of 3-D Finite Element Analysis on 380 diesel crankshaft" International Conference on Computational and Information Sciences, pp 1269-1272

[10]. M. Jian, L. Yong-qi, L. Rui-xiang, Z. Bin, (2010), "3-D Finite Element Analysis on 480 diesel crankshaft " Institute of Electrical and Electronic Engineers

[11]. S. Ai-ling, Y. Wen-hua, D. Zhou, L. Yu-mei, (2010), "Assessment of Residue Fatigue Life of Crankshaft Based on Theory of Fatigue Damage" International Conference on Optoelectronics and Image Processing [12]. X. Zhou, G. Cai, Z. Zhang, Z. Cheng, (2010), "The Whole Crankshaft Model for Dynamic Simulation Analysis of the Diesel Engine" Institute of Electrical and Electronic Engineers.

[13]. C. Xiaoping, Y. Xiaoli, J. Binwei, (2010), "Study of crankshaft Strength based on iSIGHT Platform and DOE Methods" International Conference on Measuring Technology and Mechatronics Automation, pp 548-551

[14]. Z. Guangming, J. Zhengfeng, (2009), "Study on Torsional Stiffness of Engine Crankshaft" International Forum on Computer Science Technology and Applications, pp 431-435